

SOUS LE HAUT PATRONAGE DE SA MAJESTÉ LE ROI MOHAMMED VI



الجمعية المغربية للعلوم الميكانيكية

Société Marocaine des Sciences Mécaniques



FSAC

UNIVERSITÉ HASSAN II DE CASABLANCA
FACULTÉ DES SCIENCES AÏN CHOCK

La Société Marocaine des Sciences Mécaniques

et

**La Faculté des Sciences Aïn Chock de l'Université Hassan II
de Casablanca**

Organisent

15^{ème} CONGRÈS DE MÉCANIQUE
15th CONGRESS OF MECHANICS

Tome II : Mécanique des Fluides

24 - 27 Mai 2022, FSAC-Casablanca-Maroc

May 24 - 27 2022, FSAC-Casablanca-Morocco

<https://smsm.org.ma/cmm/>

Sous le Haut Patronage de Sa Majesté le Roi Mohammed VI



الجمعية المغربية للعلوم الميكانيكية

Société Marocaine des Sciences Mécaniques



UNIVERSITÉ HASSAN II DE CASABLANCA
FACULTÉ DES SCIENCES AÏN CHOCK

ème
15 CONGRÈS
DE MÉCANIQUE

Mai 24-27, 2022, Casablanca

Tome II : Mécanique des Fluides

Comité d'Honneur

Pr. Abdellatif MIRAOUI	Ministre de l'Enseignement Supérieur, de la Recherche scientifique et de l'Innovation
Pr. Omar FASSI-FEHRI	Secrétaire Perpétuel de l'Académie Hassan II des Sciences et Techniques, Président Fondateur de la Société Marocaine des Sciences Mécaniques
Pr. Mohammed TALBI	Président de l'Université Hassan II de Casablanca
Pr. Omar SADDIQI	Doyen de la Faculté des Sciences Aïn Chock - Casablanca
Pr. Jaâfar KHALID NACIRI	Ex Président de la Société Marocaine des Sciences Mécaniques
Pr. Nouredine DAMIL	Ex Président de la Société Marocaine des Sciences Mécaniques
Pr. Abdellatif MASLOUHI	Ex Président de la Société Marocaine des Sciences Mécaniques

Comité d'Organisation

Présidents :

Pr. ANISS S., FSAC, Université Hassan II de Casablanca

Pr. KHATYR R., FSAC, Université Hassan II de Casablanca

Pr. ABDOUN F.	ENSAM, Université Mohammed V, Rabat
Pr. AIT ALI M.E.	EMI, Université Mohammed V Rabat
Pr. AZIM A.	ENSA, Université Chaib Doukkali, El Jadida
Pr. AZRAR L.	ENSAM, Université Mohammed V, Rabat
Pr. BOUCHGL J.	ISPM, Agadir
Pr. EL HAD K.	ISEM, Casablanca
Pr. LAMINE M.	FS Ain Chock, Université Hassan II de Casablanca
Pr. MORDANE S.	FS Ben M'Sik, Université Hassan II de Casablanca
Pr. OUHIMMOU S.	FS Ain Chock, Université Hassan II de Casablanca
Pr. OUSSOUADDI O.	FS, Université Moulay Ismail, Meknès
Pr. RACHID H.	FS Ain Chock, Université Hassan II de Casablanca
Pr. RAHMOUNE M.	ESTM, Université Moulay Ismail, Meknès
Pr. SEHAQUI R.	FS Ain Chock, Université Hassan II de Casablanca
Pr. TAIBI M.	FS Ain Chock, Université Hassan II de Casablanca
Pr. ZENASNI M.	ENSA, Université Mohamed Premier, Oujda

Comité Scientifique

Présidents :

Pr. MORDANE S., FS Ben M'Sik, Université Hassan II de Casablanca

Pr. RAHMOUNE M., EST, Université Moulay Ismail, Meknès

Pr. ANISS S.	FS Ain Chock, Université Hassan II de Casablanca, Maroc
Pr. ABBADI M.	ENSA, Université Mohammed Premier, Oujda, Maroc
Pr. ABDELBAKI A.	FS, Université Cadi Ayyad, Marrakech, Maroc
Pr. ABED MERAIM F.	ENSAM, Metz, France
Pr. AFSHARI A.	Université d'Aalborg, Danemark
Pr. AGOUZOUL M.	EMI, Université Mohammed V, Rabat, Maroc
Pr. ASSARAR M.	IUT, Université Reims Champagne-Ardenne, Troyes, France
Pr. ATALLA N.	Université de Sheerbrooke, Canada
Pr. AZAIEZ M.	INP, Bordeaux, France
Pr. AZARI Z	ENIM, Université Lorraine, Metz, France
Pr. AZIM A.	ENSA, Université Chaïb Doukkali, El Jadida
Pr. AZRAR L.	ENSAM, Université Mohammed V, Rabat
Pr. BHADAURIA B. S.	Université Babasaheb Bhimrao Ambedkar, Inde
Pr. BOTTON V.	Insa, Lyon, France
Pr. BOU-ALI M.	MGEP, Université Mondragon, Arrasate, Espagne
Pr. BRAIKAT B.	FS Ben M'Sik , Université Hassan II de Casablanca, Maroc
Pr. CHAARI F.	École nationale d'ingénieurs de Sfax, Tunisie
Pr. CHAMPMARTIN S.	ENSAM Angers, France
Pr. CHARIF D'OUAZZANE S.	ENSMR, Mines, Rabat, Maroc
Pr. CHARRIER-MOJTABI M.C.	IMFT, Université Paul Sabatier Toulouse III, France
Pr. CHEDDADI A.	EMI, Université Mohammed V, Rabat, Maroc
Pr. CORDIER L.	ENSMA, Université Poitiers, Poitiers, France
Pr. DALKILIC A. S.	YTU, Mechanical Engineering Faculty, Istanbul, Turkey
Pr. DAMIL N.	FS Ben M'Sik, Université Hassan II, Casablanca, Maroc
Pr. DRAOUI A.	FST, Université Abdelmalek Essaâdi, Tanger, Maroc
Pr. ECHCHELH A.	FS, Université Ibn Tofaïl, Kénitra, Maroc
Pr. EL HAD K.	ISEM, Casablanca, Maroc
Pr. EL KHLIFI M.	FST, Université Hassan II, Mohammedia, Maroc
Pr. EL KISSI N.	CNRS, Université Joseph Fourier, Grenoble, France
Pr. ELBIKRI K.	ENSAM, Université Mohammed V, Rabat , Maroc
Pr. ELGHORBEL E.	Université de Cergy-Pontoise, Cergy-Pontoise, France
Pr. ERCHIQUI F.	UQAT, Université Rouyn-Noranda, Québec, Canada
Pr. FABRE D.	IMFT, Université Paul Sabatier Toulouse III, France
Pr. FARAJI M.	FS Ain Chock, Université Hassan II de Casablanca, Maroc
Pr. FOURNIER M.	IMFT, Université Paul Sabatier Toulouse III, France
Pr. GANGHOFFER J. F	Université de Lorraine, France
Pr. GUINES D.	INSA de Rennes, Rennes, France
Pr. HADDAR M.	École nationale d'ingénieurs de Sfax, Tunisie
Pr. HARIRI S.	École des mines de Douai, Douai, France
Pr. HASNAOUI M.	FS Semlalia, Université Cadi Ayyad, Marrakech, Maroc
Pr. HATTABI M.	Université Hassan II de Casablanca, Maroc
Pr. HU H.	Université de Wuhan, China
Pr. HENRY D.	École centrale de Lyon, France
Pr. HIFDI A.	FS Ain Chok , Université Hassan II de Casablanca, Maroc
Pr. HJIAJ M.	INSA de Rennes, Rennes, France
Pr. ICHCHOU M.N.	École Centrale de Lyon, Lyon, France
Pr. JAMIL A.	EST, Université Sidi Mohamed Ben Abdellah, Fès, Maroc
Pr. KOURTA A.	Polytech, Université Orléans, Orléans, France
Pr. LAHJOMRI J.	FS Ain Chok, Université Hassan II de Casablanca, Maroc
Pr. LAKSIMI A.	Université de Technologie de Compiègne, Compiègne, France
Pr. LIMAM A.	INSA de Lyon, France
Pr. MASLOUHI Ab.	FS, Université Ibn Tofaïl, Kénitra, Maroc
Pr. MASLOUHI Ah.	FGM, Université Sherbrooke, Québec, Canada

Pr. MOJTABI K.	IMFT, Université Paul Sabatier Toulouse III, France
Pr. MUTABAZI I.	Université le Havre Normandie, France
Pr. NACIRI J.K.	FS Ain Chok, Université Hassan II de Casablanca, Maroc
Pr. NOUARI M.	St Diès-des-Vosges, Université Lorraine, Nancy, France
Pr. NTAMACK G.E.	FS, Université Ngaounderé, Ngaounderé, Cameroun
Pr. OUAZZANI M.T.	FS Ain Chok, Université Hassan II de Casablanca, Maroc
Pr. OUAZZANI M.T.	EMI, Université Mohamed V, Rabat, Maroc
Pr. OUBARRA A.	FS Ain Chok, Université Hassan II de Casablanca, Maroc
Pr. OUSSOUADDI O.	FS, Université Moulay Ismail, Meknès, Maroc
Pr. PLOURABOUE F.	IMFT, Université Paul Sabatier Toulouse III, France
Pr. RACHIK M.	Université de Technologie de Compiègne, Compiègne, France
Pr. RAPHAEL E.	ESPCI, Paris, France
Pr. SAADANI R.	EST , Université Moulay Ismail, Meknès, Maroc
Pr. SARSRI D.	ENSA , Université Abdelmalek Essaâdi, Tanger, Maroc
Pr. SEHAQUI R.	FS Ain Chok, Université Hassan II de Casablanca, Maroc
Pr. SIMOES F.M.F.	Université Lisbonne, Lisbonne, Portugal
Pr. SKALI LAMI S.	LEMTA -ENSEM , Université Lorraine, Nancy, France
Pr. TAHA JANAN M.	ENSAM, Université Mohammed V, Rabat , Maroc
Pr. TRI A.	ISEM, Casablanca, Maroc
Pr. YAO K.	Luxembourg Institute of Science and Technology, Luxembourg
Pr. ZAGZOULE M.	IMFT, Université Paul Sabatier, Toulouse, France
Pr. ZAHROUNI H.	Université Lorraine, Metz, France
Pr. ZENASNI M.	ENSA, Université Mohamed Premier, Oujda
Pr. ZHANG C.	Shandong University, China
Pr. ZIDI M.	Université Paris-Est Créteil, France
Pr. ZRIKEM Z.	FS Semlalia, Université Cadi Ayyad, Marrakech, Maroc

Thème 06 :

Milieux poreux et milieux granulaires

T06013	Z. KAHOT Analyse fiabiliste de la stabilité des barrages en terre après un vidange rapide	2
T06087	M. JAOUHARI, M. ECHCHADLI, S. ANISS Stability of two viscous miscible fluid layers saturating a porous medium heated from below	5
T06094	R. ABABOU Infiltration in unsaturated porous media : phenomenology and modeling	8
T06097	TAZI, RIDA, ECHCHELH. ADIL Numerical simulation of a granular flow on a smooth inclined plane	11

Thème 07 :

Mécanique des fluides, instabilités hydrodynamiques, transition et turbulence

T07011	KHALIL ATLASSI, MOHAMED NABHANI, MOHAMED EL KHLIFI Dynamic characteristics of finite journal bearings lubricated with ferrofluids	15
T07019	M. OUZI, B. BAHRAR, H. SAMRI, M. TAMANI Modeling homogenous two-phase water hammer flow	18
T07021	A. SERRAR, M. EL KHLIFI, A. KOURTA Numerical investigation of the flow mechanisms in a fluidic oscillator and a sweeping jet actuators	21
T07025	A. AIT HRA, S. MORDANE, F. HDIDOU, Z. SAHLAOUI Modélisation de la prévision des poussières atmosphériques et étude de cas	24
T07026	MOHAMED MADI, KHALID SOUHAR, ABDESSAMADE RAFIKI, SEYED MOHAMMAD TAGHAVI Local hydrodynamic stability of the plane Poiseuille flow of an upper convected Maxwell fluid through a periodic channel	27
T07039	O. RKIBI, B. BAHRAR, K. GUERAOUI Numerical Modelling of Cavitation in elastic Pipe	30
T07057	N.ACHAK, W.RKIBI, B.BAHRAR, K. GUERAOUI Numerical modeling of non-Newtonian fluid flow	33
T07065	A. GÓMEZ, J. BERASATEGI, M.M. BOU-ALI Development of magnetoactive polymeric materials	36
T07066	A. GÓMEZ, I. ALCORTA, J. BERASATEGI, T. GOMEZ-ARENAS , M.M. BOU-ALI Experimental analysis of the stability of magnetorheological fluids	39
T07069	M. RAMMANE, S. MESMOUDI, O. ASKOUR, A. TRI, B. BRAIKAT, N. DAMIL Bifurcation analysis of incompressible fluid flow by a high order mesh-free indicator with Padé approximants	42
T07072	M. KARDOUDI, M. NABHANI, M. EL KHLIFI MHD non-Newtonian effects on lubricated contacts performances	45
T07076	M. HAKIM, S. CHOUKRI Détermination de profils aérodynamiques à moyen Reynolds pour une aérostructure à fuselage intégré	48
T07080	A. AGRISS, M. ASSOUL, A. ETTOUIL, A. MEHDARI Three-dimensional numerical investigation of a new drag reduction device applied to Ahmed body	51
T07085	M. MOUDA, M. EL KHLIFI, M. NABHANI Effect of load on minimum film thickness in non-Newtonian MEHD parabolic slider bearings	54
T07088	M. ASSOUL, A. EL JAOUAHIRY, M. ECHCHADLI, S. ANISS Effect of the wall slip on the stability of a viscous interface under horizontal periodic oscillation	57
T07092	S. SOBHI, M. NABHANI, M. EL KHLIFI Numerical model of cavitation in periodic squeeze film between two discs	60
T07096	A. SAKIM, M. NABHANI, M. EL KHLIFI Viscous shear effects on the performance of elastic porous bearings	63
T07102	A. EL OMARI, M. EL KHLIFI, L. CORDIER Interpolation of parameterized reduced-order models for fluid flow prediction	66
T07104	C. TAOUZER, M. ASSOUL, S. ANISS Rayleigh-Taylor instability under vertical periodic oscillation with two frequencies	69
T07113	J. BOUCHGL, S. ANISS, S. SKALI LAMI Interfacial instability of superposed immiscible fluids through two porous layers with different porosities under horizontal periodic oscillation	72
T07114	M. LAMINE, S. ANISS, A. HIFDI Hydrodynamic stability of shear flow of viscoelastic liquids	75
T07117	R. FADIL, M. RAMMANE, S. MESMOUDI, A. TRI, B. BRAIKAT Approche sans maillage pour la simulation des écoulements de fluides compressibles	78
T07123	M. HAYANI CHOUJAA, M. RIAHI, S. ANISS, MOHAMED OUZZANI TOUHAMI, JAAFAR KHALID NACIRI Stabilité de l'écoulement pulsé d'un fluide viscoélastique de type Oldroyd-B nonlinéaire en géométrie de Taylor-Couette	81
T07124	M. HAYANI CHOUJAA, M. RIAHI, S. ANISS, MOHAMED OUZZANI TOUHAMI, JAAFAR KHALID NACIRI Stabilité d'un écoulement oscillatoire de Taylor-Couette dans un fluide de Maxwell sur-convecté	84
T07125	D. HENRY, S. MILLET, S. DAGOIS-BOHY, V. BOTTON, AND H. BEN HADID Rayleigh-Bénard flow for a shear-thinning fluid in a 3D cavity	87
T07142	Y. BANDADI, A. SBAIBI, A. LACHHEB, R.SAADANI Contrôle des instabilités d'écoulement sur plaque plane rugueuse à l'aide d'un système de bosse-creux	90
T07155	H. RACHID, M. OUZZANI, N. LAHLOU Coupling heat transfer and peristaltic transport of a Newtonian fluid through an elliptical duct	93
T07156	N. LAHLOU, M. OUZZANI TOUHAMI, S. ANISS, H. RACHID Caractérisation du comportement rhéologique des solutions aqueuses à base d'argile grise et modélisation de leur transport en conduits péristaltiques	96
T07158	S. LAGHRABLI, M. NABHANI, M. EL KHLIFI New lubrication model based on ferrohydrodynamic equations of Jenkins model	99
T07163	A. RACHIK, M. ASSOUL, S. ANISS Instabilité de Faraday en cellule de Hele-Shaw avec deux fréquences commensurables	102
T07169	MOHAMED SAKINE, MEHDI RIAHI, MOHAMED HAYANI CHOUJAA, SAÏD ANISS, ABDELMAJID DAYA Instabilité d'une couche fluide Newtonienne horizontale en présence d'une modulation périodique de température imposée à la frontière supérieure	105

Thème 08 :

Transferts thermiques, transfert de masse, énergies renouvelables et environnement

T08004	K. OUDAOU, M. FARAJI Numerical study of latent heat discharge of a phase change material shell-and-tube thermal energy storage system	109
T08007	MUSTAPHA FARAJI Investigation by LBM of heat transfer within fins equipped channel heated by a heat source	112
T08009	M. SHAIMI, R. KHATYR and J. KHALID NACIRI Analytical and numerical solutions of the Graetz problem in ducts of arbitrary cross-section	115
T08014	Y. DAHANI, M. HASNAOUI, A. AMAHMID, S. HASNAOUI, A. MANSOUR LBM simulation of the coupling of mixed convection and surface radiation in a lid-driven cavity cooled with a non-uniform temperature	118
T08016	A. MANSOUR, A. AMAHMID, M. HASNAOUI, I. FILAHI, Y. DAHANI Assessment of the magnetic field effect on thermal convection in a Bénard square porous cavity salted from its vertical walls	121
T08020	H. FETIAN, Z. NGADI, M.L. LAHLAOUTI Étude de la co-combustion du coke de pétrole et la biomasse dans les fours de cimenterie.	124
T08024	S. JENNAH, N. BELOUAGGADIA, R.LBIBB, M.EZZINE L'impact de l'amélioration des performances des modules thermoélectriques sur son rendement	127
T08029	M. DARIF, K. ZNIBER, A. OUBARRA, J. LAHJOMRI Solution analytique et numérique de l'écoulement potentiel stationnaire autour d'un cylindre chauffé avec une densité de flux de chaleur uniforme	130
T08030	KAMAL BOUZID, LAHOUCINE BELARCHE, BTISSAM ABOURIDA, ALI SIADI Numerical simulation of natural convection flow inside 2D Cavity filled with Alumina Al ₂ O ₃ nanofluid	133
T08041	A. DAIZ, A. BAHLAOU, I. ARROUB, S. BELHOUIDE, S. OUSEMRAR, A. RAJI, M. HASNAOUI Simulation numérique de la convection mixte couplée au rayonnement dans une cavité carrée à paroi mobile par la méthode de Boltzmann	136
T08044	F. TRIBAK, F. BEN NEJMA, Z. ARAOUD, O. BENDAOU, A. ASSELMAN Impacte de l'inclinaison de l'orbite sur les transferts thermiques dans un CubeSat 1U en LEO	139
T08045	MOHAMMED NIKCHI, RABIÁA IDMOUSSA, KHALID ZNIBER, JAWAD LAHJOMRI AND ABDELAZIZ OUBARRA Analytical study of heat conduction inside orthotropic media	142
T08047	Y. TIZAKAST, M. KADDIRI, M. LAMSAADI Double-diffusive mixed convection in a shallow rectangular cavity with single lid-driven boundary filled with Newtonian fluid	145
T08048	O. IMGHOURE, S. OUAHBI, N. BELOUAGGADIA, M. EZZINE, R. LBIBB, Z. YOUNSI Effet de l'insertion des matériaux à changement de phase sur l'amélioration du confort thermique dans les bâtiments	148
T08054	F. REDOINE, N. BELOUAGGADIA, K. MANSOURI Le stockage de chaleur dans les matériaux à changement de phase application CSP	151
T08055	M. BOUFKRI, A. EL MANSOURI, M. HASNAOUI, A. AMAHMID Lattice Boltzmann analysis of mixed convection in a water based photovoltaic thermal collector	154
T08056	A. BOUCHAALA, O. MERROUN, A. MIKDAM, A. SAKIM Modelling and simulation of a direct evaporative cooling system with purge zone	157
T08063	Z. AITCHFI, A. TOUZIT, M. IGOUZAL Assessment of hydrokinetic energy potential in the Sebou river estuary	160
T08064	S. ELALAOUI, A. OUBARRA, J. LAHJOMRI, K. ZNIBER Amélioration du transfert thermique par utilisation d'une conduite à parois ondulées	163
T08067	A. ERRARTE, A. MIALDUN, V. SHEVTSOVA, M. M. BOU-ALI Analysis of the thermodiffusion phenomenon by Optical Digital Interferometry	166
T08068	A. SANJUAN, A. ERRARTE, M. MOUNIR BOU-ALI, C. SANTAMARIA Validity limits of the FJO theory-thermogravitational microcolumn : Numerical analysis and experimental validation	169
T08078	K. SIOUD, A. ABDENNADHER, A. BERGEON, M.-C. CHARRIER-MOJTABI, A. MOJTABI Effet Soret et convection mixte : séparation des espèces d'un fluide binaire de façon continue.	172
T08081	Y. TIZAKAST, M. KADDIRI, M. LAMSAADI Double-diffusive mixed convection in a shallow rectangular cavity with double lid-driven boundaries filled with Newtonian fluid : opposing flow	175
T08083	O. IKEN, S.D. FERTAH, R. AGOUNOUN, I. KADIRI, M. RAHMOUNE, K. SBAL, R. SAADANI Thermal and energy performance investigation of a smart double skin facade through CFD simulations	178
T08084	ZAKARIA EL HADY, HAMZA HAMID, JAWAD LAHJOMRI, ABDELAZIZ OUBARRA Numerical study of the thermal behavior within two anisotropic media in cylindrical geometry	181
T08089	B. ELHADOU, M. KADDIRI, M. LAMSAADI, H. EL HARFI Effect of the Lewis number on natural double diffusive convection in shallow rectangular cavities filled with nanofluid	184
T08093	A. CHARRAOU, S. OUBENMOH, R. SAADANI, A. MOURID, M. RAHMOUNE, M. EL ALAMI Effet de la variation des matériaux de construction sur le comportement thermique du système de chauffage par le sol.	187
T08095	M. AIT FARIA, H. HAMZA, K. ZNIBER, J. LAHJOMRI, A. OUBARRA Numerical study of a wall protected by water film from an incident radiative heat flux	190
T08098	R.IDMOUSSA, H. HAMZA, J. LAHJOMRI, A.OUBARRA Numerical solution of heat transfer inside an anisotropic plate medium submitted to a lateral flux	194
T08099	M. EL HOU, L. MOUAKKIR, S. MORDANE Investigating wave energy potential in Casablanca-Mohammedia coast using SWAN model	197
T08111	A. CHAREF, M. FEDDAOUI, M. NAJIM, M. RAHMOUNE, R. SAADANI, M. HISSOUF Evaporation of a saltwater film inside a vertical tube	200
T08118	M. TOUIL, O. RAHMOUN, A. LACHHEB, M. DLIMI, A. CHAREF, R. SAADANI Évaluation expérimentale de l'effet du taux d'eau de gâchage sur le comportement thermique du plâtre	203
T08119	V. BOTTON, S. MIRALLES, N. EL GHANI, S. MARCELIN, B. TER-OVANEISSIAN, H. BEN HADID, D. HENRY Amélioration du transfert de masse à une interface liquide/solide à l'aide d'écoulements générés par des ultrasons	206
T08120	M. ER-RAKI, M. HASNAOUI, S. HASNAOUI, F.Z. AMATOUL Effet des paramètres externes sur les performances électriques d'un générateur photovoltaïque	209
T08129	Z. CHARQUI, M. BOUKENDIL, L. EL MOUTAOUAKIL, R. HIDKI, Z. ZRIKEM, A. ABDELBAKI Modélisation et simulation des transferts thermiques à travers un mur trombe en régime stationnaire	212
T08130	Z. CHARQUI, M. BOUKENDIL, L. EL MOUTAOUAKIL, R. HIDKI, Z. ZRIKEM, A. ABDELBAKI Simulation numérique du comportement thermique d'un mur trombe en régime variable	215
T08131	R. HIDKI, L. EL MOUTAOUAKIL, M. BOUKENDIL, Z. CHARQUI, Z. ZRIKEM, A. ABDELBAKI Convection naturelle dans une cavité carrée contenant deux blocs générateurs de chaleur	218
T08132	R. HIDKI, L. EL MOUTAOUAKIL, M. BOUKENDIL, Z. CHARQUI, Z. ZRIKEM, A. ABDELBAKI Mixed convection in a ventilated square cavity with a circular heat-generating cylinder	221
T08133	A. LACHHEB, M. TOUIL, Y. BANDADI, R. SAADANI, A. SBAIBI, M. RAHMOUNE Effet de la finesse et du taux de remplacement des cendres sur les propriétés physiques de mortier de ciment	224

T08136	M. EI HAMMA, M.TAIBI, A. RTIBI, K. GUERAOU, H. RACHID Modélisation théorique et simulation numérique de phénomène de la convection thermosolutale, tenant compte de l'effet Soret et Dufour, dans une cavité poreuse cylindrique remplie d'un nanofluide	227
T08137	H. SGHIOURI EL IDRISSE, A. SAMAOUALI, M. TAIBI, S. OUHIMMOU Étude expérimentale des propriétés thermiques des Schistes Bitumineux de TIMAHDIT-Maroc	230
T08139	M. DLIMI, M. TOUIL, R. AGOUNOUN, R. SAADANI, M. RAHMOUNE Optimisation des performances énergétiques d'un bâtiment marocain isolé avec un matériau biosourcé	233
T08143	AMIR BOUSSOUF, NABILA LABSI, M'BAREK FEDDAOUI, YOUNB KHALED BENKAHLA, KARIMA SELLAMI Étude numérique de la condensation d'éthanol en film liquide dans une membrane hydrophile le long d'un canal vertical	236
T08159	S. LAHLOU, R. SEHAQUI Étude du comportement thermique des nanofluides (NTC/Glycérol) à partir de leurs propriétés thermo-physiques	239
T08162	H. ALLOUHI, A. ALLOUHI, A. JAMIL A novel design of solar dryer based-ETC technology for agricultural applications : Thermal performance assessment	242
T08167	M. DARIF, E. ESSAGHIR, Y. HADDOUT, A. OUBARRA, J. LAHJOMRI Convection de l'écoulement de Jeffery-Hamel à travers un canal convergent avec conduction radiale et un saut de température à la paroi	245

Thème 10 :

Interaction fluide-structure

T10033	A. JABRALI, R. KHATYR, J. KHALID NACIRI Modelling of DEXA wave energy converter under irregular wave conditions	249
T10036	ABDELKADER LAHLALI, ZAKARIA EL MASKAOUI, LAHBIB BOUSSHINE Optimisation de la forme du bulbe d'étrave d'un bateau dépollueur	252
T10046	H. AKARNI, L. EL AARABI, S. MORDANE Investigation numérique de l'interaction houle-plaque-courant par la méthode des éléments frontières	255
T10115	KISSI BENAÏSSA, GUÉMIMI CHAFIK, M. ANGEL PARRON VERRA Detection of erosion at the fluid/soil interface during the Hole Erosion Test	258
T10141	P. CATHALIFAUD, M. ZAGZOULE Compliant coaxial flow waves : distensibility modulation and fluid sheets analogy	261

Index des Auteurs-Tome II

264

Thème 06 :
Milieux poreux et Milieux granulaires

Analyse fiabiliste de la stabilité des barrages en terre après un vidange rapide Reliability analysis of slope stability in earthen dams following rapid drawdown

Z. KAHOT¹

1. Université Privée de Marrakech – zkahot08@gmail.com

Abstract

This work was dedicated to assessing reliability of slope stability in earthen dams following rapid drawdown. Modeling of the problem physics was based on saturated - unsaturated seepage flow through the nonlinear Richards equation, under the assumption of two-dimensional approximation in steady state and transient state. The modeling of seepage taking place inside the earth dam infill materials included soil parameters, the negative pore water pressure - hydraulic conductivity relationship as well as the negative pore water pressure - volumetric water content. Slope stability was analyzed by using safety factor as evaluated by the Bishop's simplified method. Solution of the governing equations was performed by using the modules SEEP/W and SLOPE/W of the Finite Element based software package GeoStudio.

Résumé

Ce travail a été consacré à l'évaluation de la fiabilité de la stabilité des talus des barrages en terre suite à un vidange rapide. La modélisation du problème physique était basée sur un écoulement d'infiltration saturé - non saturé à travers l'équation de Richards non linéaire, sous l'hypothèse d'une approximation bidimensionnelle en régime permanent et en régime transitoire. La modélisation des infiltrations ayant lieu à l'intérieur des matériaux de remplissage du barrage en terre comprenait les paramètres du sol, la relation pression interstitielle négative - conductivité hydraulique ainsi que la pression interstitielle négative - teneur en eau volumétrique. La stabilité de la pente a été analysée en utilisant le facteur de sécurité évalué par la méthode simplifiée de Bishop.

Mots clefs: *Sol insaturé-saturé ; barrage en terre; équation de Richards; Stabilité des talus; fiabilité.*

Keywords : *Unsaturated-saturated soil; earthen dam; Richards equation; slope stability; reliability.*

1. Introduction

Les problèmes liés à l'infiltration des eaux souterraines sont d'un intérêt pratique majeur pour l'ingénierie géotechnique et géo-environnementale. La défaillance

des barrages en terre se produit fréquemment en raison de l'infiltration d'eau à travers les matériaux de remplissage poreux. La défaillance peut prendre diverses formes telles que l'érosion interne, les défauts de fondation et l'instabilité des pentes. Les barrages en terre peuvent fonctionner correctement pendant de nombreuses années, mais ils nécessitent une conception et un entretien appropriés. Ils échouent rapidement s'ils sont exposés à un événement susceptible de les faire s'effondrer. Le rabattement rapide est connu pour être une condition dangereuse dans laquelle la stabilité de la face amont peut être critique. Par conséquent, la stabilité des pentes des fronts de remblai doit être assurée dans toutes les circonstances d'existence du barrage. Ce type d'instabilité a été connu dans le passé, comme dans le barrage de San Luis, en Californie, à la suite d'un prélèvement d'eau ordinaire du réservoir en septembre 1981.

L'infiltration des eaux souterraines a fait l'objet de nombreuses études. Lemacha et al. [1] ont utilisé l'équation de Richards non linéaire qui traite des écoulements saturés - insaturés à travers des milieux poreux et ont résolu le problème d'écoulement au moyen d'une procédure itérative numérique basée sur la méthode des différences finies implicites dans les directions alternatives. Botos [2] a mené une recherche pour déterminer les caractéristiques d'infiltration pour différentes stratégies d'exploitation des réservoirs d'eau. Il a représenté en conséquence le modèle de zone saturée lorsque le niveau d'eau maximum est maintenu constant pendant une longue période de temps. Mao et al. [3] ont étudié la modélisation inverse de l'écoulement à travers des milieux à saturation variable. Ils ont donné les conditions nécessaires pour que ce problème soit bien défini en termes d'existence d'une solution unique en milieu poreux à la fois homogène et hétérogène.

Andreaa [4] a présenté l'analyse de l'infiltration et de la stabilité réalisée en régime non saturé à travers le barrage de Maneciu en Roumanie. Il a pris en compte l'analyse des infiltrations à la fois en régime permanent et en régime transitoire. Il a utilisé l'infiltration d'eau telle que donnée par la courbe caractéristique sol-eau du modèle de van Genuchten [5]. Une rupture de pente peut se produire dans un barrage en terre même s'il est conçu

avec un facteur de sécurité élevé, car les géomatériaux sont principalement dominés par les incertitudes [6]. Pour intégrer cet effet des incertitudes inhérentes aux paramètres du sol dans le problème de la stabilité des pentes, l'analyse de fiabilité de la rupture des pentes du sol est considérée dans ce travail. Cette question probabiliste a reçu une attention considérable dans divers domaines scientifiques au cours des dernières décennies et de nombreuses méthodes de fiabilité ont été proposées dans la littérature [7]. Celles-ci incluent la méthode de fiabilité du premier ordre (FORM), la méthode des moments du premier ordre (FOSM), la méthode de fiabilité du deuxième ordre (SORM) et la simulation Monte Carlo (MCS) [8]. Les méthodes de surface de réponse (RSM) ont également été utilisées [9]. Le RSM utilise une représentation informatique efficace pour se rapprocher plus efficacement du modèle physique d'origine. Ceci permet d'effectuer une analyse de fiabilité explicitement basée sur la fonction de performance correspondant au RSM. En ce qui concerne la stabilité des pentes, de nombreux travaux précieux ont été publiés ces dernières années, tels que FOSM [10], MCS [11] et RSM [7] qui ont été appliqués pour évaluer la fiabilité de la conception de la stabilité des pentes.

Dans ce travail, la stabilité de la pente de la face amont du barrage en terre après un rabattement rapide du réservoir est évaluée par une approche d'analyse de fiabilité. L'objectif est d'analyser le flux d'infiltration saturé-insaturé se produisant à l'intérieur des matériaux de remplissage du barrage et de caractériser la stabilité comme prédisposée par les incertitudes inhérentes. Une approche efficace d'analyse de la fiabilité du système basée sur les méthodes MCS et RSM est utilisée pour évaluer la probabilité de rupture de pente. La modélisation probabiliste de la variabilité intrinsèque des propriétés du sol en termes de paramètres de résistance du sol est réalisée par des distributions lognormales. Ces incertitudes sont ensuite propagées par les conditions d'écoulement d'infiltration et la résistance au cisaillement dans les matériaux de remplissage du barrage en terre au facteur de sécurité réel. La modélisation physique utilise l'équation de Richards qui régit l'infiltration dans les milieux poreux saturés et insaturés. La relation entre la pression interstitielle négative et la conductivité hydraulique ainsi que la pression interstitielle négative et la teneur en eau volumétrique sont considérées à travers le modèle de van Genuchten. Pour la stabilité des pentes, la méthode simplifiée de Bishop est utilisée. Cette méthode suppose des forces horizontales entre les tranches et satisfait l'équilibre des forces verticales pour chaque tranche et l'équilibre des moments pour l'ensemble de la masse. L'état stable et l'état transitoire sont simulés à l'aide des logiciels GeoStudio SEEP/W et SLOPE/W [12]. Un barrage en terre homogène typique est étudié et une analyse de fiabilité est effectuée

au vu des incertitudes qui peuvent affecter les paramètres clés intervenant dans le problème de stabilité de la pente.

2. Méthodes

2.1 Equations gouvernantes du modèle

Dans ce qui suit, l'infiltration des eaux souterraines est supposée avoir lieu dans un barrage en terre homogène et isotrope. L'eau s'écoule à travers le milieu poreux remplissant le barrage et le modèle d'infiltration dépend de la composition du sol, du type d'écoulement, des propriétés du liquide et du gradient hydraulique. L'infiltration est régie par l'équation de Darcy dans des conditions saturées et non saturées. Cette équation exprime le fait que le débit volumique d'eau par unité de surface est directement proportionnel au taux de variation de la charge hydraulique. L'écoulement transitoire de l'eau dans le sol dans des conditions isothermes peut être dérivé en appliquant le principe de conservation de la masse. Le principe de conservation de la masse stipule que pour un volume élémentaire de sol donné, le taux de perte ou de gain d'eau est prudent et est égal au flux net d'entrée et de sortie. La conservation de la masse d'eau dans les milieux poreux saturés-insaturés donne l'équation de Richards [15] qui s'écrit sous la forme suivante :

$$\frac{\partial}{\partial x} \left(k_x(u_w) \frac{\partial H}{\partial x} \right) + \frac{\partial}{\partial y} \left(k_y(u_w) \frac{\partial H}{\partial y} \right) + Q = m_w(u_w) \rho_w \frac{\partial H}{\partial t}$$

La pression interstitielle est liée à la hauteur manométrique totale par :

$$H = h + y$$

avec h est la hauteur de pression définie comme :

$$h = \frac{u_w}{\rho_w g}$$

2.2 Simulation numérique

Différentes méthodes ont été développées pour résoudre les problèmes d'infiltration et de stabilité. Ces méthodes peuvent être classées comme analytiques, expérimentales ou numériques. Dans ce travail, il est fait appel au logiciel éléments finis GeoStudio à travers ses modules SEEP/W et SLOPE/W. GeoStudio est une suite de produits logiciels qui peuvent être utilisés pour évaluer la performance des barrages et des digues avec différents niveaux de complexité. Des analyses en régime permanent à long terme ou des analyses transitoires détaillées peuvent être effectuées pour prendre en compte les réponses dépendantes du temps. Les pressions interstitielles et les contraintes peuvent être incluses dans une analyse de stabilité avancée. La réponse de la structure à la charge sismique, au gel/dégel du sol ou à d'autres interactions terre-climat peut également être étudiée.

Dans ce qui suit, pour analyser la pente amont lors du rabattement du barrage en terre, une simulation est effectuée à l'aide de GeoStudio SLOPE/W. Cet outil

intégré au logiciel de la suite GeoStudio permet de calculer le coefficient de sécurité des talus en terre et en rocher. SLOPE/W peut analyser efficacement des problèmes complexes pour une variété de formes de surface de glissement, de conditions de pression interstitielle, de propriétés du sol, de méthodes d'analyse et de conditions de charge. Ici SLOPE/W est utilisé pour analyser la stabilité de la pente selon le critère de Mohr-Coulomb.

Résultats

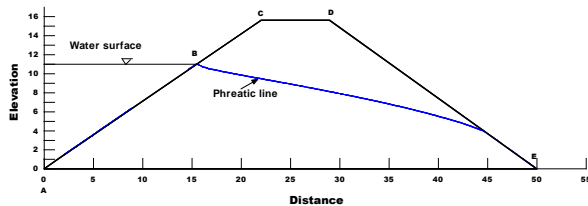


Figure 1 : Ligne phréatique après un vidange rapide

La figure 1 montre la variation des lignes phréatiques à 0 jusqu'à 5 jours après un rabattement rapide. L'eau commence à s'écouler du talus amont après le rabattement.

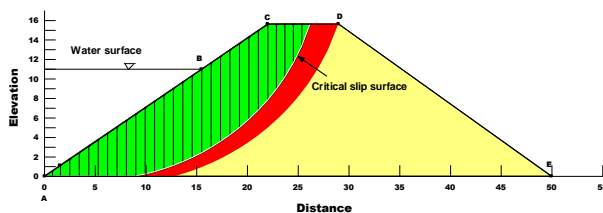


Figure 2 : Surface de glissement traversant la bande de cisaillement pour matériau argilo-limoneux

La figure 2 donne la surface de glissement. La surface de rupture a été divisée en 30 tranches car cela s'est avéré donner une convergence asymptotique en termes de facteur de sécurité. La figure 2 montre que le cercle de sécurité traverse la bande de cisaillement.

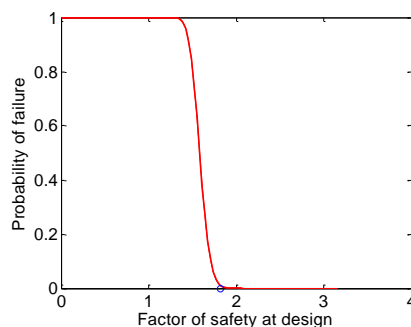


Figure 3 : Probabilité de défaillance en fonction du facteur de sécurité

Sur la figure 3, le cercle bleu est associé à une probabilité de défaillance égale à 0. Le facteur de sécurité associé est égal à 1,823.

Conclusion

Ce travail s'est concentré sur l'analyse de la stabilité des pentes des terres barrage suite à un rabattement rapide. Il a été constaté que, dans l'enquête au niveau des paramètres, seuls les deux facteurs : cohésion du sol et le coefficient de frottement interne expliquent la variabilité du facteur de sécurité.

Références

- [1] Lemacha, H., Maslouhi, A., Razack, M., Modeling of Transient Two Dimensional Flow in Saturated-Unsaturated Porous Media, European Scientific Journal Vol. 13 (2017) pp. 195-213.
- [2] Botos, M.L, Study of seepage for small homogeneous earth dams, Applied Numerical Mathematics and Scientific Computation, Proceedings of the 1st International Conference on Civil Engineering, Water Resources, Hydraulics & Hydrology (CEWHH 2014), Atena, Greece, November 28-30, pp. 142-146.
- [3] Mao, D., Yeh, T.-C. J., Wan, L., Hsu, K.-C., Lee, C.-H., J.-C. Wen J.-C, Necessary conditions for inverse modeling of flow through variably saturated porous media, Advances in Water Resources Vol. 52 (2013) pp.50-61.
- [4] Andreea, C, Unsaturated Slope Stability and Seepage Analysis of a Dam, Energy, Procedia Vol. (2015) pp. 93-98.
- [5] van Genuchten, M.T, A Closed-form Equation for Predicting the Hydraulic Conductivity of Unsaturated Soils, Soil Science Society of America Journal Vol. 44, (1980) pp. 892-898.
- [6] Johari, A., Lari, A.M., System probabilistic model of rock slope stability considering correlated failure modes, Comput Geotech Vol. 81 (2016) pp.26-38.
- [7] Li, D.S., Zhenga, D., Cao, Z.J., Tang, X.S., Phoon, K.K., Response surface methods for slope reliability analysis: Review and comparison, Engineering Geology Vol. 203 (2013) pp. 3-14.
- [8] Zhang, J., Huang, H.W., Juang, C.H., Li D.Q., Extension of Hassan and Wolff method for system reliability analysis of soil slopes, Engineering Geology Vol.160 (2013) pp. 81-88.
- [9] Jiang, S.H., Li, D.Q., Cao, Z.J., Zhou, C.B., Phoon, K.K., Efficient system reliability analysis of slope stability in spatially variable soils using Monte Carlo simulation, J. Geotech. Geoenviron Vol.141, (2015) pp. 04014096.
- [10] Low, B.K., FORM, SORM, and spatial modeling in geotechnical engineering,, Structural Safety, Vol. 49 (2014), pp.56-64.
- [11] Wang, Y., Uncertain parameter sensitivity in Monte Carlo simulation by sample reassembling,, Comput Geotech, Vol. 46, (2012) pp.39-47.
- [12] GeoStudio. Headquartered in Calgary, Alberta, Canada. <https://www.geoslope.com/>

Stability of two viscous miscible fluid layers saturating a porous medium heated from below

M. JAOUHARI¹, M. ECHCHADLI¹, S. ANISS¹

1. University of Hassan II, Faculty of Sciences Ain-Chock, Laboratory of Mechanic, B.P.5366 Maarif, Casablanca, Morocco. meryem.jaouhari12@gmail.com

Abstract :

We present a linear stability analysis of a system composed of two miscible, viscous fluid layers confined in a porous medium, between two parallel horizontal walls, in the presence of a temperature gradient opposite to the gravitational acceleration. The numerical procedure is based on the spectral methods, using the Chebychev collocation points. This method makes it possible to obtain a matrix formulation of the linear stability problem which corresponds to an eigenvalue problem. In this study, we examine the influence of the buoyancy number on the variation of Rayleigh number as a function of Darcy number. Numerical results show that small values of the Darcy number have a stabilising effect.

Keywords : Porous media; convection; two fluid layer

1 Introduction

Several works have been devoted to the thermal instability in two chemically stratified fluid layer and these studies have shown that the onset of instability can be either steady or oscillatory [1]. Other theoretical and experimental works have shown that the addition of an intrinsic density difference, due to chemical differences between layers, can lead to an oscillatory convection which can also occur separately in the two superimposed fluid layers without deformation of the interface. For instance, the situation corresponding to the earth mantles where the interface remains flat and a stationary convection develops in two superimposed layers, was studied by several researchers [2].

Le Bars and Davaille [3] focused on the regime where the interface is widely deformed and the movements propagate throughout the depth of the system. They used linear stability analysis and laboratory experiments, in the case where the viscosity ratio is greater than ($\gamma = 1$), to further study the stability and existence of two convection regimes, one stationary and the other one oscillatory, depending on viscosity ratio, depth and density ratio.

In the present study, we perform a linear stability analysis of a system, consisting of two layers of miscible fluids, confined in a porous medium heated from below. The

objective of this work is to study the effect of the Darcy number and other parameters such as the buoyancy number on the convection threshold.

2 Mathematical formulation

2.1 Governing equations

Consider a system of two viscous and incompressible fluid layers of thicknesses d_i , densities ρ_i and kinematic viscosities ν_{fi} ($i = 1,2$), saturating a porous medium of constant porosity and impermeability. The indices 1 and 2 designate respectively the lower layer and the upper one. It is assumed that there is no surface tension (miscible fluids) or chemical diffusion at the interface between the two fluids. The lower and upper walls are maintained at the uniform temperatures T_1 and T_2 respectively.

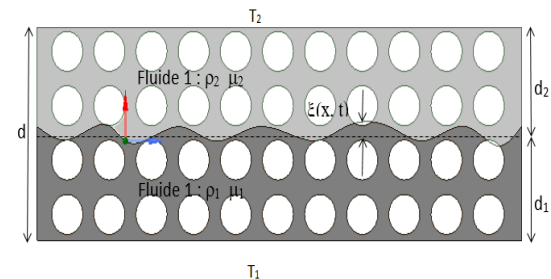


FIG. 1. System of two layers of incompressible, viscous and miscible fluids confined in a porous medium heated from below.

The conservation equations are the conservation of mass equation, the Brinkman equation of motion and the energy equation, which are written as :

$$\nabla \cdot \mathbf{u}_i^* = 0 \quad (1)$$

$$\rho_i (\phi^{-1} \frac{\partial \mathbf{u}_i^*}{\partial t^*} + \phi^{-2} (\mathbf{u}_i^* \cdot \nabla) \mathbf{u}_i^*) = -\nabla p_i^* - \frac{\mu_{fi}}{K} \mathbf{u}_i^* + \mu_{ei} \nabla^2 \mathbf{u}_i^* + \rho_i \mathbf{g} \quad (2)$$

$$(\rho c)_{pi} \frac{\partial T_i^*}{\partial t^*} + (\rho c_p)_{fi} (\mathbf{u}_i^* \cdot \nabla) T_i^* = \lambda_{pi} \nabla^2 T_i^* \quad (3)$$

where ρ_i , is the density, μ_{fi} , is the dynamic viscosity of the fluid i , μ_{ei} , is the effective viscosity of the fluid

i , ϕ , is the porosity, K , is the permeability, $(\rho c)_{pi} = (1 - \phi)(\rho c)_{si} + \phi(\rho c)_{fi}$, is the total heat capacity of the porous medium, $\lambda_{pi} = (1 - \phi)\lambda_{si} + \phi\lambda_{fi}$, is the total thermal conductivity of the porous medium. The indices f and s , are related to the fluid and solid phases respectively. Note that p_i^* , is the hydrodynamic pressure, and \mathbf{u}_i^* , is the filtration rate for each fluid.

2.2 Dimensional analysis and scales

We only consider the case where the system is initially stable, assuming that the bottom fluid layer is the densest. After having determined the basic solution which corresponds to a conductive regime in the two fluid layers and a zero velocity field, we proceed to carry out a linear stability study of this basic state by superimposing on it infinitesimal perturbations in terms of velocity field, temperature, pressure and density. The elimination of pressure from the equation of motion using the continuity equation leads to two equations of motion in w_i ($i = 1, 2$) and two equations of energy in T_i .

These equations are associated with four conditions at the bottom and top walls. On the upper wall, we use the non-slip boundary condition and a zero temperature disturbance for the fluid 1. On the lower wall, we impose the same boundary conditions for the fluid 2. Note that these conditions are reduced to six boundary conditions. At the interface, we consider the continuity of velocities (2 boundary conditions), the continuity of temperature and heat flux (2 boundary conditions) and continuity of normal and tangential stresses (2 boundary conditions). In total, we obtain 12 boundary conditions.

The resulting system is thus written in a dimensionless form by introducing the following dimensionless variables for each fluid ($i=1, 2$)

$$(x, y, z) = \left(\frac{x^*}{d}, \frac{y^*}{d}, \frac{z^*}{d}\right); (u_i, v_i, w_i) = \left(\frac{u_i^*}{\vartheta}, \frac{v_i^*}{\vartheta}, \frac{w_i^*}{\vartheta}\right)$$

$$T_i = \frac{T_i' - T_1}{T_2 - T_1} = \frac{T_i'}{\Delta T}; \quad t = \frac{t^*}{d}; \quad p_i = \frac{p_i'}{\rho_{02} \nu_{f2} \vartheta}$$

where $d = d_1 + d_2$ and $\vartheta = \frac{\beta g (T_1 - T_2) d^2}{\nu_{f2}}$ is the Stokes velocity which has been chosen as the characteristic scale of the velocity of the problem [3]. We designate by α the thermal diffusivity and β the thermal expansion coefficient considered the same for the two fluid layers.

The system (1)-(3) is written in adimensional form as follows :

$$\nabla \cdot \mathbf{u}_i = 0 \quad (4)$$

$$\phi^{-1} Pr^{-1} S Ra \frac{\partial \mathbf{u}_i}{\partial t} = -\nabla p_i - \frac{\nu_{fi}}{\nu_{f2} Da} \mathbf{u}_i + \Lambda \frac{\nu_{fi}}{\nu_{f2}} \nabla^2 \mathbf{u}_i - T \mathbf{k} \quad (5)$$

$$Ra \left(\frac{\partial T_i}{\partial t} - w_i \right) = \nabla^2 T_i' \quad (6)$$

Thus, the following dimensionless numbers are introduced : the ratio of kinematic viscosities, $\gamma = \frac{\nu_{f1}}{\nu_{f2}}$, the

Rayleigh, $Ra = \frac{\beta d^3 g \Delta T}{\nu_{f2} \alpha}$, the buoyancy number which is the ratio between the difference density of compositional origin and temperature difference, $B = \frac{\rho_{01} - \rho_{02}}{\rho_0 \beta \Delta T}$,

the Prandtl number, $Pr = \frac{\nu_{f2}}{\alpha}$, the Darcy number which represents the relative permeability effect of the medium with respect to its cross section, $Da = \frac{K}{d^2}$, and the ratio of the heat capacities of the fluid and $\Lambda = \frac{\mu_{ei}}{\mu_{fi}}$ is the ratio of the effective viscosity and the viscosity of the fluid, equal to unity [5] and [6]

The solutions of the two equations of motion in w_i ($i = 1, 2$) and two equations of energy in T_i are written in normal modes : $(w_i, T_i) = (\tilde{w}_i(z), \tilde{T}_i(z)) \exp(\sigma t + qx)$ where $\sigma = \sigma_r + i\omega$, the imaginary part of σ , denoted ω , is the phase velocity of the perturbation. The growth rate of the perturbation is σ_r , the real part of σ . We designate by q the wavenumber in the horizontal direction.

3 Numerical procedure and validation

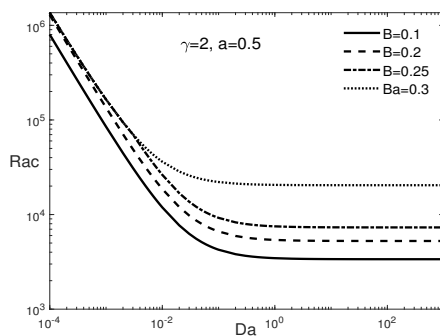
To solve the linearized equations obtained in section 2. 2, we use the spectral methods based on the Gausse-Lobatto collocation points for spatial resolution [4] to obtain a matrix formulation corresponding to an eigenvalue problem. To validate our numerical procedure, we consider the case where the medium is not porous ($Da = 1000$) and we restrict our validation to the stationary marginal stability assuming that $\omega = 0$ (marginal stability) and $\sigma_r = 0$ (stationary stability). This configuration has been studied at first in [3], and corresponds to a system of two fluid layers of equal dynamic viscosities ($\gamma = 1$) and equal depth ratio $a = 0.5$. For a buoyancy number $B \neq 0$, we obtain the results by Le Bars and Davaille [3]: $Ra_c = 17610$ and $q_c = 5.36$.

4 Results and discussions

We present in FIG. 1 the evolution of the critical Rayleigh number, Ra_c , as a function of the Darcy number, Da ,

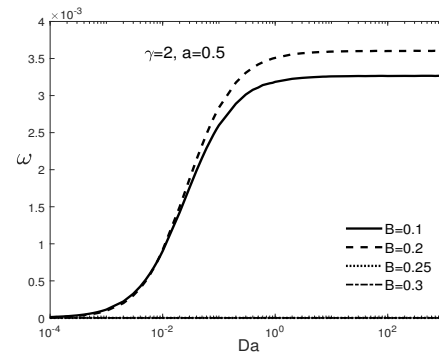
for different values of the buoyancy number B . It can be seen that as the Darcy number increases, a situation where convection is accompanied by less resistance, the convection threshold decreases and reaches a constant value independent of the Darcy number, which is the value of the critical Rayleigh number of a two-layer fluid system in a non-porous medium [3]. The significant decrease in the critical Rayleigh number occurs for Darcy numbers below 1. Above this value, the Darcy number has no effect and the values of the critical Rayleigh numbers reach asymptotic values corresponding to those of non-porous media. This leads to confirm that the porous medium is stabilizing, through the small values of the Darcy number, compared to a non-porous medium. Moreover, By increasing the buoyancy number, the curves for the critical Rayleigh number versus the Darcy number undergo upward translations. Thus, the buoyancy number stabilizes the system (see FIG. 2).

We also observe that for Darcy values lower than 10^{-3} the curves corresponding to $B = 0.25$, $B = 0.3$, are almost merged, this is due to the existence of two regimes, the oscillatory regime and the stratified one. The first one depends on the value of the buoyancy number and has a non-zero temporal frequency (see figure FIG. 3). When we increase the buoyancy number, the convection threshold and the temporal frequency also increase. The stratified regime is independent of the buoyancy number and the temporal frequency is zero, thus the curves merge. For $\gamma = 6.7$ and for Darcy values higher than 1, we find similar results to those of Le Bars and Davaille



[3].

FIG. 2. Variation of the rayleigh number as a function of the Darcy's number for buoyancy number, $B = 0.1$, $B = 0.2$, $B = 0.25$, $B = 0.3$, $\gamma = 2$, $Pr = 10^4$ and $\phi =$



0.6.

FIG. 3. Variation of the temporal frequency as a function of the Darcy's number for buoyancy number, $B = 0.1$, $B = 0.2$, $B = 0.25$, $B = 0.3$, $\gamma = 2$, $Pr = 10^4$ and $\phi = 0.6$.

5 Conclusion

In this work, we studied the effect of the Darcy number on the instability threshold of two superimposed layers confined in a porous medium and heated from below. We examined the effect of the Darcy number on the convection threshold. It turns out that for Darcy number smaller than unity, we have a stabilising effect on the convection threshold. For large values of the Darcy number, we obtain the values of the Rayleigh number corresponding to a non-porous medium [3].

References

- [1] F. M. Richter, et C. E Johnson, "Stability of a chemically layered mantle.", *J. Geophys. Res.* 79 1635-1639 (1974).
- [2] S. Rasenat, F. H. Busse, et I. Rehberg, "A theoretical and experimental study of double-layer convection". *J. Fluid Mech.* 199 519-540 (1989).
- [3] M. Le Bars, A. Davaille, "Stability of thermal convection in two superimposed miscible viscous fluids", *J. Fluid Mech.* vol. 471, pp. 339-363 (2002).
- [4] A. C. Weideman, S. C. Reddy, *A MATLAB differentiation matrix suite*, *ACM Trans. Math. Softw.* 26, 465-519, 2000.
- [5] T. S. Lundgren, "Slow flow through stationary random beds and suspensions of spheres.", *J. Fluid Mech.*, 51:273-299 (1972).
- [6] G. Neale, and W. Nader, "Practical significance of Brinkman's extension of Darcy's law: Coupled parallel flows within a channel and a bounding porous medium.", *Canad. J. Chem. Engng.*, 52:475-478 (1974).

INFILTRATION EN MILIEUX POREUX NON-SATURES : PHENOMENOLOGIE ET MODELISATION

INFILTRATION IN UNSATURATED POROUS MEDIA : PHENOMENOLOGY AND MODELING

R. ABABOU ⁽¹⁾

⁽¹⁾ IMFT - Institut de Mécanique des Fluides de Toulouse, France. ababou@imft.fr

Abstract

We review infiltration models for unsaturated porous media, including the piston flow model of Green & Ampt, and a recent extension called « Moving Multi-Front (MMF) » which propagates a discrete number of « fronts » or pressure iso-values (Alastal & Ababou 2019). The MMF model is quasi-analytical, gridless. It reduces the Richards PDE to a system of ODE's governing the positions of the fronts. We compare it to a finite volume model (BIGFLOW3D) and a finite element model (HYDRUS1D) for infiltration to a water table, capillary rise, etc. The results are encouraging in terms of CPU time and precision. Finally, we analyze other infiltration phenomena with the general Richards PDE solver BIGFLOW3D : dynamics of 2D/3D wetted zone during localized infiltration ; effects of heterogeneities (perched water, discontinuous humidity profiles, etc.).

Résumé

On passe en revue certains modèles d'infiltration en milieux poreux non saturés, incluant l'écoulement piston de Green & Ampt, et une extension récente dite "Moving Multi-Front (MMF)" qui propage un nombre discret de "fronts" ou iso-valeurs de pression (Alastal & Ababou 2019). Le modèle MMF est quasi-analytique, "sans grille". Il réduit l'EDP de Richards à un système différentiel régissant les positions des "fronts". On le compare à des modèles volumes finis (BIGFLOW3D) ou éléments finis (HYDRUS1D), pour une infiltration vers une nappe, une ascension capillaire, etc. Les résultats sont encourageants en temps calcul et en précision. Enfin, on analyse d'autres phénomènes d'infiltration avec le code BIGFLOW3D (solveur général de l'EDP de Richards) : dynamique du bulbe humide en infiltration localisée 2D/3D ; effet des hétérogénéités (eaux perchées, profils discontinus d'humidité, etc.).

Mots clefs: *Écoulements non saturés ; milieux poreux non saturés ; infiltration ; méthode multi-front (MMF)*

Keywords : *Unsaturated flow ; Unsaturated porous media ; Infiltration ; Multi-Front Method (MMF)*

1. Introduction

On présente d'abord une classe particulière de modèles d'écoulements en milieux poreux non saturés, le modèle Moving Multi-Front (MMF) (Alastal & Ababou 2019), qui peut être vu comme une extension du modèle classique d'écoulement piston de Green & Ampt. Le modèle MMF est quasi analytique, sans grille, de type Lagrangien. Il est comparé aux modèles numériques classiques Eulériens (volumes finis, éléments finis) pour une série de problèmes d'écoulements verticaux (infiltration vers le bas, mais aussi, ascension capillaire). Les comparaisons indiquent que le calcul avec le modèle MMF est très avantageux sur le plan temps calcul (CPU), à précision égale.

En second lieu, on complète cette présentation par une étude d'autres phénomènes d'infiltration, qui requièrent un outil de modélisation plus général, tel que le code de calcul BIGFLOW3D (Ababou & Bagtzoglou 1993) qui est un solveur général de l'EDP de Richards en milieux hétérogènes à saturation variable (non saturés ou partiellement saturés). On présente certains aspects phénoménologiques de l'infiltration multidimensionnelle dans les sols non saturés (dynamique du bulbe d'humidité en infiltration localisée), ainsi que certains phénomènes causés par les hétérogénéités et les interfaces en milieux souterrains multicouches ou stratifiés (eaux perchées, distribution discontinue des teneurs en eaux, ...).

2. Le modèle d'infiltration MMF (Moving Multi-Front)

2.1 Présentation du modèle MMF

Le modèle Moving Multi-Front consiste à résoudre l'EDP de Richards pour un écoulement vertical 1D en discrétisant la variable pression « h » en un nombre discret d'iso-valeurs, ou « fronts ».

L'équation verticale de Richards (1931) est une équation de conservation de masse qui peut s'écrire comme suit avec « z » orienté vers le bas (profondeur):

$$\frac{\partial \theta(h)}{\partial t} = \frac{\partial}{\partial z} \left(K(h) \frac{\partial h}{\partial z} - g K(h) \right)$$

Cette EDP de Richards est accompagnée de la loi de Darcy exprimant la densité de flux q_z [m/s] non saturée

(le 2nd membre ci-dessus est la divergence de $-qz$). La variable h [m] est la pression interstitielle relative à la pression atmosphérique, θ [m³/m³] est la teneur en eau volumique, K [m/s] est la conductivité hydraulique. La relation $\theta(h)$ est la courbe de rétention d'eau du sol, et la relation $K(h)$ est la courbe de conductivité hydraulique du sol : ces deux relations constitutives caractérisent le milieu poreux non saturé.

Le nouveau modèle dit « MMF » consiste à résoudre le système différentiel décrit ci-dessous, qui régit les positions des M « fronts » mobiles. Le milieu poreux est découpé en zones mobiles, et la loi de Darcy est appliquée à chaque zone mobile (i) avec $1 \leq i \leq M$:

$$q_{i-1/2}(t) = -K_{i-1/2} \left[\frac{\partial h}{\partial z} - g \right] = K_{i-1/2} \left[\frac{\psi_{F_i} - \psi_{F_{i-1}}}{Z_{F_i}(t) - Z_{F_{i-1}}(t)} + 1 \right]$$

...où ψ représente la succion ($\psi = -h$, opposé de la pression). Les flux mid-frontaux $q_{i-1/2}(t)$ sont définis aux positions intermédiaires entre les fronts mobiles $Z_{F_{i-1}}(t)$ et $Z_{F_i}(t)$. Chaque flux $q_{i-1/2}(t)$ est une approximation du flux dans la zone mobile $[Z_{F_{i-1}}(t), Z_{F_i}(t)]$. Enfin, la conservation de masse s'applique autour de chaque front mobile $Z_{F_i}(t)$:

$$q_{i+1/2}(t) - q_{i-1/2}(t) = (\theta_{i+1/2} - \theta_{i-1/2}) \frac{dZ_i}{dt}$$

Finalement, le système différentiel de taille M à résoudre pour les $Z_{F_i}(t)$ s'obtient en insérant ci-dessus les flux mid-frontaux $q_{i-1/2}(t)$

La **Figure 1** illustre graphiquement la discrétisation en iso-valeurs ou « fronts ».

Lorsque $M = 1$, ce modèle se réduit au modèle d'écoulement piston de Green & Ampt. A l'opposé, lorsque $M \rightarrow \infty$ on s'attend à ce que le modèle MMF converge vers l'EDP de Richards, avec comme coefficients les courbes non linéaires $K(h)$ et $\theta(h)$, propriétés du milieu non saturé.

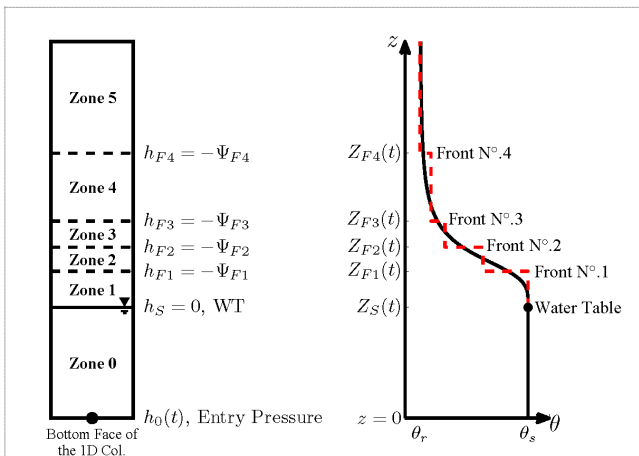


Figure 1 Illustration graphique de la discrétisation d'un profil vertical de teneurs en eau en iso-valeurs ('fronts').

2.2 Applications et tests du modèle MMF

On présente ci-dessous des tests du modèle d'infiltration multi-front (MMF), modèle Lagrangien sans grille, par rapport à des modèles numériques Eulériens classiques volumes finis (BIGFLOW3D) ou éléments finis (HYDRUS1D), qui ont un maillage fixe. Les problèmes

testés dans Alastal & Ababou (2019) incluent divers types d'écoulements verticaux :

- infiltration vers une nappe profonde;
- infiltration vers une nappe peu profonde;
- ascension capillaire à partir d'une nappe fixée;
- ascension capillaire due à la montée graduelle de la surface libre d'une nappe.

Un de ces tests est montré dans la **Figure 2**.

De plus, les tests ont été analysés quantitativement en termes de temps calcul CPU et en termes de précision, en fonction du nombre M de fronts discrets pris en compte dans le modèle MMF (par exemple $M = 1, 2, 4, 8, 16, 31, 64$). Pour les simulations avec les codes classiques, on a pris comme base de comparaison un maillage très fin de 1000 nœuds verticalement. Ces résultats sont encourageants. En résumé, on a trouvé que l'ordre de précision est en $1/M^2$ sur la teneur en eau (et en $1/M$ sur le flux). En pratique la précision constatée est excellente si $M \geq 32$, mais elle reste bonne même pour un nombre réduit de fronts ($M = 8$). Rappel : le modèle piston de Green & Ampt correspond au cas $M = 1$.

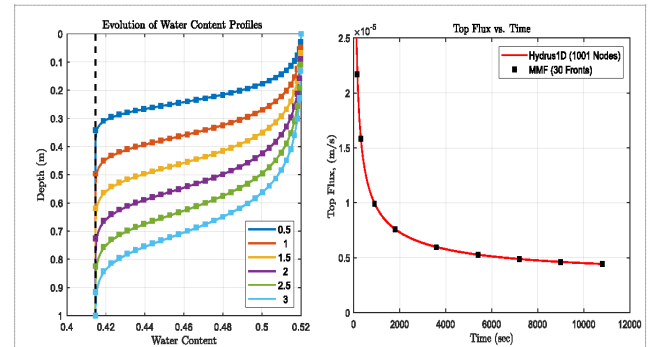


Figure 2 Test de la méthode MMF pour un problème d'infiltration vers une nappe profonde pour le sol Guelph Loam (GL). Simulation par le code éléments finis HYDRUS1D avec 1001 nœuds fixes, comparée avec la méthode MMF avec $M = 30$ fronts mobiles représentés par des symboles carrés. A gauche : profils de teneur en eau à différents temps t [heures]. A droite : évolution temporelle du flux d'infiltration en surface (t en secondes).

3. Phénomènes d'infiltration en milieux hétérogènes multicouches (cas 1D)

La **Figure 3** illustre le cas d'une infiltration verticale transitoire vers une nappe fixée, à travers un bi-couche (sol fin limoneux « Guelph Loam », au-dessus d'un sol plus grossier « Fine Sand »). La nappe fixe est à 5 m de profondeur. L'interface entre les deux couches est à 2.5 m de profondeur. La condition initiale est une condition d'équilibre hydrostatique. L'infiltration dure 3.5 jours, sous condition de saturation totale de la surface de la couche supérieure de sol (limon GL).

On remarque notamment :

- la « forte discontinuité » de teneur en eau à l'interface entre les deux couches ; et
- l'établissement aux temps longs d'un régime d'écoulement gravitationnel ($\partial\theta/\partial z \approx 0$) dans une grande partie de la couche inférieure constituée du sol sableux (Fine Sand).

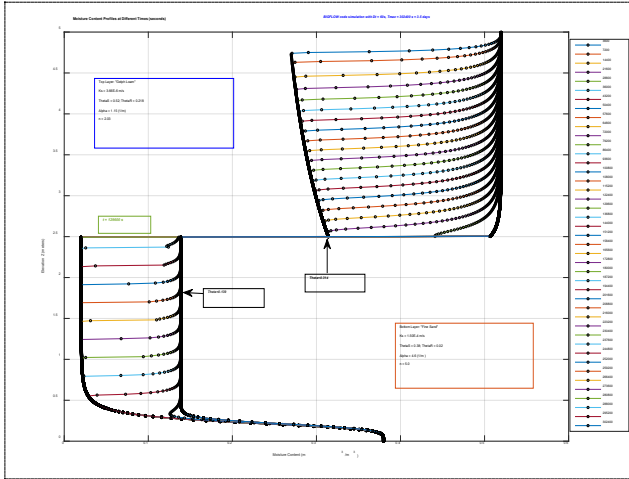


Figure 3 Infiltration dans un sol bi-couche GL/FS (limon/sable) : profils de teneur en eau $\theta(z,t)$ sur 3.5 jours

4. Infiltration multidimensionnelle

4.1 Dynamique des bulbes d'humectation en irrigation localisée 2D/3D

La **Figure 4** ci-dessous illustre l'effet sur l'extension du bulbe d'humectation du débit Q [l/h/m] d'une source linéaire, e.g., une ligne de goutteurs en micro-irrigation. Noter la formation d'une flaque saturée en surface, surtout pour le plus fort débit.

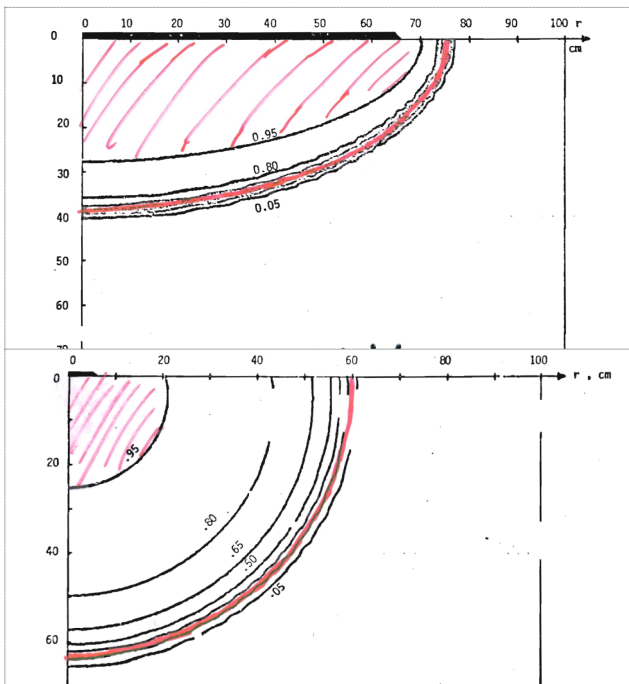


Figure 4 Infiltration localisée linéaire en sol homogène (Loess MC4). Effet du débit de la ligne d'irrigation Q [l/h/m] pour un même volume total d'apport d'eau (72 litres/m). En haut : débit fort $Q = 3.0$ l/h/m ($t=24$ h). En bas : débit faible $Q = 0.5$ l/h/m ($t=144$ h). [Ababou 1981].

Ce résultat a été obtenu par simulation numérique aux différences finies de l'EDP de Richards, transformée par la transformation de *Kirchhoff*, résolue par la méthode ADI (*Alternate Directions Implicit*), soit en symétrie

axiale (x,z) comme ici pour une ligne d'irrigation, soit en symétrie cylindrique (r,z) pour un goutteur individuel (Ababou 1981). NB : des problèmes d'infiltration localisée 3D sont également simulés en milieux hétérogènes par le code Volumes Finis BIGFLOW3D.

4.2 Distribution de teneur en eau en milieu hétérogène : formation d'une nappe perchée

La **Figure 5** présente une modélisation de la genèse d'une nappe perchée au-dessus d'une lentille d'argile, qui bloque temporairement le passage de l'eau lors d'une infiltration par submersion (inondation). Cette simulation a été obtenue par résolution de l'EDP de Richards partiellement saturée avec le code BIGFLOW3D.

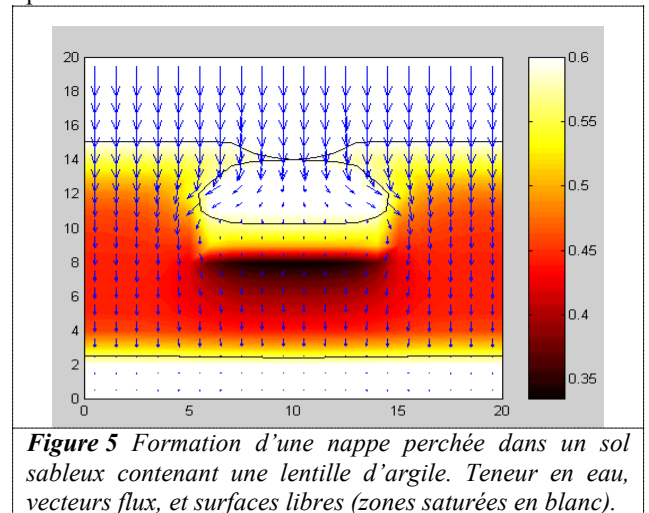


Figure 5 Formation d'une nappe perchée dans un sol sableux contenant une lentille d'argile. Teneur en eau, vecteurs flux, et surfaces libres (zones saturées en blanc).

Références

- [1] Ababou R. (2018). *Capillary Flows in Heterogeneous and Random Porous Media 1*. ISTE Ltd and John Wiley & Sons, Inc. ISBN 978-1-84821-528-3. Vol. 1, First Print, Nov. 2018, 371 pp.
- [2] Ababou R., Bagtzoglou A.C. (1993). *BIGFLOW: A Numerical Code for Simulating Flow in Variably Saturated, Heterogeneous Geologic Media (Theory and User's Manual, Version 1.1)*. Report NUREG/CR-6028, U.S. Nuclear Regul. Comm., Gov. Printing Office, Washington D.C., USA, 139p.
- [3] Ababou R. (1981). *Modélisation des Transferts Hydriques dans le Sol en Irrigation Localisée*. Thèse de Dr-Ing., Institut de Mécanique de Grenoble (IMG), USMG & Institut National Polytechnique de Grenoble (INPG), France, 1981, 218 pp.
- [4] Alastal K., R. Ababou (2019). Moving Multi-Front (MMF): a generalized Green-Ampt approach for vertical unsaturated flows. *J. of Hydrology* 579 (2019) 124184 (17 pp.). <https://doi.org/10.1016/j.jhydrol.2019.124184>
- [5] Green W.H., G.A. Ampt (1911). Studies on soil physics. *The Journal of Agricultural Science*, 4(1): 1-24. DOI:10.1017/S0021859600001441
- [6] Richards L.A. (1931). Capillary Conduction of Liquids through Porous Mediums. *Physics*, 1(5): 318-333. DOI:10.1063/1.1745010
- [7] Simunek J., Sejna M., Saito H., Sakai M., Van Genuchten M.T. (2013). *The HYDRUS-1D Software Package for Simulating the One-Dimensional Movement of Water, Heat, and Multiple Solutes in Variably-Saturated Media, V.4.17*. June 2013. Univ. California, Riverside, CA. Report, 308 pp.

NUMERICAL SIMULATION OF A GRANULAR FLOW ON A SMOOTH INCLINED PLANE

TAZI. Rida ^{*,*}, ECHCHELH. Adil ^a

^a Laboratory of Electronic systems information processing, Mechanics and energetics, Faculty of sciences, Ibn Tofail University, Kenitra, Morocco

* Corresponding e-mail address: Tazirida1993@gmail.com

ABSTRACT

The granular flow on a rough inclined plane, can be described by empirical models, model of Saint-Venant and Savage-Hager consider the substance as a dense fluid (hydrodynamic description). However, these two models don't take into account the effects of the forces which appear between the particles during the flow. In order to highlight these forces, we use the method of cellular automata. Indeed, this type of model considers the studied system as a set of cells, where each cell contains one or more particles and the rules are given to change the state of the system. In this work, we are interested in the macroscopic parameters which is the origin of clogging phenomenon on a rough substrate. The numerical simulation which carried out on PYTHON based on the Discrete Element Method (DEM) that made it possible to highlight the effect of the parameters: substance-substance / substance-substrate friction, particle density, the forces between the particles (Forces of Hertz, Visco-elastic force...) on the flow velocity of particles on a plane with variable angles of inclination (0° to 45°).

KEYWORDS

Granular flow, Friction, Cohesion, angle of inclination, temperature.

INTRODUCTION

Granular materials are the set of macroscopic solid particles (powders, sand, seeds, etc.) and an interstitial fluid. They can be formed either as solids or as fluids, depending on how they are stressed, like the hourglass [1, 2]. The field of application is very variable, we find industrial processes (transport, silo storage) and in geophysics (fast gravity flow, movement of sand dunes, etc.). The domain which interests us is the granular flow, it is the nowadays objects that consist of many sensitive parameters such as flow, roughness of the walls, shape of the grains and coupling with the interstitial fluid [3, 4].

In our work, we will focus on dry grains, without cohesion. Implies the neglect of the possible influence of the interstitial fluid and the interaction between the grains must be in direct contact (collisions, friction). In this regime, the strength of the material is essentially characterized by an angle of internal friction, and the

Mohr-Coulomb criterion is distinguished by the stability of the rupture, which occurs by sliding along planes (also called shear band [5, 6, 7]). The work's objective which carried out, is to produce precise information on the dense flows of dry grains, either by experience or by numerical simulations, and to compare it with the predictions of several models. finds the description of Saint-venant [8,9,10,11] which considers a dry granular flow, without cohesion and it takes into account the substance-substrate friction, it proposes a simplified form of the granular flow, which depends on the angle of inclination of the substrate upon the horizontal plane and the thickness of the substance H. It makes it possible to visualize the variation of speed according its position on the thickness of the substance (for each point of the thickness going from 0 to H we have a well-defined speed) . Thanks to this speed, we can deduce the average speed of the substance on an inclined plane, The description of Savage-Hatter makes reference to Saint-venant, we designate the Height of the substance is variable all along the inclined plane and we also have the method of cellular automata, this type of modeling considers the system studied is a set of cells, each cell contains one or more particles and we give some rules to evaluate the state of the system. Cellular automata makes it possible to test the mechanisms that would be at the origin of the phenomenon studied. These simulations were created ,by considering each grain individually, and called Discrete Element Methods (DEM).

The objective of this study is to determine the parameters that create the phenomenon of clogging in a granular flow, for this reason, we use molecular dynamics (one of the DEM simulations) on Python, since it is based on the interactions of particles during shocks (the laws of repulsion and friction). The first part of this work is devoted to a brief study of molecular dynamic theory, which will be the basis of our algorithm. Then, we will carry out simulation tests of the granular flow in order to verify the logic of the program. In the second part, several simulation tests were carried out in order to find the clogging situation and to determine the parameters which cause this state (Clogging). The results demonstrate that the reasons of colmatage phenome are the force

between particles and its density .Besides, the effect of the friction of the substrate-substance and substance-substance has no serious impact on release of particles.

MATHEMATICAL MODEL

The granular flow in an inclined plane can be simulated as a set of spheres linked together by interaction forces, in order to determine the equation that controls the movement of the particles, we considered a sphere located at the substrate level and surrounded by other spheres (compact medium), figures (1).

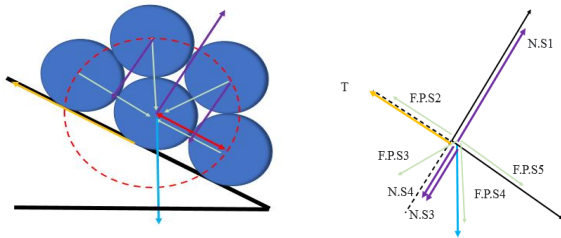


Figure (1) : Description of the forces applied on the sphere studied

The description of movement of spheres on an inclined plane is obtained by projecting the equation of motion (1) onto the 'z' axis and the 'x' axis, equations (2) and (3).

$$(3). m_i \frac{d\vec{v}_i}{dt} = \sum_{contact \{ij\}} \vec{F}_{j \rightarrow i} + m_i \vec{g} \quad (1)$$

Suivant l'axe 'OZ':

$$0 = N.S1 - mg.\cos(\theta) - N.S4 - N.S3 - |F.P.S3|. \sin(\alpha) - |F.P.S3|. \sin(180 - \alpha) \quad (2)$$

Suivant l'axe 'OX':

$$M.S1\ddot{X} = mg.\sin(\theta) - T - F.P.S2 + F.P.S5 - |F.P.S3|. \cos(\alpha) + |F.P.S3|. \cos(180 - \alpha) \quad (3)$$

Avec θ : Angle of inclination, α : Angle of positions of the spheres which surrounded the studied sphere. F.P.S2, F.P.S5, F.P.S3, F.P.S4, N.S4, N.S3 et N.S1 : the forces applied to the studied sphere, figure (1)

During the variation of the angle of inclination, we have two states Rolling and Slipping.the determination of the tangential force which is going to be inserted in the equation (3) different from one state to another. For rolling case of the spheres, we use the Moment of inertia, equation (4) and the condition of non-slip equation (5) to determine the tangential force and for the sliding, we base it on the projection "OZ" , equation (2) and regularize Amontons-Coulomb law (6).

$$I_{\Delta}\ddot{\theta} = R|T| \quad (4)$$

With :

$$I_{\Delta} = \frac{1}{2}mR^2$$

$\ddot{\theta}$: vitesse angulaire

$$\dot{X} = R\dot{\theta} \quad (5)$$

$$|T| = \mu.N.S1 \quad (6)$$

Result is discussed:

In order to validate the program, we visualized the effect of the variation of the angle of inclination, we present the positions of the spheres at two different times ($T=t_0$ and $T=t$) in figure (2). We note that at each instant 't', we have new positions for our particles.

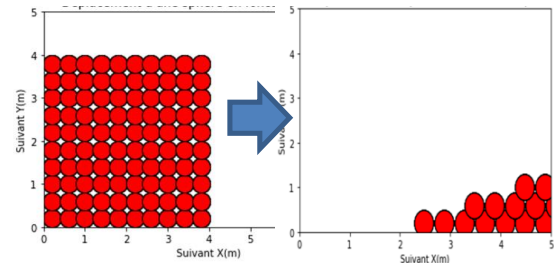


Figure (2): Particle positions at different times t_0 and t

After having noted the correct operation of the program under the imposed conditions, we tried to visualize the effects of the density of the spheres and the modulus of the forces which applied between spheres on the flow in a variable inclination plane (from 0° to 30°) as a function of time. During the first simulation, we vary the density of the spheres ($\rho_1 = 19,09$, $\rho_2 = 76,39 \text{ kg/m}^3$) we set the substrate substance friction coefficient ($\mu_1 = 0,4$), the substance-substance friction coefficient is ($\mu_2 = 0,7$) and the model of the interaction forces after projection on the "OX" axis ($|Total \text{ force}|=5$). It is noted that the spheres placed at the free surface and in the center, the speed increases from the first moment contrary to the spheres in contact with the substrate their speed remains null until the moment $t=2$ seconds. during The transition from the rolling state to the sliding state, the speed decreases before resuming its increase until the plan is fixed at the maximum angle of inclination which is worth 30° for stabilization at a constant value all the duration of the simulation. The spheres which are in contact with the substrate move only by sliding, figure (3). If the density is increased, the spheres in contact with the substrate are released at the same time as the other upper spheres and the transition effect (rolling-sliding) is minimized, figure (4).

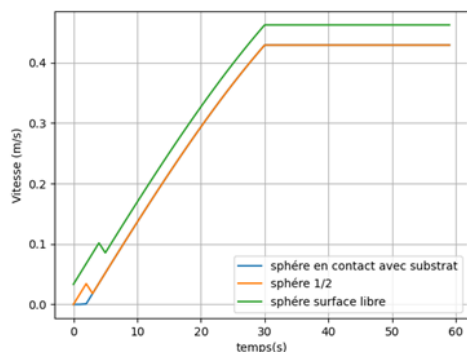


Figure 3: variation in the speed of the spheres, for a density of 19.09 kg/m³

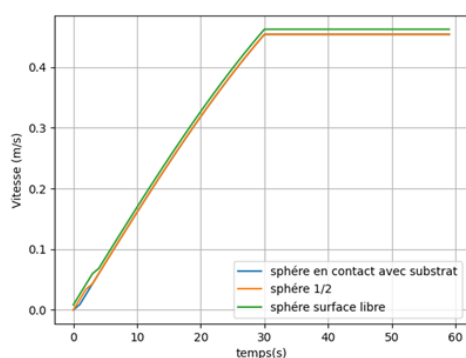


Figure 4 : variation in the speed of the spheres, for a density of 76.39 kg/m³

In the second simulation, we kept the previous conditions such as the substance-substance and substance-substrate friction coefficient and we increase the interaction force modulus ($|Total\ force|=20$) for spheres of density $\rho=19.09\text{ kg/m}^3$, figure (5). It is shown that the velocity of middle sphere is canceled after the phase of rolling which lasts 2 seconds, the spheres are free only after the plan of inclination waits for the angle 8° , contrary to the figure (4). We see the effect of the interaction forces on the movement of the spheres, the greater their modulus, the greater the braking of the spheres and the non-flow of the substance (CLOGGING EFFECT).

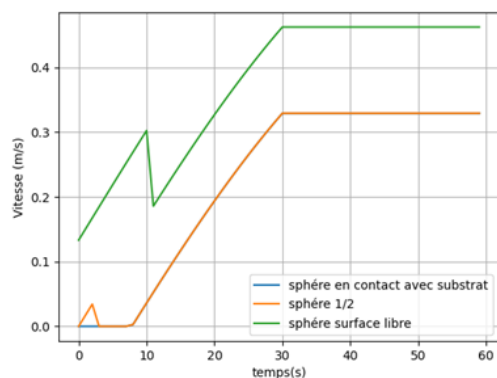


Figure 5 : variation of the speed of the spheres on an inclined plane

Conclusion :

The description of the dry granular flow that is proposed by the method model of cellular automata, made it possible to visualize the effect of the density and the interaction applied forces on the behavior of the spheres on a plane with a variable inclination. It is noted that to avoid the clogging of the particles, it is necessary that the density of the particles is important and the forces of interaction are minimized. In the next work, we will try to determine the types of major forces that apply to the particles during a flow, in order to develop solutions that will be applicable in real life for avoiding the clogging of substances on the particles substrates in the industrial field, such as truck bodies and storage silos.

REFERENCES

1. R. L. Brown et J. C. Richards Principles of powder mechanics Pergamon Press, Oxford (1970).
2. J. Duran Sables, poudres et grains. Eyrolles Sciences, Paris (1997).
3. S. B. Savage Flow of granular materials In Theoretical and applied mechanics edited by P. Germain, M. Piau and D.Caillerie, 241-266, North Holland, Amsterdam (1989).
4. K. Hutter & K. R. Rajagopal On flows of granular materials Continuum Mechanics and Thermodynamics, 6, 81-139 (1994).
5. J. Desrues, R. Chambon, M. Mokni & F. Mazeroll Void ratio evolution inside shear bands in triaxial sand specimens studied by computed tomography, Volume 46 Issue 3, September 1996, pp.529-546.
6. Minglong Fei , Qicheng Suna , Deyu Zhong, Gordon G.D. Zhou, Simulations of granular flow along an inclined plane using the Savage–Hutter model. Particology 10 (2012) 236– 241
7. Pierre ROGNON, Jean-Noël ROUX, Mohamed NAAIM, François CHEVOIR, Bidisperse granular flows on an inclined plane: Friction law and two-layer model, 2007
8. Michaël Prochnow. Ecoulements denses de grainssecs. Mécanique [physics.med-ph]. Ecole des Ponts ParisTech, 2002. Français.
9. J. Duran Sables, poudres et grains. Eyrolles Sciences, Paris (1997).
10. S. B. Savage Flow of granular materials In Theoretical and applied mechanics edited by P. Germain, M. Piau and D.Caillerie, 241-266, North Holland, Amsterdam (1989).
11. J. Duran Sables, poudres et grains. Eyrolles Sciences, Paris (1997).

Thème 07 :

Mécanique des fluides, instabilités hydrodynamiques, transition et turbulence

Dynamic characteristics of finite journal bearings lubricated with ferrofluids

Khalil Atlassi^a, Mohamed Nabhani^a and Mohamed El Khlifi^a

^a Hassan II University of Casablanca, Faculty of Sciences and Techniques, P.O. Box 146, Mohammedia 28806, Morocco.

E-mail addresses: khalil.atlassi@etu.fstm.ac.ma (K. ATLASSI), nabhanimohamed@yahoo.fr (M. NABHANI), mohamed.elkhlifi@fstm.ac.ma (M. EL KHLIFI).

Abstract

This work is concerned by the dynamic behavior of a rigid shaft supported by a finite hydrodynamic journal bearing lubricated with ferrofluids based on the Shliomis model. The applied magnetic field is created by a wire located out of the journal bearing system. The pressure field is determined by solving the modified Reynolds equation of this type of fluids to determine the dynamic characteristics performance. The calculation of dynamic coefficients is based on the use of the mathematical perturbation method. Dynamic coefficients are calculated to obtain the whirling frequency and the critical mass to determine the stability zone of the journal bearing.

Keywords: Ferrofluid; Rotational viscosity; Dynamic characteristics; Journal bearing.

1. Introduction

The journal bearing is a very important element of rotating machines, because it ensures the rotational guidance of the rotor. The rotor stability, in dynamic conditions, is conditioned and may affect the proper functioning of the machine, hence the advantage of determining the instability zones of the journal bearings in order to avoid them and preserve the duration of life of systems working in dynamic regime. Several interesting works have focused on analyzing journal bearing dynamic characteristics [1-5].

This work deals with a numerical study of stability conditions of finite hydrodynamic journal bearings lubricated by ferrofluids. The Shliomis model [6,7] is used to consider the effects of the rotation of ferromagnetic particles and their magnetic moment. The applied magnetic field is created by a finite wire of electric current placed at an optimum angle outside the journal bearing [8,9]. The modified Reynolds equation developed here is numerically solved using the finite difference method. The stiffness and damping coefficients are determined by applying the mathematical perturbation method, with the aim of specifying the stability map corresponding to each relative eccentricity.

2. Analysis

The modified Reynolds equation for journal bearings operating under dynamic conditions is obtained here as follows [10]:

$$\frac{1}{R^2} \frac{\partial}{\partial \theta} \left(\frac{h^3}{\eta + \eta_{sx'}} \frac{\partial p}{\partial \theta} \right) + \frac{\partial}{\partial z} \left(\frac{h^3}{\eta + \eta_{sz'}} \frac{\partial p}{\partial z} \right) = 6\omega \frac{\partial h}{\partial \theta} + \frac{1}{R} \frac{\partial}{\partial \theta} \left(\frac{h^3}{\eta + \eta_{sx'}} f_{mx'} \right) + 12\dot{h} \quad (1)$$

with $\eta_{sx'} = \frac{1}{4} \mu_0 \bar{\tau} M_0 H$ is the rotational viscosity in the x' direction, $\eta_{sz'} = \frac{1}{4} \mu_0 \bar{\tau} \frac{M_0}{H} H_y^2$ is the rotational viscosity in the z' direction, $f_{mx'} = \mu_0 \frac{M_0}{H} H_{x'} \frac{\partial H_{x'}}{\partial x'}$ is the magnetic force induced per unit of volume, $\bar{\tau} = \frac{6\eta\phi}{nK_B T(1+\xi \coth \xi)}$, $M_0 = n\mu(\coth \xi - 1/\xi)$ is the equilibrium magnetization given by the Langevin formula [6].

The externally applied magnetic field adopted in this study is that created by a finite wire of an electric current located at a distance R_0 , greater than the radius R of the journal bearing, and parallel to the axial direction z' from the bearing center O . The strength of this magnetic field is simply given by [8]:

$$H(\theta, z') = \frac{I_c}{4\pi r} \left\{ \sin \tan^{-1} \left(\frac{L/2 + z'}{r} \right) + \sin \tan^{-1} \left(\frac{L/2 - z'}{r} \right) \right\} \quad (2)$$

where $r = \sqrt{R^2 + R_0^2 - 2RR_0 \cos(\Psi - \theta)}$.

The calculation of dynamic coefficients, therefore of the pressure field, requires the use of numerical methods. In dynamic regime the movement of the center of the shaft is described by the amplitudes Δx and Δy measured from its static equilibrium position. Using the fundamental assumption of small journal displacements, the film thickness can be written in the following form:

$$h = h_0 + \Delta x \cos(\theta + \Phi) + \Delta y \sin(\theta + \Phi) \quad (3)$$

where Φ is the attitude angle h_0 is the film thickness for the static regime given by:

$$h_0 = c(1 + \varepsilon \cos \theta) \quad (4)$$

The shaft center displacements remaining small, we have:

$$\dot{h} = \Delta \dot{x} \cos(\theta + \Phi) + \Delta \dot{y} \sin(\theta + \Phi) \quad (5)$$

The shaft center movement causes a perturbation in the pressure field p_0 in static regime:

$$p = p_0 + p_x \Delta x + p_y \Delta y + p_x' \Delta x' + p_y' \Delta y' \quad (6)$$

Injecting Eq. (3) and (5) into Reynolds equation (1), and using the following approximation:

$$h^3 \approx h_0^3 + 3h_0^2(\Delta x \cos(\theta + \Phi) + \Delta y \sin(\theta + \Phi)) \quad (7)$$

we obtain the five following equations:

$$L_0(p_0) = 6\omega \frac{\partial h_0}{\partial \theta} + \frac{1}{R} \frac{\partial}{\partial \theta} \left(\frac{h_0^3}{\eta + \eta_{sx'}} f_{mx'} \right) \quad (8)$$

$$L_0(p_x) = -6\omega \sin(\theta + \Phi) + \frac{1}{R} \frac{\partial}{\partial \theta} \left(\frac{3h_0^2 \cos(\theta + \Phi)}{\eta + \eta_{sx'}} f_{mx'} \right) + L_x(p_0) \quad (9)$$

$$L_0(p_y) = 6\omega \cos(\theta + \Phi) + \frac{1}{R} \frac{\partial}{\partial \theta} \left(\frac{3h_0^2 \sin(\theta + \Phi)}{\eta + \eta_{sx'}} f_{mx'} \right) + L_y(p_0) \quad (10)$$

$$L_0(p_x) = 12 \cos(\theta + \Phi) \quad (11)$$

$$L_0(p_y) = 12 \sin(\theta + \Phi) \quad (12)$$

with

$$L_0(\cdot) = \frac{1}{R^2} \frac{\partial}{\partial \theta} \left(\frac{h_0^3}{\eta + \eta_{sx'}} \frac{\partial}{\partial \theta} (\cdot) \right) + \frac{\partial}{\partial z'} \left(\frac{h_0^3}{\eta + \eta_{sz'}} \frac{\partial}{\partial z'} (\cdot) \right)$$

$$\begin{cases} L_x(\cdot) \\ L_y(\cdot) \end{cases} = -\frac{1}{R^2} \frac{\partial}{\partial \theta} \left(\frac{3h_0^2}{\eta + \eta_{sx'}} \begin{cases} \cos(\theta + \Phi) \\ \sin(\theta + \Phi) \end{cases} \frac{\partial}{\partial \theta} (\cdot) \right)$$

$$- \frac{\partial}{\partial z'} \left(\frac{3h_0^2}{\eta + \eta_{sz'}} \begin{cases} \cos(\theta + \Phi) \\ \sin(\theta + \Phi) \end{cases} \frac{\partial}{\partial z'} (\cdot) \right)$$

3. Pressure boundary conditions

The hydrodynamic pressure and the magnetic force have a symmetry with respect to the median plane of the journal bearing ($z'=0$). Based on this symmetry and to save computation time, equations (8) to (12) are only solved on half of the bearing. Reynolds boundary conditions in dimensionless form accounting for bearing cavitation phenomena are used:

$$\bar{p}(\theta, \bar{z}' = 1/2) = 0 \quad (13)$$

$$\frac{d\bar{p}}{d\bar{z}'}(\theta, \bar{z}' = 0) = 0 \quad (14)$$

$$\bar{p}(\theta = 0, \bar{z}') = 0 \quad (15)$$

$$\bar{p}(\theta_c, \bar{z}') = \frac{d\bar{p}(\theta_c, \bar{z}')}{d\theta} = 0 \quad (16)$$

with θ_c represents film breakage or reformation angle.

4. Stiffness coefficients K_{ij}

The calculation of \bar{p}_x and \bar{p}_y are used to determine the dimensionless stiffness coefficients \bar{K}_{ij} .

$$\bar{x} = \frac{x}{c}, \bar{y} = \frac{y}{c}, \bar{p} = \frac{p}{\eta_0 \omega} \left(\frac{c}{R} \right)^2, \bar{K}_{ij} = K_{ij} \frac{c}{W_0}$$

$$\bar{K}_{xx} = -2 \int_0^{0.5} \int_0^{2\pi} \frac{1}{W_0} \bar{p}_x \cos(\theta + \Phi) d\theta d\bar{z}' \quad (17)$$

$$\bar{K}_{yx} = -2 \int_0^{0.5} \int_0^{2\pi} \frac{1}{W_0} \bar{p}_x \sin(\theta + \Phi) d\theta d\bar{z}' \quad (18)$$

$$\bar{K}_{xy} = -2 \int_0^{0.5} \int_0^{2\pi} \frac{1}{W_0} \bar{p}_y \cos(\theta + \Phi) d\theta d\bar{z}' \quad (19)$$

$$\bar{K}_{yy} = -2 \int_0^{0.5} \int_0^{2\pi} \frac{1}{W_0} \bar{p}_y \sin(\theta + \Phi) d\theta d\bar{z}' \quad (20)$$

5. Damping coefficients C_{ij}

The calculation of \bar{p}_x and \bar{p}_y are used to determine the dimensionless damping coefficients \bar{C}_{ij} .

$$\bar{x} = \frac{\dot{x}}{c\omega}, \bar{y} = \frac{\dot{y}}{c\omega}, \bar{p} = \frac{p}{\eta_0 \omega} \left(\frac{c}{R} \right)^2, \bar{C}_{ij} = C_{ij} \frac{c\omega}{W_0}$$

$$\bar{C}_{xx} = -2 \int_0^{0.5} \int_0^{2\pi} \frac{1}{W_0} \bar{p}_x \cos(\theta + \Phi) d\theta d\bar{z}' \quad (21)$$

$$\bar{C}_{yx} = -2 \int_0^{0.5} \int_0^{2\pi} \frac{1}{W_0} \bar{p}_x \sin(\theta + \Phi) d\theta d\bar{z}' \quad (22)$$

$$\bar{C}_{xy} = -2 \int_0^{0.5} \int_0^{2\pi} \frac{1}{W_0} \bar{p}_y \cos(\theta + \Phi) d\theta d\bar{z}' \quad (23)$$

$$\bar{C}_{yy} = -2 \int_0^{0.5} \int_0^{2\pi} \frac{1}{W_0} \bar{p}_y \sin(\theta + \Phi) d\theta d\bar{z}' \quad (24)$$

The calculation of the dynamic coefficients allows to calculate the dimensionless whirling frequency (DWF), dimensionless critical mass (DCM) and the dimensionless threshold speed (DTS) :

$$\text{DWF} = \sqrt{\frac{(\bar{K}_{xx} - \bar{K}_o)(\bar{K}_{yy} - \bar{K}_o) - \bar{K}_{xy}\bar{K}_{yx}}{\bar{C}_{xx}\bar{C}_{yy} - \bar{C}_{xy}\bar{C}_{yx}}} \quad (25)$$

$$\text{DCM} = \frac{\bar{K}_o}{\text{DWF}^2} \quad (26)$$

$$\text{DTS} = \sqrt{\text{DCM}} = \frac{\sqrt{\bar{K}_o}}{\text{DWF}} \quad (27)$$

where

$$\bar{K}_o = \frac{\bar{K}_{xx}\bar{C}_{yy} + \bar{K}_{yy}\bar{C}_{xx} - \bar{K}_{xy}\bar{C}_{yx} - \bar{K}_{yx}\bar{C}_{xy}}{\bar{C}_{xx} + \bar{C}_{yy}} \quad (28)$$

6. Numerical simulation

In this work, for each given relative eccentricity a first calculation is made to determine the angle corresponding to the maximum pressure using a non-ferrofluids lubricant. This angle will correspond to the angular position Ψ of the electric current wire generating the applied magnetic field. The second stage consists in calculating the pressure p_0 corresponding to the static regime for determining the attitude angle Φ . The latter will serve as the basis for the calculation of the different dynamic coefficients.

The algebraic system, obtained after discretization by the finite difference method of the flow equations on a rectangular mesh of constant grid size, is solved

numerically by the iterative method of Gauss Seidel with a relaxation coefficient for an accelerated convergence of the solution. The resolution of all the equations is ensured by a FORTRAN program specially developed in this work.

7. Example of results

It's found in this work that the use of a ferrofluid as a lubricant has a destabilizing effect for a circular journal bearing for low loads (low relative eccentricities), this unstable zone is more significant with high loads (high relative eccentricities). Consequently, the zone corresponding to the dimensionless whirling frequency will be widened with the use of ferrofluid as a journal bearing lubricant.

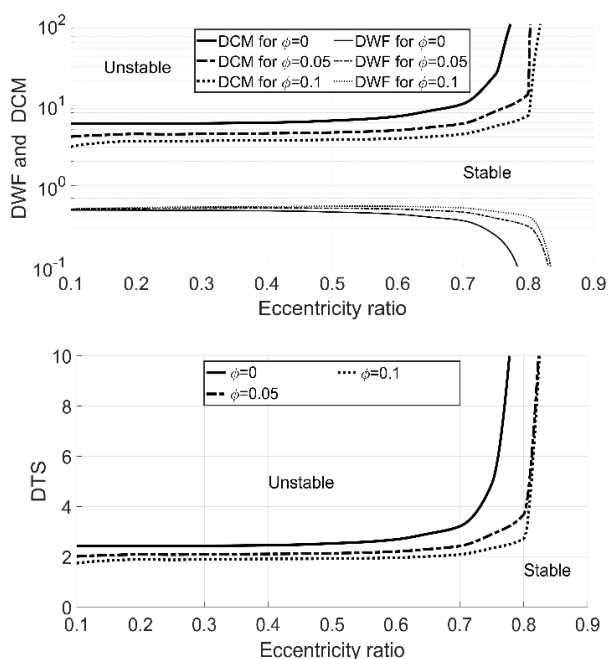


Fig. Stability map for different concentrations of ferromagnetic particles for $\omega = 500$ RPM.

More details on the used model and the obtained results can be found in [10].

8. Conclusions

Based on the mathematical method of film thickness perturbation, the dynamic characteristics of a journal bearing lubricated by ferrofluids were presented to finally bring out the stability map as a function of the concentration of ferromagnetic particles. The applied external magnetic field is that created by a finite wire displaced out of and parallel to the journal bearing. Using this method, five governing equations are derived and resolved using finite differences and Successive Over Relaxation methods.

This work shows that the use of ferrofluid as a lubricant reduces the limits of stability during operation

of a finished plain bearing, despite its considerable contribution to the improvement of pressure and load capacity. Therefore, this study allows lubrication specialists to find a compromise between improving pressure and load capacity and ensuring operating stability.

References

- [1] Patel NS, Vakharia D, Deheri G (2017) Hydrodynamic journal bearing lubricated with a ferrofluid. *Indus Lubri and Tribo* 69(5):754-760. <https://doi.org/10.1108/ILT-08-2016-0179>
- [2] Wang X, Li H, Lu W (2017) Stiffness and damping properties of (semi) floating ring bearing using magnetorheological fluids as lubricant. *Journal of Tribology* 139(5):051701-1-11. <https://doi.org/10.1115/1.4035773>
- [3] Montazeri H (2009) Numerical analysis of hydrodynamic journal bearings lubricated with ferrofluid. *Intelli Mater Syst and Struc* 222(1):51-60. <https://doi.org/10.1243/13506501JET314>
- [4] Niklas M (1987) Influence of magnetic field on Taylor vortex formation in magnetic fluid. *Z.Phys. B-Condensed Matter* 68:493-501. <https://doi.org/10.1007/BF01471080>
- [5] Lin JR, Li PJ, Hung TC (2013) Effects of non-newtonian ferrofluids on the performance characteristics of long journal bearings. *Flui Dyna and Mater Proc* 9(4):419-434. <https://doi.org/10.3970/fdmp.2013.009.419>
- [6] Shliomis MI (1972) Effective viscosity of magnetic suspensions. *Soviet Physics* 34(6):1291-1294
- [7] Shliomis MI (1974) Magnetic fluids. *Sov Phys* 17:153-169. <https://doi.org/10.1070/PU1974v017n02ABEH004332>
- [8] Laghrabli S, El Khelifi M, Nabhani M, Bou-saïd B (2017) Static characteristics of ferrofluid finite journal bearing considering rotational viscosity effect. *Lubri Sci* 29(4):203-226. <https://doi.org/10.1002/lis.1364>
- [9] Laghrabli S, El Khelifi M, Nabhani M, Bou-saïd B (2017) Ferrofluid lubrication of finite journal bearings using Jenkins model. *Lubri Sci* 29(7):441-454. <https://doi.org/10.1002/lis.1379>
- [10] Atlassi K, Nabhani M, El Khelifi M (2021) Rotational viscosity effect on the stability of finite journal bearings lubricated by ferrofluids. *J Braz. Soc. Mech. Sci. Eng.* 43:548. <https://doi.org/10.1007/s40430-021-03264-2>

Modeling Homogeneous Two-phase Water Hammer Flow

M.OUZI¹, B. BAHRAR², H. SAMRI³, M. TAMANI⁴

1. Nanostructures and Advanced Materials, Mechanics and Thermofluid, FST Mohammedia, 28804, Morocco. mohamed.ouzi-etu@etu.univh2c.ma
2. Team of Fluid Mechanics and Environment ENSETM Nanostructures and Advanced Materials, Mechanics and Thermofluid FST Mohammedia, Hassan II University, Casablanca, 28804, Morocco. bahrarb@yahoo.com.
3. Team of Mechanics, Energetics & Environment, ENSET of Mohammedia, Hassan II University, CasablancaLEM2A, FS-ESTM Laboratory. samrih1@yahoo.fr.
4. ESTM, Moulay Ismail University of Meknes, 50040, Morocco. m.tamani@umi.ac.ma

Abstract

The present study deals with the computational simulation of a two-phase water hammer using the homogeneous model. Governing equations are presented. The experimental validation example chosen allowed the use of the equations in conservative, semi-conservative and non-conservative forms. The Two steps Lax-Wendroff scheme stabilized with the flux limiter: Flux Corrected Transport (FCT) [1] is devoted to numerical methods. Present results are compared with the considered experimental ones. The comparisons show that the present method gives adequate results which are in good agreement with those of the experiment.

Keywords : *Flux Corrected transport, Lax-Wendroff, Transient flow, Water Hammer.*

1. Introduction

Variations in flow velocities in the industry are a common and even unavoidable occurrence. Water hammer is generally one of the worst consequences. Single-phase models of water hammers have been discussed by several authors [2], [3], [4], the simulation associated with these studies has improved over the years [5]. Several two-phase water hammer flow models have been proposed [6], [7], but it should be noted the difficulty of the latter compared to single-phase flows. The capture of the position and shape of interfaces and of bubbles, and the number of possible bubbles or gas pockets are some examples. The homogeneous model is the simplest to apply and simulate the two-phase flows, especially for a small amount of dissolved gas. It can be treated as single equivalent fluid.

This study is based on the two-phase flow experimentally realized by Chaudhry et al. [8], which consists of a flow whose quality is assumed to be constant. For more details, see Chaudhry et al. [8]. The particularity of this example is that it can be treated by three approaches: conservative, semi-conservative and non-conservative forms for the flow conservation equations.

2. Homogeneous flow model

The main assumptions used in this study are: unidirectional unsteady flow, the flow is homogeneous and isothermal, and the pipe is elastic. Gas bubbles are assumed uniformly distributed in the flowing liquid through the pipe. The pipe cross-section under a pressure p can be written as: [9]:

$$\frac{dA}{A\sqrt{A}} = \frac{2c}{Ee\sqrt{\pi}} dp \quad (1)$$

The pipe section area can then be expressed in terms of the average absolute pressure as:

$$A(p) = \frac{A_o}{1 - \frac{D_o c}{2Ee}(p - p_o)^2} \quad (2)$$

A_o is the initial pipe section area at the standard pressure p_o , E is the Young's modulus for the pipe material, D_o is the pipe diameter, c is the pipe constraint factor, and e is the wall thickness. For all analysis, the variables with subscript o refer to steady-state.

2.1 Two-phase mixture density formulation

Neglecting heat transfer between the two phases of liquid involves to consider isothermal flow, therefore the pressure p is related to gas density as:

$$\frac{p}{\rho_g} = \frac{p_o}{\rho_{go}} = cte \quad (3)$$

The quality θ is assumed to be constant, thus the mixture density ρ can be expressed in terms of the density of each phase as:

$$\rho = \left(\frac{\theta}{\rho_g} + \frac{1-\theta}{\rho_l} \right)^{-1} \quad (4)$$

Assuming the bulk liquid modulus of elasticity k_l to be constant leads the liquid phase density ρ_l to be expressed as:

$$\rho_l = \rho_{lo} e^{\frac{p-p_o}{k_l}} \quad \text{Thus} \quad \rho = \left[\frac{\theta p_o}{\rho_{go} P} + \frac{1-\theta}{\rho_{lo}} e^{\frac{(p_o-p)}{k_l}} \right]^{-1} \quad (5)$$

3. Governing equations

3.1 Conservative form

The mass and momentum conservation laws are represented by the following equations [2]:

- Mass conservation equation:

$$\frac{\partial}{\partial t}(\rho A) + \frac{\partial}{\partial x}(\rho AV) = 0 \quad (6)$$

- Momentum conservation equation

$$\frac{\partial V}{\partial t} + V \frac{\partial V}{\partial x} + \frac{1}{\rho} \frac{\partial p}{\partial x} = -g \sin \beta - \frac{f |V| V}{2D} \quad (7)$$

Where V is the cross-section averaged fluid velocity, β is the pipe slope angle, f Darcy-Weisbach friction coefficient and, g is the acceleration of gravity.

By using Eq. (5), the last equation Eq. (7) can be written as outlined below [9]:

$$\frac{\partial V}{\partial t} + \frac{\partial F}{\partial x} = S \quad (8)$$

Where: $S = -g \sin \beta - \frac{f |V| V}{2D}$

$$F = \frac{V^2}{2} + \frac{\theta p_o}{\rho_{go}} \ln \left(\frac{p}{p_o} \right) - \frac{k_1 (1-\theta)}{\rho_l}$$

Eqs. (6), and (8) can be written in a vectorial form as:

$$\frac{\partial \mathbf{V}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} = \mathbf{S}$$

$$\text{Where: } \mathbf{V} = \begin{bmatrix} \rho A \\ V \end{bmatrix}, \mathbf{F} = \begin{bmatrix} \rho AV \\ F \end{bmatrix} \text{ and } \mathbf{S} = \begin{bmatrix} 0 \\ S \end{bmatrix}$$

3.2 Semi-conservative form

In this subsection the mass equations can be expressed in non-conservative form as follows [11]:

$$\frac{\partial}{\partial t}(\rho A) = \rho A \left[\frac{2c}{Ee\sqrt{\pi}} \sqrt{A} + \rho \left(\frac{\theta}{\rho_g p} + \frac{1-\theta}{K_l \rho_l} \right) \right] \frac{\partial p}{\partial t} \quad (9)$$

$$\frac{\partial p}{\partial t} = -\frac{1}{w(p)} \frac{\partial}{\partial x}(\rho AV) \quad (10)$$

Where:

$$w(p) = \rho(p) A(p) \left[\frac{2c}{Ee\sqrt{\pi}} \sqrt{A(p)} + \rho(p) \left(\frac{\theta}{\rho_g p} + \frac{1-\theta}{K_l \rho_l} \right) \right]$$

In the following, Eq. (6) and Eq. (7) are written in respectively, non-conservative and conservative form by the use of the vectorial notation as: (semi-conservative form)

$$\frac{\partial \mathbf{P}}{\partial t} + \mathbf{A} \frac{\partial \mathbf{F}}{\partial x} = \mathbf{S}$$

$$\text{Where: } \mathbf{P} = \begin{bmatrix} p \\ V \end{bmatrix}, \mathbf{A} = \begin{bmatrix} 1 \\ w(p) \end{bmatrix}, \mathbf{a} = \begin{bmatrix} 1 \\ \epsilon \end{bmatrix}$$

3.3 Non-conservative form

Herein, both of the two conservation equations are considered in their non-conservative forms using primitive variables. Therefore, the mass and momentum conservation

equations are used directly from Eq. (10), and, Eq. (7), respectively.

$$\frac{\partial p}{\partial t} = -\frac{1}{w(p)} \frac{\partial}{\partial x}(\rho AV) \quad (11)$$

$$\frac{\partial V}{\partial t} = -V \frac{\partial V}{\partial x} - \frac{1}{\rho} \frac{\partial p}{\partial x} - g \sin \beta - \frac{f |V| V}{2D} \quad (12)$$

Comparison of the solutions obtained numerically by considering conservative, semi-conservative and non-conservative forms will be covered in the following.

4. Numerical methods

The numerical simulation was performed using a cell-centered finite difference method. Governing equations classically discretized are solved using Lax-Wendroff scheme. In order to suppress spurious oscillations, (FCT) algorithm is invested in this work. Approximations are done, namely in non-conservative forms. The main approximation consist of replacing derivative with numerical derivative approximations as follows:

$$\frac{1}{\rho} \frac{\partial p}{\partial x} \Big|_i^n = \frac{1}{\Delta x} \left(\frac{p_{i+1}^n}{\rho(\bar{p})} - \frac{p_i^n}{\rho(p_{i+\frac{1}{2}}^n)} \right) \text{ With } \bar{p} = \frac{1}{2} \left(p_{i+\frac{1}{2}}^n + p_{i-\frac{1}{2}}^n \right)$$

4.1. Initial Conditions

The pressure $p^o(x)$ along the pipe in steady-flow, can be determined using the Runge-Kutta algorithm. Once $p^o(x)$ is obtained, one can compute $V^o(x)$ Further details can be found in [10], [11].

4.2. Boundary Conditions

As can be seen in Fig.1, the upstream boundary was a constant-level reservoir $p_o^{n+1} = \rho g H_{res}$, where H_{res} denoted the main reservoir head, while the pressure history at upstream was given by the reported data from experiment [8]. For the (FCT) algorithm, transmissive boundary conditions are chosen herein. Further details can be found here [11], [12].

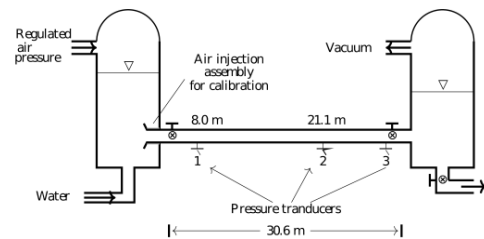


Fig.1 Experimental facility, Chaudhry et al.[8]

5. Results and validation.

All conditions experiments can be found in [8].

The ratio e/c appearing in Eq. (1) is taken equal to:

$e/c = 8,93.10^{-5}$ in order to agree with the value $715 m.s^{-1}$ of wave speed corresponding to the single phase water flow given by Chaudhry [8]. The numerical method used in this work shows stability for the Courant-Friedrichs-Lewy (CFL)

number, from $CFL=0,1$ to $CFL=0,8$ for conservative form of equations, but the best value which fit with all experiment results for semi and non-conservative form is $CFL=0,45$ [11].

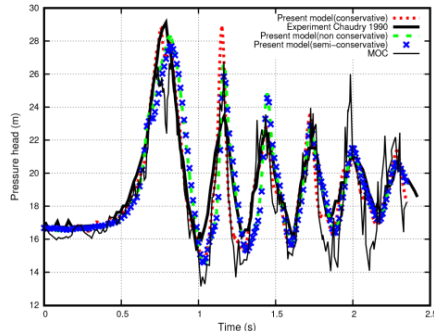


Fig. 2. Comparison between computed and experimental absolute pressure at $x = 8$ m for $\theta_1 = 3,2 \cdot 10^{-6}$

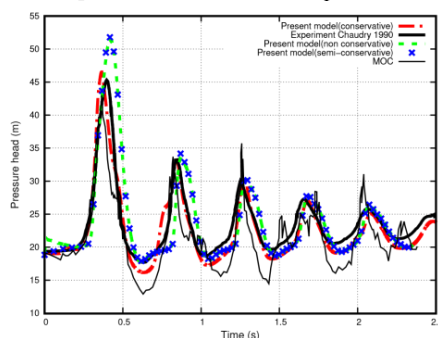


Fig. 3 . Computed and experimental absolute pressure at $x = 8$ m for $\theta_2 = 7,4 \cdot 10^{-6}$.

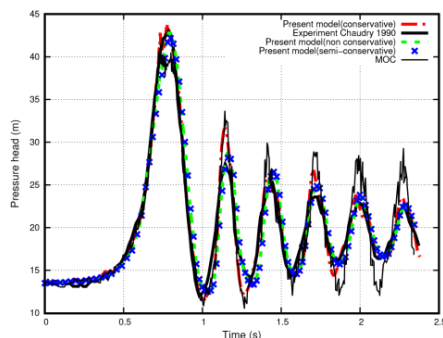


Fig. 4 . Comparison between computed and experimental absolute pressure at $x = 21$ m for $\theta_1 = 3,2 \cdot 10^{-6}$.

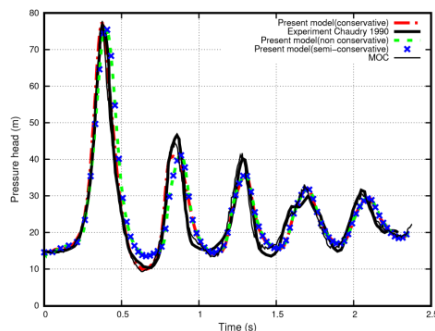


Fig.5 . Computed and experimental absolute pressure at $x = 21$ m for $\theta_2 = 7,4 \cdot 10^{-6}$

6. Conclusion

The homogeneous flow model to simulate water hammers was associated with a Lax-Wendroff scheme. The numerical diffusion has been suppressed thanks to the algorithm (FCT). The numerical results obtained are in good agreement with experimental results from the literature as it can be seen in Figs. 2 to 5. It's important to note the total absence of spurious oscillations generated by the numerical diffusion. The main conclusion of this work is that the usage of an accurate second-order numerical scheme combined with a flux limiter type (FCT) gives good results even when the equations are taken in their non-conservative forms.

REFERENCES

- [1] J. P Boris, D. L Book, *Flux-corrected transport. I. SHASTA, a fluid transport algorithm that works*, Journal of Computational Physics, Volume 11, Issue 1,(1973) , Pages 38-69.
- [2] M.H. Chaudhry *Applied hydraulic transients*. Editions Van Nostrand Reinhold Company Inc New York, 1987.
- [3] L.V. Streeter, Wylie, E, *Hydraulic transients*, FEB Press, 1982.
- [4] M.S. Ghidaoui, Zhao, Ming, McInnis, A. Duncan, Axworthy, David H. *A Review of Water Hammer Theory and Practice*, Applied Mechanics Reviews vol 58 (1) (2005) pages: 49-76.
- [5] A. Bergant, A. Tijsseling, J.P. Vítkovský, D. Covas, A.R. Simpson, M.F. Lambert, *Parameters affecting water hammer wave attenuation, shape and timing. Part 2: Case studies*, Journal of Hydraulic Research, Vol. 46, No. 3, (2008) pp. 373-381.
- [6] M. Tamani, B. Bahrar, K. Guerraoui,, *Modeling Vaporous Cavitation in Transient Pipe Flow Using the Zielke's Friction Model*, International Review of Mechanical Engineering Vol 8. N 1,(2014) pp 111-115.
- [7] Z.Xue, L.Zhou,, B.Karney, D.Liu and P.Wang, *Primitive form Godunov type scheme for two-phase homogeneous water hammer flows*, American Society of Civil Engineers.Vol 146 No 4 (2020).
- [8] H. Chaudhry, M. Murty, S. Bhallamudi , S. Martin, M. Naghash, *Analysis of Transient Pressures in bubbly Homogeneous gas-liquid Mixtures*, ASME J. Fluids Eng. (1990).
- [9] E. Hadj Taieb, L. Lili, *Validation of Hyperbolic model for water-hammer in Deformable pipes*, ASME J. Fluids Eng.(2000).
- [10] E. Hadj taieb, L. Lili, *Transient Flow of Homogeneous gas Liquid Mixtures in Pipelines*, International Journal of Numerical Methods for Heat and Fluid Flow 8 No 3, (1997).
- [11] M.Ouzi, B.Bahrar, M. Tamani, , *Modeling Two-Phase Water Hammer Flow Using Shock-Capturing Scheme*, International Review of Mechanical Engineering, 15 (8), (2021), pp. 424-433.
- [12] F. Eleuterio Toro, *Riemann solvers and numerical methods for fluid dynamics*. Springer-Verlag Berlin Heidelberg. 2009.

Numerical investigation of the flow mechanisms in a fluidic oscillator and a sweeping jet actuators

A. SERRAR^{1*}, M. EL KHLIFI¹, A. KOURTA²

¹Hassan II University of Casablanca, Faculty of Sciences and Techniques, BP 146, Mohammedia 28806, Morocco

²University of Orléans, INSA-CVL, PRISME, EA 4229, 45072, Orléans, France

* abderrahime.serrare-etu@etu.univh2c.ma

Abstract

We present a detailed numerical study of the characteristics of a fluidic oscillator and a sweeping jet operating under the same working conditions. Unsteady flow analysis was performed using two-dimensional Reynolds-averaged Navier-Stokes equations with a realizable $k - \epsilon$ turbulence model. This study focuses on visualizing the evolution of the flow field associated with each actuator in order to elucidate the mechanism of the jet oscillation in order to use them for separation turbulent flow control. When the injected fluid is swept sideways, the features and dimensions of the vortices formed in the mixing chamber and feedback channel (FBC) change with time. The sweeping jet's frequencies are higher than those produced by the fluidic oscillator, which may help to improve aerodynamic performance while lowering power supply costs.

Keywords: *Sweeping jet, Fluidic oscillator, Actuators, Turbulent flow control*

1. Introduction

Nowadays, in the aeronautics field, one of the main objectives that settle the different lines of investigation and research around the world, is the maximisation of the flight efficiency, rather than the velocity or power, as it had been some years before. A way to accomplish this objective of improvement is to use flow control devices (passive or active) in different aerodynamic body surfaces of transport vehicles (airplane or road vehicle) in order to control the boundary layer separation or the laminar to turbulent transition and hence to improve the aerodynamic performances. This is why the fluidic oscillators become more and more attractive due to its design simplicity and the fact that they have no moving parts [1]. Cattafesta and Mark [2] presented a detailed review of fluid oscillators for active flow control. Pandey and Kim [3] carried out simulations of the internal flow of a sweeping jet and showed that the inlet width had a significant effect.

The primary goal of this study was to analyze the dynamic properties of the two actuators that were chosen: the fluidic oscillator and the sweeping jet.

Through numerical research, we focused in particular on the oscillation frequency and the jet oscillation mechanism.

2. Computational method

2.1 Geometrical model

Fluidic and sweeping jet oscillators are devices which emit a spatially oscillating jet solely based on the internal flow dynamics. The basic principle is depicted in Figure 1. A continuous supply of pressurized fluid at the inlet forms a jet which enters the mixing chamber (MC). It attaches to one chamber wall side (left or right) due to the Coanda effect. Each internal attachment causes a change in the deflection of the external jet. Part of a fluid goes into the feedback channel at the same side as can be seen in figure 1. Arriving at the inlet nozzle, it feeds a separation bubble located between the wall and the jet. The bubble grows and pushes the jet over the opposite side. The same process is repeated at this side. More details of this sustained mechanism is provided by Serrar et al. [4]. In addition, the geometrical dimensions of the actuators are presented in table (1).

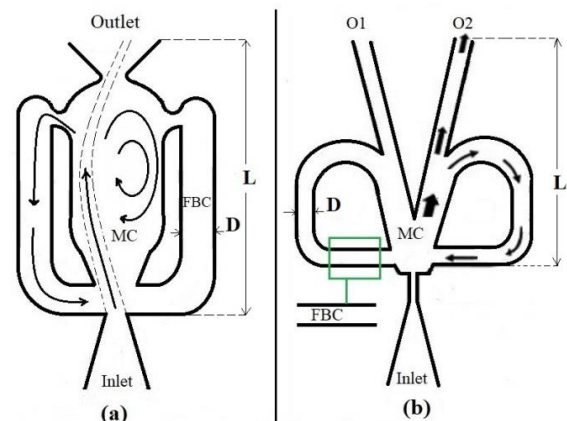


Figure 1: (a) Sweeping jet, (b) Fluidic oscillator

Actuator	Size (mm)				
	Inlet	Outlet	O1=O2	D	L
Fluidic oscillator	9	-	0.5	1.26	35
Sweeping jet	7.6	11,396	-	3.8	26.03

Table : Geometric sizes of the actuators

2.2 Methodology and Boundary condition

All simulations were carried out using Ansys Fluent 19. A second order implicit discretization in time has been adopted for the 2D unsteady calculation. The flow equations were discretized by a second-order upwind scheme. Spatial gradients were reconstructed by a least-square cell-based method. The time step was set as 10^{-6} s. The density-based solver is used to be able to simulate a compressibility effects (high-speed flows). The system of equations include the energy equation resulting in a single matrix problem to solve. Compressibility and viscous dissipation are also properly integrated into the model. The equations solved are the two-dimensional Reynolds-averaged Navier-Stokes equations with a realizable $k - \epsilon$ turbulence model.

At the inlet of the fluidic oscillator and the sweeping jet, an absolute total pressure value of 0.25 MPa and a temperature of 298 K were imposed. At the outlets O1 and O2 of the fluidic oscillator and at the outlet of the sweeping jet, an absolute static pressure of 0.1 MPa and a temperature of 298 K were imposed. The outer and inner sides of the oscillators were considered as walls. The operating fluid was air, an ideal gas with the following properties: molar mass $M = 28.9$ g/mol, specific heat capacity at constant pressure $c_p = 1005$ J/kg.K, dynamic viscosity $\mu = 1.8 \cdot 10^{-5}$ Pa.s.

3. Results and discussion

Figures (2,a1) and (2,b1) show the complete attachment of the jet at the left end, and figures (2,a2) and (2,b2) at the right end in the mixing chamber for two operating times of these oscillators, which is beneficial for the formation of the periodic oscillating jet. When the main jet adheres to one of the inner walls of the mixing chambers, it can induce less consumption of the jet momentum and a lower pressure loss inside the oscillators.

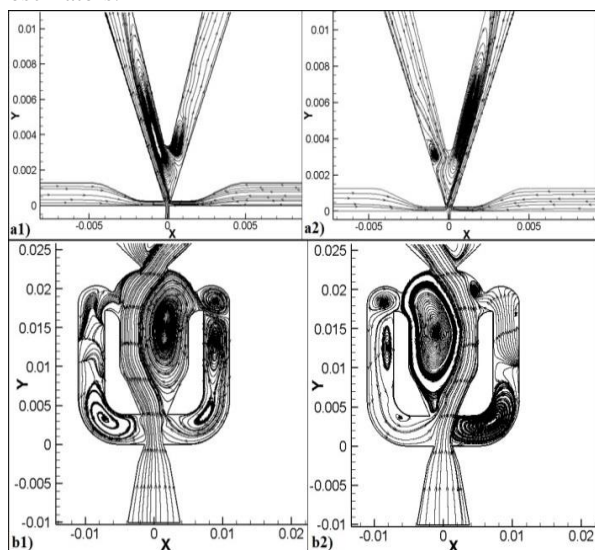


Figure 2: Instantaneous streamlines
a) Fluidic oscillator, b) Sweeping jet

Figure 3 shows the Mach number contours for two operating times of the two oscillators. When the jet completely adheres to one of the left or right mixing chamber walls, the flow velocity reaches the sound speed of the fluidic oscillator and the Mach number is then equal to 1. In the case of the sweeping jet, when the jet deflects from the left to the right and attaches completely to the wall of the mixing chamber, the Mach number becomes greater than or equal to 1. This number logically increases until it reaches the value of 1.3 at the level of the throat. It can be stated that the Coanda effect helps to generate the pressure difference inside the mixing chamber, to tilt the incoming jet [4].

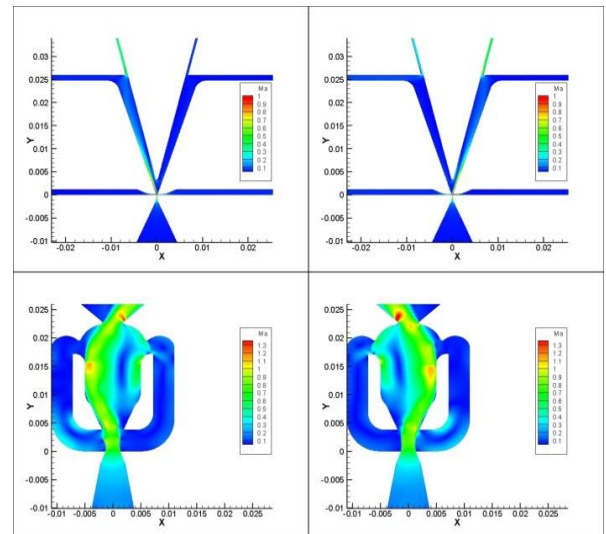


Figure 3: Mach number contours within the oscillators illustrating the jet switching process

At the outlets, a Fast Fourier Transform (FFT) of the oscillating jet mass flow was performed for the fluidic and sweeping jet oscillators [4]. The dominant jet frequencies are reported on table (2).

Actuators	Dominant Frequencies	
	Peak 1	Peak 2
Fluidic oscillator	298 Hz	-
Sweeping jet	1.67 kHz	16.7 kHz

Table 2: Dominant outlet jet frequencies (FFT analysis)

The frequency of the fluidic oscillator obtained is 298 Hz, while that of the sweeping jet is higher. It reaches 1.67 kHz. This means that the sweeping jet will perform better if applied to flow control for separated turbulent flows where higher frequencies are required. Actuators with higher frequency and low amplitude will in fact contribute to decrease the power supply of the actuator

and hence the cost. As has been pointed out by Gregory et al. [5], this result is connected to the change in oscillator size, geometry and fluid properties as observed. In this figure, another peak at higher frequency can be observed for this actuator (16.7 kHz). This frequency is related to a standing wave in the feedback channels. The comparison of the frequency extracted from the simulations with that obtained by the experiments of Wang et al. [6] shows a good agreement (8.7% difference). The difference observed in the present case can be explained by the imposed symmetry of the computational domain geometry.

4. Conclusion

The internal flow in a fluidic oscillator and a sweeping jet were investigated in this study using 2-D unsteady simulations to regulate high Reynolds number separated flows. Unsteady compressible flow analysis was performed using two-dimensional Reynolds-averaged Navier-Stokes equations with a realizable $k - \varepsilon$ turbulence model.

The pressure difference at both ports determines the flow direction of the fluid in the FBC. Because of the interaction with the fluid in the FBC, the main jet is pushed to switch and connect to the left and right side walls of the mixing chamber. After interacting with the main jet, the fluid pouring out of the FBC generates distinct vortices in the mixing chamber depending on the flow direction.

One important result of the present study is related to the oscillations frequencies obtained for the two actuators studied. The sweeping jet's frequencies were higher than those produced by the fluidic oscillator. This property is promoted to apply fluidic actuators for separated turbulent flow control. This can contribute to improve aerodynamic performance with low power supply costs.

References

- [1] F. Ostermann, R. Wozidlo, C.N. Nayeri, C.O. Paschereit, *Properties of a sweeping jet emitted from a fluidic oscillator*, Journal of Fluid Mechanics (2016) pp. 216-238.
- [2] L. Cattafesta, S. Mark, *Actuators for active flow control*, Annual Review of Fluid Mechanics (2011) pp. 247-272.
- [3] R.J. Pandey, K.Y. Kim, *Numerical modeling of internal flow in a fluidic oscillator*, Journal of Mechanical Science and Technology (2018) pp. 1041-1048.
- [4] A. Serrar, M. El Khlifi, A. Kourta, *Characterisation and comparison of unsteady actuators: a fluidic oscillator and a sweeping jet*, International Journal of Numerical Methods for Heat and Fluid Flow (2021), <https://doi.org/10.1108/HFF-07-2021-0474>
- [5] J.W. Gregory, J.P. Sullivan, G. Raman, S. Raghu, *Characterization of the microfluidic oscillator*, AIAA J (2007) pp. 568-576.
- [6] S. Wang, A. Batikh, L. Baldas, A. Kourta, N. Mazellier, *On the modelling of the switching mechanisms of a coanda fluidic oscillator*, Sensors and Actuators A: Physical (2019), <https://doi.org/10.1016/j.sna.2019.111618>

Modélisation de la prévision des poussières atmosphériques et étude de cas

A. Ait Hra^{1,2*}, S. Mordane², F. Hdidou^{1,2}, Z. Sahlaoui^{1,2}

1 LPPSMM, Faculté des Sciences Ben M'Sik, Université Hassan II, abdelaziz.aithra@gmail.com

2 CNRM, Direction Générale de la Météorologie, sahlaoui@yahoo.fr/hdidou@gmail.com

Résumé

Dans ce travail, nous allons décrire et évaluer le NMMB/BSC-Dust, un modèle de cycle des aérosols de poussière utilisé largement par les prévisionnistes de la Direction Générale de la Météorologie. Ce modèle à échelle globale est destiné à fournir des prévisions météorologiques de poussière de courte et moyenne échéance. Cette article décrit les paramétrages utilisés dans le modèle pour simuler le cycle de la poussière, y compris les sources, le transport et le dépôt. Nous étudions une situation type d'une violente tempête de poussière en comparant les prévisions du modèle avec les observations en surface (AERONET : AErosol RObotic NETwork) et en altitude (Satellite MSG d'EUMETSAT : European Organisation for the Exploitation of Meteorological Satellites).

Mots clefs : *poussières atmosphériques, aérosol, AOD, NMMB/BSC-Dust, paramétrisation.*

1. Introduction

Les tempêtes de poussière sont un type de temps désastreux. Elles intéressent principalement les zones désertiques et génèrent des quantités importantes de poussières dans l'atmosphère. Les sources des poussières atmosphériques les plus importantes et les plus persistantes sont situées dans l'hémisphère Nord, principalement dans la "ceinture de poussière" qui s'étend de la côte ouest de l'Afrique du Nord à la Chine, en passant par le Moyen-Orient, l'Asie centrale et l'Asie du Sud.

Les particules de poussière soulevées se mélangent verticalement, atteignant jusqu'à plusieurs kilomètres, d'où elles sont transportées sur des distances de milliers de kilomètres par des vents forts en altitude.

Les modèles de poussière prévoient principalement l'émission de poussière, leur transport dans l'atmosphère et leur dépôt. Au cours des deux dernières décennies, plusieurs modèles de cycle de poussière ont été développés et couplés avec des modèles de prévisions météorologiques à court et moyen terme ou avec des modèles climatiques.

Dans ce travail, nous présentons et évaluons le NMMB/BSC-Dust, un modèle de poussière atmosphérique multi-échelle en ligne conçu et développé au Barcelona Supercomputing Center (BSC). Nous présentons une étude de cas pour montrer la performance de ce modèle dans la prévision du phénomène étudié.

2. Le modèle multi-échelle non hydrostatique de NMMB

Le modèle multi-échelle non hydrostatique NMMB est un modèle atmosphérique unifié pour une large gamme d'échelles spatiales et temporelles. Son noyau dynamique non hydrostatique unifié permet des simulations régionales et mondiales. Ce modèle a été développé dans le cadre de modélisation du système terrestre (ESMF : Earth System Modeling Framework) des NCEP (National Centers for Environmental Prediction) en suivant la philosophie de modélisation générale du modèle régional non hydrostatique à méso-échelle WRF (WRF-NMM : Weather Research and Forecasting) [1].

Les équations primitives du modèle prennent leurs origines des lois physiques agissant sur un fluide en mouvement. Elles s'écrivent comme suit :

- L'équation de continuité :

$$\frac{d(\frac{\partial P}{\partial \eta})}{dt} = -\frac{\partial P}{\partial \eta} \left(\text{div}(\vec{V}) + \frac{\partial \eta}{\partial \eta} \right) \quad (1)$$

avec P la pression hydrostatique, \vec{V} est la composante horizontale du vecteur vent, et η la cordonnée verticale hybride suivant la pression de surface.

- L'équation du mouvement sur le plan horizontal :

$$\frac{d\vec{v}}{dt} = -2\vec{\Omega} \wedge \vec{V} - \vec{\nabla} \phi - RT\vec{\nabla} \log(P) + \vec{F}_{\vec{v}} \quad (2)$$

avec ϕ le géopotential, $\vec{\Omega}$ la vitesse angulaire de la rotation de la terre, T la température, R la constante des gaz parfaits et $\vec{F}_{\vec{v}}$ correspond à l'effet de la partie physique sur le vent (incluant le frottement).

- L'équation hydrostatique :

$$\frac{\partial P}{\partial z} = -\rho g \quad (3)$$

ρ correspond à la masse volumique de l'air.

- L'équation thermodynamique :

$$\frac{dT}{dt} = \frac{RT}{C_p P} \omega + F_T \quad (4)$$

$\omega = \frac{dP}{dt}$ la dérivée totale de la pression hydrostatique, C_p la chaleur spécifique à pression constante de l'air, F_T la contribution de la partie physique à la température (incluant le rayonnement, la convection...etc.).

- L'équation de l'humidité spécifique :

$$\frac{dq}{dt} = F_q \quad (5)$$

q étant l'humidité et F_q la contribution de la partie physique à l'humidité.

La partie physique correspond aux processus physiques non traités par la dynamique, c'est à dire par une

modélisation d'une atmosphère adiabatique et sans frottement.

3. Le modèle de poussière : BSC-Dust

Le BSC-Dust est un modèle de cycle d'aérosols de poussière intégré au modèle NMMB, il résout l'équation du bilan massique de la poussière en tenant compte des processus suivants : (1) génération de poussière et soulèvement par le vent de surface et la turbulence, (2) advection horizontale et verticale, (3) diffusion horizontale et transport vertical par turbulence et convection (4), dépôt sec et sédimentation gravitationnelle et (5) élimination humide qui comprend le balayage dans et sous les nuages des nuages convectifs et stratiformes [2].

Les émissions d'aérosols désertiques suivent fondamentalement un processus à effet de seuil. En effet, il a été constaté que les particules de poussière sont mises en mouvement quand la vitesse du vent en surface atteint ou dépasse une valeur minimale, appelée « vitesse seuil ».

3.1 Paramétrisation de la vitesse de friction seuil

Le flux massique des particules de saltation G dépend du surplus de la vitesse de friction u^* par rapport à la vitesse de friction seuil u_{*t} , c'est-à-dire : les émissions sont plus fréquentes lorsque u_{*t} est dépassée. Les quantités soulevées sont d'autant plus importantes que la vitesse de friction u^* est grande devant la vitesse seuil u_{*t} . Le seuil d'érosion est principalement fonction du diamètre des grains de poussières, de la rugosité de la surface et de l'humidité du sol. D'où l'expression suivante :

$$u_{*t}(D_p, Rug, w, \dots) = u_{*t}(D_p) f_R(Rug) f_w(w) \quad (6)$$

où $u_{*t}(D_p)$ exprime la dépendance de la vitesse de friction des diamètres des grains de poussières D_p , f_R et f_w sont des fonctions de pondération de l'influence de la rugosité de la surface et de l'humidité du sol.

3.2 Flux horizontal de saltation

Le flux horizontal représente la masse des particules traversant par unité de temps une surface verticale perpendiculaire au sens de l'écoulement de hauteur infinie et de largeur unité, il est essentiellement constitué de particules se déplaçant en saltation et en reptation.

La théorie de White (1979) [3] est utilisée pour estimer le flux massique horizontal G . Cette théorie suppose une linéarité entre le flux massique des particules en saltation et le surcroît de friction de vent $u_* - u_{*t}$. Cette supposition est largement vérifiée par les modèles de saltation microphysiques aussi bien que par les études de tunnel de vent.

Elle est donnée par la formule suivante :

$$G = \frac{C_s \rho u_*^3}{g} \left(1 - \frac{u_{*t}}{u_*}\right) \left(1 + \frac{u_{*t}^2}{u_*^2}\right) \quad (7)$$

Avec $C_s=2.61$, ρ est la densité de l'air, u^* est la vitesse de friction du vent, g est l'accélération de la pesanteur et u_{*t} est la vitesse de friction seuil

4. Flux vertical

Le flux horizontal de saltation G est converti en un flux vertical F d'aérosols par l'introduction de l'efficacité α appelée efficacité de production des particules d'aérosols. Pour des teneurs en argile comprises entre 0 et 20%, Marticorena et Bergametti [4] ont ajusté une relation empirique pour α qui est fonction de la teneur du sol en argile :

$$\alpha = \frac{F}{G} = 100 \exp[(13,4M_{clay} - 6,0)\ln 10] \quad (8)$$

Où M_{clay} est la fraction massique de l'argile.

4.1 Sédimentation et dépôt sec

La sédimentation ou dépôt gravitationnel est le processus d'élimination le plus efficace pour les gros aérosols. Nous résolvons implicitement la sédimentation pour le rapport de mélange de poussière (χ) du haut ($L=1$) au bas de la colonne ($L=LM$) sur une période dt comme suit :

$$\chi_L^{n+1} = \frac{\chi_{L-1}^{n+1} dt v_g}{\Delta Z_{L-1}} + \chi_L^n \quad (9)$$

où n est l'indice de temps, ΔZ est la profondeur de couche, L le numéro de couche et v_g est la vitesse de sédimentation gravitationnelle à chaque couche, qui est calculée selon l'approximation de Stokes-Cunningham :

$$v_g = \frac{d^2 g (\rho_k - \rho_a) C_c}{18 \nu}$$

Où d est le diamètre de la poussière, ρ_k est la densité de la poussière, ρ_a est la densité de l'air et g est la constante gravitationnelle, et C_c le facteur de correction de Cunningham qui explique la résistance réduite de la viscosité.

4.2 Balayage humide et mélange convectif

Le piégeage humide de la poussière par les précipitations est calculé séparément pour les précipitations convectives et à l'échelle de la grille (stratiformes). Il représente le procédé le plus efficace pour le dépôt des plus petites particules de poussière. Le modèle comprend des paramétrisations pour le balayage dans les nuages, c'est-à-dire le processus par lequel les particules tombent après le balayage par nucléation ; et pour le balayage sous-nuage ou sous les nuages, c'est-à-dire le processus par lequel les particules sont lessivées par collecte par précipitation. Ces processus sont schématisés est traitée suivant la paramétrisation de Pruppacher et Klett [5].

5. La situation météorologique étudiée

La situation du 23 Février 2020 a connu de forte tempêtes de sable sur les provinces Sud du Maroc ainsi que sur les Iles Canaries. Nous évaluons dans cette section la performance du modèle BSC-Dust en comparant ses sorties avec les données de l'observation.

Cette situation a été caractérisée par la présence d'un bas géopotential à 500 hpa avec un flux de sud à sud-est. A la surface des rafales de vents assez fortes de l'ordre de 36 nœuds (~72 km/h) ont été observées. Cette situation a généré des tempêtes de sable spectaculaires affectant la

majorité des provinces Sud Marocaines, et touchant par la suite les Iles Canaries.

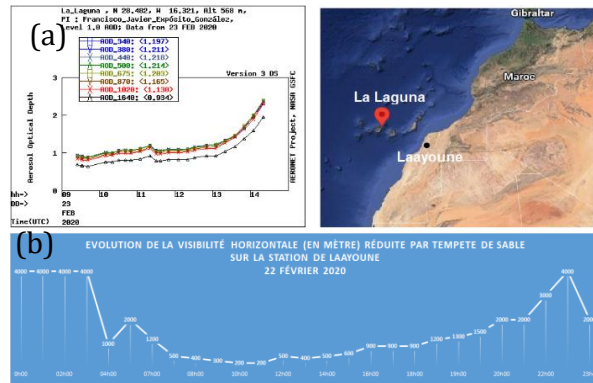


Figure 1 : (a) Evolution de l'épaisseur optique des aérosols (AOD) sur les Iles Canaries, et (b) l'évolution de la visibilité horizontale associée aux tempêtes de sable sur la station météorologique de Laayoune pendant la journée du 23-02-2020.

La figure 1 illustre ce phénomène par la réduction de la visibilité horizontale à des niveaux très bas (200 mètres) avec présence des poussières dans l'atmosphère dans la ville de Laayoune (figure 1.b). Ce phénomène a débuté la matinée pour atteindre les Iles Canaries l'après-midi. La figure 1.a met en évidence l'augmentation de la profondeur optique AOD (Aerosol Optical Depth) dans une station de mesure du projet AERONET (<https://aeronet.gsfc.nasa.gov/>) placée sur les Iles Canaries. L'AOD mesure l'extinction de la lumière par effet de présence des aérosols dans l'atmosphère. Cette augmentation montre l'arrivée des tempêtes de sable dans cette région.

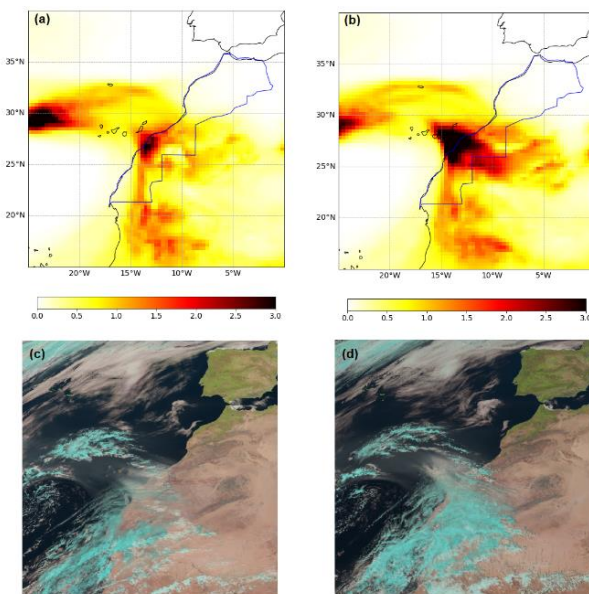


Figure 2 : Prédiction de l'AOD par le modèle BSC-Dust le 23-02-2020 à (a) 12h00 et (b) 15h00, et les images satellitaires observées pendant les mêmes périodes (c) et (d) respectivement.

La figure 2 montre la performance du modèle *BSC-Dust* dans la simulation et la prévision des tempêtes de sable avec une bonne précision. À 12h00, le modèle prévoit un nuage de sable sur les provinces de Laayoune, Tarfaya, TanTan et Guelmim (figure 2.a), avant de s'intensifier davantage tout en se déplaçant vers l'Ouest pour atteindre les Iles canaries vers 15h00 (figure 2.b). Cette configuration se réalise en réalité et est bien visible sur les images satellitaires correspondantes (figure 2.c et 2.d respectivement). Une légère surestimation est marquée à 15h00, le modèle prévoit plus que ce qui est observé, ceci peut être dû à la résolution assez lâche. Afin d'améliorer la prévision des tempêtes de sable sur notre région avec une résolution spatio-temporelle fine, nous devons intégrer la paramétrisation de la génération et transport des aérosols désertiques dans nos modèles opérationnels.

6. Conclusion et perspectives

Nous avons présenté le NMMB/BSC-Dust, un modèle de poussière multi-échelle en ligne préparé pour les domaines de simulation régionaux et mondiaux. Nous avons comparé ensuite ses sorties pour la situation du 23 Février 2020 avec les données d'observations. Trois types d'observations sont utilisées : visibilité horizontale, AOD du réseau de photomètres AERONET et image satellite MSG d'EUMETSAT. Nous avons montré que le modèle reproduit de manière significative la situation étudiée avec une légère surestimation.

Afin de fournir à nos prévisionnistes un modèle local de prévision de poussière sur des échelles spatio-temporelles très fine, il faut étudier la paramétrisation des poussières dans nos modèles AROME et ALADIN. Ensuite vérifier l'impact de l'inclusion de ces paramétrisations sur l'efficacité globale des modèles.

Références :

- [1] William C., et al: *A Description of the Advanced Research WRF Version 2*, NCAR TECHNICAL NOTE, June 2005.
- [2] Pérez, .C et al.: *Atmospheric dust modeling from meso to global scales with the online NMMB/BSC-Dust model-Part 1: Model description, annual simulations and evaluation*, Atmos. Chem. Phys., (2011) 11, 13001-13027.
- [3] White, B. R.: *Soil transport by winds on Mars*, J. Geophys. Res., (1979) 84, 4643-4651.
- [4] Marticorena, B. and Bergametti, G.: *Modeling the atmospheric dust cycle: 1. design of a soil-derived dust emission scheme*, J. Geophys. Res., (1995) 100, 16415-16430.
- [5] Pruppacher H., Klett J.: *Microphysics of Clouds and Precipitation*. Kluwer Academic Publishers, 2000.

Local hydrodynamic stability of the plane Poiseuille flow of an upper convected Maxwell fluid through a periodic channel

MOHAMED MADI¹, KHALID SOUHAR¹, ABDESSAMADE RAFIKI², SEYED MOHAMMAD TAGHAVI³

1. Laboratory of Energy Engineering Materials and Systems, ENSA ,Ibn Zohr University, Agadir, Morocco.
E-mail: madi.mohamed122@gmail.com
2. Department of Physics and Chemistry, Polydisciplinary Faculty of Ouarzazate, Ibn Zohr University, Morocco.
3. Department of Chemical Engineering, Laval University, QC, G1V 0A6, Quebec, Canada.

Abstract:

The linear stability of the plane Poiseuille flow of an upper convected Maxwell (UCM) fluid through a periodic channel is studied numerically. The system of dispersion equations, governing the stability analysis, is solved using a spectral method, which is based on the Chebyshev collocation discretization. We investigate the combined effects of periodic channel modulation and UCM fluid's elasticity on the local stability of the flow. This investigation proves the existence of dangerous solution corresponding to $x_c = 3\pi/2n$ for a wavenumber $n = 0.1$. Also, we found that the effect of the channel amplitude ϵ , has a stabilizing or destabilizing effect on the flow according to the channel section x and the elasticity number E or Weissenberg number w_e .

Keywords: *Viscoelastic flow; Upper convected Maxwell; Periodic channel; Spectral method; Linear stability analysis.*

1 Introduction

Viscoelastic flows through porous media are present in many disciplines of engineering research and technology, including methods for increased oil recovery in industry and blood flow modeling in Hemodynamics. This has motivated researchers to study viscoelastic flows using periodic geometries, which represent a class of mixed kinematic flows [1].

Hydrodynamic stability is a regular phenomenon from a practical point of view. These viscoelastic flows lose their stability if one or more control parameters change. The instability with high Reynold number Re is frequently studied for non-Newtonian fluids, this change is invariably followed by a modification of properties [2].

The most consistent viscoelastic fluid behavior law to describe these occurrences is Maxwell's law of fluid which allows for the effects of solvent viscosity to be taken into consideration. The linear instability of the plane Poiseuille flow of the non-Newtonian fluids in a periodic channel were investigated in the post critical zone [3].

Rafiki and al [4] have investigated, the combined

effects of periodic channel modulation and fluid elasticity for the second-order model and the second-grade model on flow stability, which used Chebyshev's spectral collocation method to solve the eigenvalue problem.

Introduction of the elasticity number $E = w_e/Re$, representing the ratio of fluid relaxation time to a viscous diffusion time based on the half height of the channel, The elastic effects on flow instabilities can be classified into two flow regimes in the plane ($Re - w_e$), the purely elastic regime ($Re \equiv 0$), and the inertial regime ($Re \gg 0$) [5].

In the present work, we investigate the local linear hydrodynamic stability of UCM fluid Through a periodic channel flow explored in the inertial regime. Our study focuses on the combined effect of periodic modulation and fluid elasticity on this flow stability. We examine the impact of geometric and rheological parameters using the Chebyshev spectral method based on Gauss-Lobatto collocation points to solve the eigenvalue problem [6].

2 Formulation of the problem

We investigate our study in a dimensionless system, we consider the Poiseuille flow of an incompressible viscoelastic fluid between the periodic walls of the channel, defined by the equation 1 in a dimensionless Cartesian coordinate system (x, y) :

$$y_w = \pm(1 + \epsilon \sin(nx)) \quad (1)$$

Where ϵ is the amplitude of the channel walls, n is the wavenumber.

The equations governing the fluid motion in a dimensionless form and the corresponding boundary conditions are written as :

$$\begin{cases} Re \left(\frac{\partial U}{\partial t} + U \cdot \nabla U \right) = -\nabla P + \nabla \cdot (\tau) \\ \nabla \cdot U = 0 \\ U(x, y = \pm y_w) = 0 \end{cases} \quad (2)$$

Where P is a dimensionless Pressure field, $U = (u, v)$ is the dimensionless velocity vector and τ is the stress tensor, defined by :

$$\tau + w_e \frac{D\tau}{Dt} = \frac{1}{Re} A_1 \quad (3)$$

With A_1 is the strain rate, and $\frac{D}{Dt}$ is the Jaumann

derivative. The control parameters are $Re = \frac{\rho u_0 H}{\mu}$; $w_e = \frac{\lambda_1 u_0}{H}$; $E = \frac{W_e}{Re}$ denote the Reynolds number, the Weissenberg number and the elasticity number, respectively.

3 Linear stability analysis

The present analysis is valid for small values of ϵ and it is based on the important hypothesis of the local parallel flow approximation, to establish the basic solution of the problem, we consider a stationary flow, taking into account of equation (Eq.3). The system of the solutions which satisfy the boundary condition are :

$$\begin{cases} \mathbf{u}_b = \left(\frac{1}{y_w} - \frac{y^2}{y_w^3} \right); \mathbf{v}_b = \mathbf{0} \\ \tau_{bxx} = \frac{8w_1 y^2}{Re y_w^6}; \tau_{byy} = \mathbf{0}; \tau_{bxy} = \frac{-2y}{Re y_w^3} \end{cases} \quad (4)$$

In this study, we considered a wavenumber n of the order of 10^{-1} , furthermore we worked in the context of the low amplitudes of the order of 10^{-2} , taking into account ($v_b \ll u_b$) and ($\tau_{byy} \ll \tau_{bxx}$) thus assuming that v_b and τ_{byy} are neglected.

Following the usual terminology of linear stability analysis, let the perturbed flow be written as a basic stable flow plus a time-dependent perturbation, assumed to be infinitesimal, we perturb the equilibrium solution to find the solution of the form :

$$(\mathbf{u}; \mathbf{v}; P; \tau_{ij}) = (\mathbf{u}_b + \mathbf{u}'; \mathbf{v}'; P_b + P'; \tau_{bij} + \tau'_{ij}) \quad (5)$$

Where u' , v' , P' the small perturbations of the velocity and pressure component. And τ'_{ij} are the small perturbations of normal and tangential stresses.

It is common to employ the current function $\psi(x, y, t)$, while working on a two-dimensional issue. The normal stresses τ'_{xx} and τ'_{yy} , as well as the tangential stresses τ'_{xy} , are decomposed into Fourier modes :

$$[\psi; \tau'_{xx}; \tau'_{yy}; \tau'_{xy}] = [\phi; \tau_{11}; \tau_{22}; \tau_{12}] \exp[i\alpha(x-ct)] \quad (6)$$

Taking into account the equations (Eq.5) and (Eq.6), the dispersion system of equations (Eqs.2,3) are written as :

$$\begin{cases} i\alpha[u_b(D^2 - y_w^2 \alpha^2) - y_w^2 u_b'] \phi - i\alpha y_w D^1 (\tau_{11} - \tau_{22}) - (D^2 + y_w^2 \alpha^2) \tau_{12} = i\alpha c (D^2 + y_w^2 \alpha^2) \\ y_w^2 Re [1 + i\alpha w_e u_b] \tau_{11} - w_e Re (i\alpha \tau'_{11} y_w^2 + 2i\alpha \tau_{11} y_w D^1 + 2\tau_{12} D^2) \phi \\ - 2 Re \tau_{12} u_b w_e y_w^2 - 2i\alpha y_w D^1 \phi = y_w^2 Re i\alpha w_e c \tau_{11} \\ y_w Re [1 + i\alpha w_e u_b] \tau_{22} - 2w_e \alpha^2 y_w Re \tau_{12} \phi + 2i\alpha D^1 \phi = i\alpha c Re y_w w_e \tau_{22} \\ y_w^2 Re [1 + i\alpha w_e u_b] \tau_{12} - w_e Re (i\alpha y_w^2 \tau'_{12} + \alpha^2 y_w^2 \tau_{11}) \phi - w_e Re \tau_{22} y_w^2 u_b' - (D^2 + y_w^2 \alpha^2) \phi = i\alpha Re w_e c \tau_{11} y_w^2 \end{cases} \quad (7)$$

Where $\frac{\partial^m}{\partial y^m}$ is a differential operator that replaces by D^m with $m = 1, 2$.

The boundary conditions for numerically solving of the dispersion system of equations (Eq.7) are defined by :

$$\phi(\pm 1) = D^1 \phi(\pm 1) = \mathbf{0} \quad (8)$$

4 Numerical solutions

The equations and the associated boundary conditions describe completely the evolution of an infinitesimal

perturbation in time and space. It consists to reducing an initial value problem into an eigenvalue problem such that the equation system (7) is written as :

$$A \cdot \mathbf{f} = cB \cdot \mathbf{f} \quad (9)$$

Where :

$$\mathbf{f} = [\phi, \tau_{11}, \tau_{22}, \tau_{12}] \quad (10)$$

A and B are two operator matrices depending on α , Re , w_e .

The eigenvalue problem (Eq.9) is solved numerically using the spectral method based on Chebyshev polynomials evaluated in N Gauss-Lobatto collocation points.

In order to validate our code of calculation, the code is tested, for validation, with the results obtained by Sureshkumar [1] for a plane Poiseuille flow of an UCM fluid. Their results are in good agreement with ours presented in Table 1.

Table 1 : The variation of the critical values of Re and α for different elasticity number E at $\epsilon = 0$.

E [10^{-3}]	Present implemenation			Sureshkumar [1]		
	Re_c	α_c	w_e	Re_c	α_c	w_e
1	2310.62	1.30	2.31	2310	1.30	2.31
2	1721.47	1.40	3.44	1722	1.40	3.55
3	1985.32	1.20	5.95	1990	1.20	5.97

In Figure 1, we reproduced with precision the graph $\sigma_i = f(\sigma_r)$. This eigenspectrum is virtually the same as that reported in Figure 9 of Sureshkumar work [1].

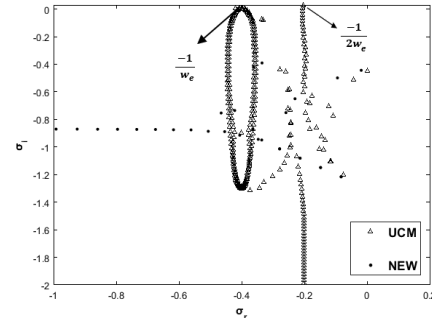


Figure 1 : The Newtonian and UCM fluids eigenspectrums for $N = 120$, $Re = 2013$, $\alpha = 1.3$, $w_e = 2, 31$ at $\epsilon = 0$, $x = 0$.

5 Results and discussion

After validating our code, we present the instability results describing the combined effects of E , w_e and the periodic channel modulation on the stability.

In Figure 2, we plot the variation of Re_c as a function of x for the two wall amplitudes $\epsilon = 0.01$ and $\epsilon = 0.03$. The stability thresholds in each region of the channel for the different UCM Fluid elasticity values, as shown for the two values of elasticity $E = 0.001$ and $E = 0.002$, Moving from the relaxed zone $x = 0$ to the enlarged section $x = \pi/2n$. The Re_c increases and the flow becomes more stable. From section $x = \pi/2n$ to the narrower section $x = 3\pi/2n$, the flow's stability decreases and then it regains stability at

section $x = \pi/2n$. Moreover, It can be observed that, for $E > 0.003$, Re_c decreases when the section are enlarged and vice-versa.

As a result, the most dangerous station, the most unstable flow area for $E < 0.002$ is $xc = 3\pi/2n$ and for $E = 0.003$ is $xc = \pi/2n$, which corresponds to the narrowest or largest sections when Re_c is minimal.

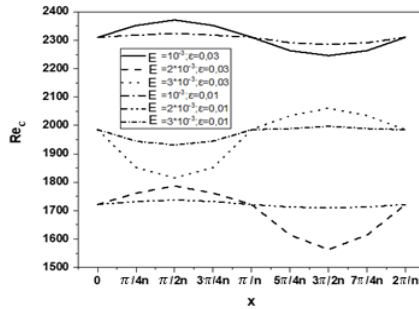


Figure 2 : The evolution of Re_c versus x for different values of E .

In Figure 3, we present the variation of Re_c as a function of ϵ . For the values ($E = 0.001$) and ($E = 0.002$), Re_c increases in the larger section and decreases in the narrower region. When $E = 0.03$, the Re_c drops in the broadest station while increasing in the narrowest section.

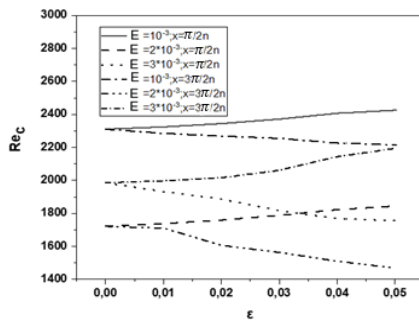


Figure 3 : The evolution of Re_c versus ϵ .

This result allows to conclude that the channel magnitude ϵ has a stabilizing or destabilizing effect on the flow according to the channel section x at different values of elasticity E .

Table 2 : Influence of E on Weissenberg numbers w_e for $\epsilon = 0.01$ at sections $x = \pi/2n$ and $x = 3\pi/2n$.

x	$\pi/2n$			$3\pi/2n$		
	Re_c	α_c	w_e	Re_c	α_c	w_e
$E \cdot 10^{-3}$						
1	2324.01	1.31	2.32	2285.34	1.34	2.28
1.5	1959.60	1.40	2.93	1907.89	1.39	2.86
2	1737.53	1.42	3.47	1710.16	1.40	3.42
2.5	1694.60	1.30	4.23	1669.48	0.98	4.17
3	1931.77	1.22	5.79	1996.99	1.23	5.99
3.5	1915.56	1.60	6.70	1848.12	1.61	6.47

We report in Table 2, the stability thresholds in each elasticity number $E > 0.0035$ corresponding to the

wall amplitude $\epsilon = 0.01$ of the UCM fluid for the two important sections are $x = 3\pi/2n$ and $x = \pi/2n$.

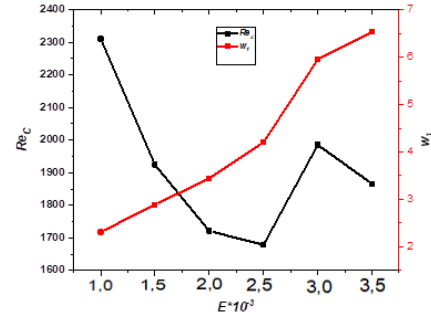


Figure 4 : Variation of Re_c and w_e versus the elasticity parameter E of the UCM fluid.

In figure 4, we show that Re_c decreases with the growth of w_e until $E < 0.0025$ and flow becomes more unstable.

6 Conclusion

Local linear stability of the plane Poiseuille flow of an upper convected Maxwell fluid was investigated, through a periodical channel, for multiple selected values of ϵ and E . The critical values of Re_c , α_c , w_e and c were determined numerically. We conclude that the elasticity of the UCM fluid has a combined impact of destabilizing or stabilizing influence on the local Poiseuille flow. Furthermore, the channel's amplitude has a stabilizing or destabilizing impact on the flow for UCM fluid with the existence of a dangerous station which corresponds to the narrowest section.

References

- [1] R.Sureshkumar, *Local linear stability Characteristics of viscoelastic periodic channel*, J. non-Newtonian Fluid Mech 97 (2001) 125-148.
- [2] H.J.Wilson, M. Renardy, *Structure of the spectrum in zero Reynolds number shear flow of the UCM and Oldroyd-B liquids*, J. non-Newtonian Fluid Mech 80 (1999) 251-268.
- [3] K.Zeeshan, R.Haroon, N.Sahib, K.Waris, *Analytical solution of UCM viscoelastic liquid with slip condition: The galerkin approach*, J. Hindawi mathematical problems in Engineering 7 (2020).
- [4] A.Rafiki, A.Hifdi, Y.Haddout, Y.Belkassmi, *Hydrodynamic stability of a periodic channel flow of viscoelastic fluid*, J. of theoretical and applied mechanics, Sofia 51 (2021) 301-316.
- [5] B.Sadanandan, R.Sureshkumar, *Numerical eigenspectrum of non-viscometric viscoelastic flows: results for the periodic channel flow*, J. non-Newtonian Fluid Mech 108 (2002) 143-161.
- [6] C.Canuto, M.Y.Hussaini, A.Quarteroni, T.A.Zang, *Spectral methods in fluid dynamics*, New york: Springer-verlag (1988).

Numerical Modelling of Cavitation in elastic Pipe

O. RKIBI¹, B. BAHRAR³, K. GUERAOU²

1. Equipe de Modélisation et Simulation en Mécanique et Energétique, ouafae.rkibi@gmail.com
2. Equipe de Modélisation et Simulation en Mécanique et Energétique, kgueraoui@yahoo.fr
3. Laboratoire Nanostructures et Matériaux Avancés, Mécanique et Thermo- fluide, bahrarb@yahoo.fr

Abstract –This study provides a theoretical and numerical modeling of transient vaporous cavitation in a horizontal pipeline, anchored to the upstream reservoir. The model approach is, essentially, based on that of the column separation model (CSM). The basic system of partial differential equations to solve is a hyperbolic type and adapts perfectly to the method of characteristics. This code, allows us, taking into account the unsteady part of the friction term, to determine at any point of the pipe, and at each instant, the average piezometric head, the average discharge and the change in volume of the vapor cavity. This study illustrates the effect of the presence of air pockets, resulting in cavitation, on the amplitude of the pressure wave. The calculation results are in good agreement with those reported in the literature.

Keywords: *Cavitation model, Column separation, Method of characteristics, Transient flow, Unsteady Friction model, Vapor pressure.*

I. Introduction

In transient pipe flows, transient vaporous cavitation occurs when the pressure drops below the saturating vapor of the fluid. Air bubbles appear in the fluid. These cavities can cause implosion and damage to the various components of the hydraulic system [1, 2]. There are two types of vapor cavitation, localized vapor cavitation (high vacuum) and distributed vapor cavitation (low vacuum). In some regions where evaporation is produced by pressure drop, the continuous medium is ruptured by creating columns separated from the fluid, this is the Column Separation Model (CSM). Several studies have been made using this model, notably the work of Anton Bergant and al [3], Paquette[4], Courant [5]. In this article we will study the discrete vapor cavity model (DVCM) with consideration of the shear stress of Darcy-Weisbach that reflects the unsteady friction term introduced by Zielke Vardy-Brown support and including an operating valve. The flow is axisymmetric and longitudinal gradients of velocity are assumed to be small compared to transverse gradients.

Given these assumptions, the averaged equations of the flow in a cross section of pipe can be written in traditionally a one-dimensional formulation, reflecting the relations of conservation of mass and momentum averaged over a section of a cross section, as hyperbolic system that is suitable for characteristic methods:

[6]. We then examine the evolution of the pressure head and the effect of velocity of the flow at the valve and at the midpoint of the pipeline, and the changes in the volume of air bubbles

II. Assumptions and basic equations

The basic equations are derived from the classical laws of conservation of mass, momentum for the fluid and the pipe wall in the case of isentropic transformations. We assume also that the fluid is barotropic Newtonian and in the expression of the viscous stress, the longitudinal velocity gradients are negligible compared to transverse gradients. The material of the pipe wall behaves like linear Geometrically, the pipe is assumed to be cylindrical horizontal, the cylindrical coordinates are (x, r, θ) . One of its end is rigidly attached to a reservoir upstream, which imposes a constant pressure and the other is on a fixe

In summary, the system of equations to be solved is:

$$\frac{1}{\rho_f} \frac{dP}{dt} + a^2 \frac{\partial V}{\partial x} = 0 \quad (1)$$

$$\rho_f \frac{dV}{dt} + \frac{\partial P}{\partial x} + \rho_f g \frac{\partial z}{\partial x} - \frac{4\tau_f}{D} = 0 \quad (2)$$

$$a = \left(\rho_f (1/\kappa + \alpha D_m E / e) \right)^{-1/2}$$

α : parameter characterizing the type of anchoring the pipe and which, in the case of a pipe anchored longitudinally, is written. [8]:

In these equations, $\rho_f, A_f, V, P, g, z, \tau_f, D$ are respectively, the density of the fluid, the inner section of the pipe, average pressure and velocity of the flow, acceleration of gravity, elevation of the pipe, friction stress and inside diameter of the pipe.

The deformation, ε , of the wall pipe was considered as the sum of two terms :

If we introduce the relationship which links the pressure to deformation:

$\varepsilon^e = \alpha(p(x,t) - p(x,0)) D_m J(0) / 2e$, in linear elastic case,

The shear stress $\tau_f = \tau_q + \tau_u$, [1,2], is composed of the quasi-stationary part τ_q related to the quasi-steady friction λ and depending on the flow regime and the relative roughness of the pipe which in the case of turbulent flow, is derived from the Colebrook equation:

$$\frac{1}{\sqrt{\lambda}} = -2 \log_{10} \left(\frac{2.51}{\text{Re} \sqrt{\lambda}} + \frac{\varepsilon/D}{3.71} \right) \quad (3)$$

Where Re is the Reynolds number; ε is the roughness of the pipe wall. The unsteady part τ_u linked to the convolution product is:

$$\tau_u = \frac{4\mu}{D S} \int_0^t W(t-t') \frac{\partial V(t')}{\partial t'} dt' \quad (4)$$

μ is the dynamic viscosity of water, W the weighting function related to Zielke's model.

By introducing the water hammer velocity [1,2]

$$C = \left(\frac{\partial \rho}{\partial P} + \frac{\rho}{S} \frac{\partial S}{\partial P} \right)^{\frac{1}{2}}$$

The system of equations (1) and (2) transformed along the characteristic curves of slopes

$$\left. \frac{dx}{dt} \right|_{C^{\pm}} = \pm C$$

We obtain at each calculation node at each instant the algebraic system:

$$P_{i,t} - P_{i-1,t-\Delta t} + \rho C ((V_u)_{i,t} - (V_d)_{i-1,t-\Delta t}) = \rho C \Delta t T_f \quad (4)$$

$$P_{i,t} - P_{i+1,t-\Delta t} + \rho C ((V_d)_{i,t} - (V_u)_{i+1,t-\Delta t}) = -\rho C \Delta t T_f \quad (5)$$

The continuity equation for the volume of the discrete vapor cavity is described by:

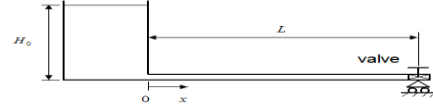
$$(\psi_g)_{i,t} = (\psi_g)_{i,t-2\Delta t} + (\Psi((V_d)_{i,t} - (V_u)_{i,t}) + (1 - \Psi((V_d)_{i,t-2\Delta t} - (V_u)_{i,t-2\Delta t}))) S_x 2\Delta t \quad (6)$$

III. Initial and boundary conditions

The initial conditions are those for steady flow and the balance for the pipe wall. The boundary conditions are in addition to the pressure imposed by the tank on the upstream and the instantaneous closing of a free valve on the downstream, the conditions of fluid-conduit interfaces requiring, in viscous flow, equal velocities and stresses as well as:

$$P(0,t) = \rho g H_0, \quad (12)$$

IV. Application and Results



1: diagram of the system studied

In this application, we consider a turbulent flow, for two steady-state flow values $V_0=0.3$ m/s and $V_0=1.4$ m/s. The flow is in the horizontal copper pipe anchored to the upstream to a tank filled with water and of height H_0 , ending at the downstream to a valve that closes abruptly. The parameters of the fluid and the pipe are summarized in table 1:

Tank height H_0 (m)	26
Internal diameter of the pipe (mm)	22.1
Pipe length (m)	37.2
Pipe thickness (mm)	1.63
Kinematic viscosity of water at 20°C (m^2/s)	1.11E-6
Poisson Coefficient	0.34
Modulus of elasticity by volume of water (GPa)	2.2
Young's modulus of copper (GPa)	120
Vapor pressure at 20°C (m)	0.23
Density of water at 20°C (Kg/m^3)	1000

The figure 1 and 2 show, respectively, the temporal variation of the water head pressure at the valve in the case of the classical water hammer without or with cavitation for the velocity value $V_0=0.3$ m/s corresponding to the steady-state flow.

The figures 3 and 4 show, in the same conditions, for a steady-state velocity $V_0=1.4$ m/s, the time evolution of the pressure at the valve without or with cavitation. In the results section, we illustrate an attenuation of the pressure wave due to the friction of the fluid on the pipe wall. This study shows that, for low velocity of the quasi-stationary flow, the phenomenon of the classical water-hammer is predominant, whereas for high speeds, the first over pressure; the most dangerous for the hydraulic system, is caused by the instantaneous closing of the valve, and the

other peaks are due to the collapse of the air pockets present in the fluid. In the case of cavitation, the presence of air pockets leads to greater attenuation of the pressure wave amplitude compared to the case of a conventional water-hammer. During cavitation, the formation of the air pocket, its volume increases, and by collapse, an over pressure takes place that can affect the internal surface of the pipe wall causing its wear.

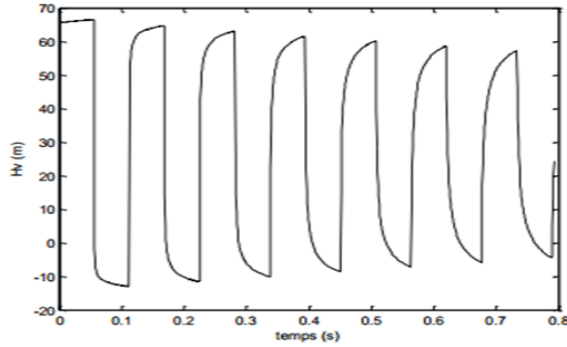


Fig. 2. Pressure in the middle of the pipe in the Case of the water hammer for $V_0=0.3\text{m/s}$

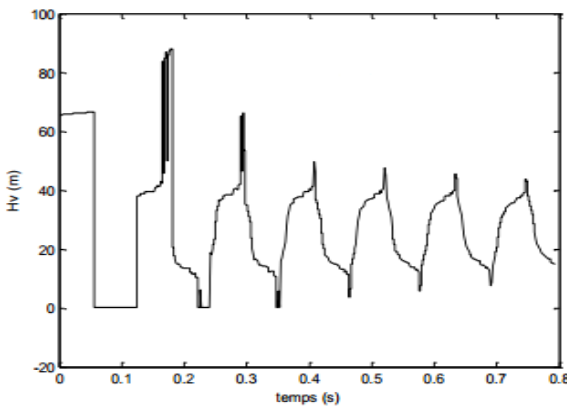


Fig. 3. Pressure in the middle of the pipe in The model (DVCM) for $V_0=0.3\text{m/s}$

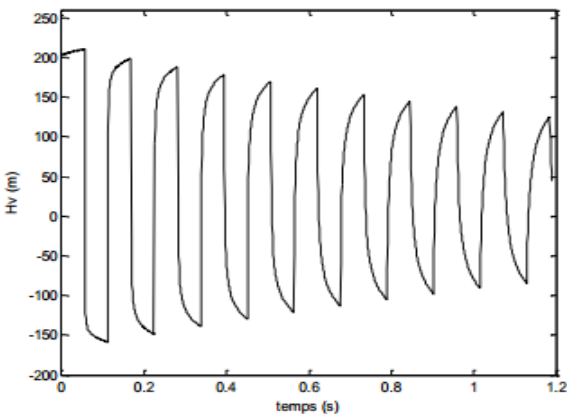


Fig. 4. Pressure in the middle of the pipe in The case of water hammer for $V_0=1.4\text{m/s}$

The table 2 shows of the maximum pressures and maximum volumes of the air pockets for both .

V_0 $=1.4\text{m/s}$	$V_{max} (\text{m}^3)$	1 st collapse (s)	$H_{V_{max}}$ (m)	t ($H_{V_{max}}$) (s)
N=32	4.093 E-5	0.4172	214.7657	0.0565=2 L/C

V_0 $=0.3\text{m/s}$	$V_{max} (\text{m}^3)$	1 st collapse (s)	$H_{V_{max}}$ (m)	t ($H_{V_{max}}$) (s)
N=32	7.8105 E-7	0.1261	87.9842	0.1817

Table II

Conclusion

This paper presents the study of transient cavitation flow in a horizontal straight copper pipe, and an initial data set under turbulent conditions was collected during transient events caused by rapid valve closure. This study, clearly, shows the possibility of a simple numerical treatment of cavitation. It has highlighted, in addition to the local nature of cavitation, the importance of the turbulent flow regime, the value of the velocity of the permanent regime of the flow, and the air pockets resulting of cavitation. In view of a practical level where the important is the knowledge of the maximum pressure likely to occur, it is logical to think that this computer code is a relevant understanding of the tool in transient cavitation flow in pipes.

References

- [1] Joukowsky, Y. ,*Über der hydraulischen Stoss in Wasserleitungen* [Over the hydraulic shock in water pipes]. Mémoires de l'Académie Impériale de St. Petersbourg, Bd. IX, N.5, 1900.
- [2] E.B. Wylie, V. L. Streeter, *Fluid Transients*. New York: Mac Graw-Hill, 1978.
- [3] . Bergant A., Vitkovsk'y J. P., Simpson A., Lambert M., Tijsseling A., Discrete vapor cavity model with efficient and accurate convolution type unsteady friction. term. Int. J. Kurokawa, Proceedings 23rd IAHR Symposium on Hydraulic Machinery and Systems, Yokohama, Japan, Paper 109 IAHR, (2006)
- [4] Y. Paquette, *Fluid-Structure Interaction and Cavitation Erosion*. PhD thesis. Universit'e Grenoble Alpes (2017), HAL Id: hal-02066203
- [5] A.R. Courant and D. Hilbert, *Method of Mathematical Physics*, Vol. 1 and 2, Interscience Publishers, N.Y., 1962.

Numerical modeling of Non-Newtonian fluid flow

N.Achak^{1,a*}, W.Rkibi^{1,b}, B.Bahrar^{2,c}, and K. Gueraoui^{1,d}

1 *Team of Modeling and simulation of Mechanical and Energetic, Faculty of sciences, Mohammed V University, Rabat, Morocco.

2 Team of Fluid Mechanics & Environment, ENSET Mohammedia, Condensed Matter Physics Laboratory, FST Mohammedia, Hassan II University Casablanca, Morocco.

a achaknawal3@gmail.com, c ouafae.rkibi@mail.com, d bahrarb@yahoo.fr, c kgueraoui@yahoo.fr

Abstract

The water hammer phenomena is a sudden change in flow, causing large pressure oscillations in the piping systems, In the present study an unsteady flow of a viscoplastic fluid, caused by instantaneous valve closing is examined. The governing equations of the problem are presented and then, they are solved by a numerical approach. The power law is used, to simulate the non-Newtonian behavior of the fluid. The Difference scheme and Characteristic methods are used for spatial discretization, and the fourth order Runge - Kutta scheme is based for temporal integration . The present study shows a remarkable deviation in pressure history.

Keywords : *transient pipe flow, Non-Newtonian fluid, viscoplastic fluid flow*

1. Introduction

The transient flows associated with the water hammer phenomenon, which means sudden changes in the flow conditions of the pressurized pipes, caused by the closing of the valve, are mainly found in natural and engineered systems, such as hydraulic system, human arterial network . This phenomenon can cause significant pressures leading to harmful forces [1]. In this study, we consider non-Newtonian shear-thinning fluid .For these types of fluids, viscosity can be described using the power law, or Cross models. The capacity of these models has been studied by several researchers, Bird and al. [2] with experimental and numerical studies., Chhabra and al.[3], notably Pinho and Whitelaw [4], Toms [5] .

In this type of fluid, the strain rate must be evaluated, and it requires a two-dimensional analysis to have the profile of unstable pressure and velocity. Several studies in this context have been developed by many researchers [6] to [7].Pezzinga gave a quasi two-dimensional model of the unsteady turbulent flow, of a pipeline network with better results than the 1D models [8] and [9]. Vardy and Brown [10] are also made significant studies in unsteady non-Newtonian pipe flows, particularly the modeling of fluids with time-dependent viscosities .

More recently, there is Wahba [11] who compared thinning and shear fluids using the power law model.

This article, studies an unstable pipe flow of a viscoplastic fluid, it can be considered as a new extension, of the classical model of Newtonian fluid in an elastic pipe.

After having carried out these simulation for the power law and the Cross models, there are alternative works [12] on the transients of flow fluids which can be proposed for future research. ex. Ahmadi and Keramat [12], [13], Soares and al. [14], Hadj-Taïeb [15]. Pezzinga and al.

And the transient in pressurized polymer pipes using a two-dimensional (2D) KelvinVoigt viscoelastic model [16], they deduced that viscoelastic models represent faster decay of pressure oscillations as well as velocity profiles. Brunone and al. [17] , Kim [18] present the pressure and energy dissipation, as well as the unstable friction in laminar transient flows, comparing their numerical results with those of the experiments. Meniconi and al. studied the turbulent flow of rapidly decelerating pipes, so they proposed a new approach to estimate energy dissipation as well as pressure decay [19] and [20], than other very recent studies are done [21], [22], [23]. In the present study, the non-Newtonian transient laminar flow, of pipes is simulated using the Power law model. To this object, the water hammer equations for non-Newtonian fluids are given, and solved by appropriate numerical methods based on the Finite Difference, Characteristics and Runge-Kutta .

2. Governing Equations

2.1 Basic equations

To derive unsteady flow equations for a non-Newtonian fluid some assumptions and simplifications to be stated in the following are made.

The flow is quasi-two dimensional. The term ‘quasi’ indicates that $v_\theta = 0$, $v_r = 0$ meaning that v_z is the only velocity component that varies along radial and axial directions. The convective terms are neglected

The system equations that can be describing the phenomenon

$$\begin{cases} \frac{1}{\rho a^2} \frac{\partial P}{\partial t} + \frac{\partial V}{\partial x} = 0 \\ \rho \frac{\partial V}{\partial t} = -\frac{\partial P}{\partial x} + T_p \end{cases} \quad (1)$$

$$\begin{cases} \frac{1}{\rho a^2} \frac{\partial P}{\partial t} + \frac{\partial v}{\partial x} = 0 \\ \rho \frac{\partial v}{\partial t} = -\frac{\partial P}{\partial x} + \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial \tau}{\partial r} \right) \end{cases} \quad (2)$$

where V is average velocity, P is presser head averaged over the cross-sectional area of flow, a is wave speed, ρ is density, t is time and T_p is steady friction

And v is the axial velocity, P is presser head, a is wave speed, ρ is density, t is time and τ is shear stress.

2.2 Non-Newtonian Fluid Equations

The shear stress of the fluid is :

$$\tau = \mu \dot{\gamma} \quad (3)$$

Where μ and $\dot{\gamma}$ are apparent viscosity and shear rate respectively.

The power law is described by the following equation [22] [23], with the fewest possible parameters.

$$\mu = m(\dot{\gamma})^{n-1} \quad (4)$$

where m and n are two empirical curve fitting parameters, also known the fluid consistency coefficients and the flow behavior index respectively. So, if n equals one and m is set to μ_0 , the Newtonian fluid is obtained. In our study, m is fixed to μ_0 , and quantity of n is selected.

2.3 Initial Conditions

The initial conditions corresponds to the steady state flow. These conditions can be written as:

$$\frac{\partial v}{\partial x} = 0 \quad (5)$$

$$\frac{\partial P}{\partial x} = -\frac{2\tau_0}{R} \quad (6)$$

2.4 Boundary Conditions

The velocity is set to zero after the valve closure, at the valve boundary. At the reservoir, a constant pressure head is associated. In contact, the flow boundaries with the pipe wall have zero velocity. These boundary equations can be reduced as follows:

$$v_{(r=R)} = 0 \quad (7)$$

$$v_{(\text{valve})} = 0 \quad (8)$$

$$H_{\text{reservoir}} = \text{cte} \quad (9)$$

3. Numerical method

The equations of the proposed unsteady flow system is solved using the Finite Difference and Characteristics

methods for spatial discitisation. The system of equations over time, is solved using a fourth-order Runge-Kutta scheme .

The systems (Eqs. 1) and (Eqs. 2) are given in matrix form (Eq.10)

$$\frac{\partial \{W\}}{\partial t} = -[B] \frac{\partial \{W\}}{\partial z} + \{c\} \quad (10)$$

with the unknown vector : $\{W\} = \{H \ V\}$ (11)

Where $\frac{\partial \{W\}}{\partial t} = f(W, t)$ (12)

$$\{W\}^{N+1} = \{W\}^N + \frac{1}{6} [\{K_1\} + 2\{K_2\} + 2\{K_3\} + \{K_4\}] \quad (13)$$

4. Numerical results

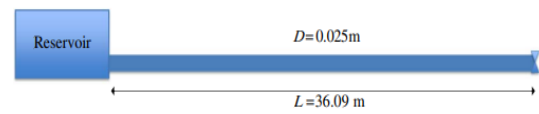


Fig 1. Reservoir-pipe-valve system to run fluid transients tests [22]

In the present study the test is initiated by a sudden closing of the valve, which causes excision of the fluid flow to the valve, creating oscillations of pressure and velocity, that propagate along the pipe, a representation of the head pressure over time in midpoint and valve is given (Fig. 1).

The copper pipe has an internal diameter of 0.025m and a length of 36.09 m, with a laminar flow conditions, the fluid used with high viscosity oils ($\mu = 0.03484 \text{ N}\cdot\text{s}/\text{m}^2$).

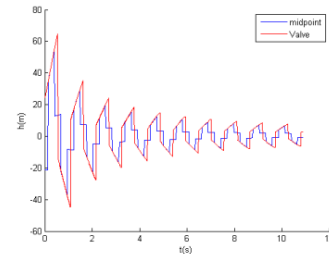


Fig.2 corresponds to the pressure heads.

Fig2. shows significantly the effect of the discrete leak on the pressure response at the valve and at the midpoint, in a simple hydraulique reservoir-pipeline-valve system as well as energy dissipation due to the effect of friction in time, on water hammer phenomenon, which has good correlation with literature results at valve and at midpoint .

5. Conclusion

In these study, the governing equations of water hammer phenomena, are numerically solved, Finite Difference and Characteristics scheme, were used for the discretization in space, the fourth order Runge-Kutta method was proposed also for the approximation in time.

The non-Newtonian power law was used, the pressure history shows a reduced pressure drop over time.

Références

- [1] E. B. Wylie, V.L.A. Streeter, L. Suo., *Fluid Transients in Systems*. Prentice Hall, Upper Saddle River (1993)
- [2] R. B. Bird, R.C. Armstrong, O. Hassager. *Dynamics of Polymeric Liquids*, Wiley Interscience Publication, Hoboken, 2nd ed., vol. 1,(1987)
- [3] R. P. Chhabra, J. F. Richardson. *Non-Newtonian Flow and Applied Rheology: Engineering Applications*, Elsevier Science, Oxford, 2nd ed., (2011)
- [4] F. T. Pinho, J. H. Whitelaw. Flow of non-Newtonian fluids in a pipe. *Journal of Non-Newtonian Fluid Mechanics*, vol. 34, no. 2. p. 129-144 ,(1990)
- [5] B. A. Toms. Some observation on the flow of linear polymer solutions through straight tubes at large Reynolds numbers. *Proceedings of 1st International Congress on Rheology*, vol. 2, p. 135-141. (1948)
- [6] M. S. Ghidaoui, , D.A. McInnis, D.H. Axworthy, , M. Zhao. A review of water hammer theory and practice. *Applied Mechanics Review*, (2005)
- [7] B. Brunone, B. Karney, M. Mecarelli , M. Ferrante., Velocity profiles and unsteady pipe friction in transient flow. *Journal of Water Resources Planning and Management*, vol. 126, no. 4, p. 236-244, (2000)
- [8] G. Pezzinga, Quasi-2D Model for Unsteady Flow in Pipe Networks. *Journal of Hydraulic Engineering*, vol. 125, no. 7, p. 676-685, (1999)
- [9] G. Pezzinga, Evaluation of Unsteady Flow Resistances by Quasi-2D or 1D Models. *Journal of Hydraulic Engineering*, vol. 126, no. 10. p. 778-785, (2000)
- [10] A. E. Vardy, J. M. B. Brown. Laminar pipe flow with time dependent viscosity. *Journal of Hydro informatics*, vol. 13, no. 4, p. 729–740, (2011)
- [11] E. M. Wahba. Non-Newtonian fluid hammer in elastic circular pipes: Shear-thinning and shear-thickening effects. *Journal of Non-Newtonian Fluid Mechanics*, vol. 198, no. 0, p. 24-30, (2013)
- [12] A. Keramat, A. S. Tijsseling, , Q. Hou, A. Ahmadi., Fluid structure interaction with pipe-wall viscoelasticity during water hammer. *Journal of Fluids and Structures*, vol. 28, p. 434– 455, (2012)
- [13] A. Ahmadi, , A. Keramat, Investigation of fluid–structure interaction with various types of junction coupling. *Journal of Fluids and Structures*, vol. 27, no. 7-8, p. 1123-1141, (2010)
- [14] A.K. Soares, D.I.C. Covas, N.J.G. Carriço., Transient vaporous cavitation in viscoelastic pipes. *Journal of Hydraulic Research*, vol. 50, no. 2, p. 228-235, (2012)
- [15] L. Hadj-Taïeb, , E. Hadj-Taïeb. Numerical simulation of transient flows in viscoelastic pipes with vapour cavitation. *International Journal of Modelling and Simulation*, vol 29, p. 206–213, (2009)
- [16] G. Pezzinga, B. Brunone, D. Cannizzaro, M. Ferrante, S. Meniconi, A. Berni, Two-Dimensional Features of Viscoelastic Models of Pipe Transients. *Journal of Hydraulic Engineering*, vol. 140, no. 8, p. 04014036, (2014).
- [17] B. Brunone, M. Ferrante, , M. Cacciamani., Decay of Pressure and Energy Dissipation in Laminar Transient Flow. *Journal of Fluids Engineering*, vol. 126, no. 6, p. 7, (2005)
- [18] S. Kim, Holistic Unsteady-Friction Model for Laminar Transient Flow in Pipeline Systems. *Journal of Hydraulic Engineering*, vol. 137, no. 12, p. 1649-1658, (2011).
- [19] S. Meniconi, , H. Duan, B. Brunone, , M. Ghidaoui, , P. Lee, M. Ferrante., Further Developments in Rapidly Decelerating Turbulent Pipe Flow Modeling. *Journal of Hydraulic Engineering*, vol. 140, no. 7, p. 04014028, (2014)
- [20] S. Meniconi, , B. Brunone, , M. Ferrante, C. Massari., Energy dissipation and pressure decay during transients in viscoelastic pipes with an in-line valve. *Journal of Fluids and Structures*, vol. 45, no. 0, p. 235-249, (2014)
- [21] E. M. Wahba, Runge–Kutta time-stepping schemes with TVD central differencing for the water hammer equations. *International Journal for Numerical Methods in Fluids*, vol. 52, no. 5, p. 571-590, (2006).
- [22] A. Majd, , A. Ahmadi, , A. Keramat, Investigation of non-Newtonian fluid effects during transient flows in a pipeline. *Journal of Mechanical Engineering*, 62(2), 105-115. (2016).
- [23] A. Khamoushi, , A. Keramat, , A. Majd, One-Dimensional Simulation of Transient Flows in Non-Newtonian Fluids. *Journal of Pipeline Systems Engineering and Practice*, 11(3), 04020019. (2020)

DEVELOPMENT OF MAGNETOACTIVE POLYMERIC MATERIALS

A. GÓMEZ¹, J. BERASATEGI¹, M.M. BOU-ALI¹

1. Mechanical and Industrial Production Department, Faculty of Engineering, Mondragon Unibertsitatea, Loramendi 4, Arrasate-Mondragón, 20500, Spain. agomezp@mondragon.edu

Abstract

Nowadays, it is well known the magnetostrictive property of some metallic materials. However, due to the high magnetic fields required, their fragility and the low deformations obtained, their application is being limited. To address these disadvantages, significant progress has been made in the development of magnetorheological elastomers, whose magnetostrictive properties are comparable to those of magnetostrictive materials, requiring lower magnetic fields. However, in order to promote the development and use of these materials, it is necessary to achieve higher deformations, in order to cover new application areas. In this context, a new generation of magnetoactive elastomers has been developed, which, in addition to the virtues of magnetorheological elastomers, exhibit a high deformation capacity in the presence of a magnetic field

Keywords: *Magnetostriction, Elastomers, Deformation, Magnetorheological*

1. Introduction

Magnetorheological elastomers (MREs) are within the family of magnetorheological smart materials, which can be controlled by the application of a magnetic field, [1]. Magnetorheological elastomers are composed of particles with magnetic properties dispersed in a polymeric medium. They are materials that offer great application possibilities in different fields, thanks to the elastomeric matrix (high elasticity, flexibility, strength, etc.) that allows their integration in flexible devices or devices with complex geometries.

In the presence of the magnetic field, the magnetic particles tend to align in the direction of the magnetic field, resulting in changes in the mechanical properties and magnetostriction of the material (Figure 1).

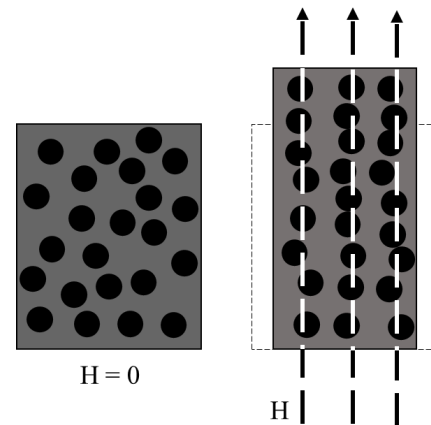


Figure 1. Effect of magnetic field on MREs

Due to their mechanical properties, ferromagnetic composites with an elastomer matrix have conquered different applications, for example, controllable vibration absorbers and damping components, noise barrier systems and sensors [2].

However, one of the properties of MREs that has generated most interest recently is magnetostriction, which is the change in shape of a body when it is subjected to a magnetic field (Figure 1).

The phenomenon of magnetostriction in EMRs can be explained as follows. Each iron particle has a magnetic moment due to dipole interaction with the adjacent particle. In the absence of an externally applied magnetic field, the directions of the moments corresponding to each iron particle (magnetisation vectors) are randomly aligned in such a way that the total sum of the vectors is zero. When a magnetic field is applied, the iron particles in the sample are influenced, and tend to align their magnetic moments in the direction of the applied field, inducing a dipole-dipole interaction between them. This interaction takes place in the polymeric matrix and leads to the displacement and rotation of the iron particles causing the change of dimension of the sample, known as magnetostriction. The magnetostriction obtained in MREs (3000 ppm) [3] is comparable to the maximum deformation obtained with the magnetostrictive material for excellence, Terfenol-D.

Until now, the magnetostrictive material has been conceived as a powerful actuator and transducer, although there have been many factors that have limited its applications: fragility, requirement of large magnetic fields to induce deformation.... The development of polymer matrix composites aims to alleviate each of these drawbacks through the introduction of a polymer material: The polymer improves the fracture toughness of the material and allows higher loads to be applied.

The introduction of non-magnetostrictive materials in the composite can significantly reduce the field strength needed to obtain large magnetostrictions.

Another advantage of composites is that they are more manageable than magnetostrictive materials and can be moulded into specific sizes and shapes, resulting in materials with greater durability. Research results confirm that magnetostrictive composites respond in the same way as monolithic materials, in terms of deformation response under the application of magnetic fields [4].

The capability of rapid and reversible deformation, although limited, has made these materials promising candidates for next-generation actuators [5], however, higher magnitude deformations need to be achieved to promote their development and application.

In order to maximise the deformability, advances in the development of a new generation of magnetoactive elastomers are presented below.

2. Experimental

Based on a polymeric matrix and ferromagnetic particles dispersed inside it, we have formulated a new generation of elastomers that show a high deformation under external magnetic excitation.

The particles and the polymeric liquid matrix in combination with other additives are mixed and cured in a mould with the desired shape. The ferromagnetic particles give the material the magnetic properties necessary for an active response to a magnetic field. On the other hand, the polymeric matrix together with the additives provide the necessary flexibility so that the material deforms when a magnetic field is applied and recovers its initial shape after the magnetic field is removed.

These new elastomers have been formulated in two different formats: on the one hand, thin elastomer layers

have been developed, and on the other hand, in tube shape (Figure 2).



Figure 2. Magnetoactive samples of different geometry. On the left tube shape sample, and on the right elastomer layer.

When a magnetic field is applied perpendicular to the elastomer support surface, this new material deforms by elevating two of its sides symmetrically, adopting a curved shape (Figure 3). Moreover, the degree of deformation is controllable by controlling the intensity of the applied magnetic field, thus allowing dynamic deformations to be obtained by modifying the intensity of the applied magnetic field.

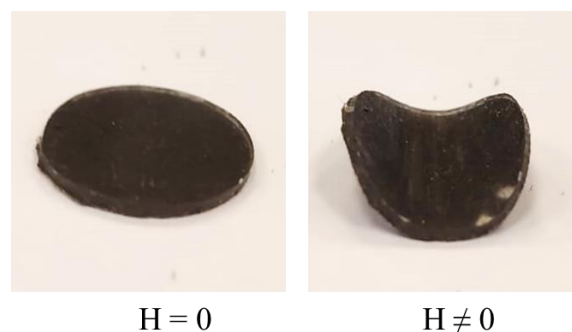


Figure 3. Magnetoactive film appearance without and with external magnetic field applied

In the case of the tubular format, when a magnetic field is applied in the direction parallel to the support surface, the material shows an instantaneous active response, joining

its two sides in the direction of the magnetic flux lines (Figure 4), almost closing the tube section.

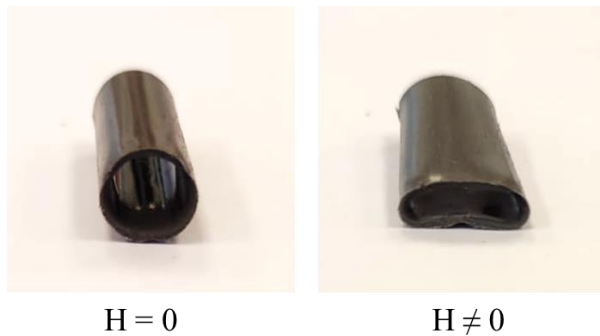


Figure 4. Magnetoactive tube appearance without and with external magnetic field applied

GOM ARAMIS 3D [6] camera systems have been used to analyse the deformation of materials (Figure 5 a). The measurement technique is based on the comparison of two images corresponding to the surface of a test sample. The first one corresponds to a reference state (no magnetic field) and the second one corresponds to a deformed state (under magnetic field). The surface of the sample must show a random pattern. In general, it is obtained by spraying black and white paints that must be thin enough so that the displacement of the dots matches that of the sample surface (Figure 5 b).

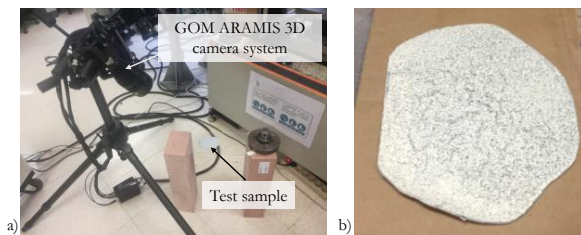


Figure 5. GOM ARAMIS 3D deformation measurement method. a) Set-up of the measurement system, b) magnetodeformable elastomer sample painted with a random pattern.

As shown in Figure 5, the sample is placed on a platform under which the magnetic field is applied using a permanent magnet. The camera is positioned so that the measurement area covers the entire sample and tilted to follow the displacement of the painted surface throughout the entire path.

With this method the displacement of the elastomer has been measured, achieving displacements of up to almost 6 mm (Figure 6).

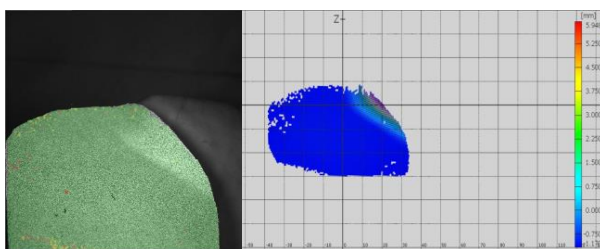


Figure 6. 3D scanning of the magnetodeformable elastomer using GOM ARAMIS 3D.

The GOM ARAMIS 3D camera system has made it possible to characterise the deformation capacity of the material without limitations regarding the intensity of the magnetic field. This method will make it possible to characterise the deformation of the sample as a function of the applied magnetic field, achieving control over the deformation of the material.

3. Conclusions

In this work has been demonstrated in both the sheet elastomers and the synthesised tubular elastomers, the change in geometry observed when the magnetic field is applied is very different from the usual magnetostriction that occurs in magnetorheological elastomers. In contrast to the magnetostrictive response of MR elastomers, whereby a change in dimensions occurs at the microscale, this new generation of elastomers, known as magnetoactive, has the capacity for large displacements. The possibility of measuring the deformation capacity of the material as a function of the geometry and the magnetic field will make it possible to control the latter according to the desired deformation. All these advances, both the development of a new magnetodeformable material and the possibility of obtaining a controlled deformation depending on the applied magnetic field, will lead to the development of practical applications based on these new materials.

References

- [1] Y. Li, J. Li, W. Li, and H. Du, *A state-of-the-art review on magnetorheological elastomer devices*, Smart Mater. Struct. vol. 23, no. 12, 2014.
- [2] A. G. Díez, C. R. Tubio, J. G. Etxebarria, and S. Lanceros-Mendez, *Magnetorheological Elastomer-Based Materials and Devices: State of the Art and Future Perspectives*, Advanced Engineering Materials, vol. 23, no. 6. 2021.
- [3] Ginder, J. M., Clark, S. M., Schlotter, W. F., & Nichols, M. E., *Magnetostrictive phenomena in magnetorheological elastomers*, International Journal of Modern Physics B, vol. 16, no. 17, 2002.
- [4] T. A. Duenas and G. P. Carman, *Large magnetostrictive response of Terfenol-D resin composites*, J. Appl. Phys., vol. 87, no. 9 II, pp. 4696–4701, 2000.
- [5] L. E. Faidley, Y. Han, K. Tucker, S. Timmons, and W. Hong, *Axial strain of ferrogels under cyclic magnetic fields*, Smart Mater. Struct., vol. 19, no. 7, 2010.
- [6] GOM mbH, ARAMIS Commercial software. Aramis 6.0.2; GOM mbH: Braunschweig, Germany, 2007.

EXPERIMENTAL ANALYSIS OF THE STABILITY OF MAGNETORHEOLOGICAL FLUIDS

A. GÓMEZ¹, I. ALCORTA¹, J. BERASATEGI¹, T. GOMEZ-ARENAS², M.M. BOU-ALI¹

1. Mechanical and Industrial Production Department, Faculty of Engineering, Mondragon Unibertsitatea, Loramendi 4, Arrasate-Mondragón, 20500, Spain. agomezp@mondragon.edu
2. Instituto de Acústica, CSIC, Calle Serrano 144, Madrid, 28006, Spain

Abstract

Nowadays, magnetic particles of micrometric size have been widely used for the formulation of magnetorheological (MR) fluids. However, in MR fluids stability problems appear after long waiting times, leading to the need of developing methods to measure the sedimentation rate of MR fluids.

In this work, an experimental method based on ultrasound for determining the stability of magnetic fluids has been developed. The results show that the new procedure has advantages over other techniques, since it has not limitation of concentration of particles and does not modify thermophysical properties of the fluid. This method has proved its effectiveness measuring the MR fluids but will be useful also to measure other non-magnetic colloids.

Keywords: *Gravitational stability, Settling, MR fluids*

1. Introduction

Magnetorheological fluids are fluids composed of micro- or nano-sized particles capable of change their physical properties when an external magnetic field is applied [1]. Their active behaviour under magnetic field conditions is due to the fact that these fluids are suspensions of magnetic particles in a carrier fluid, stabilised by a surfactant and other additives. In the absence of a magnetic field, the particles are randomly distributed in the fluid; however, when a magnetic field is applied, the particles tend to polarise and align themselves with the field, forming a chained fibrillar structure within a few milliseconds (Figure 1). Thus, the viscosity of these fluids increases as a function of the applied field strength. This increase is not unlimited, but eventually exhibits a gradual magnetic saturation.

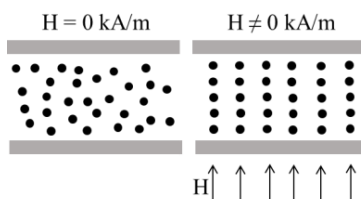


Figure 1. Effect of the magnetic field on MR fluids.

The success of Industry 4.0 depends on the successful development of smart components and materials that can incorporate the ability to power the IIoT into the DNA of the material itself, during its manufacture, assembly, transport and lifetime. In this context, magnetorheological fluids are a promising option for digital transformation in applications such as: damping, through active suspension systems for both machines and civil engineering, in bioengineering [2], in braking systems [3] and in machine tools, for example in clamping systems [4].

However, one of the characteristics that is limiting the development of MR devices is the low gravitational stability of the fluids. Due to the disparity in density between the particles and the carrier fluid, the particles tend to settle. After performing the synthesis of the MRFs, the entire sample has the original concentration, however, as time passes the fluid begins to settle, generating four clearly delimited zones in height. In the top, a zone is formed in which practically all of the carrier fluid is present, followed by a zone in which the original concentration is maintained. Further down, there is a region in which the concentration varies over time, and finally, in the bottom of the cylinder, most of the sediment is concentrated.

So, measuring the settling velocity of MR fluids is key to developing and promoting the use of MR devices, as once settling occurs the performance of the devices can be negatively affected.

Several methods have been developed to measure the stability of magnetorheological fluids, however, all of them have some limitation in terms of particle concentration, precision or feasibility for development. Others may also induce changes in the thermophysical properties of the fluid.

Optical methods are limited by particle concentration, since above a certain concentration the sample becomes completely opaque [5].

Methods based on inductance or magnetic permeability control require the application of magnetic fields that could induce particle structuring [6,7].

Thermal conductivity monitoring requires applying heat to the sample, which may change the viscosity of the

sample and therefore induce changes in the sedimentation rate [8].

Finally, measuring X-ray and gamma-ray absorption requires expensive equipment [9].

After analysing the available methods and the limitations of each of them, we have developed a new method based on ultrasound transmission with which the variation of particle concentration over time is measured, without limitations and avoiding the disadvantages previously described.

2. Theoretical fundamentals

The new stability measurement method is based on the theory of acoustic transmission in fluids.

The transmission velocity of sound (c) varies depending on the medium to be crossed (Equation 1). The speed of sound in a fluid varies with the density (ρ_{ef}) and compressibility (β_{ef}) of the sample.

$$c = \frac{1}{\sqrt{\rho_{ef}\beta_{ef}}} \quad (1)$$

As for the requirements of the method, to avoid the effects of scattering it is necessary to work within the Rayleigh region. Working within this region allows us to neglect scattering, because within this region the wavelength of the acoustic signal is much larger than the particle size, so using absorption theory would be sufficient [10]. In non-scattering systems the effective density and compressibility of the mixture can be calculated by multiplying the fraction (φ) of each of the components (solid or liquid) by its respective property (Equation 2,3).

$$\rho_{ef} = \rho_s\varphi + \rho_l(1 - \varphi) \quad (2)$$

$$\beta_{ef} = \beta_s\varphi + \beta_l(1 - \varphi) \quad (3)$$

To measure the transmission velocity, a time-of-flight (*TOF*) acoustic technique has been used, which consists of producing and detecting a pressure pulse transmitted through the sample to be analysed. Therefore, determining the transition of the transmission velocity during the gravitational settling process would help to identify the stability level of the MR fluid.

3. Experimental procedure

3.1 Device set-up

The measurements were carried out using a pulse-echo configuration, that is, the same transducer is in charge of emitting and receiving the signal. The transducer is placed in one of these housings, which emits a pulse that passes through the sample and bounces off the opposite wall, passing through the fluid again before being detected.

To contain the magnetorheological fluid and position the transducer, it was necessary to design a specific container.

The container retaining the sample has been designed with the following technical specifications in mind: to be removable for easy cleaning, to have transparent walls through which the sedimentation process can be visualised, to have the possibility of measuring at different heights by placing the transducer in the different housings and to have a large impedance difference between the sample and the walls, to better reflect the signal.

The transducer (Figure 2) is immersed in the sample and is connected to an Epoch 650 wave receiver generator. This in turn is connected to a computer, from which the device is controlled, and data reception and processing is carried out.

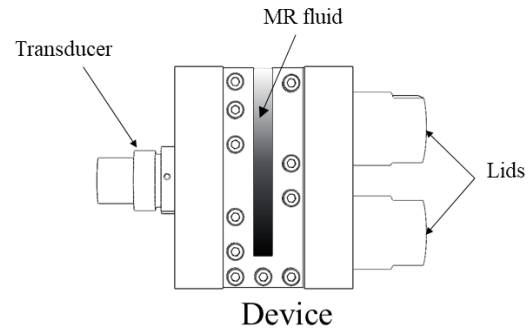


Figure 2. Schematic representation of the container with transducer

3.2 Methodology and results

Once the device has been designed and the measurement set-up has been developed, it is necessary to perform a calibration to relate the signal propagation speed to the particle concentration of the sample. For this purpose, we have prepared MR fluids with different particles (solid phase) concentrations.

In Figure 3, an example of signal transmitted in the characterised samples is presented. With this method, as the transducer itself is responsible for emitting and receiving the signal, some noise is recorded before the useful signal (see Figure 3). This noise is mainly due to the reflection of the ultrasonic wave. To measure the TOF, the noise has been omitted and only the time elapsed from the generation of the pulse to the reception of the transmitted signal has been considered.

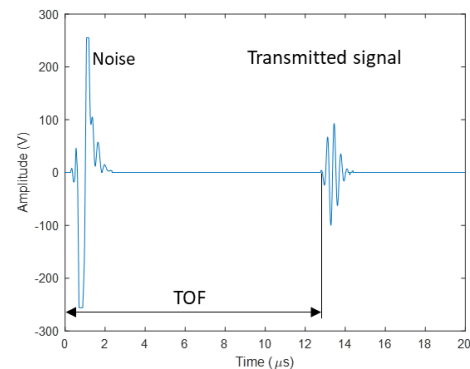


Figure 3. Example of signal transmitted through a fluid

We have measured the time of flight of the signal at different concentrations and calculated the transmission speed considering that the signal must cross the gap of the device twice.

After obtaining the acoustic transmission velocity of the samples at different concentrations, we proceeded to carry out a sedimentation test. During the test we captured a signal every 40 s and placed the transducer in the highest housing.

Interpolating the transmission speed with the concentrations obtained in the previous calibration we have measured the variation of the concentration of the sample in time at the measuring height. The measured initial concentration agrees with the formulated MR fluid concentration, and as the slurry line passes through the measurement area, it decreases to almost 0% (Figure 4). These results are in agreement with the observed and expected concentrations, thus verifying the effectiveness of this new method for measuring the sedimentation of MR fluids.

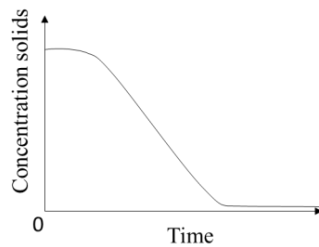


Figure 4. An example of a colloid settling process measurement

4. Conclusions

The developed ultrasonic-based stability measurement method has proven to be an effective system for monitoring the gravitational stability of MR fluids with high repeatability and with no alteration of the thermo-physical properties of the sample. Experimental results confirm the theoretical basis of the method, as the particle concentration increases, the wave propagation velocity decreases due to the increased density of the suspension. Furthermore, this method is not limited by particle size and is suitable for measuring micro- and nano-sized particle suspensions. Finally, it should be remarked the usefulness of this method for measuring gravitational stability of other non-magnetic colloids when there is a difference in density between the carrier liquid and the particles.

References

- [1] C. D. Mesquida and J. L. Lässig, *Introducción a los fluidos magnetoreológicos con aplicaciones en ingeniería*, Rev. Iberoam. Ing. Mecánica, vol. 16, no. 1, pp. 27–36, 2012
- [2] Sohn, J. W., Kim, G. W., & Choi, S. B., *A state-of-the-art review on robots and medical devices using smart fluids and shape memory alloys*. Applied Sciences, 8(10), 1928, 2018.
- [3] Kim, W. H., Park, J. H., Kim, G. W., Shin, C. S., & Choi, S. B. *Durability investigation on torque control of a magneto-rheological brake:*

- experimental work*. Smart Materials and Structures, 26(3), 037001, 2017.
- [4] Ma, J., Zhang, D., Wu, B. et al. *Stability improvement and vibration suppression of the thin-walled workpiece in milling process via magnetorheological fluid flexible fixture*. Int J Adv Manuf Technol 88, 1231–1242, 2017.
 - [5] J. L. Viota, J. De Vicente, J. D. G. Durán, and A. V. Delgado, *Stabilization of magnetorheological suspensions by polyacrylic acid polymers*, J. Colloid Interface Sci., vol. 284, no. 2, pp. 527–541, 2005.
 - [6] G. R. Iglesias, L. F. Ruiz-Morón, J. I. Monesma, J. D. G. Durán, and A. V. Delgado, *An experimental method for the measurement of the stability of concentrated magnetic fluids*, J. Colloid Interface Sci., vol. 311, no. 2, pp. 475–480, 2007.
 - [7] X. Guan, J. Li, and J. Ou, *Design and fabrication of a novel settled and laminated testing instrument for magnetorheological fluid*, vol. 5762, pp. 267–274, 2005.
 - [8] H. Cheng, X. Zhang, G. Liu, W. Ma, and N. M. Wereley, *Measuring the sedimentation rate in a magnetorheological fluid column via thermal conductivity monitoring*, Smart Mater. Struct., vol. 25, no. 5, p. 0, 2016.
 - [9] J. S. Abel, G. C. Stangle, C. H. Schilling, and I. A. Aksay, *Sedimentation in flocculating colloidal suspensions*, J. Mater. Res., vol. 9, no. 2, pp. 451–461, 1994.
 - [10] Álvarez-Arenas, T. G., Segura, L. E., & de Sarabia, E. R. F. *Characterization of suspensions of particles in water by an ultrasonic resonant cell*. Ultrasonics, 39(10), 715-727, 2002.

Bifurcation analysis of incompressible fluid flow by a high order mesh-free indicator with Padé approximants

M. RAMMANE¹, S. MESMOUDI², O. ASKOUR³, A. TRI^{4,5}, B. BRAIKAT¹, N. DAMIL¹

1. Laboratoire d'Ingénierie et Matériaux, Faculté des Sciences Ben M'sik, Hassan II University of Casablanca, B.P 7955 Sidi Othman, Casablanca, Morocco
2. Hassan First University of Settat, Ecole Nationale des Sciences Appliquées, LISA Laboratory, Berrechid 26100, Morocco
3. Laboratoire Matériaux Procédés Environnement Qualité (LMPEQ), Université Cadi Ayyad, Ecole Nationale des Sciences Appliquées de Safi, Morocco
4. Laboratoire de Mécanique, Faculté des Sciences Ain Chock, Hassan II university of Casablanca, Morocco
5. Institut Supérieur des études Maritimes (ISEM), Km 7 Route d'El Jadida, Casablanca, Morocco

Abstract :

In this work, a mesh-free indicator is proposed to detect the bifurcation points of incompressible fluid flow in a sudden expansion. This indicator is mainly based on the high order mesh-free algorithm (HO-MFA) to solve the nonlinear problems obtained from the linear stability analysis. The Padé approximants are then used to improve the validity range of the HO-MFA and to determine the roots of the indicator. The combination of these mathematical tools allows us to determine the zero of the indicator corresponding to the critical point. The Lyapunov-Schmidt reduction is used to search all the bifurcated branches from the critical point. To show the accuracy of our model, comparisons of the obtained results with those of the literature are made.

Keywords: *Bifurcation point, mesh-free method, indicator, Padé approximants, Lyapunov-Schmidt reduction*

1 Introduction

Many experimental and numerical studies have been carried out to study the bifurcation phenomenon of laminar flows in a sudden expansion [1–6]. It is well known that these flows can provide symmetrical or non-symmetrical profiles, for example, a Pitchfork bifurcation (symmetry breaking) is encountered at the critical Reynolds number. Indeed, the stability of the system changes; the flow is initially symmetrical and becomes asymmetric beyond the critical value [1–6]. Several numerical models were proposed to detect numerically these bifurcation points. We can find the direct simulation model [2], the linear stability analysis using either the Eigenvalue model [3] or the bifurcation indicator model [4]. In all these models the finite element method is the most used, due to its large panel of application in mechanical engineering. However several difficulties are encountered in these models, such as numerical integration, mesh generation, compu-

tational cost due to determination of the Eigenvalues or steps used in the classical indicator. Recently a Geometric Progression model [5, 6] was proposed to reduce the time of computation consumed in the classical indicator models. We propose in this concept, a high order mesh-free indicator that uses the Padé approximants (HO-MFI-PA) to overcome the difficulties encountered in the classical models. In this approach, a linear perturbation is used for the stability analysis. The HO-MFA [6] is used to solve the nonlinear problem, to avoid the numerical integration, the mesh-generation and the computational cost consumed by the iterative algorithms [2,3]. Then, the Padé approximants [4, 7] are used to increase the validity range of the HO-MFA. The Padé approximants allow to compute directly the null values of the indicator, which for the bifurcation point is detected. This is practically achieved by analyzing the roots of its numerator, this is why this approximation is chosen in this model. To compute all post bifurcated branches that cross at the critical point, the Lyapunov-Schmidt reduction [5, 6] is used in this model. The obtained results will be compared and verified by those presented in some works of the literature [1–6]. The rest of this paper is organized as follows. In section 2, we describe the proposed indicator to solve stationary incompressible Navier-Stokes equations. Then, numerical examples, concerning classical flow in a sudden expansion are reported and a list of references is given in the end of this paper.

2 Description of the mesh-free indicator

The steady Navier-Stokes equations for a Newtonian and incompressible fluid flow are the following :

$$\begin{cases} \rho(\mathbf{V} \cdot \nabla \mathbf{V}) &= \mu \Delta \mathbf{V} - \nabla p & \text{in } \Omega \\ \nabla \cdot \mathbf{V} &= 0 & \text{in } \Omega \\ \mathbf{V} &= \lambda \mathbf{V}_d & \text{on } \partial \Omega_V \end{cases} \quad (1)$$

where \mathbf{V} is the field velocity, ρ is the mass density, μ is the dynamic viscosity, p is the pressure and \mathbf{V}_d is the imposed velocity. λ is a dimensionless control parameter, used here to search numerically the path solution of velocity variation as function of Reynolds number defined by $Re = (\lambda v_d \rho D) / \mu$ [4-6]. In the constraint (1)b, the penalty technique [4-6] is used to ensure the incompressibility condition by introducing the penalty parameter ξ and to express the the pressure as function of the velocity ($p = \xi \nabla \cdot \mathbf{V}$). The previous equations can be rewritten in the following compact form:

$$L(\mathbf{V}) + Q(\mathbf{V}, \mathbf{V}) = \lambda F \quad (2)$$

where $L(\mathbf{V})$ and $Q(\mathbf{V}, \mathbf{V})$ are the linear and nonlinear operators respectively, F is the second member obtained from the boundary on $\partial\Omega_V$. In the linear stability analysis, the solution is sought as linear combination of a perturbation $\delta\mathbf{V}$ and a stationary solution \mathbf{V}^λ (such as $\mathbf{V} = \mathbf{V}^\lambda + \delta\mathbf{V}$). By introducing this combination in (2) and by considering only the responses to the 1st order of perturbation, we obtain:

$$\begin{cases} L(\mathbf{V}^\lambda) + Q(\mathbf{V}^\lambda, \mathbf{V}^\lambda) = \lambda F \\ L(\delta\mathbf{V}) + \tilde{Q}(\mathbf{V}^\lambda, \delta\mathbf{V}) = \eta^s f \end{cases} \quad (3)$$

where $\tilde{Q}(\mathbf{V}^\lambda, \delta\mathbf{V}) = Q(\mathbf{V}^\lambda, \delta\mathbf{V}) + Q(\delta\mathbf{V}, \mathbf{V}^\lambda)$. Thus, the velocity perturbation introduces a perturbation denoted by $\eta^s f$, where the scalar η^s is the proposed bifurcation indicator and f is a random function [4]. The computation of a bifurcation requires solving the equation (3) by using the HO-MFA presented in [6], where the unknowns of the problem are sought into a Taylor series expansion, truncated at the order k , with respect to the parameter a as follows:

$$\mathbf{u} = \mathbf{u}_0 + \sum_{i=1}^k a^i \mathbf{u}_i \quad (4)$$

where $\mathbf{u}^t = \langle \mathbf{V}^\lambda, \lambda, \delta\mathbf{V}, \eta^s \rangle$ and the parameter a is defined by the arc-length parametrization [4-6] : $a = \langle \mathbf{V}^\lambda, \mathbf{V}_1^\lambda \rangle + \lambda \lambda_1$. By using the HO-MFA techniques of [6] and an orthogonality condition to the solution (chosen as $\langle \delta\mathbf{V} - \delta\mathbf{V}_0, \delta\mathbf{V}_0 \rangle = 0$), we obtain the following linear systems:

$$\begin{cases} \mathbf{K}_t(\mathbf{V}_0^\lambda) \mathbf{V}_i^\lambda = \lambda_i \mathbf{F} - \sum_{r=1}^{i-1} \mathbf{Q}(\mathbf{V}_r^\lambda, \mathbf{V}_{i-r}^\lambda) \\ \langle \mathbf{V}_i^\lambda, \mathbf{V}_1^\lambda \rangle + \lambda_i \lambda_1 = \delta_{i1} \\ \mathbf{K}_t(\mathbf{V}_0^\lambda) \delta\mathbf{V}_i = \eta_i^s \mathbf{f} - \sum_{r=1}^{i-1} \tilde{\mathbf{Q}}(\mathbf{V}_r^\lambda, \delta\mathbf{V}_{i-r}) \\ \langle \delta\mathbf{V}_i, \delta\mathbf{V}_0 \rangle = 0 \end{cases} \quad (5)$$

It should be noted that the previous linear systems (5) have the same tangent operator. First, the unknowns of (5) are calculated which gives $(\mathbf{V}_i^\lambda, \lambda_i)$. Then, the other

unknowns are computed $(\delta\mathbf{V}_i, \eta_i^s)$. Classically, the continuation procedure is used by considering the end of each step $\mathbf{u}(a_{max})$ as the new starting point for the next branch \mathbf{u}_0 where $a_{max} = \left(\epsilon \frac{\|\mathbf{u}_1\|}{\|\mathbf{u}_k\|} \right)^{\frac{1}{k-1}}$ [4-6].

The asymptotic expansions of (4) are replaced by equivalent rational approximations called Padé approximants:

$$\mathbf{u}_{Padé,k} = \mathbf{u}_0 + \sum_{i=1}^{k-1} \frac{\mathcal{D}_{k-1-i}(a)}{\mathcal{D}_{k-1}(a)} a^i \mathbf{u}_i \quad (6)$$

where \mathcal{D}_i is a polynomial of degree k defined by :

$$\begin{cases} \mathcal{D}_i = 1 + \sum_{n=1}^i a^n d_n \\ d_i = -\frac{\beta_{k(k-i)}}{\beta_{(k-i)(k-i)}} - \sum_{m=1}^{i-1} \frac{\beta_{(k-m)(k-i)}}{\beta_{(k-i)(k-i)}} d_m \\ \beta_{ii} = \sqrt{\langle \mathbf{u}_i^p, \mathbf{u}_i^p \rangle} ; \quad \beta_{im} = \langle \mathbf{u}_i, \mathbf{u}_m^* \rangle \\ \mathbf{u}_i^p = \mathbf{u}_i - \sum_{l=1}^{i-1} \mathbf{u}_l^* ; \quad \mathbf{u}_i^* = \mathbf{u}_i^p / \beta_{ii} \end{cases} \quad (7)$$

In general, the representation (6) has a validity range " a_{max}^p " greater than the polynomial approximation (4). The validity range of (6) is determined by [4, 7]:

$$\varepsilon_d = \frac{\|\mathbf{u}_{Padé,k}(a_{max}^p) - \mathbf{u}_{Padé,k-1}(a_{max}^p)\|}{\|\mathbf{u}_{Padé,k}(a_{max}^p) - \mathbf{u}_0\|} \quad (8)$$

Once the parameter a_{max}^p is calculated, a new starting point can be defined by introducing the value a_{max}^p in the expression (6) allowing to calculate the solution \mathbf{u} for any Reynolds number. Finally, a stationary bifurcation point is determined when $\eta^s = 0$. Once, the point of bifurcation is obtained \mathbf{u}_c , we propose to compute the bifurcated solutions by using \mathbf{u}_c as a new starting point of (4). In theory, the tangent operator evaluated at the bifurcation point is singular, for that the Lyapunov-Schmidt reduction [5] is combined with the HO-MFA to compute all the post-bifurcated branches as in [6].

3 Numerical results

The classical flow in a sudden expansion is considered to illustrate the accuracy of the proposed model. The domain, the boundary conditions and the properties of the fluid are presented in figure 1.

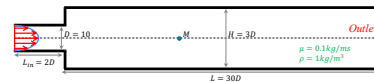


Figure 1: Geometry and boundary conditions

In figure 2, we present the evolution of the indicator as a function of the Reynolds number. It shows that the indicator reaches a null value at $Re_c = 82.87$. In the literature investigations, the value of Re_c are : $Re_c > 81$

in the experimental studies of [1], $Re_c = 79.95$ in [2] using direct simulation model, $Re_c = 82.97$ in [3] using the Eigenvalue model, $Re_c = 80.46$ in [4] using the indicator model with FEM, $Re_c = 82.08$ in [5] using 3D investigation by FEM with Geometric Progression and $Re_c = 82.87$ in [6] using mesh-free approach with Geometric Progression. The obtained results are very similar to those obtained in the literature and the closest to the experimental results of [1] using 2D investigations.

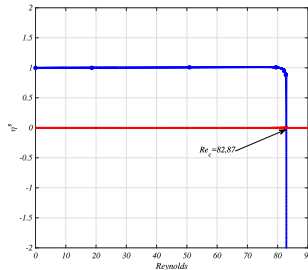


Figure 2: Curve of the bifurcation indicator

In table 1, we present the truncation order influence on the critical point detected using the classical series continuation (4), the Padé approximants (6) and the indicator model of [4]. The results show that the Padé approximants have a precise detection and by using lower truncation order, continuation steps and time of computation.

Order k		15	20	25	30
Padé (6)	Re_c	82.85	82.86	82.87	82.87
	Step	7	7	6	6
	CPU	55s	63s	42s	49s
Series (4)	Re_c	—	—	—	82.94
	Step	—	—	—	10
	CPU	—	—	—	101s
Indicator model of [4]	Re_c	—	—	—	82.79
	Step	—	—	—	11
	CPU	—	—	—	3200s

Table 1: Influence of k on Re_c

Figure 3 presents the diagram of bifurcation obtained by the proposed model with respect to the streamlines solution corresponding to each bifurcated branch solution. The diagram represents a Pitchfork bifurcation characterized by the asymmetrical flow (where one being the mirror image of the other) and a second fundamental solution where the flow remains symmetrical. We note that these results are similar to those of the literature [1–6].

4 Conclusion

We have proposed in this paper an algorithm to compute stationary bifurcation points in fluid mechanics. This model gave the results similar to those of literature. It is an efficiency approach to detect precisely the critical

Reynolds points and to visualize the bifurcating solutions, consuming a less matrix inversions and less expensive computation time compared to the classical indicator.

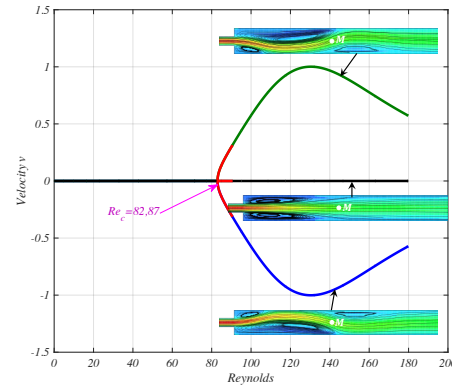


Figure 3: Bifurcation diagram

References

- [1] W. Cherdron, F. Durst, J. H. Whitelaw. *Asymmetric flows and instabilities in symmetric ducts with sudden expansions*. Journal of Fluid Mechanics, 84(1) (1978) : 13–31.
- [2] D. Drikakis. *Bifurcation phenomena in incompressible sudden expansion flows*. Physics of Fluids, 9(1) (1997) : 76–87.
- [3] N. Alleborn, K. Nandakumar, H. Raszillier, and F. Durst. *Further contributions on the two-dimensional flow in a sudden expansion*. Journal of Fluid Mechanics, 330 (1997) : 169–188.
- [4] Y. Guevel, G. Girault, and J. M. Cadou. *Parametric analysis of steady bifurcations in 2D incompressible viscous flow with high order algorithm*. Computers & Fluids, 100 (2014): 185–195.
- [5] M. Medale and B. Cochelin. *High performance computations of steady-state bifurcations in 3D incompressible fluid flows by Asymptotic Numerical Method*. Journal of Computational Physics, 299 (2015): 581–596.
- [6] M. Rammane, S. Mesmoudi, A. Tri, B., Braikat, N. Damil. *Bifurcation points and bifurcated branches in fluids mechanics by high order mesh-free geometric progression algorithms*. International Journal for Numerical Methods in Fluids, 93(3) (2020): 834–852.
- [7] A. Elhage-Hussein, M. Potier-Ferry, N. Damil. *A numerical continuation method based on Padé approximants*. International Journal of Solids and structures 37.46-47 (2000): 6981-7001.

MHD non-Newtonian effects on lubricated contacts performances

M. KARDOUDI, M. NABHANI, M. EL KHLIFI

Hassan II University of Casablanca, Faculty of Sciences and Techniques
P.O. Box 146, 28806 Mohammedia, Morocco
mohammed.kardoudi-etu@etu.univh2c.ma
nabhanimohamed@yahoo.fr
mohamed.elkhlifi@fstm.ac.ma

Abstract

A numerical study of infinite Magneto-HydroDynamic (MHD) journal bearings lubricating with non-Newtonian couple stress fluid is presented in this work. The modified Reynolds equation type is derived based on the MHD and Stokes theories. The film pressure is iteratively computed using a numerical differentiation approach. Various bearing characteristics are analyzed using different couple stress parameter and Hartman number values. The obtained numerical results show that the couple stress and MHD effects significantly provide the journal bearings performances.

Keywords: *Magneto-HydroDynamic; Infinite journal bearings; Non-Newtonian couple stress fluid.*

1. Introduction

The study of MHD bearings has attracted more attention of many researchers due to its wide range of industrial applications¹⁻². Comparing to classical bearings, the MHD bearings provide a significant increase in load-carrying capacity and can function under high temperatures.

Journal bearings are widely designed for supporting high speed rotating machinery such as compressors and turbines because of their low friction and high load-carrying capacity. Many authors have investigated the infinite MHD journal bearings performances³⁻⁵. However, these studies are limited to Newtonian lubricants.

Owing to developing of industry, the use of fluids mixed with long-chain polymer additives has attracted a considerable importance because of load-carrying capacity increase and friction parameter reduction effects. To describe the rheological behavior of these kinds of non-Newtonian fluids, the Stokes micro-continuum theory⁶ has been proposed.

In this present work, a numerical investigation of infinite MHD journal bearings operating with non-Newtonian lubricants is presented. These later effects are described using Stokes theory. The modified Reynolds equation is obtained and discretized with finite difference method.

The algebraic system of equations is computed by Gauss-Seidel technique with successive over-relaxation.

2. Physical problem position

The physical configuration of an infinite journal bearing is composed of an inner cylinder (journal) and an outer cylinder (bearing). The gap between two cylinders is separated by a lubricant fluid film of thickness h . The lubricant is an incompressible, isothermal, electrically conducting and non-Newtonian couple stress fluid of constant properties. The journal has a radius R , length L , eccentricity e and rotates with a uniform angular velocity U about its axis in opposite clockwise direction. A radial magnetic field $B_r = -B_0 R/r$ is applied to the journal bearing with B_0 is the magnetic field strength. The journal and the bearing are both made of non-conductive materials.

Based on the assumption of hydrodynamic lubrication and in this case the body couples, body forces (except for Lorentz force) and inertia force are small in comparison with viscous forces, hence the continuity and momentum equations of laminar flow of an electrically conducting non-Newtonian lubricant are:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

$$\frac{\partial^4 u}{\partial y^4} - \frac{\mu}{\eta} \frac{\partial^2 u}{\partial y^2} + \frac{\sigma B_0^2}{\eta} u = - \left(\frac{1}{\eta} \frac{\partial p}{\partial x} + \frac{\sigma B_0 E_z}{\eta} \right) \quad (2)$$

$$\frac{\partial p}{\partial y} = 0 \quad (3)$$

where u and v are the velocity components in x and y directions respectively, p is the film pressure, E_z stand for the electric field components in z axis, μ is the dynamic viscosity, η represents a material constant responsible for couple stresses due to polar additives in the lubricant and σ is the fluid electrical conductivity.

Considering zero net current flow and insulating surfaces condition, the electric field is calculated by:

$$E_z = - \frac{B_0}{h} \int_0^h u(y) dy \quad (4)$$

Solving equation (2) and using the no-slip and no couple

stress conditions on journal and bearing surfaces, the velocity component u is obtained.

Using this last equation, the velocity component u deduced is inserted into equation (1), and integrating across h , the modified Reynolds equation in dimensionless form is derived as the follows:

$$\frac{1}{6} \frac{\partial}{\partial \theta} \left\{ F^* \frac{\partial p^*}{\partial \theta} \right\} = \frac{\partial h^*}{\partial \theta} \quad (5)$$

with

$$F^* = \frac{6}{M^2} \frac{(A^{*2} - B^{*2})h^{*2}}{l^* \left(\frac{A^{*2}}{B^*} \operatorname{tgh} \left(\frac{B^*}{2l^*} h^* \right) - \frac{B^{*2}}{A^*} \operatorname{tgh} \left(\frac{A^*}{2l^*} h^* \right) \right)}$$

$$A^* = \frac{\sqrt{1 + \sqrt{1 - 4M^2 l^{*2}}}}{\sqrt{2}}, \quad B^* = \frac{\sqrt{1 - \sqrt{1 - 4M^2 l^{*2}}}}{\sqrt{2}}$$

where p^* is the dimensionless film pressure, θ is the bearing angle across circumferential direction, $l^* = \sqrt{\eta/\mu}/c$ is the couple stress parameter, $M = B_0 c \sqrt{\sigma/\mu}$ stands for the Hartmann number and c is the radial clearance.

The dimensionless film thickness for infinite journal bearings is:

$$h^*/c = 1 + \varepsilon \cos \theta \quad (6)$$

where ε is the journal eccentricity ratio defined by $\varepsilon = e/c$.

Equation (5) is discretized using finite differences. The algebraic equations are solved iteratively using the successive over-relaxation scheme. When the relative error on p^* between two successive iterations is less than 10^{-5} , the numerical scheme converges. Once the convergence is achieved, the load-carrying capacity is computed by the trapezoidal rule.

3. Results Analysis

Figure 1 shows the dimensionless film pressure p^* along the circumferential direction for two values of couple stress parameter l^* and Hartmann number M at $\varepsilon = 0.6$. The non-Newtonian effects ($l^* = 0.1$) yields an increase in the film pressure values as compared with Newtonian case ($l^* = 0$). Physically, this is due to the presence of additive of long-chain molecules which will resist and oppose the lubricant fluid motion and, as a consequence, enhance the pressure in the film fluid. It is also observed that the MHD influence ($M = 5$) increases the film pressure compared to the non-magnetic field case ($M = 0$); the application of magnetic field to an electrically conducting Non-Newtonian lubricant produces a Lorentz force, which acts in opposite direction of fluid motion and provide further higher values of film pressure. The influence of eccentricity ratio on maximum film pressure

with $l^* = 0.2$ and $M = 2$ is shown in Figure 2. It is observed that as the eccentricity ratio increases, maximum film pressure increases due to the decrease in the minimum film thickness.

Figure 3 shows the dimensionless load-carrying capacity varying with Hartmann for two values of couple stress parameter with $\varepsilon = 0.8$. The load capacity is observed to increase with increasing values of the Hartmann number. Since the load-carrying capacity is obtained by integration of film pressure acting on the journal surface with respect to θ , the augmentation in film pressure due to the MHD and non-Newtonian fluids will result in increase of the load-carrying capacity. Also, a high load-carrying capacity is obtained at larger values of ε for $l^* = 0.2$ and $M = 2$ (Figure 4).

4. Conclusion

A calculation program was developed in this work to simulate the combined effects of MHD and non-Newtonian couple stress fluid on infinite journal bearings performances. The obtained numerical results demonstrate that these effects are significant. This means that journal bearings lubricated with an electrically conducting lubricant containing additives molecules in presence of a magnetic field can sustain a higher load-carrying capacity compared to the classical infinite journal bearings.

References

- [1] T.V.V.L.N. Rao, A.M.A. Rani, N.M. Mohamed, H.H. Ya, M. Awang and F.M. Hashim, *Analysis of magnetohydrodynamic partial slip laser bump texture slider and journal bearing*, Part J: Journal of Engineering Tribology, 233 (2019) pp. 1921–1938.
- [2] M. Mouda, M. Nabhani and M. El Khelifi, *Effect of non-Newtonian magneto-elastohydrodynamic on performance characteristics of slider-bearings*, Industrial Lubrication and Tribology, 71 (2019) pp. 1158–1165.
- [3] D.C. Kuzma, *The magnetohydrodynamic journal bearing*, Journal of Basic Engineering, 85 (1963) pp. 424–427.
- [4] S. Kamiyama, *Magnetohydrodynamic journal bearing (report 1)*, Journal of Lubrication Technology, 91 (1969) pp. 380–386.
- [5] T. Sasada, Y. Kurosaki, K. Honda and K. Kamijo, *MHD journal bearing in a magnetic field perpendicular to its axis*, Bulletin of JSME, 17 (1974) pp. 1645–1651.
- [6] V.K. Stokes, *Couples Stresses in Fluids*, Phys Fluids 9 (1966) pp. 1709 - 1715.

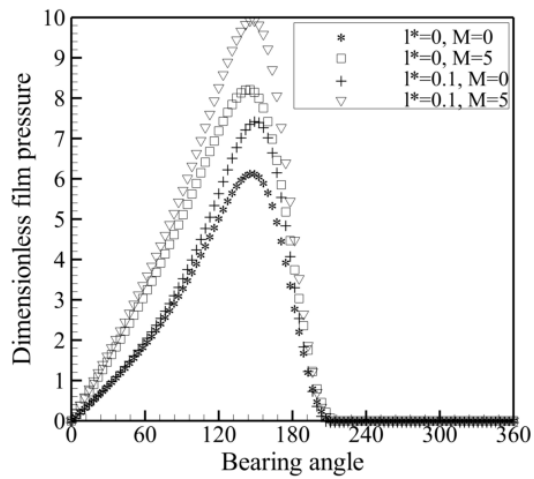


Figure 1: Dimensionless film pressure versus circumferential direction at $\varepsilon = 0.6$

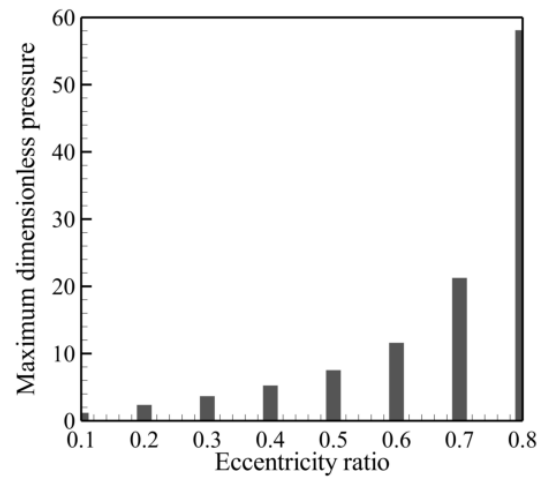


Figure 2: Maximum dimensionless film pressure versus eccentricity ratio at $l^* = 0.2$ and $M = 2$

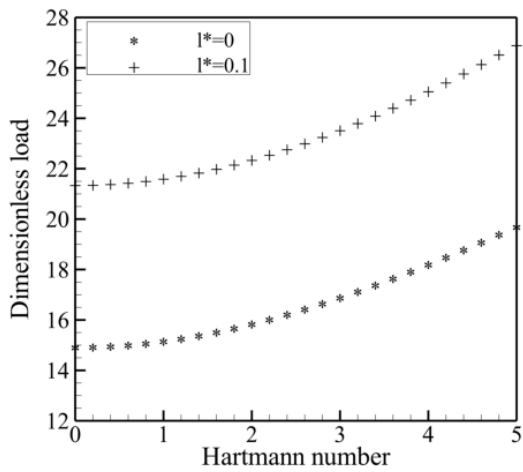


Figure 3: Dimensionless load versus Hartmann number at $\varepsilon = 0.8$

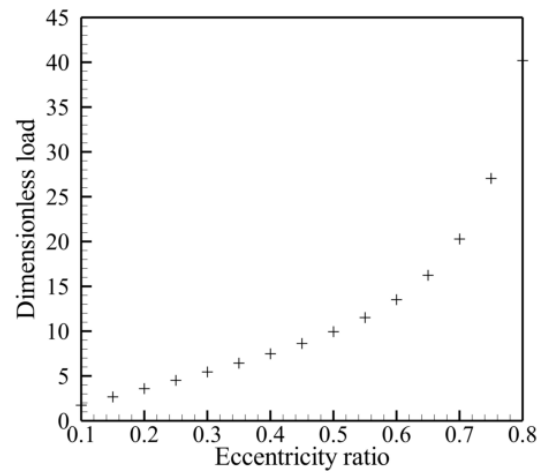


Figure 4: Dimensionless load versus eccentricity ratio at $l^* = 0.2$ and $M = 2$

Détermination de profils aérodynamiques à moyen Reynolds pour une aérostructure à fuselage intégré

M. HAKIM¹, S. CHOUKRI²

1. ERG2ME. Ecole Mohammadia des ingénieurs, Université Mohammed V de Rabat. Mohamed_hakim@um5.ac.ma
2. ERG2ME. Ecole Mohammadia des ingénieurs, Université Mohammed V de Rabat. choukri@emi.ac.ma

Abstract

The aim of this work is to determine medium Reynolds aerodynamic profiles for an integrated fuselage aerostructure. For this purpose, a weighted scoring method is adopted to select profiles that meet the criteria of our integrated fuselage aerostructure among 32 airfoils existing in the literature. Thus, a comparison of the effects of the selected airfoils on the aerodynamic properties at the full plane level, is illustrated, by the use of a multi-fidelity analysis tool named XFLR5 to derive these aerodynamic properties and trim characteristics during level flight.

Résumé

L'objectif de ce travail vise la détermination de profils aérodynamiques à moyen Reynolds pour une aérostructure à fuselage intégré. Pour cela, une méthode de notation pondérée est adoptée pour sélectionner, parmi 32 profils existant dans la littérature, des profils qui satisfont des critères prédéfinis. Ainsi, une comparaison des effets des profils aérodynamiques sélectionnés sur les propriétés aérodynamiques au niveau de l'avion complet, est illustrée, par l'utilisation d'un outil d'analyse multi-fidélité nommé XFLR5 pour dériver ces propriétés aérodynamiques et les caractéristiques de compensation pendant le vol en niveau.

Mots clefs : *Aérodynamique, aérostructure, drone, fuselage intégré, nombre de Reynolds, stabilité aérodynamique.*

Keywords: *Aerodynamics, aerostructure, drone, integrated fuselage, Reynolds number, aerodynamic stability.*

1. Introduction

De nos jours, le domaine de la construction aéronautique se diversifie et s'étend ; à des applications en aviation de plaisance, aux drones avec différents types de missions civiles ou militaire [1] et suscite un intérêt grandissant ; scientifique, industriel et commerciale. En effet, l'absence de pilote et de passagers dans les drones, autorise un libre choix des formes en plan (planform) et d'aérostructures

non conventionnelles [2]. Ajouter à cela la diversité de missions civiles potentielles pour les mini drones basses altitudes. Il est à noter que dans l'aviation conventionnelle, où les aéronefs évoluent à des Reynolds supérieurs à 10^6 en écoulements quasi turbulents avec des règles de conception relativement figées et bien établies, les drones civils évoluent à des régimes de Reynolds autour de 10^5 , qualifiés de critiques pour un bon nombre de profils aérodynamiques réputés. Dans ce travail nous présentons une méthodologie et des solutions de conception préliminaires qui répondent aux spécificités aérodynamiques d'un mini drone à fuselage intégré et ses contraintes de stabilité.

2. Le fuselage intégré

Au cours des dernières années, un nouveau concept d'avion a commencé à être envisagé, le concept d'aile à fuselage intégré fig. 1, qui est une forme hybride entre l'avion à aile volante et l'avion classique. Les recherches menées sur ce concept innovant montrent une réduction du poids de la structure, de la traînée avec une finesse plus grande et donc une baisse de la consommation en carburant comparé à la génération actuelle.

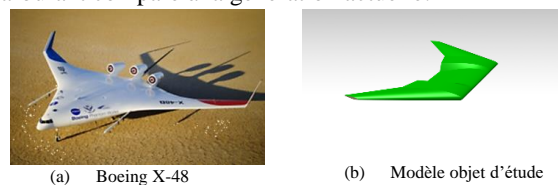


Figure 1 : Aile à Fuselage intégré

Indépendamment de tous ces avantages déclarés, ce concept d'avion sans empennage pose généralement des contraintes de stabilité surtout la stabilité longitudinale. L'équation qui gouverne la stabilité longitudinale de l'aile seule d'un avion conventionnel [3] est définie par :

$$C_{m_{cg}} = C_{m_{ac}} + C_L \left(\frac{x_{cg}}{\bar{c}} - \frac{x_{ac}}{\bar{c}} \right) \quad (1)$$

Où $C_{m_{cg}}$, $C_{m_{ac}}$, C_L , x_{cg} , x_{ac} et \bar{c} sont respectivement le coefficient du moment de tangage par rapport au centre de gravité, le coefficient du moment de tangage par rapport au centre aérodynamique, le coefficient de la portance, la position du centre de gravité, la position du centre aérodynamique et la corde moyenne.

D'après l'équation (1), on remarque que le centre aérodynamique doit se trouver en arrière du centre de gravité pour que la pente de la courbe du coefficient de moment de tangage en fonction d'angle d'attaque, $C_{m\alpha}$, soit négative (condition de la stabilité longitudinale statique).

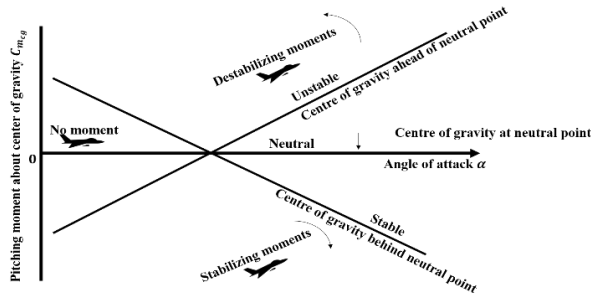


Figure 2 : Coefficient du moment de tangage en fonction de l'angle d'attaque pour différentes positions du centre de gravité

3. Aérodynamique du mini drone

La portance d'une aile est le résultat d'intégration d'une distribution de portances locales générées par les profils constituant la section courante de son envergure. Le comportement aérodynamique d'un profil d'aile dépend essentiellement de la nature de la couche limite (laminaire, transitoire ou turbulente). Le nombre de Reynolds, qui est le rapport entre les forces d'inertie et les forces visqueuses ou la longueur caractéristique multipliée par la vitesse divisée par la viscosité cinématique, est généralement utilisé pour mettre les véhicules à l'échelle. Les recherches [4-5] sur la classification d'objets volants naturels et artificiels montrent que la catégorie des mini drones a pour masse (aérostructure + charge utile) des valeurs de 10 kg à 100 kg, une vitesse de vol de 10 m/s à 100 m/s, envergure < 10 m et un nombre de Reynolds aux environs de 10^5 , avec des altitudes de vols pouvant aller jusqu'à 5000 m. Les travaux réalisés sur l'étude des profils conventionnelles aux faibles nombres de Reynolds ont montré que leurs performances se détériorent rapidement à un nombre de Reynolds critique autour de 10^5 . Même si ces profils sont les plus performants à des nombres de Reynolds élevés ($> 10^6$).

4. Sélection des profils aérodynamique

4.1 La méthode de notation pondérée

A partir de 32 profils aérodynamiques existants dans la littérature consultée et couvrant cette gamme des Reynolds, nous nous proposons d'identifier une collection plus réduite de profils adaptés à notre application. Pour cela nous avons eu recours à la méthode de notation pondérée (WSM) que nous décrivons dans ce qui suit.

Nous commençons par identifier les critères sur les caractéristiques aérodynamiques 2D des profils ; tels que l'épaisseur maximale qui dicte les capacités d'emport et de résistance, ainsi que les coefficients de tangage, de

portance, de trainée, et l'angle d'attaque maximale (décrochage). Ces caractéristiques contribuent à la stabilité et la finesse de l'aéronef qui sont des éléments importants dans la conception de ces derniers. Nous poursuivons par l'affectation de pondérations qui traduisent la sensibilité des performances à chaque critère. Nous attribuons ensuite des notes numériques à chaque critère et nous terminons par la somme des notes pondérées pour chaque profil. Comme notre modèle est sans empennage, nous avons accordé au coefficient de tangage la pondération la plus importante pour assurer la stabilité. Le coefficient de portance, épaisseur maximale et angle de décrochage sont d'égale importance dans notre modèle donc nous leur attribuons la même pondération. Du fait que la trainée n'influence pas directement la stabilité nous lui accordons une pondération faible. Le tableau 1 résume les critères et pondérations adoptés.

N°	Coef	Critère	Poids %
1	h_{max}	$15\% < h_{max} < 20\%$	20%
2	c_{mac}	$-0.05 < c_{mac} \leq 0$	30%
3	α_{max}	$\geq 15^\circ$	20%
4	c_l	$c_l \geq 0$ pour $\alpha \geq 0$ à $Re = 10^5$	20%
5	c_{dmin}	< 0.05	10%

Tableau 1: Critères de sélection des profils d'ailes volantes.

Les performances de chaque profil, pour un Reynolds de 10^5 , sont analysés par XFOIL ; implémenté dans XFLR5 [6]. Le programme traite de l'aérodynamique 2D des profils par interaction d'un écoulement potentiel externe avec la couche limite laminaire et sa transition turbulente. Ce programme a montré un excellent résultat par rapport aux autres outils [7]. L'autre programme, XFLR5, traite l'aérodynamique 3D des aérostructures par l'utilisation du XFOIL, les méthodes vortex lattice (VLM) et les panneaux 3D. La validation de XFLR5 est discuté dans [8].

4.2 Résultats de la notation pondérée

Les profils retenus ont eu une note pondérée $\geq 10/20$ et sont listés dans le tableau 2. Les profils du tableau 2.b sont des profils minces (épaisseur < 10%) qu'on a les modifiée au niveau de l'épaisseur et de la cambrure afin de répondre aux exigences de notre modèle en se basant sur les analyses par XFOIL.

N°	Profil	N°	Profil	N°	Profil
1	AS5045 (15%)	4	LIEBECK LA5055	7	NACA 64(2)/15
2	CLARK YM-15	5	LS417	8	NLF5(1)-0115
3	GOE 222	6	NACA 63(3)-018	9	S8036 (16%)

Tableau 2.a : Profils non modifiés.

N°	Profil	Modification
10	HS 522	$h_{max} = 18\%$
11	MH 45	$h_{max} = 18\%$
12	NACA 64A56	$h_{max} = 18\%$, $c_{max} = 5\%$ à $P_{cmax} = 40\% c$
13	S1210	$h_{max} = 18\%$
14	TL 54	$h_{max} = 18\%$ à $Ph_{max} = 33.2\%$, $c_{max} = 5\%$ à $P_{cmax} = 33.6\% c$

Tableau 2.b : Profils modifiés.

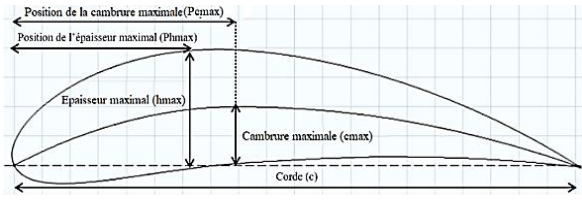


Figure 3 : Paramètres géométriques d'un profil aérodynamique

5. Modèle d'aile à fuselage intégré

Le tableau 3 montre les paramètres utilisés pour la conception de notre mini drone.

Paramètre	Valeur	Paramètre	Valeur
Poids	15 kg	Allongement	3.44
Envergure	3 m	Effilement	3.33
Corde aérodynamique moyenne	0.93 m	Fleche du 1/4 de la corde	41.46°
Surface de référence	2.62 m ²		

Tableau 3 : Paramètres de la conception de mini drone.

Les performances recherchées à ce niveau sont l'équilibre et la vitesse du vol en croisière. La méthode des panneaux 3D de XFLR5 est choisie pour évaluer l'impact des profils sur les performances de notre mini drone. La figure 4a montre que certains avions ont un $C_{m_0} < 0$ ce qui implique un angle d'attaque et une portance négatifs, et entraîne donc une vitesse de croisière infinie. Ces profils nécessiteraient un vrillage trop excessif pour l'équilibre. Alors malgré qu'ils aient eu une note pondérée élevée, ces profils ne sont pas adaptables à notre mini drone. Par contre, les configurations formées par les profils restants pourraient être stables statiquement avec une différence au niveau de $C_{m\alpha}$.

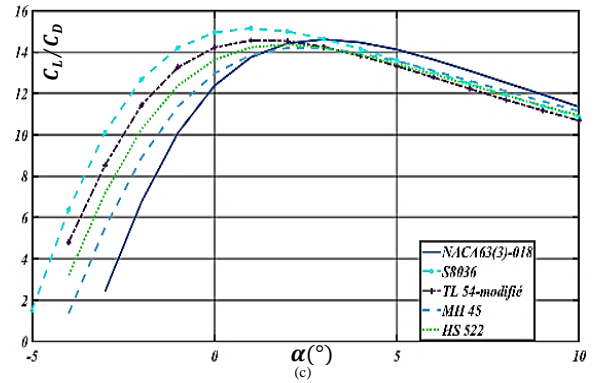
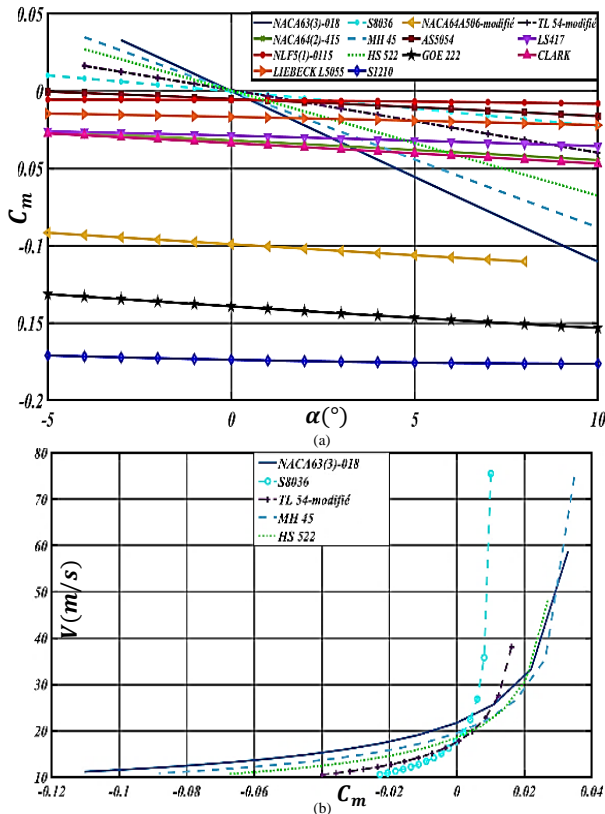


Figure 4 : Comparaison entre les profils sélectionnés : (a) coefficient du moment de tangage en fonction d'angle d'attaque, (b) Vitesse du vol en fonction du coefficient du moment de tangage et (c) la finesse en fonction d'angle d'attaque

Les figures 4b et 4c illustrent respectivement la vitesse et la finesse pour les cinq mini drones construisaient par les profils restants qui respectent le critère de stabilité. On remarque qu'en vol de croisière les vitesses sont autour de 20 m/s et que les finesesses sont entre 12 et 15 pour $\alpha = 0$.

6. Conclusion

Dans ce travail, nous nous sommes intéressés à la détermination de profils aérodynamiques à moyen Reynolds donnant les meilleures performances pour notre aérostructure à fuselage intégré. A partir de 32 profils nous n'avons retenu que cinq profils dont deux initialement épais et trois ont été modifiés dans ce sens. Malgré notre contrainte sur le coefficient de tangage de profil, il s'est avéré que certains profils ne permettraient pas l'équilibre de l'avion en croisière. Au fait l'équilibre fait intervenir des paramètres autre que le $c_{m\alpha}$ tels que le vrillage et la position du point neutre. Nous visons pour la suite d'approfondir cet aspect ainsi que d'étudier la stabilité dynamique.

Références

- [1] P. Brisset, *Drones civils Perspectives et réalités*, Ecole Nationale de l'Aviation Civile, 2004
- [2] P. Bowers, *Unconventional Aircraft*, Tab Books, 1990.
- [3] R. C. Nelson, *Flight stability and automatic control*, Vol. 2, WCB/McGraw Hill, New York, 1998.
- [4] P. B. S. Lissaman, *Low-Reynolds-number airfoils*, Annual review of fluid mechanics 15.1 (1983) p 223-239.
- [5] J. H. McMasters, M. L. Henderson, *Low Speed Single-Element Airfoil Synthesis*, Technical Soaring 6.2 (1980) 1-21.
- [6] XFLR5, <http://www.xflr5.tech/xflr5.htm>, (2022).
- [7] J. Morgado et al., *XFOIL vs CFD performance predictions for high lift low Reynolds number airfoils*, Aerospace Science and Technology 52, 207-214 (2016)
- [8] M. Hakim, S. Choukri, *An integrated preliminary approach elaboration for the analysis of a blended wing body aérostructure concept*, submitted to 12th International Conference on Integrated Design and Production

Three-dimensional numerical investigation of a new drag reduction device applied to Ahmed body

A. AGRISS^{1,*}, M. AGOUZOUL¹, A. ETTAOUIL¹, and A. MEHDARI¹

1–Mohammed V University in Rabat, Mohammadia School of Engineers, Avenue Ibn Sina, P.O. Box 765–Agdal, Rabat, Morocco

Mechanical and Energetic Engineering Research Team: Modeling and Experimentation ERG2(ME)

*email address: amineagriss@research.emi.ac.ma

Abstract :

In this work, a new technique of drag reduction applied to the Ahmed body is developed. This consists of perforating a rectangular conduit in the body. The purpose of this conduit is to lead part of the airflow from the front and inject it in the recirculation region behind the body. Three-dimensional numerical simulations have been performed for a Reynolds number $Re = 2.85 \times 10^6$ using the computational fluid dynamics (CFD) software ANSYS FLUENT. The influence of changing the conduit dimensions and its position from the lower slant edge on drag coefficient is presented. This study is a continuation of the two-dimensional numerical study of this drag reduction technique.

Keywords : *drag reduction, Ahmed body, conduit, three-dimensional, ANSYS FLUENT.*

1 Introduction

In order to reduce toxic gas emissions and fuel consumption, the automotive industry try to reduce drag by designing more aerodynamically efficient vehicles. In this context, lot of flow control techniques have been developed in order to reduce drag. These techniques are tested on simplified car models (i.e. the Ahmed body) before using them in reality. Among passive flow control techniques applied on the Ahmed body, the use of vertical splitter plates placed at the body rear could reduce drag [1]. Using flaps on the slant edges of the body could reduce the pressure drag [2]. Among active flow control techniques, using a synthetic jet to control the wake flow could reduce drag [3]. The use of a continuous jet at the top of the slant part of the body reduces drag [4]. All these flow control methods are described in detail in scientific literature.

In this paper, we present a 3D numerical study of the drag reduction device treated in the 2D case [5]. Numerical simulations are performed using the CFD software ANSYS FLUENT 17.0. A rectangular conduit is perforated into the Ahmed body. It leads part of the airflow from the body front to be inject in the recirculation zone. Various conduit configurations are tested. For each position of the

conduit from the lower edge of the slant, various dimensions of the conduit are tested. Finally, the conduit is narrowed at the exit in order to accelerate the flow. Various configurations are tested based on the reduction ratio of the conduit dimensions. Effects of all these modifications on drag coefficient are evaluated.

2 Numerical modeling

In this study, we consider an incompressible viscous flow. Reynolds-averaged Navier-Stokes (RANS) equations for a turbulent flow are defined by:

$$(\mathbf{U} \cdot \nabla) \mathbf{U} = \nu \nabla^2 \mathbf{U} - \frac{1}{\rho} \nabla p + \nabla \tau \quad (1)$$

where ν is the kinematic viscosity, ρ is the density, p is the pressure, $\mathbf{U} (u_x, u_y, u_z)$ is the mean velocity vector and τ is the Reynolds stresses.

Turbulent models are used in order to capture the effects of turbulence and determine the Reynolds stresses.

In this study, the SST $k-\omega$ turbulent model is used. More information about this model, developed by Menter, is found in literature [6].

The Reynolds number is given by:

$$Re = \frac{UL_c}{\nu} \quad (2)$$

where U is the freestream velocity and L_c is the characteristic length taking as the length of the body.

The drag coefficient is defined by:

$$C_d = \frac{F_d}{\frac{1}{2} \rho U^2 S} \quad (3)$$

where F_d is the drag force and S is the frontal projected area of the body.

3 Test configuration

Figure 1 represents the geometry of the Ahmed body. A rectangular conduit is perforated along the body (Figure 2a). L is its length and l is its width (Figure 2b). For various distances D from the slant lower edge, dif-

ferent tests are performed for various conduit lengths and widths. The conduit is narrowed to accelerate the flow at the exit. L and l are reduced by the same ratio R . For the suitable distance D , various conduit configurations depending on the ratio R are tested. Dimensions are in mm.

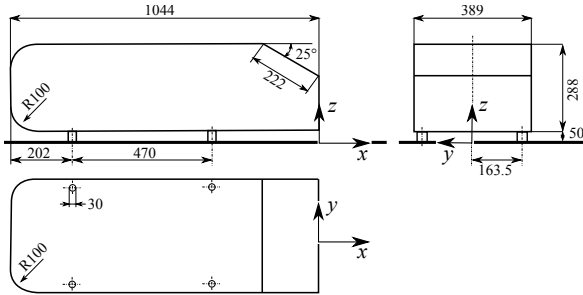


Figure 1. Ahmed body geometry

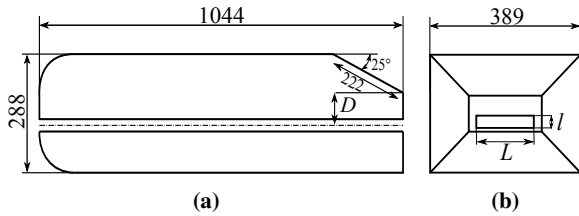


Figure 2. Description of the conduit (a) Conduit location (b) Conduit characteristics

Our numerical simulations are performed using the computational domain described in the figure 3. The blockage ratio is equal to 1.71%. This ensures suitable air growth around the body. Boundary conditions are detailed in the figure 4. The symmetry condition is used to cut the model in half and reduce the computational time. An unstructured tetrahedral mesh of 1 502 900 elements is used. In this study, we consider a viscous and incompressible fluid (air) with a velocity of 40 m/s, a density of 1.225 kg/m³ and a viscosity of 1.7894 × 10⁻⁵ kg/m s.

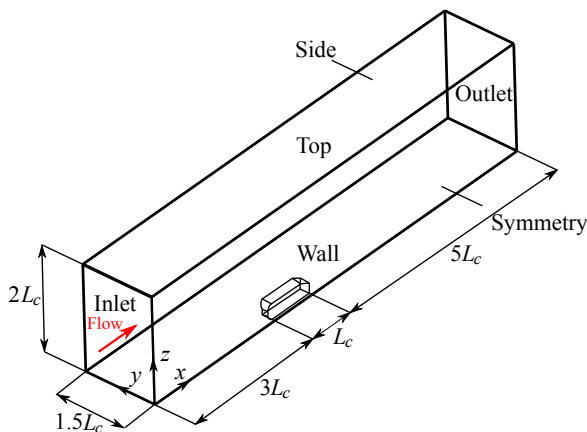


Figure 3. Description of the flow domain

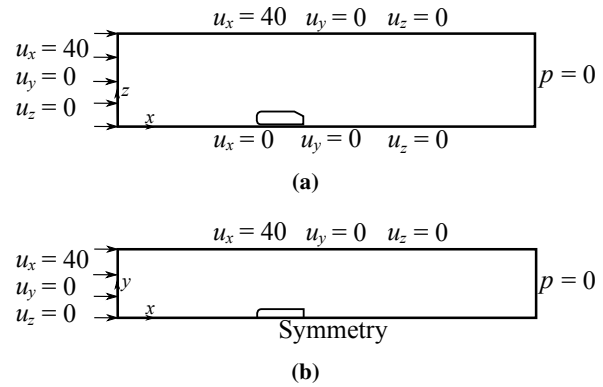


Figure 4. Boundary conditions description (a) Front view (b) Top view

Our simulations are considered in the steady state. The SST $k-\omega$ turbulence model is used. The coupled scheme for the pressure-velocity coupling method is used. Gradients are solved with the Green-Gauss node-based discretization. The pressure is solved with the second order scheme. The momentum and turbulence equations are solved by the second order upwind scheme.

4 Results and discussion

A numerical simulation on the Ahmed body without considering the conduit is performed in order to validate our numerical model with literature results.

Table 1 shows that the results obtained by our simulations are in adequation with previous results.

Table 1. Drag coefficient validation

	Present study	Experimental result [7]	Numerical result [8]
C_d	0.287	0.285	0.3074

In the case of uniform conduits placed at different distances D with respect to the lower edge of the slant, almost all configurations increase the drag coefficient of the Ahmed body. The more the dimensions of the conduits increase, the more the drag coefficients increase for all distances. The distance $D = 0.03$ m is the favorable distance. It allows to have drag coefficients lower than the other distances for all configurations of the conduit (Figure 5). For this, this distance is considered in the next simulations. In these simulations, the effect of narrowing the exit of the conduit on drag coefficient is studied. Several configurations are tested by changing the reduction ratios of the conduit dimensions R (Figure 6). $R = 4$ represents the most favorable ratio especially for the conduit of dimensions ($L = 0.06$ m, $l = 0.02$ m). It allows a maximum drag reduction up to 3%.

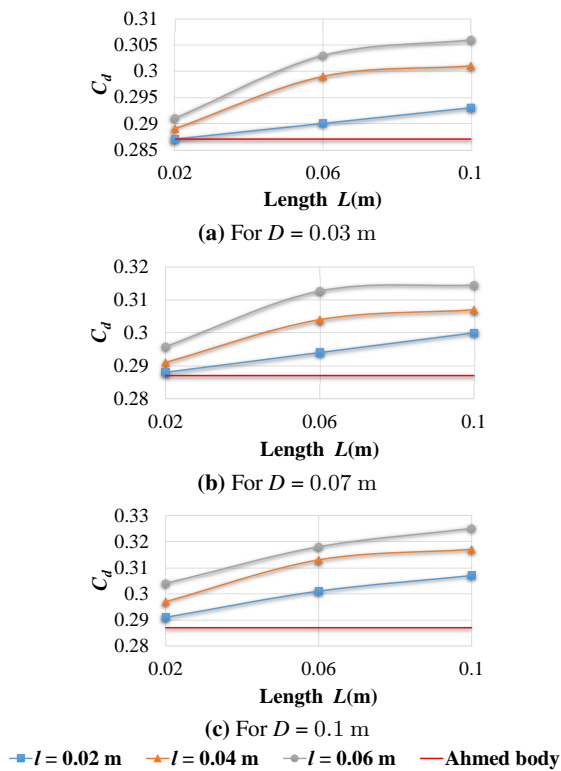


Figure 5. Drag coefficient evolution with the distance D from the slant lower edge for various conduit dimensions

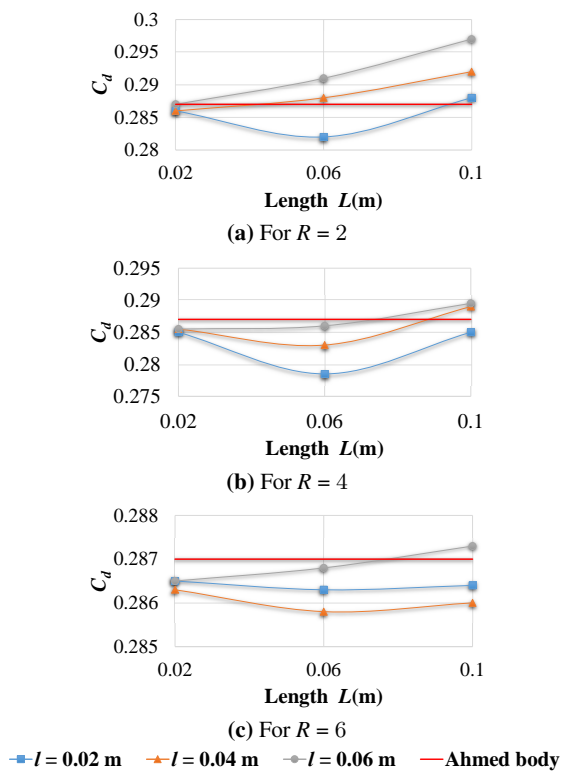


Figure 6. Drag coefficient evolution with the reduction ratio R of the conduit dimensions

5 Conclusion

In this study, numerical simulations have been carried out in order to study the effects of changing the dimensions and the position of a rectangular conduit perforated along the Ahmed body. The aim of this conduit is to benefit from part of the airflow from the front to be injecting behind the body in the recirculation zone. The three-dimensional simulations have been performed using the CFD software ANSYS FLUENT.

Results found in this study lead us to conclude that in certain configurations of the conduit, drag coefficient could be reduced. The favorable configuration allows a maximum drag reduction of 3%.

This work is a continuation of a two-dimensional previous study of this new drag reduction technique. In future works, the focus is on studying the effects of the conduit outlet inclination on drag reduction.

References

- [1] P. Gilliéron and A. Kourta, *drag reduction by vertical splitter plates*, Exp. Fluids 48 (2010) 1–16.
- [2] J. Tian, Y. Zhang, H. Zhu, and H. Xiao, *Aerodynamic drag reduction and flow control of Ahmed body with flaps*, Adv. Mech. Eng. (London, U. K.) 9 (2017) 168 781 401 771 139.
- [3] A. Kourta and C. Leclerc, *Characterization of synthetic jet actuation with application to Ahmed body wake*, Sensors and Actuators A: Physical 192 (2013) 13–26.
- [4] M. Rouméas, P. Gilliéron, and A. Kourta, *Drag reduction by flow separation control on a car after body*, Int. J. Numer. Methods Fluids 60 (2009) 1222–1240.
- [5] A. Agriss, M. Agouzoul, A. Ettaouil, and A. Mehdari, *Numerical study of new techniques drag reduction: application to aerodynamic devices*, Int. J. Simul. Multidisci. Des. Optim., 12 (2021) 16.
- [6] F. R. Menter, *Two-equation eddy-viscosity turbulence models for engineering applications*, AIAA J. 32 (1994) 1598–1605.
- [7] S. R. Ahmed, G. Ramm, and G. Faltin, *Some Salient Features Of The TimeAveraged Ground Vehicle Wake*, SAE Technical Paper (1984) 840 300.
- [8] E. Guilmineau, *Computational study of flow around a simplified car body*, Journal of Wind Engineering and Industrial Aerodynamics 96 (2008) 1207–1217.

Effect of load on minimum film thickness in non-Newtonian MEHD parabolic slider bearings

M. MOUDA¹, M. EL KHLIFI², M. NABHANI²

1. University of Cadi Ayyad, ENSA Safi, 46000 Safi, Morocco

¹moudamouhcine4@gmail.com

2. Hassan II University of Casablanca, Faculty of Sciences and Techniques, Mohammedia 20650, Morocco

Abstract

In this paper, the minimum oil film thickness at a given load is computed for Magneto-elastohydrodynamic (MEHD) parabolic slider bearings lubricated with couple stress fluid. The modified MHD Reynolds equation considering elastic deformation of finite and infinite parabolic slider bearings is derived. This governing equation is solved numerically using finite difference scheme. The resulting algebraic equations are iteratively solved using the Gauss–Seidel method. The results show that the value of minimum film thickness is decreasing with load. In addition, the elastic deformation can influence the minimum film thickness of MHD infinite and finite parabolic slider bearing lubricated with couple stress fluid.

Keywords:

Magneto-elastohydrodynamics; Parabolic Slider Bearings; non-Newtonian lubricant

1. Introduction

MEHD pad thrust bearings are commonly used in to support axial loads in engineering applications. Many authors have investigated the MHD performance of slider bearings with conventional Newtonian lubricant in the presence of applied magnetic field. Snyder¹ was the first to study the characteristics of infinite inclined plane slider bearings. Lin *et al.*² studied the static and dynamic characteristics of one-dimensional slider bearings lubricated with electrically conducting fluid. Hughes³ analysed the characteristics of step slider bearings using an electrically conducting liquid metal lubricant. These studies conclude that the increase in the strength of applied magnetic field increases the load capacity of slider bearings compared to nonmagnetic case.

In reality, applications of non-Newtonian fluids as lubricants have received much interest. Couple stress fluid is a special type of a non-Newtonian fluid, whose particle size is taken into account. Since the classical continuum theory ignore the particle size effects, a micro-continuum theory has been developed by Stokes⁴ to take into account these effects. Several researchers are interested in the study of the combined effect of MHD

and couple stress, such as infinitely long slider bearings by Nabhani and El Khlifi⁵⁻⁶, different types of finite plates by Fathima *et al.*⁷ and non-parallel squeeze film bearings by Mouda *et al.*⁸⁻⁹. They concluded that the effect of couple stress and MHD increases load capacity and reduce friction.

In this research, the effect of load on dimensionless minimum film thickness was investigated for infinite and finite parabolic slider bearings. The two-dimensional MEHD Reynolds equation is derived. The bearing deformation is calculated using the elastic thin layer model. The present results are validated by comparison against available Newtonian published data.

2. Governing equations and numerical resolution

The geometry of parabolic slider bearings (Figure 1), of length L and width B are considered. The lubricant in the

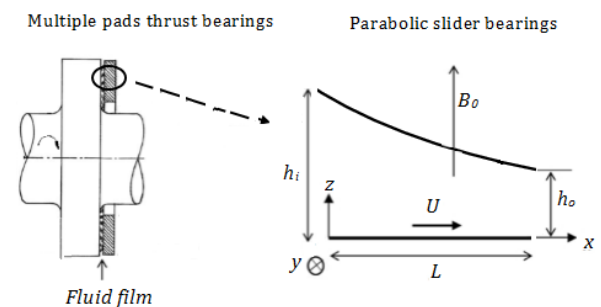


Figure 1. MEHD parabolic bearing configuration.

film region is considered as conducting Stokes couple stress fluid. The lower surface is plane, rigid and has a sliding velocity U while the upper surface is fixed and elastic. Perpendicular to the bearing a uniform transverse magnetic field B_0 is applied. The laminar flow of an isothermal, incompressible electrically conducting fluid is considered in the fluid film region. Assume that the fluid film is thin, the body force is negligible except for the Lorentz force, and the induced magnetic field is much less than the applied field as described by Zikanov and Thess¹⁰. Therefore the dimensionless modified Reynolds equation governing the fluid pressure within finite slider bearings is obtained as follows:

$$\frac{\partial}{\partial x^*} \left(F^* \frac{\partial p^*}{\partial x^*} \right) + \frac{1}{\beta^2} \frac{\partial}{\partial y^*} \left(G^* \frac{\partial p^*}{\partial y^*} \right) = 6 \frac{\partial h^*}{\partial x^*} \quad (1)$$

with:

$$F^* = \frac{6}{M^2} \left(-2h^* + \frac{1}{l^*} \frac{(A_1^{*2} - B_1^{*2})h^{*2}}{A_1^{*2} \tanh\left(\frac{B_1^* h^*}{2l^*}\right) - \frac{B_1^{*2}}{A_1^*} \tanh\left(\frac{A_1^* h^*}{2l^*}\right)} \right)$$

$$G^* = \frac{G}{h_o^3} = h^{*3} - 12l^{*2}h^* + 24l^{*3} \tanh\left(\frac{h^*}{2l^*}\right)$$

in which

$$M = B_0 h_o \sqrt{\frac{\sigma}{\mu}}, \quad l^* = \frac{l}{h_o}, \quad l = \sqrt{\frac{\eta}{\mu}}, \quad x^* = \frac{x}{L}$$

$$A_1^* = \frac{\sqrt{1 + \sqrt{1 - 4M^2 l^{*2}}}}{\sqrt{2}}, \quad B_1^* = \frac{\sqrt{1 - \sqrt{1 - 4M^2 l^{*2}}}}{\sqrt{2}}$$

where p^* is the film pressure, l^* is the couple stress parameter, M represents the Hartmann number, h_o is the minimal film thickness, σ is the electrical conductivity of the fluid, μ is the dynamic viscosity coefficient and η is the material constant of couple stress fluids.

The total dimensionless film thickness h^* , considering the dimensionless elastic deformation of upper surface calculated according to the Winkler model¹¹, can be formulized as follows:

$$h^* = \alpha + (\alpha - 1)(x^{*2} - 2x^*) + C_s p^* \quad (2)$$

in which

$$\alpha = \frac{h_i}{h_o}, \quad C_s = \frac{\mu d L U (1 + \nu)(1 - 2\nu)}{E h_o^3 (1 - \nu)}$$

where E , ν , d , h_i and α are the Young's modulus, the Poisson ratio, thin layer thickness, inlet film thickness and the profile parameter of a parabolic slider bearing respectively and C_s represents the dimensionless compliance coefficient.

In order to get the dimensionless load capacity W_{cal}^* calculated by integration of the dimensionless pressure through the fluid film for a given film thickness h^* , the finite difference method is adopted to discretize Eq. (1). Starting by an initial guess of h_o equal to $2\mu\text{m}$, W_{cal}^* is compared with the imposed load W_{imp}^* on the slider bearing. If the difference between the calculated dimensionless load and the imposed load doesn't meet the accuracy requirement of numeric calculation, Eq. (3) is used to set a new initial dimensionless minimal film thickness h_o^* ($= h_o$ normalized by a value of $2\mu\text{m}$).

$$h_o^* = h_o^* + \omega(W_{cal}^* - W_{imp}^*) \quad (3)$$

ω is under-relaxation factor.

3. Results and discussion

With the aim to validate the present computer program, the load capacity is given in Tables 1 to compare with Naduvinamani et al.¹² in the case of Newtonian MHD parabolic infinite ($\beta \rightarrow \infty$) slider bearing.

$l^* = 0$	Naduvinamani et al. ¹²	Present analysis
$M=0$	0.1722	0.1702
$M=2.5$	0.2087	0.2063
$M=5$	0.2885	0.2849

Table 1. Dimensionless load capacity of parabolic slider bearings for different given values of M with $l^* = 0$ and $C_s = 0$.

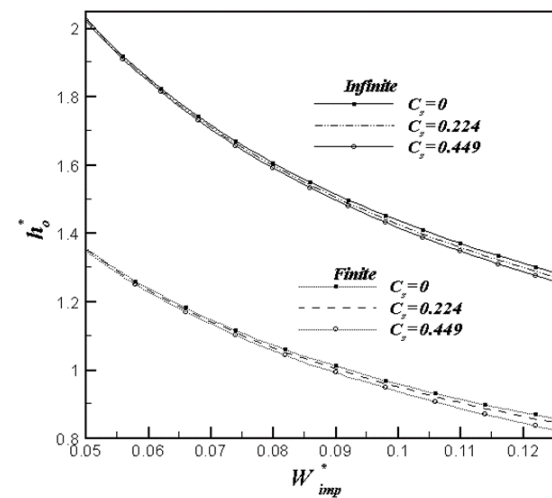


Figure 2. Dimensionless minimum film thickness versus imposed load at $M=2$ and $l^*=0.1$ for different compliance coefficients.

The effect of load on dimensionless minimum film thickness for magneto-elastohydrodynamic infinite and finite parabolic slider bearings lubricated with non-Newtonian couple stress fluids in the presence of transverse externally applied magnetic fields has been studied in this work by varying three dimensionless parameters, namely, the compliance coefficient C_s , Hartmann number M and the couple stress parameter l^* .

Figure 2 presents the variation of dimensionless minimum film thickness of finite and infinite parabolic slider bearings against the imposed load for different values of C_s using a non-Newtonian couple stress lubricant $l^* = 0.1$ and Hartmann number $M = 2$. It can be seen from this figure that increasing the imposed load on the bearing reduces the dimensionless minimum film thickness for all the considered values of C_s . In addition, the results show that the dimensionless minimum film thickness decreases with increasing values of C_s compared to rigid parabolic slider bearings case ($C_s = 0$) for any given value of the imposed load. Thus, considering surface deformation is necessary to predict

more correctly slider bearings performance. In addition, the minimum film thickness of the infinite parabolic slider bearings is higher than that of the finite slider bearings. This is useful for bearings design in industrial applications because higher minimum film thickness will protect lubricated surfaces from direct contact in addition to the fact that infinite parabolic slider bearings supports higher load capacity.

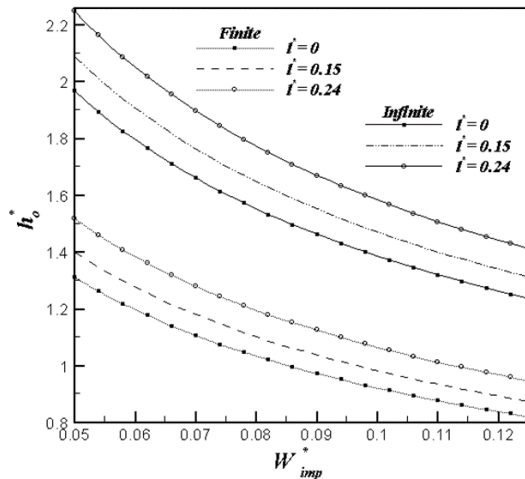


Figure 3. Dimensionless minimum film thickness versus imposed load at $C_s=0.224$ and $M=2$ for different couple stress parameter.

The effects of imposed load on the minimum film thickness considering the couple stresses effect using a compliance coefficients $C_s = 0.224$ and Hartmann number $M = 2$ is depicted in Figure 3. It's observed that using couple stresses lubricants increase the dimensionless minimum film thickness compared to Newtonian lubricant ($l^* = 0$) for any given value of the imposed load. Thus, neglecting non-Newtonian behaviour effects could underestimate slider bearings characteristic. In addition, for each fixed imposed load value the minimum film thickness of the infinite parabolic slider bearing is higher than that of the finite slider bearing.

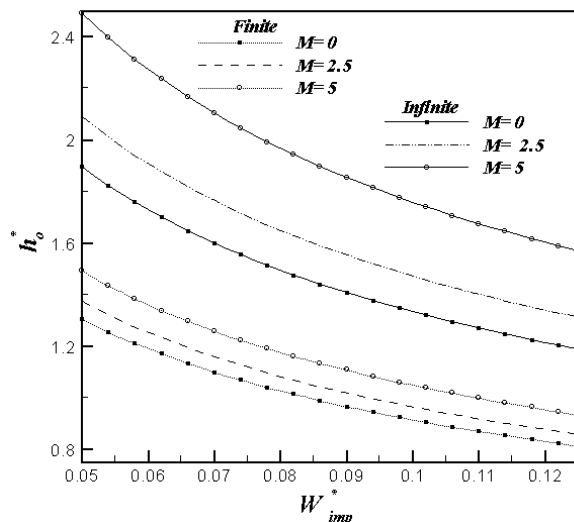


Figure 4. Dimensionless minimum film thickness versus imposed load at $C_s=0.224$ and $l^*=0.1$ for different Hartmann number.

Figure 4 presents the relationship between the load and the minimum film thickness considering the magnetic fields effects using a couple stresses lubricant ($l^* = 0.1$) with a compliance coefficient ($C_s = 0.224$). The influence of transverse magnetic field increases the minimum film thickness of MEHD parabolic slider bearings compared to Non conducting lubricant case.

4. Conclusion

The obtained results present the following conclusions:

1. The numerical simulation analysis shows that increasing the imposed load on the bearings reduces the dimensionless minimum film thickness for all the considered values of C_s , M , l^* .
2. Considering elastic deformation can obviously influence the minimum film thickness of parabolic slider bearings.
3. Using couple stresses lubricants increase the dimensionless minimum film thickness compared to Newtonian lubricant for any given value of the imposed load.
4. Under a certain load, the influence of transverse magnetic field increases the minimum film thickness of MHD parabolic slider bearings compared to non-conducting lubricant case.

Références

- [1] W.T. Snyder, J. Basic. Eng **84**,197-204 (1962)
- [2] J. R. Lin, C. R. Hung, C. H. Hsu, C. Lai, P I Mech Eng J-J Eng **223** 211- 219 (2009)
- [3] W.F. Hughes, J. Basic. Eng **85**, 129-136 (1963)
- [4] V.K. Stokes, Phys. Fluids **9**, 1709–1715 (1966)
- [5] M. Nabhani, M. El Khelifi, Tribol. Trans **57**, 515-523 (2014)
- [6] M. Nabhani, M. El Khelifi, Tribol. Trans **58** 374–383 (2015)
- [7] S. T. Fathima, N. B. Naduvinamani, B. N. Hanumagouda, J. S. Kumar, Tribol. Trans **58** 660-667(2015)
- [8] M. Mouda, M. Nabhani and M. El Khelifi, Ind Lubr Tribol **73**, 45-51 (2020)
- [9] M. Mouda, M. Nabhani and M. El Khelifi, Ind Lubr Tribol **71** 1158-1165 (2019)
- [10] O. Zikanov, A. Thess, J. Fluid Mech **358** 299–333 (1998)
- [11] E. Winkler, *Theil 1-2*, H. Dominicus (Prag, 1867)
- [12] N. B. Naduvinamani, A. Siddangouda and S. Patil, Tribol. Trans **60** 1–11 (2017)

Effect of the wall slip on the stability of a viscous interface under horizontal periodic oscillation

M. ASSOUL¹, A. EL JAOUAHIRY¹, M. ECHCHADLI¹ and S. ANISS¹

1. University of Hassan II, Faculty of Sciences Ain-Chock, Laboratory of Mechanic, B.P.5366 Maarif, Casablanca, Morocco. mouh.assoul@gmail.com

Abstract

We study the role of wall slip on the linear stability of two superposed layers of viscous, immiscible fluids of different densities. The whole system is subjected to horizontal periodic oscillation. The spectral method and Floquet's theory combined with Or's iterative method [6] are used to solve numerically the linear problem. We analyse the influence of the slip length on the marginal stability. The numerical solution shows that the wall slip has a stabilizing or a destabilizing effect on the Kelvin-Helmholtz instability as well as in the parametric resonances.

Keywords *Linear stability; slip boundaries; periodic oscillation; Kelvin-Helmholtz; parametric resonance.*

1 Introduction

The stability of a bilayer flow subjected to horizontal oscillation is a well-known topic in fluid dynamics. One of the most important studies was carried out by Talib and Juel [1] who studied theoretically and experimentally the effect of viscosities of fluid layers of finite depth. They showed that the viscous model is able to predict the occurrence of each instability mode, particularly in the limit of the high viscosity contrast. Later, Lyubimov *et al.* [2] studied the effect of viscosity on the long wave instability of a fluid interface subjected to horizontal vibrations. They found a stabilizing effect of viscosity at low frequencies and a destabilizing effect of viscosity at moderate and high frequencies.

These studies prescribed the use of the no-slip condition at the channel walls, which assumes that the relative velocity at the solid walls becomes zero. While in macroscopic flows, there is ample experimental evidence breaking the validity of the no-slip condition. Ramakrishnan and *al.* [3] studied the role of wall slip on the stability of a two-layer Poiseuille flow, they observed that slip has both a stabilising and destabilising effect on the flow system.

In this work, we extend the study of [1] by examining the effect of wall slip on the linear stability of superposed viscous layers in the presence of periodic oscillation. We focus our attention on the effect of the dimensionless slip

length on the marginal stability in terms of the amplitude of oscillation as a function of the wavenumber.

2 Formulation

Consider two superposed layers of immiscible and incompressible Newtonian fluids, having two kinematic viscosities ν_1 , and ν_2 , densities ρ_1 and ρ_2 and which are bounded below and above by slip boundaries (see Figure 1). The dense fluid occupies the lower region of height d_1 and the least dense fluid occupies the upper region of height d_2 . The system is subjected to an oscillating motion according to the acceleration of oscillation: $\rho_\beta a \omega^2 \cos(\omega t^*)$ and to the force of gravity $\rho_\beta \mathbf{g}$ ($\beta = 1, 2$), where a and ω denote respectively the displacement amplitude and frequency of the oscillatory motion. The interfacial tension between the fluids is denoted by γ .

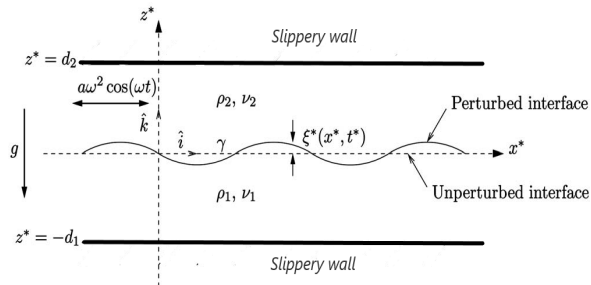


FIG. 1. Schematic diagram of the two-layer fluid system in the frame of reference of the oscillating slip boundaries.

Using d_2 , $a\omega$, ω^{-1} , $\rho_2\nu_2 a\omega/d_2$ as length, velocity, time, and pressure scales, respectively, the dimensionless equations governing the two-layer flows, in the relative frame, are :

$$\Omega \frac{\partial \mathbf{V}_\beta}{\partial t} + A \Omega (\mathbf{V}_\beta \cdot \nabla) \mathbf{V}_\beta = -R_\beta \nabla P_\beta - \frac{G_0 \Omega}{A} \hat{\mathbf{k}} + \frac{1}{N_\beta} \Delta \mathbf{V}_\beta + \Omega (\cos(t)) \hat{\mathbf{i}} \quad (1)$$

$$\nabla \cdot \mathbf{V}_\beta = 0 \quad (2)$$

where $\mathbf{V}_\beta = (U_\beta, W_\beta)$ is the velocity in each fluid layer, $\Omega = (\omega d_2^2 / \nu_2)$ and $A = a/d_2$ are the dimensionless frequency and amplitude of oscillation, respectively, $R_\beta = \rho_2 / \rho_\beta$ and $N_\beta = \nu_2 / \nu_\beta$ are the ratios of densities and viscosities, respectively, and $G_0 = g / (d_2 \omega^2) = A^2 Fr^{-1}$

where $Fr = (a\omega)^2/gd_2$ is a modified inverse Froude number. The slip boundary conditions and the impermeability conditions are given by :

$$(\mathbf{V}_1 \cdot \hat{\mathbf{i}}) = \alpha \frac{\partial(\mathbf{V}_1 \cdot \hat{\mathbf{i}})}{\partial z} \quad \text{and} \quad \mathbf{V}_1 \cdot \hat{\mathbf{k}} = 0 \quad \text{at} \quad z = -d, \quad (3)$$

$$(\mathbf{V}_2 \cdot \hat{\mathbf{i}}) = -\alpha \frac{\partial(\mathbf{V}_2 \cdot \hat{\mathbf{i}})}{\partial z} \quad \text{and} \quad \mathbf{V}_2 \cdot \hat{\mathbf{k}} = 0 \quad \text{at} \quad z = 1, \quad (4)$$

The kinematic condition, the continuity of velocity, and the balance of normal and tangential stresses at the interface $\xi(x, t)$ are given by :

$$\frac{1}{A} \frac{\partial \xi}{\partial t} + (\mathbf{V}_1 \nabla) \xi = \mathbf{V}_1 \cdot \hat{\mathbf{k}}, \quad (5)$$

$$\mathbf{V}_1 = \mathbf{V}_2, \quad (6)$$

$$(\mathbf{n} \cdot \boldsymbol{\pi}_1) \cdot \mathbf{n} - (\mathbf{n} \cdot \boldsymbol{\pi}_2) \cdot \mathbf{n} = \frac{\Omega}{WeA} \nabla \cdot \mathbf{n}, \quad (7)$$

$$(\mathbf{t} \cdot \boldsymbol{\pi}_1) \cdot \mathbf{n} = (\mathbf{t} \cdot \boldsymbol{\pi}_2) \cdot \mathbf{n}, \quad (8)$$

The flux closure condition imposes an additional constraint on the velocity :

$$\int_{-d}^{\xi} \mathbf{V}_1 \cdot \hat{\mathbf{i}} dz + \int_{\xi}^1 \mathbf{V}_1 \cdot \hat{\mathbf{i}} dz = 0. \quad (9)$$

where $d = \frac{d_1}{d_2}$, $\pi_\beta = -P_\beta I + \frac{1}{R_\beta N_\beta} [\nabla \mathbf{V}_\beta + (\nabla \mathbf{V}_\beta)^T]$ is the stress tensor in each fluid, $\alpha = \frac{\lambda}{d_2}$ is the dimensionless slip length where the dimensional slip length λ is defined as the equivalent local distance below the rigid surface where the no-slip condition can be satisfied if the flow field is extended outside the physical domain. $We = \rho_2 d_2^3 \omega^2 / \gamma$ is the Weber number, \mathbf{n} is the outward normal unit vector pointing from fluid 1 into fluid 2 and \mathbf{t} is the tangent unit vector on the interface.

3 Base flow solution

The base flow, $(\bar{V}_\beta, \bar{P}_\beta)$, is periodic, parallel to the horizontal boundaries and the interface remains unperturbed. It takes the form :

$$\bar{U}_\beta(z, t) = \Re [\Phi_\beta(z) \cdot e^{it}] \quad (10)$$

$$\bar{P}_\beta = \Re \left[-\frac{G_0 \Omega}{R_\beta A} z + x S_\beta e^{it} + C \right] \quad (11)$$

where

$$\Phi_\beta(z) = A_\beta e^{m_\beta z} + B_\beta e^{-m_\beta z} + i \left(\frac{R_\beta S_\beta}{\Omega} - 1 \right)$$

$$m_\beta = \sqrt{i \Omega N_\beta}$$

The integration constants A_β , B_β , and S_β are determined by using conditions (3),(4),(6),(7),(8) and (9) and C is

an arbitrary constant.

Note that for no-slip conditions case, $\alpha = 0$, we obtain the same profiles of velocity determined by Talib and *al.* [1].

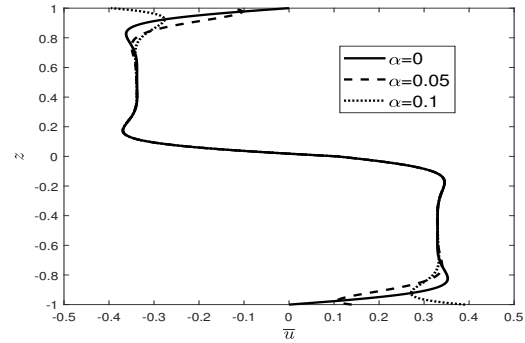


FIG. 2. Base flow solution at $t = \pi/2$, calculated for $R_1 = 0.5$, $d = 1$, $N_1 = 1$, $\Omega = 360$ and for different slip length.

4 Perturbation equations

We derive the stability equations by expressing the governing equations (1) and (2) in terms of the stream function, ψ_β , defined as :

$$u_\beta = \frac{\partial \psi_\beta}{\partial z} \quad \text{and} \quad w_\beta = -\frac{\partial \psi_\beta}{\partial x}$$

The small normal perturbation of stream function ψ_β and the interface position ξ are searched in the following form :

$$(\psi_\beta, \xi) = (\bar{\psi}_\beta, 0) + [\phi_\beta(z, t), h(t)] e^{ikx} + c.c., \quad (12)$$

where k is the dimensionless wavenumber of the disturbance and *c.c.* denotes the complex conjugate. By substituting (12) into the Navier-Stokes equations, as well as the boundary and interfacial conditions, subtracting out the base state, eliminating the pressure term, and neglecting higher order terms, we obtain the following Orr-sommerfeld equations for the two layer flows :

$$\left(\Omega \frac{\partial}{\partial t} + ikA\Omega \bar{U}_\beta \right) (\phi''_\beta - k^2 \phi_\beta) - ikA\Omega U''_\beta \phi_\beta + \frac{1}{N_\beta} (2k^2 \phi''_\beta - k^4 \phi_\beta - \phi''''_\beta) = 0 \quad (13)$$

where the prime ($'$) denotes partial differentiation with respect to z . The slip boundary conditions become :

$$\phi_1 = 0 \quad \text{at} \quad z = -d, \quad \phi_2 = 0 \quad \text{at} \quad z = 1 \quad (14)$$

$$\phi_1' = \alpha \phi_1'' \quad \text{at} \quad z = -d, \quad \phi_2' = -\alpha \phi_2'' \quad \text{at} \quad z = 1 \quad (15)$$

The linearized boundary conditions at the interface located at $z = 0$ become :

$$\frac{1}{A} \frac{\partial h}{\partial t} + ik\bar{U}_1 h + ik\phi_1 = 0 \quad (16)$$

$$\phi'_1 - \phi'_2 + h(\bar{U}'_1 - \bar{U}'_1) = 0 \quad (17)$$

$$\phi_1 - \phi_2 = 0 \quad (18)$$

$$\left[\frac{\Omega}{R_1} \frac{\partial \phi'_1}{\partial t} - \frac{ikA\Omega}{R_1} \bar{U}'_1 \phi_1 + \frac{ikA\Omega}{R_1} \bar{U}_1 \phi'_1 + \frac{3k^2}{N_1 R_1} \phi'_1 - \frac{1}{N_1 R_1} \phi'''_1 + \frac{ikG_0\Omega}{AR_1} h \right] - \left[\Omega \frac{\partial \phi'_2}{\partial t} - ikA\Omega \bar{U}'_2 \phi_2 + ikA\Omega \bar{U}_2 \phi'_2 + 3k^2 \phi'_2 - \phi'''_2 + \frac{ikG_0\Omega}{A} h \right] + \frac{ik^3\Omega}{\hat{W}_e A} h = 0 \quad (19)$$

$$\phi''_1 + h\bar{U}''_1 + k^2\phi_1 - N_1 R_1 (\phi''_2 + h\bar{U}''_2 + k^2\phi_2) = 0 \quad (20)$$

5 Numerical procedure

To solve equations (13)-(20), we use the spectral methods based on the Gauss-Lobatto collocation points for spatial resolution [3, 4]. The Chebyshev polynomials are defined in the interval $[-1, 1]$. In order to achieve high resolution near the boundaries and the interface, the lower and upper layers are mapped onto Chebyshev space, $\zeta \in [-1, 1]$, with the transformations $\zeta = \frac{2z+d}{d}$ and $\zeta = 1 - 2z$, respectively, so that the interface is placed at $\zeta = 1$. The time-dependent system was solved using a fast iterative scheme based upon the Newton-Raphson method that was developed by Or [6].

6 Results and conclusion

We present in figures 3(a) and 3(b) the marginal stability curves in terms of oscillation amplitude versus the wavenumber. In the case of a flow system with equal viscosities, $N_1 = 1$, and equal thickness ratio, $d = 1$, it is observed that the Kelvin-Helmholtz region undergoes an upward translation when increasing the slip length, α , in the case of small viscosities, $\Omega = 360$, and large viscosities, $\Omega = 25$. Thus, the slip length α has a stabilizing effect. However, in the parametric region, the effect of the slip length is only observed for large viscosities, $\Omega = 25$. On the other hand, for high viscosities, we notice that an increase in slip length acts as a weakly destabilising effect on the interface.

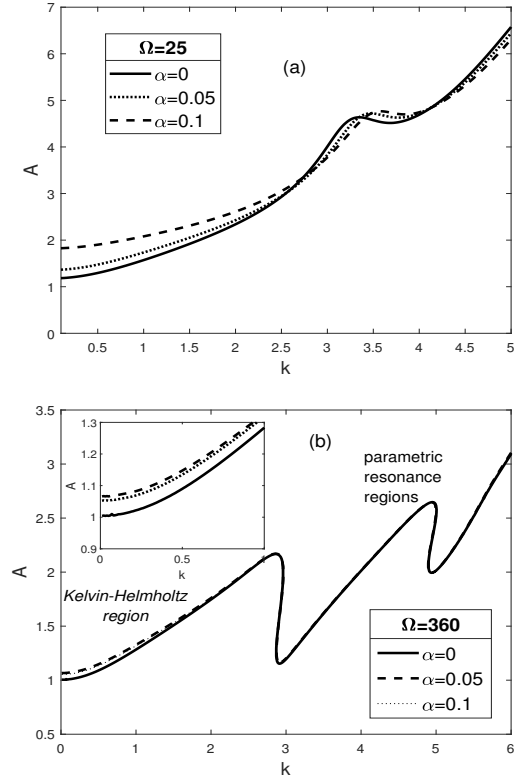


FIG. 3. Marginal stability curves showing amplitude, A , against the wavenumber, k for $\Omega = 25$ (a), $\Omega = 360$ (b) and $d = 1$. The other parameter values are $R_1 = 0.5$, $N_1 = 1$, $G_0 = 0.16$ and $W_e = 6.25$.

References

- [1] E. Talib and A. Juel, *Instability of a viscous interface under horizontal oscillation*, Phys. Fluids 19, (2007), 092102.
- [2] D. V. Lyubimov, G. L. Khilko, A. O. Ivantsov, and T. P. Lyubimova, *Viscosity effect on the longwave instability of a fluid interface subjected to horizontal vibrations*, J. Fluid Mech. 814, 24–41 (2017).
- [3] V. Ramakrishnan, R. Mushthaq, A. Roy and S. Vengadesan, *Stability of two-layer flows past slippery surfaces. I. Horizontal channels*, Phys. Fluids 33, 084112 (2021).
- [4] L. N. Trefethen, *Spectral Methods in Matlab*, SIAM, Philadelphia, 2000.
- [5] A. C. Weideman, S. C. Reddy, *A MATLAB differentiation matrix suite*, ACM Trans. Math. Softw. 26, 465-519, 2000.
- [6] A. C. Or, *Finite wavelength instability in a horizontal liquid layer on an oscillating plane*, J. Fluid Mech. 335, 213 1997.

Numerical model of cavitation in periodic squeeze film between two discs

S. SOBHI, M. NABHANI, M. EL KHLIFI

Hassan II University of Casablanca, FST, PO Box 146, 20650 Mohammedia, Morocco

Abstract

In lubrication problems, which concern thin film flow, cavitation has been considered as a fundamental element to correctly describe the characteristics of lubricated mechanisms. Cavitation effects on oscillatory squeeze film performance have often been a crucial area of interest in engineering mechanics. This work presents a numerical model to investigate cavitation effects on oscillatory squeeze film between two rigid circular discs. The mass conservative Elrod-Adams model ($p - \theta$) is used to treat cavitation, which incorporates the Reynolds equation into a single formulation. The equations are discretized using finite differences method and solved iteratively by Gauss-Siedel. The results show that frequency and amplitude have a significant influence on the generation of cavitation.

Keywords: *Cavitation, Elrod-Adams model, Oscillating squeeze film, Numerical model.*

1. Introduction

The oscillating squeeze film lubrication mechanism is omnipresent in many technical components in mechanical engineering, such as clutches, bearings, seals, shock absorbers, gaskets, computer hard disks, dampers, etc. In such cases, a positive and a negative load can act by turns, and positive and negative squeeze occurs alternatively. Work in this area has been undertaken by a number of researchers [1,2]. The analysis is easy in the case of positive squeeze; however, in the case of negative squeeze, cavitation may occur in the fluid film and the analysis becomes complex. To describe the cavitation boundary conditions the flow in the cavity region is given by the Jakobsson-Floberg-Olsson theory of cavitation [3,4], which ensures mass conservation. Elrod [5] applied this theory to develop a mass conservation algorithm that is valid throughout the bearing and implicitly and automatically finds the rupture and reformation limits. Another model to describe cavitation is the Elrod-Adams model [6]. In that work, the authors introduce the hypothesis that the cavitation region is a fluid-air mixture and an additional unknown θ appears (the saturation of fluid in the mixture). Hajjam *et al.* [7] and Zarbane *et al.* [8] presented a numerical and experimental study on the behavior of squeeze film under periodic loading. Their theoretical study was based on Elrod-Adams model. The investigations showed that when the fluid film is periodically compressed, the loading capacity is proportional to the frequency of oscillations, while being inversely proportional to the average film thickness. They also demonstrated that the shape obtained for the

cavitation zone due to the introduction of ambient air depends on the frequency, the height of the fluid film, and the geometry of the contact. The mass conservative Elrod-Adams model ($p - \theta$) is used to treat cavitation, which incorporates the Reynolds equation into a single formulation. The equations are discretized using finite differences method and solved iteratively by Gauss-Siedel. Cavitation periodic squeeze film characteristics are presented and analyzed.

2. Analysis

Figure 1 shows a schematic diagram of the physical configurations of the periodic squeeze film investigated in the present study. The two rigid discs have the same radius R and separated by a Newtonian lubricant fluid film. The upper disk oscillates while the lower disc is attached to a rigid support. The surfaces of the two discs are parallel to each other and initially separated by a distance h_0 . The upper rigid disc oscillates sinusoidally with frequency f , then the film thickness h and the squeezing velocity \dot{h} will, respectively, be given by:

$$h(t) = h_0 + h_1 (\cos(2\pi ft) - 1) \quad (1)$$

$$\dot{h} = \frac{\partial h}{\partial t} = -2\pi f h_1 \sin(2\pi ft) \quad (2)$$

where h_1 is the rigid disc oscillation amplitude.

The lubricant in the film is taken to be an incompressible Newtonian with constant properties. The flow is also assumed to be axisymmetric. Applying the hydrodynamic lubrication assumption for thin films, and neglected both local and convective inertia terms, the equations of motion in r and y direction can be expressed, respectively, as:

$$\frac{\partial p}{\partial r} = \mu \frac{\partial^2 u}{\partial r^2} \quad (3)$$

$$\frac{\partial p}{\partial z} = 0 \quad (4)$$

The continuity equation is:

$$\frac{1}{r} \frac{\partial(ru)}{\partial r} + \frac{\partial w}{\partial r} = 0 \quad (5)$$

where u and w are the fluid velocity components in cylindrical coordinates, μ is the dynamic viscosity.

The fluid particles satisfy the following conditions at the discs surfaces:

- On lower disc, $z = 0$:
 $u = 0, \quad v = 0$ (6)
- On the upper, $z = h$:

$$u = 0, \quad v = \dot{h} \quad (7)$$

Solving motion equation 3 by applying the above boundary conditions, the component of velocity vector u is derived:

$$u(r, z) = \frac{1}{2\mu} \frac{dp}{dr} (z^2 - zh) \quad (8)$$

Substituting this velocity component u into continuity equation (5) and integrating with z , accounting for the corresponding boundary conditions, one can obtain the Reynolds equation in the cylindrical coordinates:

$$\frac{1}{r} \frac{d}{dr} \left(r \frac{dp}{dr} \right) = \frac{12\mu}{h^3} \frac{dh}{dt} \quad (9)$$

When cavitation appears, the fluid film domain is divided into two sub-domains, the pressurised Ω_p and the cavitation Ω_θ . The first one is associated with a completely filled gap $\theta = 1$ and the unknown pressure p . In the second, the pressure is supposed to be $p = p_{cav}$, while θ has to be determined. The mass-conserving model in the $p - \theta$ form proposed by Elrod and Adams [6] is used. The Reynolds Eq. (9) reads, under consideration of this model, as follows:

$$\frac{1}{r} \frac{d}{dr} \left(r \frac{dp}{dr} \right) = \frac{12\mu}{h^3} \frac{d(\theta h)}{dt} \quad (10)$$

with

$$p \geq p_{cav} \quad \theta = 1 \quad \text{in full film region} \quad (11)$$

$$\theta < 1 \quad p = p_{cav} \quad \text{in cavitation region} \quad (12)$$

The boundary conditions associated to the modified Reynolds equation are:

- On the symmetrical axis, $r = 0$:

$$\frac{\partial p}{\partial r} = 0 \quad (13)$$

- At the periphery, $r = R$, the pressure is taken equal to the ambient pressure:

$$\begin{aligned} p &= p_a & \text{if } u_m &\geq 0 \\ p &= p_a - \Delta p & \text{if } u_m < 0 \end{aligned} \quad (14)$$

Here, p_a stands for the ambient pressure, which is assumed to be equal to the atmospheric pressure. The boundary condition (14) indicates that when the surrounding fluid enters the clearance gap, the fluid is accelerated radially and thereby the fluid pressure at $r = R$ becomes lower than the ambient pressure by an amount Δp . This pressure drop Δp can be expressed as:

$$\Delta p = \rho(1 + \xi) \frac{u_m^2}{2} \quad (15)$$

where ξ denotes the inlet loss coefficient and is empirically given a value of 0 to 1 [9]. In the present analysis, a value of 0.5 is assumed for ξ . Moreover, u_m is the radial velocity averaged across the film thickness at

$r = R$. It may be determined by flow rate balance across the boundaries of the clearance gap as:

$$u_m = -\frac{1}{Rh} \int_0^R wr dr = -\frac{R^2}{2Rh} \frac{\partial h}{\partial t} = -\frac{VR}{2h} \quad (16)$$

Using an implicit scheme, the following discrete form of equation (10) is obtained:

$$r_i \frac{p_{i+1}^n - p_i^n + p_{i-1}^n}{\Delta r^2} + \frac{p_{i+1}^n - p_i^n}{\Delta r} = \frac{12\mu r_i}{(h^n)^3} \left(\frac{\theta_i^n h^n - \theta_i^{n-1} h^{n-1}}{\Delta t} \right) \quad (17)$$

where h^n , p_i^n , and θ_i^n are values of h , p and θ respectively on grid point $r_i = (i-1)\Delta r$ at time $t_n = n\Delta t$, Δr is the grid spacing in r -direction, Δt is time step and n is time index.

This resulting system of algebraic equations is simultaneously solved for both cavitated and non-cavitated regions using the Gauss-Seidel iterative method with over-relaxation coefficient. Starting by atmospheric pressure in the film as initial condition, at each approximation of (θ_i^n, p_i^n) , we resolve Eq. (17) in the following way:

- if $p_i^n > p_{cav}$
then $\theta_i^n = 1$
and

$$p_i^n = -\frac{1}{\Delta r + r_i} \frac{12\mu r_i \Delta r^2}{(h^n)^3} \left(\frac{\theta_i^n h^n - \theta_i^{n-1} h^{n-1}}{\Delta t} \right) + \frac{r_i p_{i-1}^n}{\Delta r + r_i} + p_{i+1}^n \quad (18)$$

- else $p_i^n = p_{cav}$.

- if $\theta_i^n < 1$

- then $p_i^n = p_{cav}$
and

$$\theta_i^n = \frac{\Delta t}{12\mu r_i \Delta r^2 h^n} \left[r_i (h^n)^3 p_{i-1}^n + \left(\Delta r (h^n)^3 + r_i (h^n)^3 \right) p_{i+1}^n \right. \quad (19)$$

$$\left. - \left(\Delta r (h^n)^3 + 2r_i (h^n)^3 p_i^n \right) \right] + \theta_i^{n-1} \frac{h^{n-1}}{h^n}$$

- else $\theta_i^n = 1$.

At each time step, the iterative process is continued until the following conditions are satisfied:

$$\|p^m - p^{m-1}\| + \|\theta^m - \theta^{m-1}\| \leq 10^{-5} \quad (18)$$

where $\|\cdot\|$ is the Euclidean norm and m denotes the current iteration number. Once this convergence criterion is satisfied; the load capacity of the fluid film is calculated by numerical integration.

3. Results

The fluid film pressure evolution at the disc center for different amplitude values are presented in Figure 2. As shown, positive pressure is developed in the fluid film during the compression action and a negative one is developed when the surfaces separate. The figure also reveals that the pressure magnitude of the fluid film increases with the increase of the amplitude value. This is

foreseeable since the pressure of the fluid film is proportional to the squeeze velocity that is in turn proportional to the amplitude. For the amplitudes of 0.3 mm and 0.35 mm, the hydrodynamic pressure is almost symmetrical because there is no cavitation. The appearance of a horizontal step in the pressure curve for $h_1 = 0.4$ mm indicates the beginning of the cavitation phenomenon in the negative zone in the phase of separation of the upper disc from the lower disc. Likewise, Figure 3 shows the same trends on the variation of hydrodynamic force with time for different amplification values. Due to the cavitation effect, a small deviation is noticed exclusively at the level of amplitude $h_1 = 0.4$ mm.

4. Conclusion

In this study, a numerical model describing cavitation effects on the behavior of a squeeze film between a sinusoidally oscillating rigid disc and a stationary disc have been developed. It is based on Elrod-Adams algorithm into Reynolds equation modified to predict the cavities formation. With this phenomenon taken into account, the results concluded that the pressure magnitude, increases with the increase of the amplitude value. The appearance of a horizontal step in the pressure curve indicates the occurrence of the cavitation phenomenon in the phase of separation of the upper disc from the lower disc. Therefore, considering cavitation phenomena in this model will lead to a more accurate prediction of squeeze film properties and could be more useful for the analysis of the squeeze film oscillating between two rigid surfaces in engineering component design.

References

- [1] D. W. Parkins and W. T. Stanley, *Characteristics of an Oil Squeeze Film*, ASME. J. of Lubrication Tech, 104 (1982) pp. 497–502.
- [2] W. A. Wolfe, *Squeeze film pressures*, Appl. Sci. Res. 14 (1965) pp. 77–90.
- [3] B. Jakobsson, *The finite journal bearing considering vaporization*, Trans Chalmers Univ Technol, 190 (1957) pp.1-116.
- [4] K. Olsson, *Cavitation in dynamically loaded bearings*, Trans Chalmers Univ Technol, 308 (1965) pp. 1-60.
- [5] H. G. Elrod, *Cavitation algorithm*, ASME Journal of Lubricant Technology, 103 (1981) pp. 350-354.
- [6] H. G. Elrod, and M.L. Adams, *A Computer Program for Cavitation and Starvation Problems*, Proc. 1st Leeds-Lyon Symposium on Tribology, Leeds University, England, 103 (1974) pp. 37–41.
- [7] M. Hajjam, and D. Bonneau, *A transient finite element cavitation algorithm with application to radial lip seals*, Tribology International, 40 (2007) pp. 1258–1269.
- [8] K. Zarbane, T. Zeghloul, and M. Hajjam, *Experimental study of lubricant film behaviour subject to periodic loading*, Proceedings of the Institution of Mechanical Engineers, Part J:

Journal of Engineering Tribology, 227 (2013) pp. 538–547.

- [9] Y. Hori, T.Kato, and H. Narumiya, *Rubber surface squeeze film*, ASME J Lubrication Technology, 103 (1981) pp. 398–405.

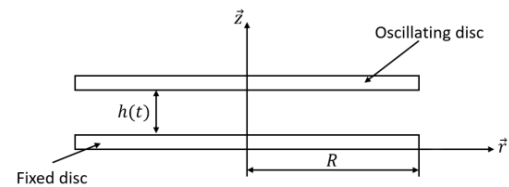


Figure 1: Simulated geometrical oscillatory squeezing configuration

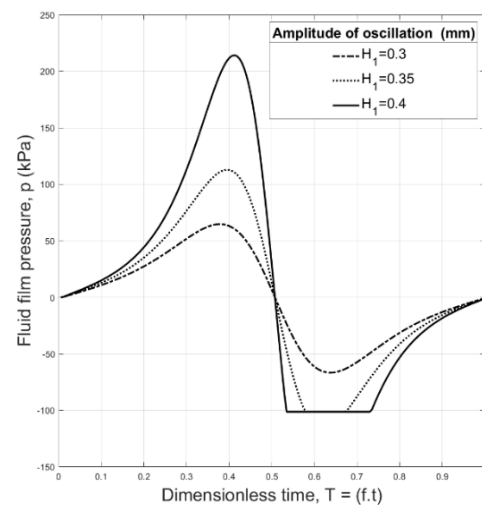


Figure 2: Effect of variation of amplitude on pressure of the fluid film at the center ($h_0 = 1.2$ mm, $f = 20$ Hz)

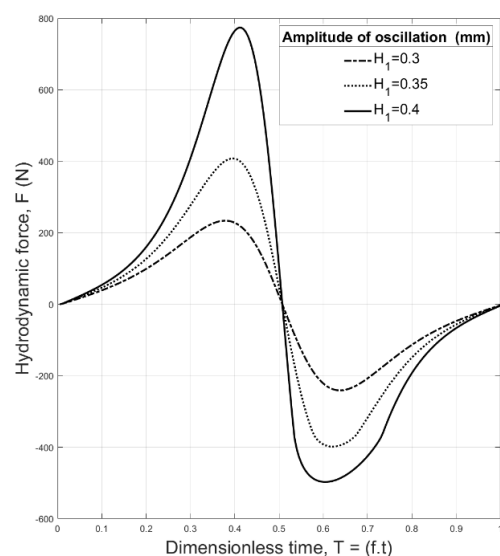


Figure 3: Effect of variation of amplitude on hydrodynamic force ($h_0 = 1.2$ mm, $f = 20$ Hz)

Viscous shear effects on the performance of elastic porous bearings

A. SAKIM¹, M. NABHANI², M. EL KHLIFI²

1. Hassan II University of Casablanca, ENSAM Casablanca, Avenue Nile 150, 20670 Casablanca, Morocco

2. Hassan II University of Casablanca, FST Mohammedia PO Box 146, 20650 Mohammedia, Morocco

Abstract

The objective of this study is to predict the viscous shear effects on finite porous elastic journal bearings lubricated with non-Newtonian couple stress fluid. Based on Stokes micro-continuum mechanics, the modified Reynolds equation including bearing deformation was derived. The porous flow was modeled by the complete Darcy–Brinkman equation. The bearing characteristics including load capacity and friction factor are compared to those obtained from Darcy model with Beavers–Joseph slip conditions (slip flow model (SFM)). It is found that the viscous shearing forces effects increase the load capacity and friction factor compared to those derived using SFM.

Keywords: *Couple stress lubricants, Porous elastic bearings, Viscous shear effects.*

1. Introduction

Porous journal bearings (PJBs) are broadly used in numerous industrial applications such as jet engine manufacturers, compressors and turbines. The most important advantage of these sorts of bearings is that they need no exterior oil supply. Low friction factor makes them very desirable in industry. Traditionally, the study of such bearings was based on the Darcy model (DM), with no-slip condition at the film-porous bearing interface. Sakim *et al.* [1] presented a primary study on finite flexible PJBs using Darcy model with Beavers–Joseph slip conditions [2]. According to their results, the deformation effects influence the porous bearing characteristics especially when they operate at high eccentricities. It was found that couple stresses lubricants improve the porous bearing characteristics. The purpose of this study is to examine how the viscous shear forces of the Brinkman model (BM) affect finite porous elastic journal bearings lubricated by non-Newtonian couple stresses fluids.

2. Analysis

The geometrical configuration showed in Figure 1 is considered. The journal of radius R is rotating with a uniform tangential velocity U about its axis within a porous bearing of initial thickness H_0 , permeability k , Young's modulus E and of Poisson's ratio ν . The bearing and the journal are supposed to be aligned, and

separated by a non-Newtonian lubricant fluid film of thickness h .

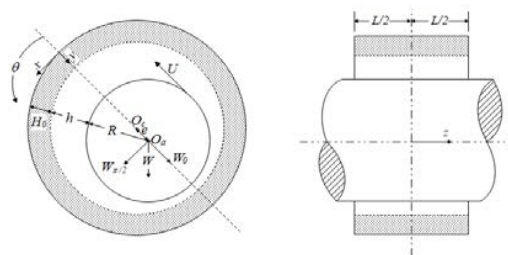


Figure 1: Porous journal bearing configuration.

This geometrical complexity is difficult to analyze, because of the deformation of the film-bearing interface. A change of variables on the radial coordinates in the film and porous bearing allows positioning the external bearing surface, the film-bearing interface and the journal interface at $\bar{r} = 0$, $\bar{r} = 1$ and $\bar{r} = 2$ respectively. The equations and boundary conditions to be solved are given below using the transforming equations:

- In the porous elastic bearing: $0 \leq \bar{r} \leq 1$

$$\frac{\partial}{\partial \theta} = \frac{\partial}{\partial \theta} - \frac{\bar{r}}{H} \frac{\partial \bar{H}}{\partial \theta} \frac{\partial}{\partial \bar{r}}$$

$$\frac{\partial}{\partial r} = -\frac{1}{H_0 \bar{H}} \frac{\partial}{\partial \bar{r}} \quad (1)$$

$$\frac{\partial}{\partial z} = -\frac{\bar{r}}{HL} \frac{\partial \bar{H}}{\partial z} \frac{\partial}{\partial \bar{r}} + \frac{1}{L} \frac{\partial}{\partial \bar{z}}$$

- In the fluid film: $1 \leq \bar{r} \leq 2$

$$\frac{\partial}{\partial x} = \frac{1}{R} \frac{\partial}{\partial \theta} - \frac{1}{CRh} \left(C \frac{\partial \bar{h}}{\partial \theta} (\bar{r} - 1) + H_0 \frac{\partial \bar{H}}{\partial \theta} \right) \frac{\partial}{\partial \bar{r}}$$

$$\frac{\partial}{\partial y} = \frac{1}{Ch} \frac{\partial}{\partial \bar{r}} \quad (2)$$

$$\frac{\partial}{\partial z} = \frac{1}{CLh} \left(C \frac{\partial \bar{h}}{\partial z} (1 - \bar{r}) - H_0 \frac{\partial \bar{H}}{\partial z} \right) \frac{\partial}{\partial \theta} + \frac{1}{L} \frac{\partial}{\partial \bar{z}}$$

where $\bar{h}(\theta, \bar{z}) = 1 + \varepsilon \cos \theta + C_0 \bar{p}$, $\bar{H}(\theta, \bar{z}) = 1 - (C_0/\Gamma) \bar{p}$ are the dimensionless film and porous thicknesses, respectively. C_0 represents the compliance coefficient and $\Gamma = H_0/C$ is the liner dimensionless thickness.

The porous elastic bearing is considered homogeneous, isotropic and fully saturated with an incompressible Newtonian fluid having the same viscosity as the fluid film lubricant. The equations for the velocity components are given by the complete Darcy–Brinkman equation [3]

and are subsequently writing in dimensionless form. Due to continuity of fluid inside the porous elastic bearing, the pressure \bar{p}^* will satisfy Laplace equation. The dimensionless equations obtained are too long and depends on $B_{i=1,\dots,9}$, $C_{i=1,2}$ and $D_{i=1,2}$ coefficients.

Additional information about these coefficients can be found in [4]. The lubricant in the film region is taken to be an incompressible non-Newtonian couple stress fluid. The Stokes micro-continuum mechanics [5] is used for modeling the lubricant rheological behavior with constant properties in absence of body forces and body couples. It is assumed that the flow is laminar, isothermal and steady. Moreover, the fluid film is thin as compared to the journal radius, and fluid inertia is small compared to the viscous shear. Under these assumptions, a reduced form of the momentum equation is obtained. Integrating this equation with the following boundary conditions: continuity of the velocity field at the porous elastic interface, vanishing of couple stresses on the journal surface and porous elastic interface, no-slip boundary conditions at the journal surface, the velocity components of the lubricant are solved. Substituting the expressions of velocity components into continuity equation and integrating across the film thickness, one has the dimensionless EHD modified Reynolds equation:

$$\begin{aligned} & \frac{\partial}{\partial \theta} \left(\bar{f}(\bar{h}, \bar{l}) \frac{\partial \bar{p}}{\partial \theta} \right) + \kappa^2 \frac{\partial}{\partial \bar{z}} \left(\bar{f}(\bar{h}, \bar{l}) \frac{\partial \bar{p}}{\partial \bar{z}} \right) = \kappa v_{\bar{r}=1}^* \\ & + \frac{\partial \left(\frac{\bar{h}}{2} (1 + u_{\bar{r}=1}^*) \right)}{\partial \theta} + \Gamma (u_{\bar{r}=1}^* - 1) \frac{\partial \bar{H}}{\partial \theta} + \kappa \frac{\partial \left(\frac{\bar{h}}{2} w_{\bar{r}=2}^* \right)}{\partial \bar{z}} \\ & + \kappa \Gamma w_{\bar{r}=1}^* \frac{\partial \bar{H}}{\partial \bar{z}} \end{aligned} \quad (3)$$

$$\text{where} \quad \bar{f}(\bar{h}, \bar{l}) = \left(\frac{\bar{h}^3}{12} + 2\bar{l}^3 \tanh \left(\frac{\bar{h}}{2\bar{l}} \right) - \bar{h}\bar{l}^2 \right)$$

In the present work the velocity components $u_{\bar{r}=1}^*$, $v_{\bar{r}=1}^*$ and $w_{\bar{r}=1}^*$ at the film-bearing interface are obtained from the continuity of velocity and viscous shear stresses. The boundary conditions associated to the system of equations obtained are:

- On the symmetrical axis, $\bar{z} = 0$:

$$\begin{aligned} & \frac{\partial \bar{p}}{\partial \bar{z}} = 0, \quad \frac{\partial \bar{p}^*}{\partial \bar{z}} = \frac{\bar{r}}{\bar{H}} \frac{\partial \bar{H}}{\partial \bar{z}} \frac{\partial \bar{p}^*}{\partial \bar{r}}, \\ & \frac{\partial u^*}{\partial \bar{z}} = \frac{\bar{r}}{\bar{H}} \frac{\partial \bar{H}}{\partial \bar{z}} \frac{\partial u^*}{\partial \bar{r}}, \quad \frac{\partial v^*}{\partial \bar{z}} = \frac{\bar{r}}{\bar{H}} \frac{\partial \bar{H}}{\partial \bar{z}} \frac{\partial v^*}{\partial \bar{r}}, \quad w^* = 0 \end{aligned} \quad (4)$$

- At edge of the bearing, $\bar{z} = 1/2$:

$$\begin{aligned} & \bar{p}^* = \bar{p} = 0, \quad \frac{\partial u^*}{\partial \bar{z}} = \frac{\bar{r}}{\bar{H}} \frac{\partial \bar{H}}{\partial \bar{z}} \frac{\partial u^*}{\partial \bar{r}}, \\ & \frac{\partial v^*}{\partial \bar{z}} = \frac{\bar{r}}{\bar{H}} \frac{\partial \bar{H}}{\partial \bar{z}} \frac{\partial v^*}{\partial \bar{r}}, \quad \frac{\partial w^*}{\partial \bar{z}} = \frac{\bar{r}}{\bar{H}} \frac{\partial \bar{H}}{\partial \bar{z}} \frac{\partial w^*}{\partial \bar{r}} \end{aligned} \quad (5)$$

- At the external bearing surface, $\bar{r} = 0$:

$$\frac{\partial \bar{p}^*}{\partial \bar{r}} = 0, \quad u^* = v^* = w^* = 0, \quad v^* = 0 \quad (6)$$

- On the film-porous elastic bearing interface, $\bar{r} = 1$:

$$u_{\bar{r}=1}^* = \bar{u}, \quad v_{\bar{r}=1}^* = -\bar{v}, \quad w_{\bar{r}=1}^* = \bar{w} \quad (7)$$

$$\frac{\partial \bar{u}}{\partial \bar{r}} - \frac{\bar{l}^2}{\bar{h}^2} \frac{\partial^3 \bar{u}}{\partial \bar{r}^3} = \frac{\alpha}{\Gamma} \frac{\bar{h}}{\bar{H}} \frac{\partial \bar{u}^*}{\partial \bar{r}} + \frac{\alpha \bar{h}}{\Gamma \left(\frac{1}{\Gamma \xi} + \frac{1}{\Gamma} + 1 - \bar{H} \bar{r} \right)} u_{\bar{r}=1}^* \quad (8)$$

$$+ \frac{\alpha \xi \kappa \bar{h}}{\Gamma \left(\frac{1}{\Gamma \xi} + \frac{1}{\Gamma} + 1 - \bar{H} \bar{r} \right)} \left(\frac{\bar{r}}{\bar{H}} \frac{\partial \bar{H}}{\partial \theta} \frac{\partial v^*}{\partial \bar{r}} - \frac{\partial v^*}{\partial \theta} \right)$$

$$\frac{\partial v^*}{\partial \bar{r}} = 0 \quad (9)$$

$$\begin{aligned} & \frac{\partial \bar{w}}{\partial \bar{r}} - \frac{\bar{l}^2}{\bar{h}^2} \frac{\partial^3 \bar{w}}{\partial \bar{r}^3} = \frac{\alpha}{\Gamma} \frac{\bar{h}}{\bar{H}} \frac{\partial \bar{w}^*}{\partial \bar{r}} \\ & + \alpha (\kappa \xi)^2 \bar{h} \left(\frac{\bar{r}}{\bar{H}} \frac{\partial \bar{H}}{\partial \bar{z}} \frac{\partial v^*}{\partial \bar{r}} - \frac{\partial v^*}{\partial \bar{z}} \right) \end{aligned} \quad (10)$$

In the circumferential direction θ , the Reynolds boundary conditions are adopted in the film fluid, and the boundary conditions in the porous elastic bearing are:

$$p^*(r, \theta = 0, z) = p^*(r, \theta = 2\pi R, z) = 0 \quad (11)$$

The EHD modified Reynolds equation in the film, those of Darcy-Brinkman equations and Laplace equation in the porous elastic bearing are interlinked at the film-bearing interface by the continuity of pressure, velocities and viscous shear stresses conditions. Since this fluid-porous bearing interaction problem necessitates numerical solution, the equations are discretized employing finite differences, and the successive over-relaxation scheme is used to solve iteratively the obtained algebraic equations. The fixed-point technique is advocated to solve sequentially this coupled problem.

3. Results

Figure 2 displays friction factor f_f with respects to various couple stresses parameters \bar{l} at $\varphi = 2.10^{-4}$ and $C_0 = 0.01$. Comparing with the SFM, due to the viscous shear stresses the BM forecasts higher values of f_f . Moreover, this later coefficient is reduced with increasing \bar{l} and the increment is enlarged at low values of ε whereas reduced at high ε . The additives in the lubricant offer larger dimensionless load capacity \bar{W} which diminishes f_f , since this later is in inverse proportion to \bar{W} . Figure 3 represents f_f variation with respects to various bearing permeability parameters φ at $\bar{l} = 0.2$ and $C_0 = 0.01$. It is shown that the viscous shear effects of the BM provide a slight increase in f_f than the results obtained from the SFM. Furthermore, increasing φ leads to an increase in f_f . This is clarified by that rising the bearing permeability leads to diminish of porous flow resistance. The variation of f_f with respects to various elastic coefficient C_0 at $\bar{l} = 0.2$ and $\varphi = 2.10^{-4}$ is depicted on Figure 4. It is observed that even for diverse C_0 , the BM still predicts a greater f_f as compared to the result obtained from SFM. In particular, the effect of C_0 is felt

at high eccentricities using BM whereas negligible for the results obtained from SFM.

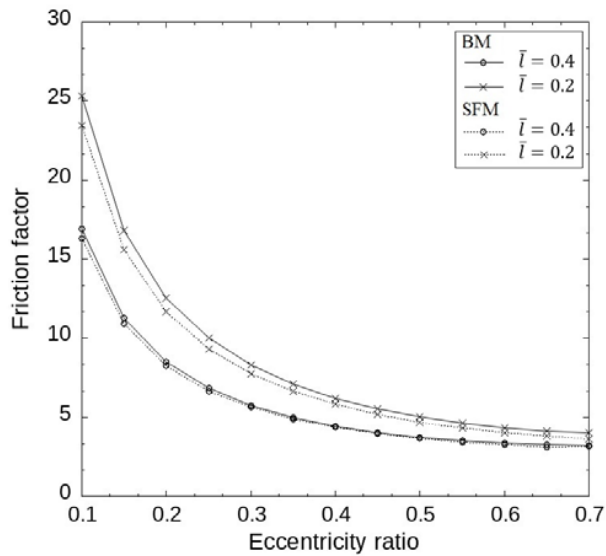


Figure 2: Couple stress effect on friction factor at $\varphi = 2.10^{-4}$ and $C_0 = 0.01$.

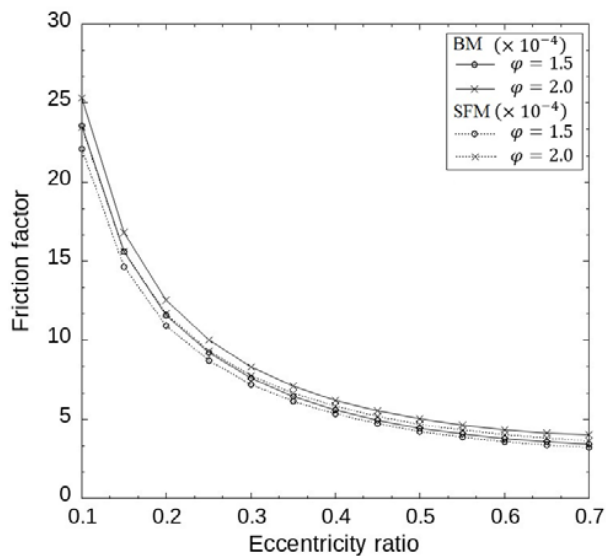


Figure 3: Permeability parameters effect on friction factor at $\bar{l} = 0.2$ and $C_0 = 0.01$.

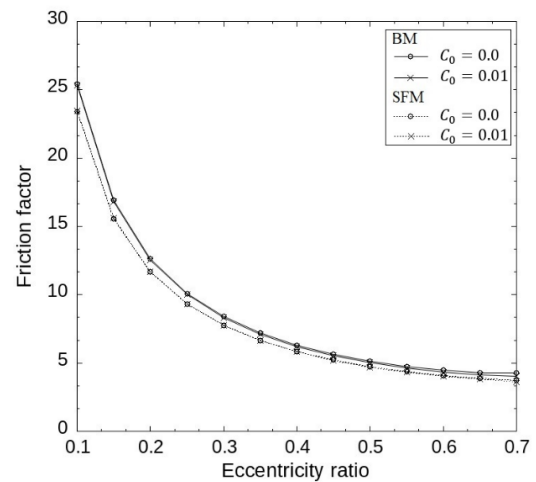


Figure 4: Elastic coefficients effect on friction factor at $\bar{l} = 0.2$ and $\varphi = 2.10^{-4}$.

4. Conclusion

In this study, viscous shearing forces effects on finite porous elastic journal bearings were numerically analyzed. Making use of the Stokes micro-continuum mechanics, and considering bearing elastic deformation determined by Winkler model, an EHD modified Reynolds equation type is derived. The porous flow was modeled by the complete Darcy-Brinkman equation. According to the results, the viscous shearing forces effects of the Brinkman model BM on the bearing performances in terms of load capacity and friction factor are not negligible. Comparing with those obtained from Darcy model with Beavers-Joseph slip conditions SFM, the viscous shearing forces effects enhance these quantities.

References

- [1] A. Sakim, M. Nabhani and M. El Khlifi, *Non-Newtonian effects on porous elastic journal bearings*, Tribology International, 2018, Vol. 120, pp. 23–33.
- [2] G.S. Beavers, and D.D. Joseph, *Boundary conditions at a naturally permeable wall*, Journal of Fluid Mechanics, 1967, Vol. 30 No. 01, pp. 197-207.
- [3] H.C. Brinkman, *A calculation of the viscous force exerted by a flowing fluid on a dense swarm of particles*, Flow, Turbulence and Combustion, 1949, Vol. 1 No. 1, pp. 27-34.
- [4] A. Sakim, M. Nabhani and M. El Khlifi, *Viscous shear effect in non-Newtonian lubrication of finite porous elastic bearings*, Industrial Lubrication and Tribology, 2019, Vol. 71, pp. 374-380.
- [5] VK. Stokes. *Couples Stresses in Fluids*, Phys Fluids 9(9), 1966, pp 1709 - 1715.

Interpolation of parameterized reduced-order models for fluid flow prediction

A. EL OMARI¹, M. EL KHLIFI¹, L. CORDIER²

1. Hassan II University of Casablanca, Faculty of Sciences and Techniques, Morocco, achraf.elomari-etu@etu.univh2c.ma
1. Hassan II University of Casablanca, Faculty of Sciences and Techniques, Morocco, mohamed.elkhlifi@fstm.ac.ma
2. Institut Pprime (CNRS, Université de Poitiers, ISAE-ENSMA), France, laurent.cordier@univ-poitiers.fr

Abstract

Reduced-order models (ROMs) can be used to reduce CPU time and memory storage needed to perform near real-time control of dynamical systems. However, the bases constructed in this way lack robustness with respect to parameter changes and therefore must usually be rebuilt for each new parameter variation. Consequently, the need of full simulations to generate a basis for each configuration is computationally expensive. To overcome this challenge, we use an accurate bases interpolation method. It consists in computing the associated bases to a set of control parameters. These bases are afterwards called to build a reduced basis adapted to a new control parameter value. This interpolation strategy involves results of Geodesics computing on Grassmann manifold.

Keywords: *Reduced Order Models (ROMs), Proper Orthogonal Decomposition (POD), bases interpolation, Grassmann manifold, cylinder wake flow*

1. Introduction

The last three decades have been marked by advances in the construction of Reduced-Order Models (ROMs) using a variety of projection methods [1]. In optimal control, the purpose of Model Reduction is to replace the full-order model by a low order model which allows to express the solution in a reduced order basis that captures the dominant information of the system to be controlled. Consequently, it is expected that only a few basis elements will provide a reasonably good description of the controlled flow. The projection of governing equations onto the selected reduced basis results in a set of differential equations with considerably smaller size than the degree of freedom arising from classical control method, and their resolution is very fast. The mostly investigated model reduction approach in optimal control is the Proper Orthogonal Decomposition (POD) [2]. Usually, a POD reduced order basis is constructed for a specific flow configuration. Consequently, the range of validity of a given POD reduced order basis is generally restricted to a neighbourhood of the control parameters set. In the control process, out of this region we are led to

construct a new reduced order basis which consequently increases the computational cost. This can possibly be avoided by using only one POD reduced order basis constructed from the snapshots corresponding to a distribution of parameters. Within the control process, instead of reconstructing a new basis for the new flow configuration determined by the new control parameter, an attractive idea would be to interpolate this basis basing on a set of precomputed bases corresponding to a distribution of control parameters. As the usual interpolation of bases does not necessarily end up with a basis, we have to define a more adequate interpolation method that can reliably approximate the desired basis. In the present work, we use an advanced interpolation method based on the calculus of a Geodesic path on the Grassmann manifold [3]. It was firstly introduced for adapting reduced order bases in the context of Aeroelasticity by Amsallem and Farhat [4,5]. The potential of the proposed interpolation method in terms of accuracy is studied in the case of the cylinder wake flow problem governed by the Navier-Stokes equations.

2. POD basis adaptation by interpolation

Let $\{\lambda_i \in \mathbb{R}\}_{i=1}^{N_p}$ be a set of parameters and $\{Y_i\}_{i=1}^{N_p}$ the corresponding set of N_p -parametrized subspaces of $\mathbb{R}^{N_x \times k}$ belonging to $G(N_x, k)$. These subspaces are non-uniquely represented by their corresponding matrices $\{\Phi_i \in \mathbb{R}^{N_x \times k}\}_{i=1}^{N_p}$ whose columns are the POD modes. Consider the problem of interpolation in which we seek an approximation of \tilde{Y} represented by a matrix $\tilde{\Phi}$ for an untrained parameter $\tilde{\lambda} \notin \{\lambda_i \in \mathbb{R}\}_{i=1}^{N_p}$. Provided that the Grassmann manifold is not a flat space, straightforward interpolation of its points does not necessarily result in a point that is included in it. Therefore, it is necessary to reformulate the interpolation process to be suitable for points in the Grassmann manifold. To provide a well-defined interpolation framework of points in the Grassmann manifold, Amsallem et al. [4,5] proposed the **ITSGM** (*Interpolation on a Tangent Space of the Grassmann Manifold*). It consists in the following steps: **step 1:** A point $Y_{i_{ref}}$ of the manifold is chosen as a reference and origin point for the interpolation

step 2: The tangent space $T_{Y_{i_{ref}}}$ and those points among $\{Y_i\}_{i=1}^{N_p}$ which lie in a sufficiently small neighbourhood of $Y_{i_{ref}}$ are now considered. More specifically, each point Y_i that sufficiently close to $Y_{i_{ref}}$ is mapped to a matrix Δ_i representing a point V_i of $T_{Y_{i_{ref}}}$ using the **Logarithm map** $Log_{Y_{i_{ref}}}$. This can be written as

$$(I - \Phi_{i_{ref}} \Phi_{i_{ref}}^T) \Phi_i (\Phi_{i_{ref}}^T \Phi_i)^{-1} = U_i \Sigma_i V_i^T \text{ (Thin SVD)}$$

$$\Delta_i = U_i \tan^{-1}(\Sigma_i) V_i^T$$

Step 3: Interpolate in the tangent space the initial velocities $\Delta_1, \Delta_2, \dots, \Delta_{N_p} \in T_{Y_{i_{ref}}}$ for the untrained

parameter $\tilde{\lambda}$ using a standard interpolation to obtain $\tilde{\Delta}$.

Step 4: Finally by the **exponential mapping** $Exp_{Y_{i_{ref}}}$, map the interpolated velocity $\tilde{\Delta} \in T_{Y_{i_{ref}}}$ back to the Grassmann manifold $G(N_x, k)$. The matrix representation of the interpolated subspace is given by

$$\tilde{\Delta} = \tilde{U} \tan^{-1}(\tilde{\Sigma}) \tilde{V}^T \text{ (Thin SVD)}$$

$$\tilde{\Phi} = \Phi_{i_{ref}} \tilde{V} \cos(\tilde{\Sigma}) + \tilde{U} \sin(\tilde{\Sigma})$$

An illustration of ITS GM algorithm can be seen on Figure 1.

3. Application: 2D cylinder wake flow

Here, we consider the channel flow past a circular cylinder [6,7,8] of diameter D (see Figure 2). Let $\Omega \subset \mathbb{R}^2$ be a two-dimensional domain filled with a Newtonian viscous fluid of density ρ and viscosity μ .

The boundaries $\partial\Omega$ of Ω are denoted by $\Gamma = \Gamma_1 \cup \Gamma_2 \cup \Gamma_3 \cup \Gamma_4 \cup \Gamma_c$ where

$\Gamma_i, i=1,2,3,4$ correspond to the exterior domain, and Γ_c to the cylinder boundary. The Navier-Stokes and continuity equations are made dimensionless by introducing

$$L_{ref} = D, u_{ref} = U_\infty, t_{ref} = D/U_\infty \text{ and } p_{ref} = \rho U_\infty^2$$

as reference quantities for length, velocity, time and pressure. The governing equations can then be cast in the following non-dimensional form:

$$\begin{cases} \frac{\partial u}{\partial t} + (u \cdot \nabla)u = -\nabla p + \frac{1}{Re} \Delta u & \text{in } \Omega \times [0, T] \\ \nabla \cdot u = 0 & \text{in } \Omega \times [0, T] \end{cases}$$

where $u = (u, v)$ are the two-dimensional velocity components, p the pressure field and $Re = \rho U_\infty D / \mu$ is the Reynolds number.

The boundary conditions are now prescribed. At the left boundary, an inflow boundary condition is applied:

$$(u, v) = (U_\infty = 1, 0) \quad \text{on } \Gamma_1$$

At the channel side-walls, zero shear stress conditions are imposed:

$$\frac{\partial u}{\partial y} = 0, \quad v = 0 \quad \text{on } \Gamma_2 \text{ and } \Gamma_4$$

Provided the boundary Γ_3 is placed at a sufficient distance downstream of the cylinder, an acceptable approximation for conditions at the outflow:

$$p = 0 \quad \text{on } \Gamma_3$$

Finally, on the surface of the cylinder the velocity is:

$$(u, v) = (0, 0) \quad \text{on } \Gamma_c$$

The Finite Element method (FEM) will be used to simulate the flow over the cylinder. Numerical simulations are performed using the time-step $\Delta t = 0.2s$ and the non-uniform mesh represented in Figure 3. To validate the simulation, the lift coefficient (C_L), drag coefficient (C_D) and Strouhal number of vortex shedding (St) are tracked. The results match well with those of the benchmark case presented in [9].

The parameter to vary is the Reynolds number Re ranged from 90 to 290 and $T=150s$.

Now, the velocity POD bases are constructed [10] by considering 100 snapshots regularly distributed between the instants $t_i = 130$ and $t_f = 150$.

For different untrained Reynolds number values, the ITS GM method is applied to find an approximation of the velocity snapshots by interpolating the POD bases. Lagrange is chosen, as standard interpolation method, to interpolate the singular values and also the initial velocities Δ_i on the tangent space.

Consider now the case of an untrained Reynolds number value $Re=195$. The quality of the corresponding interpolated velocity can be checked out by inspecting the aerodynamics coefficients and the relative error

$$\left(E_u = \frac{\|u_{exact} - u_{interpolated}\|}{\|u_{exact}\|} \right).$$

These are shown in Table 1 where a good match with the coefficients obtained by the exact solution (FEM) can be seen. The associated exact and interpolated velocity field are represented in Figures 4 and 5. From a visual point of view, the velocity field obtained by interpolation look almost identical to the original one.

4. Conclusion

In this work, the ITS GM method for model reduction of parameterized non-linear problems was applied. This strategy has the ability to accurately and quickly reproduce the flow solutions for new control parameter values. To this end, the interpolation of the cylinder wake flow solutions was considered, where the varied parameter is the Reynolds number.

References

- [1] K. Taira, S. L. Brunton, S. T. M. Dawson, C. W. Rowley, T. Colonius, B. J. McKeon, O. T. Schmidt, S. Gordeyev, V. Theofilis, L. S. Ukeiley, Modal Analysis of Fluid Flows: An Overview, AIAA Journal (2017) 1–29.
- [2] JL Lumley, The Structure of Inhomogeneous Turbulent Flows, In AM Yaglom and VI Tatarski, editors, Atmospheric turbulence and radio propagation, Nauka, (1967) 166–178.
- [3] A. Absil, R. Mahony, R. Sepulchre, Riemann geometry of Grassmann manifolds with a view

on algorithmic computation, Acta Appl. Math. 80(2) (2004) 199–220.

- [4] D. Amsallem, C. Farhat, An interpolation method for adapting reduced-order models and application to aeroelasticity, AIAA J. (2008) 1803–1813.
- [5] D. Amsallem, Interpolation on Manifolds of CFD-based fluid and finite element based structural reduced-order models for online aeroelastic predictions, Thèse, Stanford University, 2008.
- [6] M. Oulghelou, Développement de modèles réduits adaptatifs pour le contrôle optimal des écoulements, Thèse, Université de La Rochelle, 2018.
- [7] R. Mosquera, A. Hamdouni, A. El Hamidi, A. Cyrille, POD basis interpolation via inverse distance weighting on Grassmann manifolds, Discrete Contin. Dyn. Syst. Ser. S 12 (2019) no. 6, 1743–1759.
- [8] R. Mosquera, A. El Hamidi, A. Hamdouni, A. Falaize, Generalization of the Neville–Aitken interpolation algorithm on Grassmann manifolds: Applications to reduced order model, Int J Numer Meth Fluids 93 (2021) 2421–2442.
- [9] M. Schafer, S. Turek, F. Durst, E. Krause, and R. Rannacher, Computations of Laminar Flow Around a Cylinder, In Flow Simulation with High-Performance Computers II, Notes on Numerical Fluid Mechanics (NNFM), pages 547–566. Vieweg+Teubner Verlag, 1996.
- [10] M. Bergmann, L. Cordier, Optimal control of the cylinder wake in the laminar regime by trust-region methods and POD reduced-order models, J. Comput. Phys. 227 (2008) 7813–7840.

Table 1 Aerodynamics coefficients and mean relative at the untrained Reynolds number $Re = 195$.

	C_D , mean	C_L , max	St	E_U
FEM	1.30	0.67	0.17	-
ITSGM	1.32	0.69	0.17	0.16 %

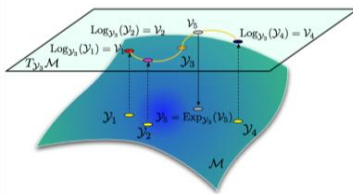


Figure 1: Illustration of bases interpolation by ITSGM

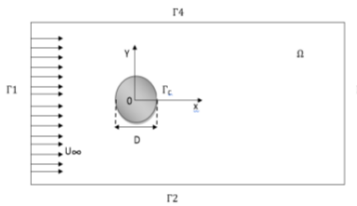


Figure 2: Simulation domain for the 2D cylinder wake flow problem

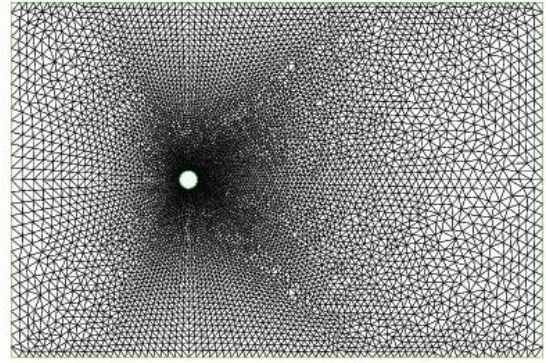


Figure 3: Non-uniform mesh used to simulate the problem of the 2D cylinder wake flow

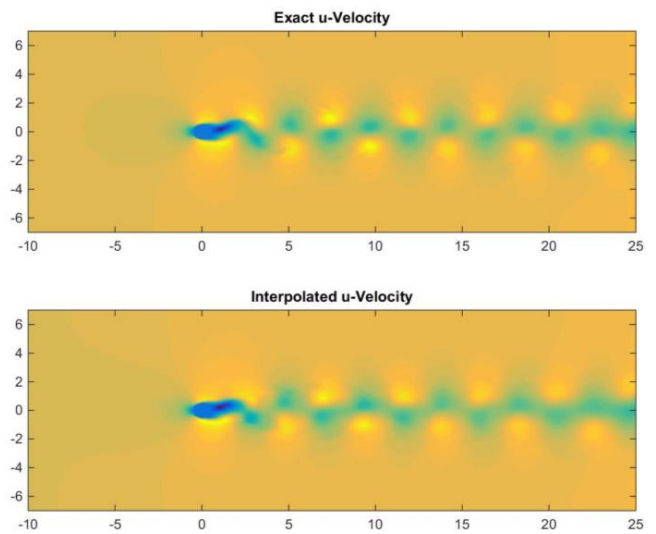


Figure 4: Comparison of exact and interpolated u -velocity for the untrained Reynolds number $Re=195$

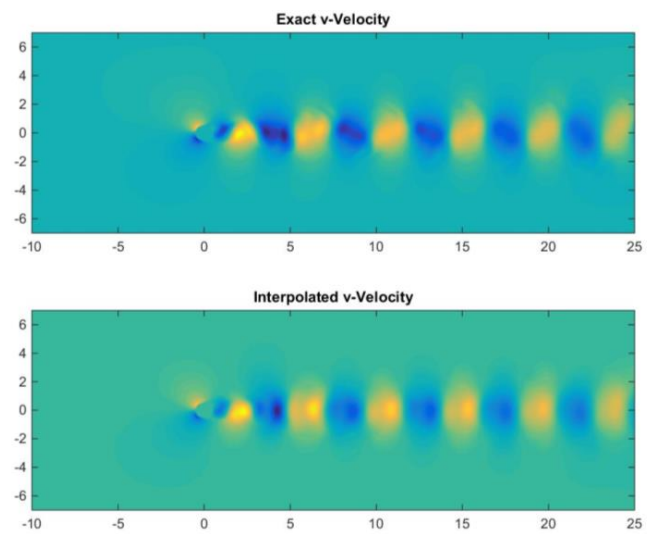


Figure 5: Comparison of exact and interpolated v -velocity for the untrained Reynolds number $Re=195$

RAYLEIGH-TAYLOR INSTABILITY UNDER VERTICAL PERIODIC OSCILLATION WITH TWO FREQUENCIES

C. TAOUZER.¹, M. ASSOUL¹, S. ANISS¹

1. University of Hassan II, Faculty of Sciences Aïn-Chock, Laboratory of Mechanics, B.P.5366 Maarif, Casablanca, Morocco. chaimaatzer@hotmail.com

Abstract :

We investigate the effect of vertical periodic oscillation on Rayleigh-Taylor instability of a system of two immiscible fluids of different densities. The system is considered of infinite extension in the horizontal directions and subjected to the gravity field. The linear stability analysis leads to the periodic Mathieu equation, which describes the evolution of the interfacial amplitude. Here, we examine the effect of oscillation, with two frequencies ω_1 et ω_2 , on the stability of the interface. The numerical solution shows that the introduction of two frequencies in oscillation allows to high dynamics, in terms of parametric resonances, than single frequency oscillation.

Keywords : *Rayleigh-Taylor instability, interfacial instability, periodic oscillation, commensurate frequencies, Floquet's theory*

1 Introduction

The Rayleigh-Taylor instability (RTI) [1] represents one of the classical and paradigmatic types of hydrodynamic instabilities, such as Kelvin-Helmholtz, Rayleigh-Bénard and Benard-Marangoni instabilities that attracted great interest over several decades. It occurs when a heavier fluid layer is supported from below in the gravity field by a lighter one. In its general context, RTI takes place when a fluid of a higher density accelerates into that with a lower density, by stellar material acceleration in astrophysics, in ballistics, in coatings of the underside of a solid substrate, in technological applications such as liquid polymer films flow, in environmental aspect, the dynamics of raindrops and their role in the biosphere, in microfluidics and lab-on-a-chip technology, and others.

Recent study [2] focused on the flow dynamics of a thin liquid film placed on the underside of a solid planar substrate, subjected to two-frequency asymmetric forcing in the tangential direction to the substrate. The effect of the amplitude ratio was examined with a linear stability analysis.

In this study, we focus on the influence of vertical periodic oscillation with two frequencies on the stability of the interface between two immiscible and incompressible fluid layers of different densities. The Fourier differentiation

matrix method (Fourdif) [4] and Floquet's theory combined with Runge-Kutta method are used to solve numerically the linear problem. After validating the results in the case of oscillation with one frequency [3], we analyse the influence of the frequencies ratio ω on the threshold of the interfacial instability.

2 Formulation

Consider an incompressible liquid film of thickness d , density ρ , kinematic viscosity ν and surface tension γ deposited on a flat and rigid substrate subjected to a periodic oscillation having two frequencies. The system is of an infinite extension in the horizontal directions and subjected to the gravity field g and to an oscillating motion in the vertical direction of acceleration $\tilde{a}(\tilde{t}) = g a(\tilde{t})$ where \tilde{t} is the time (see figure F. 1.).

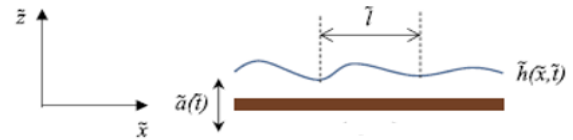


FIG. 1. Schematic illustration of the studied configuration. In the Rayleigh-Taylor configuration, gravity acts in the direction of \tilde{z} and $G < 0$.

This problem is mathematically formulated by the conservation equations associated with the boundary conditions at the wall and at the interface. The equations that describe this flow are the conservation equations which are the equation of continuity and the equations of momentum :

$$\frac{\partial \tilde{u}}{\partial \tilde{x}} + \frac{\partial \tilde{w}}{\partial \tilde{z}} = 0 \quad (1)$$

$$\frac{\partial \tilde{u}}{\partial \tilde{t}} + \tilde{u} \frac{\partial \tilde{u}}{\partial \tilde{x}} + \tilde{w} \frac{\partial \tilde{u}}{\partial \tilde{z}} = \frac{-1}{\rho} \frac{\partial \tilde{p}}{\partial \tilde{x}} + \nu \Delta \tilde{u} \quad (2)$$

$$\begin{aligned} & \frac{\partial \tilde{w}}{\partial \tilde{t}} + \tilde{u} \frac{\partial \tilde{w}}{\partial \tilde{x}} + \tilde{w} \frac{\partial \tilde{w}}{\partial \tilde{z}} \\ & = \frac{-1}{\rho} \frac{\partial \tilde{p}}{\partial \tilde{z}} + \nu \Delta \tilde{w} - g(1 + a(\tilde{t})) \end{aligned} \quad (3)$$

where \tilde{u} , \tilde{v} , g , \tilde{p} , ρ and ν designate respectively the axial and longitudinal components of the velocity field, the acceleration of gravity, the pressure, the density and the kinematic viscosity. The corresponding boundary conditions at the solid substrate $\tilde{z} = 0$ are the no slip and no penetration conditions :

$$\tilde{u} = 0, \tilde{w} = 0 \text{ en } \tilde{z} = 0 \quad (4)$$

At the free surface, the kinematic and the dynamic condition, are given respectively by :

$$\frac{\partial \tilde{h}}{\partial \tilde{t}} + \frac{\partial}{\partial \tilde{x}} \int_0^{\tilde{h}(\tilde{x}, \tilde{t})} \tilde{u} d\tilde{z} = 0 \quad (5)$$

$$(\tilde{p}_a - \tilde{p}) \mathbf{n} + \mathbf{D} \mathbf{n} = \kappa \gamma \mathbf{n} \quad (6)$$

where $\mathbf{n} = \frac{(-\frac{\partial \tilde{h}}{\partial \tilde{x}}, 1)}{\sqrt{1 + \frac{\partial \tilde{h}}{\partial \tilde{x}}^2}}$ is the unit vector normal to the interface, $\kappa = \frac{\frac{\partial^2 \tilde{h}}{\partial \tilde{x}^2}}{(1 + \frac{\partial \tilde{h}}{\partial \tilde{x}}^2)^{\frac{3}{2}}}$ is the interfacial curvature, and

$\mathbf{D} = \begin{pmatrix} 2\mu \frac{\partial \tilde{u}}{\partial \tilde{x}} & \mu (\frac{\partial \tilde{w}}{\partial \tilde{x}} + \frac{\partial \tilde{u}}{\partial \tilde{z}}) \\ \mu (\frac{\partial \tilde{w}}{\partial \tilde{x}} + \frac{\partial \tilde{u}}{\partial \tilde{z}}) & 2\mu \frac{\partial \tilde{w}}{\partial \tilde{z}} \end{pmatrix}$ is the viscous stress tensor in the liquid phase.

The dimensionless form of equations (1)-(3) and boundary conditions (2)-(6) is obtained in a dimensionless form using the following scales : d and \tilde{l} for the vertical and horizontal lengths, respectively, w_0 for velocities, and d/w_0 for time. By introducing a small film parameter :

$$\delta = \frac{d}{\tilde{l}} \ll 1$$

By using the approximation of lubrication, the dimensionless equations governing the problem in the relative frame are given in references [2, 3]:

$$\frac{\partial h}{\partial t} + \frac{\partial q}{\partial x} = 0 \quad (7)$$

$$\frac{6}{5} \text{Re} \left(\frac{\partial q}{\partial t} + \frac{17}{7} \frac{q}{h} \frac{\partial q}{\partial x} - \frac{9}{7} \frac{q^2}{h^2} \frac{\partial h}{\partial x} \right) = -\frac{3q}{h^2} + h \left(\Gamma \frac{\partial^3 h}{\partial x^3} - G(1 + a(t)) \frac{\partial h}{\partial x} \right) \quad (8)$$

The dimensionless values of the problem are :

$$\text{Re} = \frac{w_0 d}{\nu}, G = \frac{\delta^2 q d^2}{\nu w_0}, \Gamma = \frac{\delta^4 \gamma}{\rho \nu w_0}$$

These dimensionless quantities are respectively, the Reynolds, Galileo and the inverse capillary numbers. In contrast to the works [2, 3] that has focused on single frequency oscillation, in the present work we are interested

in the case of two frequencies. Therefore, we consider that the vertical non-dimensional acceleration $a(t)$ takes the form :

$$a(t) = A(\omega_1^2 \cos(\omega_1 t) + \alpha \omega_2^2 \cos(\omega_2 t)) \quad (9)$$

where $A = \frac{\tilde{A} w_0^2}{g d^2}$, $\omega_1 = \tilde{\omega}_1 \frac{d}{w_0}$, $\omega_2 = \tilde{\omega}_2 \frac{d}{w_0}$. We designate by ω_1 and ω_2 the frequencies of the vertical oscillation and A its amplitude. The parameter α , in equation (9), is the ratio of the amplitudes of the oscillatory motion related to each cosine function.

3 Linear stability analysis

Equations (7)-(8) are linearized in the neighbourhood of the basic solution, $q = 0$ and $h = 1$. The solutions of the linear problem are sought in the normal modes : $q = 0 + q_k(t) e^{i\tilde{k}x}$ and $h = 1 + \eta_k(t) e^{i\tilde{k}x}$. Equations (7)-(8) and dimensionless form of boundary conditions (2)-(6) are reduced to the Mathieu equation with damping :

$$\text{Re} \dot{\eta}_k + \left(\frac{5}{2} \right) \dot{\eta}_k + \frac{5}{6} G k^2 [1 + a(t) + \Gamma k^2] \eta_k = 0 \quad (10)$$

This differential equation corresponds to the amplitude equation of the elevation of the interface governing the Rayleigh-Taylor instability problem in the presence of vertical periodic oscillation. Equation (10) is solved numerically using Fourier differentiation matrix method [4]. Numerical results are also validated using the Floquet's theory combined with the four order Runge-Kutta method.

4 Discussion

The marginal stability curves are presented in Fig. 2. and Fig. 3. These curves allows us to determine the value of the amplitude, \tilde{A} , at which the parallel shear flow becomes unstable to standing waves of wavenumber \tilde{k} .

In Fig. 2., for $\omega = 125$ rad/s and $\alpha = 0$, we validate the work in which a single frequency was considered [3]. Fig. 2. shows the existence of two zones of marginal stability, the first one is the Rayleigh-Taylor instability which occurs for long wavelength (small wavenumber), here for $\tilde{k} < 0.37$ mm. The second appears for $\tilde{k} > 0.37$ mm and corresponds to several parametric tongue-shaped resonance zones whose origin is the presence of an oscillation. Note that the zone corresponding to the second resonance is the most unstable compared to the three other resonance zones.

In Fig. 3., numerical results are obtained with two frequencies considering an aspect ratio of the two frequencies, $\omega = \frac{5}{4}$ and restricting the ratio of amplitudes, α as : $\alpha\omega^2 = 1$. As expected the Rayleigh-Taylor instability is not affected by oscillation. However, we notice the appearance of more resonance zones in the same range of the wavenumber. It turns out that the second resonance and the sixth resonance have the same amplitude and two different wavenumbers approximately around $\tilde{k} = 1.4$ and $\tilde{k} = 1.8$, which suggests the existence of bi-critical points. Therefore, varying the frequency ratio will allow the resonance zones to be controlled in terms of amplitude and wavenumber.

5 Conclusion

In this study, we are interested in the Rayleigh-Taylor instability and we have performed a linear stability analysis on the behavior of the interface separating two viscous immiscible fluids of different densities under vertical periodic oscillations having two frequencies. The linear problem has been reduced as in [2, 3] to a periodic Mathieu equation governing the evolution of the amplitude of the interface. This preliminary study allows us then to determine the stable and unstable regions and modes taking into account the different physical parameters of the problem. The introduction of two frequencies into the oscillations generates a richer dynamic than that of a single frequency in terms of resonance zones.

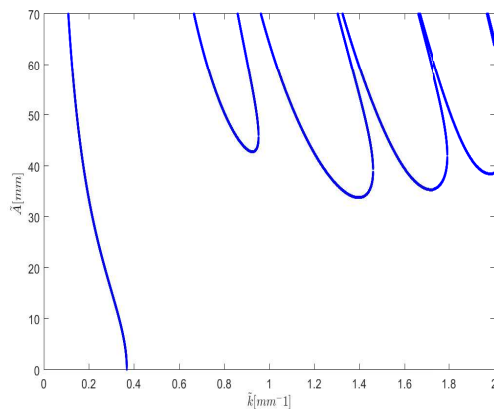


FIG. 2. Validation of the numerical procedure with results in references [3] : Marginal stability curves showing the amplitude, \tilde{A} , versus the wavenumber, \tilde{k} the film thickness $d = 0.1\text{mm}$ and the angular vibration frequency $\omega_x = 125$ rad/s and $\alpha = 0$.

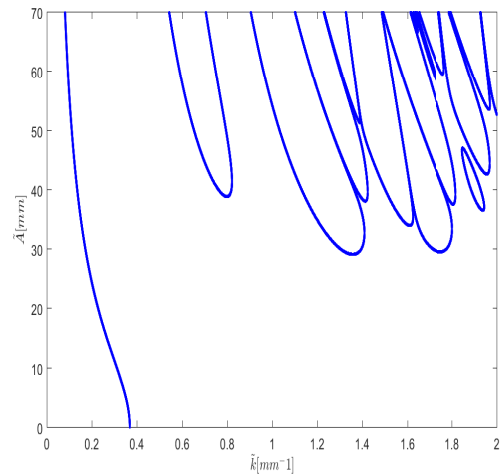


FIG. 3. Marginal stability curves showing the amplitude, \tilde{A} , versus the wavenumber, \tilde{k} for the frequency ratio $\omega = \frac{5}{4}$ and for $\alpha\omega^2 = 1$. The other parameter values are the film thickness $d = 0.1\text{mm}$ and the angular vibration frequency $\omega_x = 125$ rad/s.

References

- [1] Lord Rayleigh, *Investigation of the character of the equilibrium of an incompressible heavy fluid of variable density*, Proc. London Math. Soc.s1-14, 170 (1882).
- [2] Elad Serman-Cohen, Michael Bestehorn, and Alexander Oron, *Ratchet flow of thin liquid films induced by a two-frequency tangential forcing*, Physics of Fluids 30, 022101 (2018),14.
- [3] Elad Serman-Cohen, Michael Bestehorn, and Alexander Oron, *Rayleigh-Taylor instability in thin liquid films subjected to harmonic vibration*, Physics of Fluids 29, 052105 (2017),18.
- [4] A. C. Weideman, S. C. Reddy, *A MATLAB differentiation matrix suite*, ACM Trans. Math. Softw, 26, 465-519, 2000.

J. BOUCHGL^{1,2}, S. ANISS³, M. HMAMOU¹, S. SKALI LAMI⁴

1. Higher Institute of Marine Fisheries, Agadir, Morocco.

2. Laboratory of Energy Engineering, Materials and Systems ENSA, University Ibn Zohr, Agadir, Morocco
(jamilabouchgl@gmail.com)

3. Laboratory of Mechanics, Department of Physics, Faculty of Sciences Ain Chock, University Hassan II Casablanca,
BP 5366, Morocco

4. LEMTA -ENSEM, Université Lorraine, Nancy, France

Abstract :

We investigate the effect of horizontal periodic oscillation on the interfacial instability of two immiscible and viscous fluids of different densities in a two fully saturated porous media with different porosities and permeabilities. A linear stability analysis of the viscous and time dependent basic flow leads to a periodic oscillator describing the evolution of the interfacial perturbation amplitude. The horizontal oscillation leads to the occurrence of two types of instability, the Kelvin-Helmholtz instability and the parametric resonance. These instabilities appear at the frontier between water and petroleum and have a practical interest in oil reservoir engineering. In this investigation, we examine mainly the effect of the porosities and permeabilities of the two porous layers on the stability of the interface.

Keywords: *Kelvin-Helmholtz instability, Parametric instability, Porous media, Periodic oscillation*

1 Introduction

Several works have been carried out to study the Kelvin-Helmholtz instability arising between two immiscible fluid layers in a fully saturating porous media and flowing parallel to each other for different configurations [1-6]. The necessary and sufficient conditions for the stability of a moving interface in porous medium are first established by Saffman and Taylor [1]. Later on, Raghavan and Marsden [2] have studied the same problem presented in [1] for Darcy's type flow. They have found that the Kelvin-Helmholtz instability is possible only if the densest fluid is below the lightest one. Furthermore, this statically stable configuration has been studied by Bau [3] for Darcian and non-Darcian flows. He found that this type of instability may developed under certain conditions. Thereafter, Asthana *et al.* [4] have derived analytically a critical value of the relative velocity by using the viscous potential flow analysis. In this investigation, they have found that, the porosity of the porous medium has a stabilizing effect on the critical value of the relative velocity, the viscosity ratio have a destabilizing effect on the growth

rate while the Bond number has a stabilizing effect.

For unsteady flow, Bouchgl *et al.* [5] have investigated the effect of horizontal periodic oscillation on the interfacial instability of two immiscible and viscous fluids of different densities in a fully saturated porous media. In this situation, the oscillation leads to the occurrence of two types of instability, the Kelvin-Helmholtz's instability and the parametric resonance. They have found that an increase of the oscillation frequency destabilizes the Kelvin-Helmholtz instability and displaces the resonance regions toward the short wavelength perturbation. They have focused their analysis on the effect of the frequency of the oscillation, the physical proprieties of the fluids as well as the heights of the two fluid layers on the boundaries of the marginal stability. Also, it was shown that the porosity of the medium has a stabilizing effect on the two types of instability while the permeability has a destabilizing effect.

In this paper, a particularly interesting variation of the problem studied in [5], is the investigation of the stability of an interface between two incompressible, immiscible viscous fluid layers fully saturating two porous media with different porosities and permeabilities. In this investigation, we examine mainly how the porosities and the permeabilities of the two porous layers affect the instabilities that occur at the interface for various oscillation frequency as well as for relative heights of the two fluid layers. In the practice, the typical example, describing how the periodic oscillatory motion of the fluid layers can be generated, was investigated by Talib *et al.* [6].

2 Formulation of the problem

2.1 Governing equations

Consider two superposed layers of Newtonian, incompressible and viscous fluids in a two fully porous media of height $h = h_1 + h_2$ with both constant porosities, ε_j , and permeabilities, K_j where the subscripts $j = 1, 2$ denotes the lower and upper layers respectively (see Fig 1). Each fluid layer is characterized by a density ρ_j , a kinematic viscosity ν_j and a height h_j . The two fluid layers system is submitted to the horizontal oscillatory motion according to the law of displacement, $a \cos(\omega t)\mathbf{x}$, where \mathbf{x} is the horizontal unit vector. The surface tension between the two fluid layers is denoted by γ , the hydrodynamic pressure by

p_j and the relative fluid velocity by \mathbf{V}_{fluid_j} .

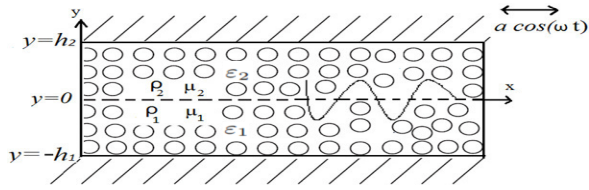


Fig1: Immiscible fluid layers, considered as infinite in the x directions, saturating two porous media with different porosities and permeabilities subjected to horizontal oscillations.

Following [5], the continuity and the momentum equations are:

$$\nabla \cdot \mathbf{V}_j = 0 \quad (1)$$

$$\frac{\rho_j}{\varepsilon_j} \left[\frac{\partial \mathbf{V}_j}{\partial t} + \frac{(\mathbf{V}_j \cdot \nabla) \mathbf{V}_j}{\varepsilon_j} \right] = -\nabla p_j - \frac{\mu_j}{K} \mathbf{V}_j + \rho_j a \omega^2 \cos(\omega t) \mathbf{x} + \rho_j \mathbf{g} \quad (2)$$

2.2 Base flow solution

In the basic state, the two fluids are separated by an interface which is initially planar and coincident to the $y = 0$ plane. The basic flow has one-component velocity field, $\mathbf{V}_j^b(t) = (U_j^b(t), 0, 0)$, which is periodic and parallel to the x -axis. Furthermore, the vertical end-walls, located at infinity, generate a counter-flowing layers in the porous media which arises when the horizontal volume-flux is conserved. This is modelled by the integral condition of balance of the displacement volume of both fluids given by:

$$\int_{-h_1}^0 \mathbf{V}_1^b \cdot \mathbf{x} \, dy = - \int_0^{h_2} \mathbf{V}_2^b \cdot \mathbf{x} \, dy \quad (3)$$

This basic flow satisfies necessarily the system of equations (1)-(2) and the integral condition (3). The steady state solution is sought in a periodic form:

$$U_j^b(t) = a\omega [F_j \cos(\omega t) + G_j \sin(\omega t)] \quad (4)$$

Using the condition (3), the functions F_j and G_j are:

$$F_1 = -HF_2 = \alpha \left[\frac{\varepsilon_1 \varepsilon_2 \left(\frac{1}{Da_2} + \frac{\mu H}{Da_1} \right)}{\sigma_2 (\varepsilon_1 + \rho H \varepsilon_2)} \right] \quad (5)$$

$$G_1 = -HG_2 = \alpha \quad (6)$$

$$\text{with } \alpha = \frac{\varepsilon_1 \varepsilon_2 H (\rho - 1)}{(\rho H \varepsilon_2 + \varepsilon_1) \left[1 + \left(\frac{\varepsilon_1 \varepsilon_2 \left(\frac{1}{Da_2} + \frac{\mu H}{Da_1} \right)}{\sigma_2 (\varepsilon_1 + \rho H \varepsilon_2)} \right)^2 \right]}$$

$H = \frac{h_2}{h_1}$, $Da = \frac{K}{h_2^2}$ is the Darcy number, $\sigma_2 = \frac{\omega h_2^2}{\nu_2}$ is the frequency number (ν_2 is the kinematic viscosity of the upper fluid), $\rho = \frac{\rho_1}{\rho_2}$ is the density ratio and $\mu = \frac{\mu_1}{\mu_2}$ is the viscosity ratio which can be expressed as a function of the contrast viscosity, $\mu = \frac{1 - A_\mu}{1 + A_\mu}$, with $A_\mu = \frac{\mu_2 - \mu_1}{\mu_2 + \mu_1}$ and $-1 \leq A_\mu \leq 1$. Remark that, for $\varepsilon_1 = \varepsilon_2$ and $Da_1 = Da_2$, the base flow solution given

by equation (4), tends to the solution in [5] given by:

$$F_1 = -HF_2 = \frac{H\varepsilon(\rho - 1)}{(\rho H + 1)(1 + \frac{\varepsilon^2}{Da^2} \frac{1}{\Gamma^4})} \frac{\varepsilon}{Da} \frac{1}{\Gamma^2} \quad (7)$$

$$G_1 = -HG_2 = \frac{H\varepsilon(\rho - 1)}{(\rho H + 1)(1 + \frac{\varepsilon^2}{Da^2} \frac{1}{\Gamma^4})} \quad (8)$$

with $\Gamma^2 = \sigma_2 \left(\frac{\rho H + 1}{\mu H + 1} \right)$. For non porous media, $\varepsilon_1 = \varepsilon_2 = 1$ and $Da_1, Da_2 \rightarrow \infty$, the base flow solution tends to the solution in [7], corresponding to the inviscid approximation.

The pressure at the equilibrium is given by:

$$P_j^b = -\rho_j g y + f(t) x \quad (9)$$

where the function $f(t)$ is not presented because it is insignificant for further consideration.

2.3 Linear stability

To perform a linear stability analysis, the base state is disturbed so that the velocity and the pressure fields in the perturbed state are written as the sum of the base flow variables and small perturbations. Also, we describe the instantaneous dimensional interface by $y = \xi(x, t)$, where $\xi(x, t)$ is an infinitesimal perturbation of the initially horizontal interface. Hereafter, we seek the solution of the problem in terms of normal modes as

$$[p_j, u_j, v_j, \xi] = [\tilde{p}_j(t, y), \tilde{u}_j(t, y), \tilde{v}_j(t, y), \tilde{\xi}(t)] e^{ikx} \quad (10)$$

where k is the wavenumber in the x direction and $i^2 = -1$. Using the impermeability condition at the horizontal walls, $v_j(y, t) = 0$ at $y = -h_1, h_2$, and the kinematic condition linearized at the interface [3, 5]

$$\varepsilon_j \frac{d\tilde{\xi}(t)}{dt} + ik U_j^b \tilde{\xi}(t) = \frac{\partial \tilde{\phi}_j(t, y)}{\partial y} \quad (11)$$

Also, the normal stress balance at the interface [4] is

$$(P_1^b + \tilde{p}_1) - (P_2^b + \tilde{p}_2) + 2(\mu_2 \frac{\partial \tilde{v}_2}{\partial y} - \mu_1 \frac{\partial \tilde{v}_1}{\partial y}) = \gamma \nabla \cdot \mathbf{n} \quad (12)$$

where \mathbf{n} is the unit vector normal to the interface, γ is the surface tension and $\nabla \cdot \mathbf{n} = -\frac{\partial^2 \xi}{\partial x^2}(x, t) = k^2 \tilde{\xi}(t)$ is the linearized form of the curvature interface. Also, the total pressure is linearized, at $y = 0$, as $P_j^b + \tilde{p}_j = P_j^b(0) + \frac{\partial P_j^b}{\partial y} \Big|_{y=0} \tilde{\xi}(t) + \tilde{p}_j(0)$. With the above assumptions, the expressions of equations (1)-(2) allow to explicit the expression of the pressure jump boundary condition given by equation (12). This condition leads to a parametric differential equation for the amplitude $\tilde{\xi}(t)$ of the interface displacement from its equilibrium position. Then, the time is scaled by $\frac{1}{\omega}$ and the spatial coordinates are scaled by h_2 . Hence, the periodic oscillator is given by:

$$\begin{aligned} & \frac{d^2 \tilde{\xi}(\tau)}{d\tau^2} + \left[\alpha_1 + \alpha_2 \cos(\tau) + \alpha_3 \sin(\tau) \right] \frac{d\tilde{\xi}(\tau)}{d\tau} \\ & + \left[\alpha_4 + \alpha_5 \cos^2(\tau) + \alpha_6 \sin^2(\tau) + \alpha_7 \sin(2\tau) \right. \\ & \left. + i\alpha_8 \cos(\tau) + i\alpha_9 \sin(\tau) \right] \tilde{\xi}(\tau) = 0 \end{aligned} \quad (13)$$

where the coefficients α_i ($i = 1, 2, \dots, 9$) are the coefficients in terms of $We = \frac{\rho_2 h_2^3 \omega^2}{\gamma}$, the Weber number, $A = \frac{a}{h_2}$, the dimensionless amplitude of oscillation, $n = kh_2$, the dimensionless wave number, $\nu = \frac{\nu_1}{\nu_2}$, the viscosity ratio and $Go = \frac{g}{h_2 \omega^2}$, the modified Froude number. Further, the differential equation (13) is solved numerically using the spectral method. This method provides a matrix formulation which corresponds to a nonlinear eigenvalues problem.

3 Results and Discussion

In this section, our leading goal is to study the interfacial instability in a two fully saturated porous media with different porosities by analyzing the influence of the physical parameters of the media such as the porosities ε_j , and the permeabilities via Darcy numbers Da_j of the two porous layers. Figure 2 illustrates the neutral curves, $A(n)$, for $\varepsilon_1 = 0.3$ and for different values of porosity ε_2 . Note that, in the situation corresponding to $\varepsilon_1 = \varepsilon_2 = 0.3$, the typical shape of the neutral curves in Fig 2 is almost similar to that obtained in [5].

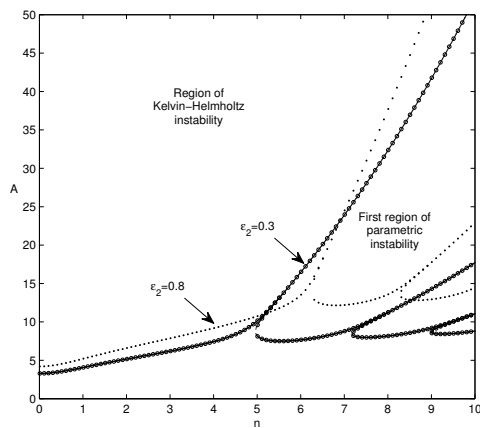


Fig2. Marginal stability curves for $Da_1 = Da_2 = 10^{-4}$, $\varepsilon_1 = 0.3$, $We = 6.25$, $Go = 0.16$, $A_\mu = 0$, $\rho = 2$, $\sigma_2 = 10^3$, $H_1 = H_2 = 1$ and for different ε_2 .

By inspecting these curves, we can see first how the Kelvin-Helmholtz instability threshold increases slightly as the porosity increases from $\varepsilon_2 = 0.3$ to 0.8 . Furthermore, the resonance zones shift to the right gradually towards the region of the short wave perturbations. Also, the results exhibit an increase of the critical wavenumber and the critical oscillation amplitude of resonances. For instance, the instability threshold of the first region for $\varepsilon_2 = 0.3$ is $A = 7.496$ at $n = 5.51$ and $A = 12.15$ at $n = 6.81$ for $\varepsilon_2 = 0.8$. Also, this displacement of the resonance zones is accompanied by an expansion of the Kelvin-Helmholtz instability region. The porosity of the upper region, ε_2 , has a stabilizing effect on the parametric instability thresholds and on the Kelvin-Helmholtz instability ones. The results in figure 3 demonstrate that the decrease in Darcy number of the upper porous layer, Da_2 , increases the stability threshold of the Kelvin-Helmholtz instability and the parametric instability one. Also, the resonance zones are displaced into the short wave regions. Moreover, this instability is suppressed, for the wavenumbers between $n = 0$ and $n = 10$, by decreasing the permeability as seen in Fig3 for $Da_2 = 10^{-5}$.

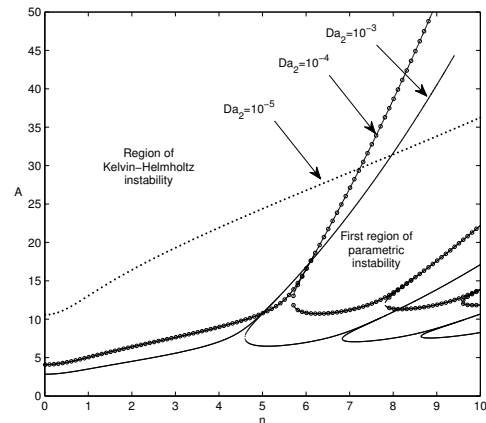


Fig3. Marginal stability curves for $Da_1 = 10^{-4}$, $\varepsilon_1 = 0.3$, $\varepsilon_2 = 0.6$, $We = 6.25$, $Go = 0.16$, $A_\mu = 0$, $\rho = 2$, $\sigma_2 = 10^3$, $H_1 = H_2 = 1$ and for different Da_2 .

4 Conclusion

In this investigation, we have performed a linear stability analysis of an interface between two viscous immiscible fluids of different densities in a fully two saturated porous media with different porosities and permeabilities under horizontal oscillation. The linear problem is reduced to a periodic oscillator governing the evolution of the amplitude of the interface. In this situation, the oscillation of the cell causes a successive parametric resonance modes. In this work, we have focused our analysis on the effect of the porosity ε_2 and the Darcy number Da_2 on the boundaries of the marginal stability. Note that ε_2 has a stabilizing effect on the stability boundaries and Da_2 has a destabilizing effect on it.

References

- [1] P. G. Saffman, G. I. Taylor, *The Penetration of a Fluid into a Porous Medium or Hele-Shaw Cell Containing a More Viscous Liquid*, Proc. S. Soc. A 245 (1958) 312-329.
- [2] R. Raghavan, S. S. Marsden, *A theoretical study of the instability in the parallel flow of immiscible liquids in a porous medium*, Quart.J. Mech. Appl. Math 26 (1973) 205-216.
- [3] H. H. Bau, *Kelvin-Helmholtz instability for parallel flow in porous media: A linear theory*, Phys. Fluids 25 (1982) 1719-1722.
- [4] R. Asthana, M.K. Awasthi, G.S. Agrawal, *Kelvin-Helmholtz instability of two viscous fluids in porous medium*, Int J. of Appl. Math. and Mech 8 (2012) 1-13.
- [5] J. Bouchgl, S. Aniss, *Effect of periodic oscillation on the interfacial instability of two superposed fluid layers in a fully saturated porous media*, IJAM Vol.13 (2021) 2150088, 1-26.
- [6] E. Talib, S. V. Jalikop and A. Juel, *The influence of viscosity on the frozen wave instability: theory and experiment*, J. Fluid Mech. Vol. 584 (2007).
- [7] M. V. Khenner, D. V. Lyubimov, T. S. Belozerova and B. Roux, *Stability of plane-parallel vibrational flow in a two-layer system*, J. Fluid Mech. Vol. 18 (1999), 1085-1101.

Hydrodynamic stability of shear flow of viscoelastic liquids

M. LAMINE¹, S. ANISS¹, A. HIFDI¹

1. Laboratory of Mechanics, Faculty of Sciences Aïn-Chock, Hassan II University, Casablanca, Morocco,
mustapha.lamine@gmail.com

Abstract:

Hydrodynamic instabilities in plane Poiseuille flow of a viscoelastic fluid are performed. A linear stability analysis has been explored based on a generalized Orr-Sommerfeld equation resulting from the conservation laws. This equation is reduced to a nonlinear eigenvalue problem, then, it is solved numerically using the Chebyshev spectral collocation method. Our main objective is focused on understanding the combined effects of the relaxation time (Deborah number, De) and the retardation time (elasticity number, E) on the onset of instability.

It turns out that, the Deborah number has a destabilizing effect while the elasticity number would suppress these instabilities. An important feature occurs when the ratio of both characteristic times, relaxation and retardation, is equal to 5772.22, corresponding to the critical Reynolds number of classical Poiseuille flow, in which the Jeffrey model fluid flow behaves as the Newtonian one.

Keywords: *Hydrodynamic instabilities; Viscoelastic fluids; Nonlinear eigenvalues problem; Spectral methods.*

1 Introduction

Since the experimental works of Osborne Reynolds in 1883, the stability of channel flows has been considering the major topics in fluid mechanics research because of its great importance in fundamentals and applications. Several experimentally, numerically and theoretically investigations have been devoted to analyzing the stability of this flow in various situations as summarized in the reviews [1, 2, 7].

In a pioneering study, Orszag [2] obtained numerically, using expansions in Chebyshev polynomials to approximate the Orr-Sommerfeld equation solutions, a critical Reynolds number of plane channel flows of Newtonian fluids, $Re_{cp} = 5772.22$. The stability analysis of non-Newtonian fluid flows are more difficult compared to the Newtonian one. This is related to the additional complexity introduced by the non-linear rheological behavior of these fluid models. An important non-Newtonian character of this fluids family is the viscoelastic behavior which combining the both viscous and elastic properties, which considering a nature of several polymer solutions partic-

ularly molten polymers, oil, toothpaste, clay...

One of the most general viscoelastic fluid models is that developed by James Gardner Oldroyd, called the Oldroyd- B model. It is considered the best model for describing physiological fluids. It represents an extension of the generalized Maxwell fluid (UCM) and it makes it possible to take into account the effects of solvent viscosity. The linear model of the Oldroyd- B fluids is known as the Jeffrey fluid. Many investigations for the stability of Jeffrey fluid flows has been one of the most intensively studied hydrodynamic problems, since, it was used to have modeled a wide range of fluids. For instance, bitumens, blood, pastes, lubricants, concrete, pastes, the earth's crust, polymers and their solutions [3] as well as aqueous solutions of polyethylene oxide at well-defined concentrations.

In this context, A. Saasen and O. Hassager [4] studied the Rayleigh-Taylor instability of the Jeffrey fluid. They showed that the elastic propriety can either delay or advances the stability of this system. They also proved that this fluid behaves like a Newtonian fluid in the short-wave regime. Later, M. Renardy et V. Tech [5] performed a nonlinear stability analysis of Jeffrey model flows. They showed that, only in the case of the linearized motion equations and for weakly elastic fluid, the flow is linearly stable with respect to infinitesimal disturbances. Leter on, Bergström [6] analyzed the transient growth of small disturbances in a pipe Jeffrey fluid flow. He showed the dependence between the fluid rheological parameters, relaxation and retardation times, and the transient amplification of the disturbance. Thereafter, V. Barra and S. Afkhami [7] studied the linear stability of the Jeffrey fluid film in interacts with van der Waals waves. It was found that this rheological model can be used to describe the linear stability of the interface in the long-wave limit. Recently, in the particular case of Jeffrey fluids corresponding to that of Maxwell linear, Lamine et al. [8] showed that the relaxation time develop long-wave instability of the shear mode.

The physical motivation of the present investigation is to assess how the combined effects of the relaxation and the retardation times interact to affect the hydrodynamic stability of a viscoelastic liquid in parallel shear flow. This paper is organized as follows. Section 2 is devoted to pre-

senting the physical model and the studied configuration, the steady basic flow solution and the formulation of linear stability and also the numerical method used to solve the stability problem. In section 3, pertinent results are discussed quantitatively and finally we conclude this investigation.

2 Problem formulation

2.1 Physical model & Governing equations

Consider a viscoelastic incompressible fluid with the density ρ in a plane channel formed by two infinite parallel rigid plates separated by a fixed distance $2h$. The fluid is set in motion by a constant pressure gradient in the \mathbf{x} direction. The following quantities are used as the scales to exprime the governing equations in their non-dimensional forms: channel half-height h for lengths, maximum streamwise velocity in basic state U for velocities, and U/h for time, ρU^2 for pressure and stresses. The flow is mathematically modeled by the continuity equation

$$\nabla \cdot \mathbf{V} = 0 \quad (1)$$

the Cauchy momentum equation

$$\frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V} \cdot \nabla) \mathbf{V} = -\nabla p + \nabla \cdot \boldsymbol{\tau} \quad (2)$$

where $\mathbf{V}(V_x, V_y)$, P and t are, respectively, the velocity vector, the pressure and the time. Further, $\boldsymbol{\tau}$ is the stress tensor of the Jeffrey fluid, whose the constitutive relation is given as follows [9]

$$\boldsymbol{\tau} + De \frac{\partial \boldsymbol{\tau}}{\partial t} = \left(\frac{1}{Re} \boldsymbol{\gamma} + E \frac{\partial \boldsymbol{\gamma}}{\partial t} \right) \quad (3)$$

where $\boldsymbol{\gamma} = \nabla \mathbf{V} + (\nabla \mathbf{V})^T$ is the rate strain, $De = \frac{\lambda_1 U}{h}$ is the Deborah number, $Re = \frac{U h}{\nu}$ is the Reynolds number and $E = \frac{\nu \lambda_2}{h^2}$ is the elasticity number. With λ_1 , λ_2 and ν represent, respectively, the relaxation time, the retardation time and the kinematic viscosity. It should be noted that Jeffrey fluid recovers linear Maxwell fluid when $E = 0$ and Newtonian fluid when $De = E = 0$. The boundary conditions at the walls ($y = \pm 1$) are

$$V_x(x, y = \pm 1) = 0 \quad (4)$$

2.2 Base state & Linear stability analysis

For the stationary and unidirectional basic flows, the basic state velocity for the Jeffrey model coincides with that of a Newtonian fluid, $V_b = 1 - y^2$.

Next, the infinitesimal perturbations $(V'(u', v'), P')$, in the normal modes form, are superimposed to the basic flow (V_b, P_b) as follows

$$\begin{cases} V_x(x, y, t) = V_b(y) + \psi_x(y) \exp(i\alpha(x - ct)) \\ V_y(x, y, t) = \psi_y(y) \exp(i\alpha(x - ct)) \\ P(x, y, t) = P_b(x) + p(y) \exp(i\alpha(x - ct)) \end{cases} \quad (5)$$

where $\psi_x(y)$, $\psi_y(y)$ and $p(y)$ are, respectively, the complex amplitudes of the perturbations u' , v' and P' . Thus, α is the wave number, c is the complex wave speed and $i^2 = -1$.

By substituting the quantities Eqs.(5) into Eqs.(1) and (2), with a linearizing process followed by elimination the pressure, we get

$$\begin{aligned} i\alpha \left[\alpha V_b(y) - c \right] (D^2 - \alpha^2) - \alpha \frac{\partial^2 V_b(y)}{\partial y^2} \psi_y(y) \\ = \alpha^2 c De \left[(c - V_b(y))(D^2 - \alpha^2) + \frac{\partial^2 V_b(y)}{\partial y^2} \right] \psi_y(y) \\ \left[(Re^{-1} - i\alpha c E)(D^2 - \alpha^2)^2 \right] \psi_y(y) \end{aligned} \quad (6)$$

The boundary conditions on $\psi_y(y)$ are

$$\psi_y(y = \pm 1) = D\psi_y(y = \pm 1) = 0 \quad (7)$$

The (6) is the Orr-Sommerfeld equation with two additional terms reflecting the intrinsic properties, λ_1 and λ_2 , of the Jeffrey fluids model. To solve this equation, we use a numerical approach based on the Chebyshev spectral collocation method, in which the function $\psi_y(y)$ is expanded as a finite sum of Lagrange polynomials discretized at N -collocation points of Gauss-Labatto [11]. Therefore, the modified Orr-Sommerfeld equation is reduced to a problem with nonlinear eigenvalues c as follows

$$\mathbf{A}\psi_y(y) + \mathbf{c}\mathbf{B}\psi_y(y) + \mathbf{c}^2\mathbf{C}\psi_y(y) = 0, \quad (8)$$

where \mathbf{A} , \mathbf{B} and \mathbf{C} are the operators characterized by (N, α, Re, De, E) and the marginal stability condition is

$$c_i(N, \alpha, Re, De, E) = 0 \quad (9)$$

The numerical code has been checked through comparison with the results of Orszag [2] for the Newtonian case and also with those obtained by Lamine *et al.* [8] for the Maxwellian case.

3 Results and Concluding remarks

The results of calculations are presented in figures 1 and 2 in which we focused our attention to understand the combined effect of the relaxation and retardation times, characterize by the Deborah and the elasticity numbers respectively, on the marginal stability ($c_i = 0$).

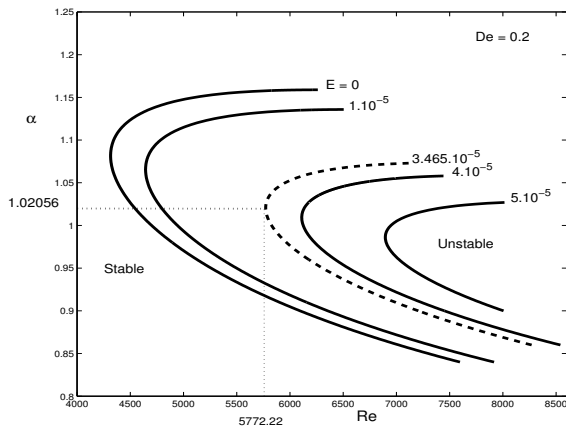


Figure 1: Typical neutral curves for various E and $De = 0.2$.

Figure 1 depicts the neutral curves for $De = 0.2$ and for different E values. The curves indicate that the critical Reynolds number, Re_c , increase and the unstable perturbations regions get smaller as the elasticity number increases. This increase in Re_c is accompanied by an increase in the critical waves number α_c . It means that the unstable disturbances get damped with retardation time and the mechanism of stability is governed by the long-wave regime. In fact, an important behavior is observed during the competition between the relaxation and retardation times to govern the flow stability. It concerns the existence of (De, E) -Couples for which the fluid model of Jeffrey behaves like that of Newton one [see the dashed line]. This feature appears when the ratio (De/E) is equal to the critical Reynolds number value of the plane flow of Newtonian fluid, $Re_{cp} = 5772.22$ [2]. The corresponding critical wave number, α_c , obtained in this case is also identical to that in [2] ($\alpha_c = \alpha_{cp} = 1.02056$).

For a specific value of the elasticity number, $E = 3.465 \cdot 10^{-5}$, the marginal stability diagram ($c_i = 0$) in the (Re, α) plane are plotted for various Deborah numbers in figure 2. It can be observed that, as De increases, the critical Reynolds number, Re_c , decreases and the critical waves number, α_c , increases with Deborah number. Indeed, the decreasing in the critical Reynolds number leads to widening of the unstable disturbances zone which extends towards the short-wave regime. It means that, the relaxation time has a destabilizing effect of the basic state.

Finally, on the basis of the results presented in this investigation, we conclude that the retardation time acts to enhance the stability of the basic flow, which behaves as a Newtonian fluid. This stabilizing effect is governed by the long-wave regime. The relaxation time leads to destabilize the basic flow with a reducing in wavelengths.

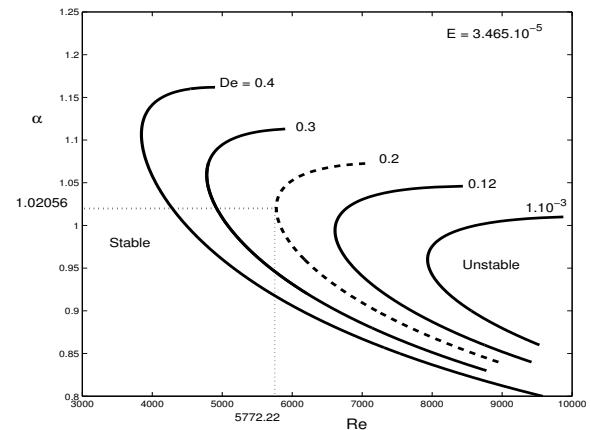


Figure 2: Typical neutral curves for various De and $E = 3.465 \cdot 10^{-5}$.

Furthermore, the Jeffrey model fluid flow behaves as the Newtonian one if the ratio of the relaxation and the retardation times is equal to the specific values $Re_{cp} = 5772.22$.

References

- [1] Peng Gao and Xi-Yun Lua, *Instability of channel flow with oscillatory wall suction/blowing*, Phys. Fluids, **18**, (2006) 034102.
- [2] S. A. Orszag, *Accurate solution of the Orr-Sommerfeld stability equation*, J. Fluid Mech., **50**, (1971) 689.
- [3] E.S. Baranovskii, *An Inhomogeneous Boundary Value Problem for the Stationary Motion Equations of Jeffreys Viscoelastic Medium*, J. Appl. Ind. Math., **7**(1), (2013) 22–28.
- [4] A.Saasen et O.Hassager, *Gravity waves and Rayleigh-Taylor instability on a Jeffrey-fluid*, Rheol Acta. **30** (1991) 301-306 .
- [5] M. Renardy et V. Tech, *Nonlinear Stability of flows of Jeffreys fluids at Low Weissenberg Numbers*, Research Report No. 94-NA-022 (1994).
- [6] L. B. Bergström, *Transient growth of small disturbances in a jeffrey fluid flowing through a pipe*, Fluid Dyn. Res., **32**, (2003) 29-44.
- [7] V. Barra et S. Afkhami, *Linear stability analysis of thin viscoelastic liquid of Jeffreys type with van der Waals interaction*, FACM, New Jersey Institute of Technology, Newark, DOI: 10.13140/2.1.1915.8409 (2014).
- [8] M. Lamine, M. Riahi & A. Hifdi, *Hydrodynamic stability of plane Poiseuille flow in Maxwell fluid with cross-flow*, Eur. Phys. J. Plus, **135**(2). (2020)
- [9] A. Fontaine, *Étude de l'hydrodynamique de fluides rhéologique complexes dans un mélangeur Maxblend par vélocimétrie laser*, (Mémoire de maîtrise, École Polytechnique de Montréal), (2012)
- [10] C. Canuto, M Y. Hussaini, A. Quarteroni and T. A. Zang, *Spectral Methods in Fluid Dynamics* New York, Springer-Verlag. **69** (1988).
- [11] J. A. C. Weideman and S. C. Reddy, *A matlab differentiation matrix suite*, ACM Transactions on Mathematical Software. **26**, (2000) 465–519.

Approche sans maillage pour la simulation des écoulements de fluides compressibles

R. FADIL¹, M. RAMMANE¹, S. MESMOUDI², A. TRI^{3,4}, B. BRAIKAT¹

rajaafadil25@gmail.com

1. Laboratoire d'Ingénierie et Matériaux (LIMAT), Faculté des Sciences Ben M'sik, Université Hassan II de Casablanca, B.P 7955 Sidi Othman, Casablanca, Maroc.
2. Laboratoire LISA, Ecole Nationale des Sciences Appliquées, Université Hassan I de Settat, Berrechid 26100, Maroc
3. Laboratoire de Mécanique, Faculté des Sciences Ain Chock, Université Hassan II de Casablanca, Maroc.
4. Institut Supérieur des Etudes Maritimes (ISEM), Km 7 Route d'El Jadida, Casablanca, Maroc.

Résumé :

Dans ce travail, nous proposons une approche sans maillage basée sur le couplage de la méthode des moindres carrés pondérées "Weighted Least Squares" (WLS) et le schéma explicite de Runge-Kutta (RK). Cette approche est dédiée à l'étude des écoulements de fluides compressibles. La formulation mathématique est basée sur les équations de Navier-Stokes écrites sous une formulation forte pour éviter toutes les difficultés de l'intégration numérique. L'application de cette approche à un exemple d'écoulements subsoniques a donné des résultats très satisfaisants par rapport aux résultats de référence.

Mots clés : WLS, Runge-Kutta, équations de Navier-Stokes, fluide compressible.

1 Introduction

La simulation des problèmes physiques décrits par des systèmes d'équations différentielles ou aux dérivées partielles est traditionnellement effectuée à l'aide de méthodes de résolution numérique de type éléments finis (MEF) ou différences finies (MDF)... Dans ces méthodes, le domaine spatial est discrétisé par des maillages dont la fonction est de définir un lien topologique entre les nœuds. Une méthode sans maillage procède en général des mêmes ingrédients que les méthodes traditionnelles (approximation, intégration, résolution d'un système) mais s'affranchit d'une définition du maillage ; la discrétisation se base alors sur un nuage de nœuds uniquement. Parmi ces méthodes, on trouve la méthode des moindres carrés mobiles "Moving Least Squares" (MLS) [1], la méthode d'interpolation polynomiale à base radiale "Radial Point Interpolation Method" (RPIM) [2], la méthode des moindres carrés pondérées "Weighted Least Squares" (WLS) [3] etc.

Pour les problèmes instationnaires, les techniques de discrétisation temporelle sont inévitables et représentent un domaine de recherche actif. Parmi ces méthodes, on trouve la méthode d'Euler, Euler amélioré, Runge-Kutta

etc.

Plusieurs méthodes sans maillage sont développées et utilisées pour résoudre les équations de Navier Stokes [1, 6].

Dans ce travail, nous proposons d'étudier numériquement les écoulements subsoniques ($Ma < 1$) de fluides compressibles, visqueux et isothermes avec un algorithme basé sur l'utilisation du schéma explicite de Runge-Kutta avec WLS. Les performances de cet algorithme seront testées sur l'exemple de l'écoulement dans une cavité entraînée.

2 Mise en équations

Les équations qui décrivent le modèle des écoulements de fluides compressibles, visqueux et isothermes sont données comme suit :

$$\begin{cases} \frac{d\rho}{dt} = \rho \nabla \cdot V & \text{dans } \Omega \\ \frac{d(\rho V)}{dt} = \mu(\Delta V + \frac{1}{3} \nabla(\nabla V)) - c^2 \nabla \rho & \text{dans } \Omega \\ V = V_d & \text{sur } \partial\Omega_d \\ c^2 \nabla \rho \cdot n = 0 & \text{sur } \partial\Omega_n \end{cases} \quad (1)$$

avec $\frac{d(\bullet)}{dt} = \frac{\partial(\bullet)}{\partial t} + \nabla(\bullet) \cdot V$. Les paramètres ρ , μ , c et V représentent respectivement la masse volumique, la viscosité dynamique, la vitesse de propagation du son [6] et le vecteur vitesse.

Afin d'appliquer l'approche proposée, nous tenons écrire les équations de Navier-Stokes (1) sous une forme quadratique comme suit:

$$M\dot{U} + L(U) + Q(U, U) = 0 \quad (2)$$

où M est l'opérateur de masse, $U = {}^T \langle u, v, \rho \rangle$ est un vecteur qui collecte les inconnues du problème, L est l'opérateur linéaire et Q représente la forme quadratique.

3 Stratégie de résolution

Dans la littérature, la résolution de ce type de problèmes repose sur une combinaison des méthodes de discrétisa-

tion spatio-temporelle. Cette combinaison représente un outil nécessaire pour transformer le problème continu en un problème discret en temps et en espace.

Dans notre travail, nous utilisons le schéma explicite de Runge-Kutta du quatrième ordre combiné avec la WLS pour la résolution de l'équation (2). Les méthodes sans maillage permettent d'éviter la génération de maillage et la connectivité entre les nœuds ce qui nécessite un temps de calcul considérable. L'approximation WLS est l'une des méthodes sans maillage, permettant ainsi d'éviter la notion de maillage.

L'approximation d'une fonction $u(x, y)$ en un point de coordonnées (x, y) est définie comme suit :

$$u(x, y) = \sum_{i=0}^m p_i(x, y) a_i = \langle p(x, y) \rangle \{a\} \quad (3)$$

où $p(x, y)$ est un polynôme d'ordre m construit à l'aide du triangle de pascal et $\{a\}^T = \langle a_0, a_1 \dots, a_m \rangle$ est le vecteur de coefficients constants. Pour déterminer le vecteur des coefficients $\{a\}$, on sélectionne n nœuds dans le domaine de support local. Alors, la détermination de ces coefficients se fait à l'aide de la minimisation d'une norme discrète pondérée définie par :

$$J = \sum_{i=1}^n W_i [u(x, y) - u_i]^2 \quad (4)$$

où u_i est l'inconnue nodale de u et W_i est la fonction de pondération de type Gaussienne associée au nœud i définie par $W_i = \frac{e^{-\left(\frac{r}{c_w}\right)^{2k}} - e^{-\left(\frac{1}{c_w}\right)^2}}{1 - e^{-\left(\frac{1}{c_w}\right)^2}}$; avec $r = \sqrt{(x - x_i)^2 + (y - y_i)^2} / rs$, $rs = kh$, rs est le rayon du support domaine, k est la valeur adimensionnelle de rayon du support domaine, h désigne la distance inter-point et c_w représente la constante de poids.

Après la minimisation de J par rapport au vecteur $\{a\}$, l'approximation des inconnues du problème par WLS s'écrit sous la forme suivante :

$$\begin{cases} V &= \sum_{i=1}^n \phi_i V_i \\ \rho &= \sum_{i=1}^n \phi_i \rho_i \end{cases} \quad (5)$$

où ϕ_i est la fonction de forme au nœud i définie par $\phi_i(\mathbf{x}) = \langle p(\mathbf{x}) \rangle [A]^{-1} [B]$; avec $[A] = \sum_{i=1}^n W_i \{p(\mathbf{x}_i)\} \langle p(\mathbf{x}_i) \rangle$ et $[B] = \sum_{i=1}^n W_i \{p(\mathbf{x}_i)\}$.

Des solveurs utilisant des schémas explicites de Runge-Kutta du quatrième ordre ont été utilisés depuis plusieurs années pour la résolution des équations de Navier-Stokes pour les fluides compressibles et incompressibles [4, 5]. Dans un schéma explicite, on utilise une discrétisation en

chaque point du domaine à l'instant courant t_n tel que :

$$M \dot{U}^n = -L(U^n) - Q(U^n, U^n) \quad (6)$$

L'évaluation des dérivées temporelles est obtenue par un schéma d'intégration en avant tel que :

$$\begin{cases} U^0 & \text{connue} \\ K_1 &= F(U^n) \\ K_2 &= F(U^n + \frac{\Delta t}{2}) K_1 \\ K_3 &= F(U^n + \frac{\Delta t}{2}) K_2 \\ K_4 &= F(U^n + \Delta t) K_3 \\ \mathcal{F} &= \frac{\Delta t}{6} (K_1 + 2(K_2 + K_3) + K_4) \\ U^{n+1} &= U^n + M_n^{-1} \mathcal{F} \end{cases} \quad (7)$$

Pour ce type de schémas, l'étude de la stabilité en temps se limite au choix du pas temporel Δt . Il est usuellement calculé par la condition de Courant-Friedrichs-Lewy (CFL-Condition) [4, 5, 6, 7]. Cette dernière est une condition de stabilité des schémas numériques instables. Elle joue un rôle important en mécanique des fluides. Pour les équations de Navier-Stokes des fluides newtoniens compressibles et isothermes, le pas du temps Δt est défini par :

$$\Delta t < \min\left(\frac{h}{c + \sqrt{u^2 + v^2}}, \frac{\rho h^2}{4\mu}\right) \quad (8)$$

4 Application numérique

L'écoulement classique dans une cavité est utilisé pour tester la validité de l'approche numérique proposée. La cavité est représentée schématiquement sur la figure 1. Le domaine est supposé carré de côté $L_x = L_y = l_0 = 1m$ où les trois côtés (AB , BC , AD) sont fixes, tandis que le côté supérieur (DC) se déplace avec une vitesse horizontale constante v_0 .

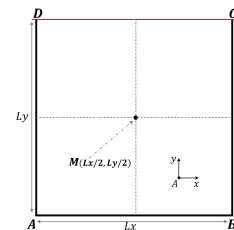


Figure 1: Domaine et conditions aux limites

Les propriétés du fluide sont choisies telles que : la masse volumique initiale du fluide est prise égale $\rho_0 = 1kg/m^3$, la viscosité dynamique $\mu = 0.1kg/ms$, on prend un nombre de Mach $Ma = 0.1$ et un nombre de Reynolds $Re = 100$ et la vitesse du son est donnée par $c = v_0/Ma$.

Pour montrer l'efficacité de notre approche, on présente dans la figure 2 l'évolution temporelle des composantes de la vitesse au centre de la cavité et une comparaison des solutions stationnaires obtenues par la WLS, la MLS et la MDF [4], en utilisant un nombre de nœuds égal à 2601, un rayon de support domaine $r_s = 3.03h$ et $h = \Delta x = \Delta y = 0.02$. D'après cette figure, on remarque que l'état stationnaire est atteint après l'instant $t = 0.9s$ et que les solutions stationnaires obtenues par les méthodes sans maillage coïncident avec la solution obtenue en utilisant la MDF couplée avec le schéma explicite de Runge-Kutta.

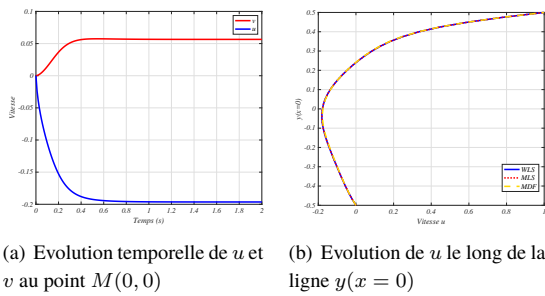


Figure 2: Comparaison des solutions obtenues par les trois méthodes WLS, MLS et MDF

Sur le tableau 1, on présente l'effet de l'ordre de la base polynomiale m des approximations WLS et MLS sur l'erreur relative des solutions stationnaires obtenues par les méthodes sans maillage avec la MDF. Le tableau montre que l'approximation WLS a une bonne convergence quelque soit l'ordre de la base, tandis que l'approximation MLS ne converge qu'avec un ordre $m = 6$. Ces résultats montrent la stabilité de l'approximation WLS par rapport à la MLS pour un même rayon de support domaine. Pour avoir des bons résultats avec la MLS, on doit augmenter le rayon de support domaine.

Ordre de la base	$m = 6$	$m = 10$	$m = 15$
MLS	10^{-6}	10^{-2}	10^{-1}
WLS	10^{-6}	10^{-6}	10^{-6}

Table 1: Influence de l'ordre de la base polynomiale des méthodes MLS et WLS sur l'erreur relative par rapport à la MDF

Sur la figure 3, on présente les lignes de courant et la distribution de la densité ρ de la solution stationnaire obtenue par notre approche. Les solutions obtenues représentent un comportement similaire à celles de la littérature [1, 6].

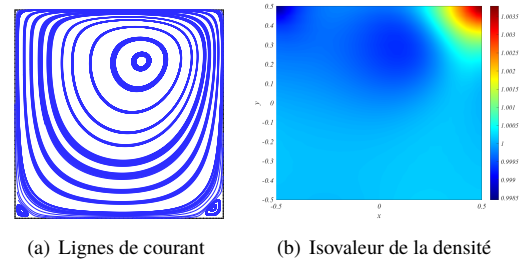


Figure 3: Lignes de courant et isovaleurs

5 Conclusion

Dans ce travail, nous avons proposé un algorithme combinant le schéma explicite de Runge-Kutta et l'approximation WLS pour résoudre les équations de Navier-Stokes décrivant l'écoulement de fluides compressibles. Les résultats obtenus ont montré que notre approche est capable de simuler les écoulements subsoniques. Des travaux sont en cours pour étendre cette technique vers l'étude des écoulements supersoniques.

References

- [1] M. Rammane, S. Mesmoudi, A. Tri, B. Braikat, N. Damil, *A dimensionless numerical mesh-free model for the compressible fluid flows*, Computers & Fluids, 221, 104845, (2021)
- [2] Liu, Gui-Rong and Gu, Yuan-Tong, *An introduction to meshfree methods and their programming*, Springer Science & Business Media (2005).
- [3] O. Elmhaia, Y. Belaasilia, O. Askour, B. Braikat, N. Damil, *An efficient mesh-free approach for the determination of stresses intensity factors*, Engineering Analysis with Boundary Elements, 133, 49-60, (2021).
- [4] W. Bao and S. Jin, *High-order i -stable centered difference schemes for viscous compressible flows*, Journal of Computational Mathematics, 101-112, (2003).
- [5] A. A. Boretti, *An explicit Runge-Kutta method for turbulent reacting flows calculations*, In Numerical Combustion, 199-210. Springer (1989).
- [6] C. L. Felner, J. H. Walther, and C. Henriksen, *Moving least squares simulation of free surface flows*, Computers & Fluids (2014), 91 :47-56.
- [7] R. Courant, K. Friedrichs, and H. Lewy, *On the partial difference equations of mathematical physics*, IBM Journal of Research and Development, 11(2) :215-234, (1967).

Stabilité de l'écoulement pulsé d'un fluide viscoélastique de type Oldroyd-B non-linéaire en géométrie de Taylor-Couette

M. Hayani Choujaa¹, M. Riahi^{1,2}, S. Aniss¹, Mohamed Ouazzani Touhami¹, Jaafar Khalid Naciri¹, Salah Skali Lami³

¹Laboratoire de Mécanique, faculté des Sciences Aïn-Chock, BP 5366 Mâarif, Casablanca, Maroc,

²Département de Mécanique, Ecole Royale de l'air, Marrakech, Maroc.

³Lemta-UMR CNRS 7563-Ensem, 2, avenue de la Forêt de Haye, BP 160, Vandoeuvre-les-Nancy, 54504, France,

Abstract

The stability of a viscoelastic fluid flow (Boger fluid model) between two concentric co-oscillating cylinders with the angular velocity $\Omega(t) = \Omega_0 \cos(\omega t)$ is investigated. In this study, we consider the Oldroyd-B fluid model. The obtained eigenvalue problem is solved, in the case of small-gap, using both Floquet theory and spectral method. This spatio-temporal resolution allows us to determine the onset of inertial instability for a large range of the frequency number. Our results reveal that the purely inertial instability observed in the case of a Newtonian fluid is destabilized under the effect of the fluid elasticity. A comparison between the dynamic of a purely polymeric solution and that of a Newtonian one is also presented.

Résumé

Le présent travail est consacré à l'étude des effets d'une modulation en phase des conditions aux limites relatives à la vitesse, imposées sur les cylindres intérieur et extérieur, sur le seuil d'instabilité au sein d'un fluide viscoélastique en écoulement de Taylor-Couette. Dans ce cadre, nous supposons que le comportement du fluide est régi par une loi de type Oldroyd-B non linéaire et que la modulation est sinusoïdale. Nous effectuons une étude de stabilité linéaire de l'écoulement de base et nous procédons à une résolution spatiale en utilisant des méthodes spectrales (méthode de collocation de Chebyshev-Gauss-Lobatto) et à une résolution temporelle avec la théorie de Floquet réalisée par la méthode de Runge-Kutta. Les résultats obtenus nous permettent de mettre en évidence les effets de la modulation et de la nature viscoélastique sur le nombre de Taylor critique.

Mots clés : Modulation, Stabilité linéaire, Taylor-Couette, Modèle d'Oldroyd-B, Théorie de Floquet.

Keywords : Linear stability, modulated Taylor-Couette flow, Oldroyd-B Model, Floquet theory.

1. Introduction

La dynamique d'une solution Newtonien en écoulement de Taylor-Couette modulé est bien étudiée par plusieurs auteurs dans la littérature [1,2]. Le cas d'une solution polymérique pure est récemment étudié dans le cadre d'un fluide de Maxwell linéaire et non linéaire [3,4]. Ce dernier est défini comme étant une substance viscoélastique ayant une viscosité constante et une élasticité désignée par E . Lorsqu'on ajoute à ce fluide un solvant de viscosité constante (solution Newtonienne) η_s on est en présence à une solution polymérique diluée. Le taux de dilution est bien mesuré par le paramètre S qui représente le rapport entre la viscosité du polymère désignée par η_p et la viscosité totale du mélange η telle que $S = \eta_p/\eta$ avec $\eta = \eta_p + \eta_s$. La substance ainsi obtenue est communément appelée fluide de Boger et sa loi rhéologique est bien décrite par une loi d'Oldroyd-B non-linéaire. L'objectif de cette communication est d'élucider l'effet de cette propriété rhéologique S sur la stabilité de cet écoulement et pour ce faire, nous considérons la solution polymérique de faible élasticité ayant $E = 0,01$. Il est à noter que le cas d'un fluide de Maxwell est retrouvé lorsque $S = 1$ et $E \neq 0$ tandis que pour $S = E = 0$ correspond au cas Newtonien.

2. Ecoulement de base

On considère un fluide viscoélastique, de densité ρ et de viscosité cinématique ν , confiné entre deux cylindres coaxiaux de rayons R_1 et $R_2 = R_1 + d$ ayant respectivement les vitesses angulaires $\Omega(R_1)$ et $\Omega(R_2)$ définies respectivement par $\Omega(R_1) = \Omega(R_2) = \Omega_0 \cos(\omega t)$, où, Ω_0 et ω sont respectivement l'amplitude et la fréquence de pulsation.

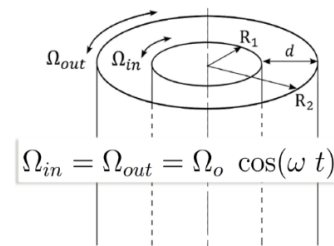


Figure 1. Géométrie de Taylor-Couette modulée.

Les équations régissant l'écoulement de base sont l'équation de conservation de la quantité de mouvement,

les équations de la loi de comportement et l'équation de conservation de la masse,

$$\left\{ \begin{array}{l} \rho \left(\frac{\partial}{\partial t} \mathbf{u} + (\mathbf{u} \cdot \nabla) \mathbf{u} \right) = -\nabla p + \nabla \cdot \boldsymbol{\tau} \\ \nabla \cdot \mathbf{u} = 0 \end{array} \right. \quad (2.1)$$

où \mathbf{u} est la vitesse du fluide, p la pression et $\boldsymbol{\tau}$ le tenseur des contraintes (tenant compte la contribution visqueuse newtonienne, et la contribution polymérique non-newtonienne). Le fluide considéré est supposé de type Oldroyd-B non-Linéaire dont le comportement est,

$$\boldsymbol{\tau} = \boldsymbol{\tau}^p + \boldsymbol{\tau}^s \quad (2.2)$$

Le premier terme représente la contribution du polymère $\bar{\boldsymbol{\tau}}^p$, tandis que le second représente celle du solvant newtonien $\bar{\boldsymbol{\tau}}^s$,

$$\boldsymbol{\tau}^s = 2\eta_s \mathbf{D} \quad (2.3)$$

où η_s est la viscosité dynamique du solvant et \mathbf{D} est le tenseur des taux de déformation,

$$2\mathbf{D} = \nabla \mathbf{u} + (\nabla \mathbf{u})^T$$

Pour la contribution polymère, nous avons,

$$\boldsymbol{\tau}^p + \lambda \overset{\nabla}{\boldsymbol{\tau}}^p = 2\eta_p \mathbf{D} \quad (2.4)$$

où λ est le temps de relaxation, η_p est la viscosité dynamique du polymère et $\overset{\nabla}{\boldsymbol{\tau}}^p$ est la dérivée convectée supérieure temporelle du tenseur des contraintes,

$$\overset{\nabla}{\boldsymbol{\tau}}^p = \frac{\partial}{\partial t} \boldsymbol{\tau}^p + \mathbf{u} \cdot \nabla \boldsymbol{\tau}^p - (\nabla \mathbf{u})^T \cdot \boldsymbol{\tau}^p - \boldsymbol{\tau}^p \cdot \nabla \mathbf{u}$$

L'écoulement de base est représenté par un champ de vitesse azimutal et un tenseur des contraintes correspondant. Nous introduisons par la suite les variables adimensionnelles suivantes,

$$r = (dx + R_1) \quad V^* = \frac{v}{R_1 \Omega_0} \quad t^* = \frac{t}{\frac{d^2}{v}} \quad \tau^* = \frac{\tau}{\frac{\rho v R_1 \Omega_0}{d}}$$

avec les conditions aux limites suivantes : $V_\theta^{0*}(0, t^*) = V_\theta^{0*}(0, t^*) = \cos(\sigma t^*)$ et nous définissons par la suite trois nombres sans dimension, à savoir le nombre de Reynolds R_e , le nombre d'Elasticité E

$$R_e = \frac{\Omega_1 R_1 d}{v}, \quad E = \frac{\lambda}{\frac{d^2}{v}}, \quad \sigma = \frac{\omega}{\frac{d^2}{v}}$$

En considérant la distance entre les deux cylindres d petite devant le rayon du cylindre intérieur R_1 , les termes d'ordre d/R_1 sont négligés. La vitesse de base adimensionnelle vérifiant les équations aux dérivées partielle (2.1)-(2.4) s'écrit sous la forme,

$$V_\theta^{0*}(x, t^*) = V_1(x) \cos(\sigma t^*) + V_2(x) \sin(\sigma t^*) \quad (2.5)$$

avec,

$$V_1(x) = \frac{1}{\cosh(\gamma\xi) + \cos(\gamma\beta)} \left(\cos(\gamma\beta x) \cosh(\gamma\xi(1-x)) + \cos(\gamma\beta(1-x)) \cosh(\gamma\xi x) \right),$$

$$V_2(x) = \frac{1}{\cosh(\gamma\xi) + \cos(\gamma\beta)} \left(\sin(\gamma\beta x) \sinh(\gamma\xi(1-x)) + \sin(\gamma\beta(1-x)) \sinh(\gamma\xi x) \right),$$

où,

$$\gamma = \sqrt{\frac{\sigma}{2}}, \quad \beta = \sqrt{\frac{1+(\sigma E)^2}{1+((1-S)\sigma E)^2} + \frac{\sigma ES}{1+((1-S)\sigma E)^2}}, \quad \xi = \sqrt{\frac{1+(\sigma E)^2}{1+((1-S)\sigma E)^2} - \frac{\sigma ES}{1+((1-S)\sigma E)^2}}$$

La forme du tenseur de contraintes suivant la composante $r\theta$ en écoulement de base, donnée par la résolution de l'équation (2.4) s'écrit sous la forme

$$\tau_{r\theta}^{0*}(x, t) = T_1(x) \cos(\sigma t) + T_2(x) \sin(\sigma t) \quad (2.6)$$

avec,

$$T_1 = \frac{S}{(1+(\sigma E)^2)} \left(\frac{\partial V_1}{\partial x} - \sigma E \frac{\partial V_2}{\partial x} \right),$$

$$T_2 = \frac{S}{(1+(\sigma E)^2)} \left(\sigma E \frac{\partial V_1}{\partial x} + \frac{\partial V_2}{\partial x} \right),$$

L'écoulement de base est caractérisé en outre par une contrainte normale suivant la direction azimutale $\theta\theta$ et qui s'écrit sous la forme,

$$\tau_{\theta\theta}^{0*}(x, t^*) = (\epsilon T_3 + T_4) \cos(2\sigma t) + (\epsilon T_5 + T_6) \sin(2\sigma t) + (\epsilon T_7 + T_8) \quad (2.7)$$

$$T_3 = \frac{We}{(1+(2\sigma E)^2)} (-V_1 T_1 + 2\sigma E V_1 T_2 + 2\sigma E V_2 T_1 + V_2 T_2),$$

$$T_4 = \frac{-We}{(1+(2\sigma E)^2)} \left(-T_1 \frac{\partial V_1}{\partial x} + 2\sigma E T_1 \frac{\partial V_2}{\partial x} + 2\sigma E T_2 \frac{\partial V_1}{\partial x} + T_2 \frac{\partial V_2}{\partial x} \right),$$

$$T_5 = \frac{We}{(1+(2\sigma E)^2)} (-2\sigma E T_1 V_1 - T_1 V_2 - T_2 V_1 + 2\sigma E T_2 V_2),$$

$$T_6 = \frac{We}{(1+(2\sigma E)^2)} \left(2\sigma E T_1 \frac{\partial V_1}{\partial x} + T_1 \frac{\partial V_2}{\partial x} + T_2 \frac{\partial V_1}{\partial x} - 2\sigma E T_2 \frac{\partial V_2}{\partial x} \right),$$

$$T_7 = -We(T_1 V_1 + T_2 V_2),$$

$$T_8 = We \left(T_1 \frac{\partial V_1}{\partial x} + T_2 \frac{\partial V_2}{\partial x} \right).$$

3. Analyse de stabilité linéaire

Nous allons nous intéresser par la suite à la stabilité de l'écoulement décrit précédemment vis à vis des perturbations axisymétriques. Nous supposons que les champs des vitesses s'écrivent sous la forme suivante :

$$\left(\begin{array}{l} u(r, z, t) = U(r, t) e^{i\sigma z} \\ v(r, z, t) = V_\theta^0(r, t) + V(r, t) e^{i\sigma z} \\ w(r, z, t) = W(r, t) e^{i\sigma z} \end{array} \right)$$

Ainsi que la pression,

$$p(r, z, t) = P^0(r, t) + P(r, t) e^{i\sigma z}$$

et le tenseur des contraintes

$$\tau_{rr}^p = \widehat{R} \widehat{R}(r, t) e^{i\sigma z} \quad \tau_{\theta\theta}^p = \widehat{\theta} \widehat{Z}(r, t) e^{i\sigma z}$$

$$\tau_{r\theta}^p = \tau_{r\theta}^0(r, t) + \widehat{R} \widehat{\theta}(r, t) e^{i\sigma z}$$

$$\tau_{\theta\theta}^p = \tau_{\theta\theta}^0(r, t) + \widehat{\theta} \widehat{\theta}(r, t) e^{i\sigma z}$$

$$\tau_{rz}^p = \widehat{R} \widehat{Z}(r, t) e^{i\sigma z} \quad \tau_{zz}^p = \widehat{Z} \widehat{Z}(r, t) e^{i\sigma z}$$

Le comportement de l'écoulement perturbé est contrôlé par trois paramètres : le nombre de fréquence σ , le nombre d'élasticité E et le nombre de Taylor Ta ou $Ta = R_e d/R_1$.

4. Approche Numérique et Résultats

Nous adoptons les méthodes spectrales de collocation de Chebyshev pour la résolution spatiale du problème de stabilité linéaire. Ces méthodes se révèlent d'un grand intérêt en raison de leur très grande précision [5]. Le système de stabilité s'écrit sous la forme d'un système de Floquet ayant la forme suivante :

$$\mathbf{B} \frac{\partial}{\partial t^*} \mathbf{X} = \mathbf{M}(t^*) \mathbf{X} \quad (4.1)$$

avec $\mathbf{M}(t^*)$ sont des matrices d'ordre 8N. Le système (4.1) est intégré en utilisant la méthode de Runge-Kutta du quatrième ordre. Enfin, on a une relation entre la fréquence γ , le nombre de Taylor Ta_c et le nombre d'onde q et qui s'écrit formellement sous la forme,

$$\mathcal{R}(Ta_c, q, \gamma, E) = 0 \quad (4.2)$$

5. Résultats et conclusion

L'évolution du nombre de Taylor-Couette critique en fonction du nombre de fréquence γ , est illustrée sur la Fig.2 pour une solution viscoélastique diluée ayant une faible élasticité où le nombre d'élasticité prend la valeur $E = 0,01$. Notons que le cas $E = S = 0$ correspond au cas Newtonien tandis que le cas où $E = 0,01$ et $S = 1$ correspond à un fluide de Maxwell non-linéaire. La dilution de cette solution est obtenue en diminuant la valeur du paramètre S . La démarche adoptée dans la description de la dynamique de cet écoulement est la suivante: partant du cas d'un fluide de Maxwell et commençant par la suite à le diluer en y ajoutant une quantité du solvant de viscosité η_s et ce en diminuant le paramètre S . D'après cette figure, on constate que l'écoulement est potentiellement instable dans le régime des fréquences intermédiaires. L'écoulement se stabilise en augmentant ou en diminuant la fréquence de modulation. Cette déstabilisation est traduite par une augmentation de la valeur du nombre de Taylor critique (le seuil critique de l'instabilité). Autrement dit, l'effet combiné de S et E est déstabilisant par rapport à un fluide Newtonien. Remarquons aussi que pour cette valeur d'élasticité ces diagrammes de stabilité sont composés de quatre branches lorsque $S=0,7$. Chaque branche correspond à un mode d'écoulement de Taylor toroidal qui se manifeste au seuil d'instabilité. Il s'agit d'une compétition de familles d'écoulements réversibles et non-réversibles [2,3,4,5]. La transition d'un mode à un autre se fait via un état bi-critique et donne lieu à un point de bifurcation de co-dimension deux. La Fig.3 montre l'évolution du nombre d'onde critique en fonction du nombre de fréquence de la modulation pour des valeurs représentatives du paramètre S . Chaque évolution se compose de branches qui dépendent du mode apparu dans une gamme de fréquence de modulation bien précise, similaires aux diagrammes du nombre de Taylor critique. En effet, pour $S = 0,7$ quatre branches sont obtenue aussi bien dans les courbes $q_c(\gamma)$ que $Ta_c(\gamma)$. La présence des

états bi-critiques entre deux modes d'écoulements tourbillonnaires de Taylor est responsable de l'apparition des discontinuités dans cette évolution, $q_c(\gamma)$.

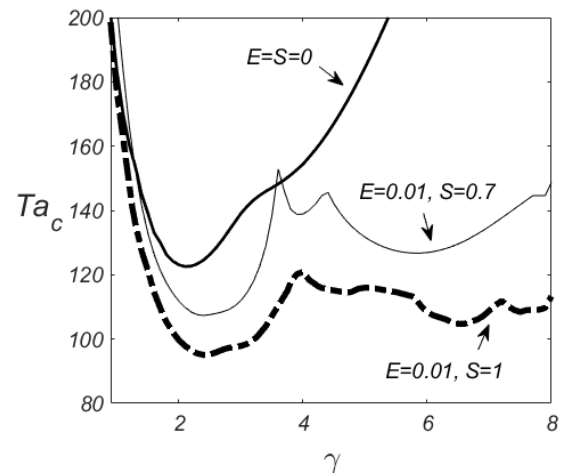


Figure 2. Nombre de Taylor critique en fonction du nombre de fréquence γ pour $E = 0,01$ et pour une valeur du paramètre $S = 0.7$

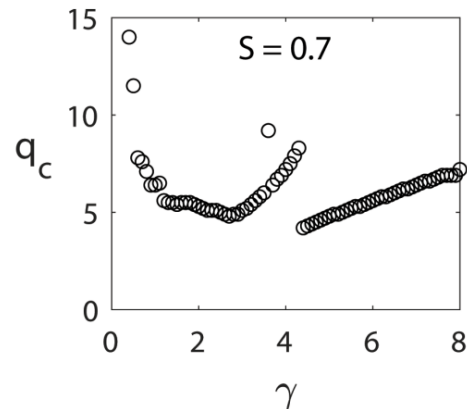


Figure 3. Nombre d'onde critique en fonction du nombre de fréquence de modulation γ pour $E=0,01$ et pour une valeur du paramètre $S = 0.7$

Références

- [1] M. Avila , M. J. Belisle , J. M. Lopez , F. Marques and W. S. Saric, "Mode competition in modulated Taylor-Couette flow", *J. Fluid Mech.* 601, 381-406 (2008).
- [2] A. J. Youd and C. F. Barenghi, "Reversing and nonreversing modulated Taylor-Couette flow at finite aspect ratio", *Phys. Rev. E* 72, 056321, (2005).
- [3] M. Hayani choujaa, S. Aniss, M. Ouazzani Touhami, J. Khalid Naciri and M. Riahi; "Stability of an oscillatory Taylor-Couette flow in an upper convected Maxwell fluid", *Phys. Fluids* 33, 074105 (2021).
- [4] M. Riahi, S. Aniss, M. Ouazzani Touhami and S. Skali Lami., "Stability of a Pulsed Taylor Couette Flow in a Viscoelastic Fluid ", *Journal of The Society of Rheology, Japan.* Vol. 42 No. 5 p. 321 327, (2014).
- [5] J. A. C. Weideman. S. C. Reddy. A MATLAB differentiation matrix suite. *ACM transaction on mathematical software*, 26, (2000)

Stabilité d'un écoulement oscillatoire de Taylor-Couette dans un fluide de Maxwell sur-convecté

M. Hayani Choujaa¹, M. Riahi^{1,2}, S. Aniss¹, Mohamed Ouazzani Touhami¹, Jaafar Khalid Naciri¹

¹Laboratoire de Mécanique, faculté des Sciences Aïn-Chock, BP 5366 Mâarif, Casablanca, Maroc,

²Département de Mécanique, Ecole Royale de l'air, Marrakech, Maroc,

Abstract

This work is devoted to the study of the effects due to the non-linear nature of a viscoelastic fluid, corresponding to an upper convected Maxwell model, on the critical instability parameters of a pulsed Taylor-Couette flow. In this framework, a linear stability analysis is performed leading to a partial differential system, which is solved using spectral method and Floquet theory. The obtained results allow us to determine a stability criteria of the system with emphasis on the effect of the non-linear rheological character of the fluid especially the first normal stress difference on the critical Taylor and wave numbers.

Résumé

Le présent travail est consacré à l'étude des effets dus à la nature non linéaire d'un fluide viscoélastique de type Maxwell convecté supérieur sur les paramètres critiques d'instabilité en écoulement pulsé de Taylor-Couette. Dans ce cadre, une étude de stabilité linéaire de l'écoulement de base a été effectuée menant à un système d'équations aux dérivées partielles qui ont été résolu par la méthode spectrale dans l'espace et par la méthode de Runge-Kutta combinée avec la théorie de Floquet dans le temps. Les résultats obtenus nous permettent de dégager un critère de stabilité de ce système toute en mettant l'accent sur l'effet du caractère rhéologique non linéaire du fluide notamment la première différence des contraintes normales sur les nombres de Taylor et d'onde critiques.

Mots clefs : Stabilité linéaire, Ecoulement modulé de Taylor-couette, Fluide d'Oldroyd-B, Théorie de Floquet.

Keywords : Linear stability, modulated Taylor-Couette flow, Oldroyd-B fluid, Floquet theory.

1. Introduction

L'écoulement de fluides viscoélastiques en géométrie de Taylor-Couette constitue un des systèmes modèles pour l'étude des instabilités viscoélastiques [1, 2]. Il est généré par la rotation de deux cylindres coaxiaux entre lesquels une couche fluide est confinée. Le cas de modulation temporelle des vitesses de rotation imposées sur les deux cylindres a fait l'objet de nombreuses études dans le cas des fluides Newtoniens [4]. Le cas d'une modulation en phase et en déphasage, en présence d'un fluide viscoélastique linéaire, a été étudiée par Riahi *et al.* [3]. Une étape majeure fut franchie quand Larson *et al.* [5] découvrirent que les effets élastiques peuvent déstabiliser l'écoulement. On parle d'instabilités purement élastiques, caractérisées au seuil par un comportement oscillant non-axisymétrique et désordonné, observables pour des nombres de Reynolds arbitrairement faibles. Larson *et al.* [5] ont établi l'existence de ce régime à la fois expérimentalement par l'étude de fluides très élastiques dits de Boger (ex.: un polyisobutylène en solution très visqueuse) et théoriquement par une étude de stabilité linéaire d'Oldroyd-B. Suite aux travaux de Larson *et al.* [5] portant sur une instabilité de nature purement élastique se produisant à de faibles valeurs du nombre de Taylor, nous nous intéressons dans le cadre de cette étude aux instabilités inertio-élastiques (couplage entre ces deux effets), en présence d'une modulation en phase, et ce lorsque le fluide est viscoélastique de type Maxwell convecté supérieur non linéaire.

2. Ecoulement de base

On considère un fluide viscoélastique, de densité ρ et de viscosité cinématique ν , confiné entre deux cylindres coaxiaux de rayons R_1 et $R_2 = R_1 + d$ ayant respectivement les vitesses angulaires $\Omega(R_1)$ et $\Omega(R_2)$ définies respectivement par $\Omega(R_1) = \Omega(R_2) = \Omega_0 \cos(\omega t)$, où, Ω_0 et ω sont respectivement l'amplitude et la fréquence de pulsation.

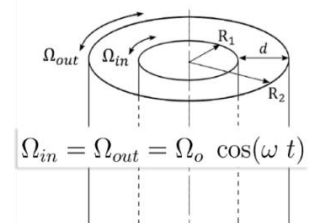


Figure 1. Géométrie

Les équations régissant l'écoulement de base sont l'équation de conservation de la quantité de mouvement,

les équations de la loi de comportement et l'équation de conservation de la masse,

$$\begin{cases} \rho \left(\frac{\partial}{\partial t} \mathbf{u} + (\mathbf{u} \cdot \nabla) \mathbf{u} \right) = -\nabla p + \nabla \cdot \boldsymbol{\tau} \\ \nabla \cdot \mathbf{u} = 0 \end{cases} \quad (2.1)$$

où \mathbf{u} est la vitesse du fluide, p la pression et $\boldsymbol{\tau}$ le tenseur des contraintes (incluant la contribution visqueuse newtonienne, et la contribution non-newtonienne). Le fluide considéré est supposé de type Maxwell convecté supérieur non-linéaire dont la loi de comportement est donnée par,

$$\boldsymbol{\tau} + \lambda \overset{\nabla}{\boldsymbol{\tau}} = 2\eta \mathbf{D} \quad (2.2)$$

où η est la viscosité dynamique du fluide et \mathbf{D} est le tenseur des taux de déformation,

$$2\mathbf{D} = \nabla \mathbf{u} + \nabla \mathbf{u}^T \quad (2.3)$$

et λ est le temps de relaxation. $\overset{\nabla}{\boldsymbol{\tau}}$ est la dérivée convectée supérieure temporelle du tenseur des contraintes,

$$\overset{\nabla}{\boldsymbol{\tau}} = \frac{\partial}{\partial t} \boldsymbol{\tau} + \mathbf{u} \cdot \nabla \boldsymbol{\tau} - \nabla \mathbf{u}^T \cdot \boldsymbol{\tau} - \boldsymbol{\tau} \cdot \nabla \mathbf{u} \quad (2.4)$$

L'écoulement de base est représenté par un champ de vitesse et un tenseur des contraintes pour lequel V_θ^0 vérifie,

$$\mathbf{u}^0 = \begin{pmatrix} 0 \\ V_\theta^0 \\ 0 \end{pmatrix} \quad \boldsymbol{\tau}^{p0} = \begin{pmatrix} \tau_{rr}^0 & \tau_{r\theta}^0 & \tau_{rz}^0 \\ & \tau_{\theta\theta}^0 & \tau_{\theta z}^0 \\ & & \tau_{zz}^0 \end{pmatrix} \quad p = \{p^0\}$$

Nous introduisons par la suite les variables adimensionnelles suivantes,

$$r = (dx + R_1) \quad V^* = \frac{V}{R_1 \Omega_0} \quad t^* = \frac{t}{\frac{d^2}{\nu}} \quad \boldsymbol{\tau}^* = \frac{\boldsymbol{\tau}}{\frac{\rho \nu R_1 \Omega_0}{d}}$$

avec les conditions aux limites suivantes : $V_\theta^{0*}(0, t^*) = V_\theta^{0*}(1, t^*) = \cos(\sigma t^*)$ et nous définissons par la suite trois nombres sans dimension, à savoir le nombre de Reynolds Re , le nombre d'Elasticité E et le nombre d'onde α : $Re = \frac{\Omega_0 R_1 d}{\nu}$, $E = \frac{\lambda}{\frac{d^2}{\nu}}$, $\sigma = \frac{\omega}{\Omega_0}$, $\alpha = \sigma d$ et $We = Re E$

En considérant la distance entre les deux cylindres d petite devant le rayon du cylindre intérieur R_1 , les termes d'ordre d/R_1 sont négligés. La vitesse de base adimensionnelle vérifiant l'équations aux équations aux dérivées partielles (2.1)-(2.4) s'écrit sous la forme,

$$\begin{aligned} V_\theta^{0*}(x, t^*) &= V_1(x) \cos(\sigma t^*) + V_2(x) \sin(\sigma t^*) \quad (2.5) \\ V_1(x) &= \frac{1}{\cosh(\gamma \xi) + \cos(\gamma \beta)} (\cos(\gamma \beta x) \cosh(\gamma \xi(1-x)) + \cos(\gamma \beta(1-x)) \cosh(\gamma \xi x)), \\ V_2(x) &= \frac{1}{\cosh(\gamma \xi) + \cos(\gamma \beta)} (\sin(\gamma \beta x) \sinh(\gamma \xi(1-x)) + \sin(\gamma \beta(1-x)) \sinh(\gamma \xi x)), \end{aligned}$$

avec les constantes,

$$\gamma = \sqrt{\frac{\sigma}{2}}, \quad \beta = \sqrt{\sqrt{1 + (\sigma E)^2} + \sigma E}, \quad \xi = \sqrt{\sqrt{1 + (\sigma E)^2} - \sigma E}$$

La forme de tenseur de contraintes suivant la composante $r\theta$ en écoulement de base, s'écrit sous la forme,

$$\tau_{r\theta}^0(x, t) = T_1(x) \cos(\sigma t) + T_2(x) \sin(\sigma t) \quad (2.6)$$

avec,

$$T_1 = \frac{1}{(1+(\sigma E)^2)} \frac{\partial}{\partial x} (V_1 - \sigma E V_2), T_2 = \frac{1}{(1+(\sigma E)^2)} \frac{\partial}{\partial x} (\sigma E V_1 + V_2),$$

L'écoulement de base est caractérisé aussi par une contrainte normale suivant la direction azimutale $\theta\theta$ et qui s'écrit sous la forme,

$$\tau_{\theta\theta}^0(x, t^*) = (\epsilon T_3 + T_4) \cos(2\sigma t) + (\epsilon T_5 + T_6) \sin(2\sigma t) + (\epsilon T_7 + T_8) \quad (2.7)$$

avec,

$$T_3 = \frac{We}{(1+(2\sigma E)^2)} (-V_1 T_1 + 2\sigma E V_1 T_2 + 2\sigma E V_2 T_1 + V_2 T_2),$$

$$T_4 = \frac{-We}{(1+(2\sigma E)^2)} \left(-T_1 \frac{\partial V_1}{\partial x} + 2\sigma E T_1 \frac{\partial V_2}{\partial x} + 2\sigma E T_2 \frac{\partial V_1}{\partial x} + T_2 \frac{\partial V_2}{\partial x} \right),$$

$$T_5 = \frac{We}{(1+(2\sigma E)^2)} (-2\sigma E T_1 V_1 - T_1 V_2 - T_2 V_1 + 2\sigma E T_2 V_2),$$

$$T_6 = \frac{We}{(1+(2\sigma E)^2)} \left(2\sigma E T_1 \frac{\partial V_1}{\partial x} + T_1 \frac{\partial V_2}{\partial x} + T_2 \frac{\partial V_1}{\partial x} - 2\sigma E T_2 \frac{\partial V_2}{\partial x} \right),$$

$$T_7 = -We(T_1 V_1 + T_2 V_2),$$

$$T_8 = We \left(T_1 \frac{\partial V_1}{\partial x} + T_2 \frac{\partial V_2}{\partial x} \right).$$

3. Analyse de stabilité linéaire

Nous allons nous intéresser par la suite à la stabilité de l'écoulement décrit précédemment vis à vis des perturbations axisymétriques. Nous considérons que le champ des vitesse est,

$$\begin{pmatrix} u(r, z, t) = U(r, t) e^{i\sigma z} \\ v(r, z, t) = V_\theta^0(r, t) + V(r, t) e^{i\sigma z} \\ w(r, z, t) = W(r, t) e^{i\sigma z} \end{pmatrix}$$

la pression est,

$$p(r, z, t) = P^0(r, t) + P(r, t) e^{i\sigma z}$$

et le tenseur des contraintes est

$$\begin{aligned} \tau_{rr}^p &= \bar{R}R(r, t) e^{i\sigma z} & \tau_{\theta z}^p &= \bar{\theta}Z(r, t) e^{i\sigma z} \\ \tau_{r\theta}^p &= \tau_{r\theta}^0(r, t) + \bar{R}\bar{\theta}(r, t) e^{i\sigma z} \\ \tau_{\theta\theta}^p &= \tau_{\theta\theta}^0(r, t) + \bar{\theta}\bar{\theta}(r, t) e^{i\sigma z} \\ \tau_{rz}^p &= \bar{R}Z(r, t) e^{i\sigma z} & \tau_{zz}^p &= \bar{Z}Z(r, t) e^{i\sigma z} \end{aligned}$$

Le comportement de l'écoulement perturbé est contrôlé par trois paramètres : le nombre de fréquence σ , le nombre de Taylor Ta et le nombre d'élasticité E .

4. Approche Numérique et Résultats

Nous adoptons la théorie des méthodes de collocation spectrale pour la résolution spatiale du problème de stabilité linéaire. Ces méthodes se révèlent d'un grand intérêt en raison de leur très grande précision [6]. Le système de stabilité s'écrit sous la forme d'un système de Floquet ayant la forme suivante :

$$\mathbf{B} \frac{\partial}{\partial t^*} \mathbf{X} = \mathbf{M}(t^*) \mathbf{X} \quad (4.1)$$

Avec \mathbf{B} et $\mathbf{M}(t^*)$ sont des matrices d'ordre $8N$ et,

$$\mathbf{X}^t = (\hat{U}_j^*, \hat{V}_j^*, \bar{R}R_j^*, \bar{R}\bar{\theta}_j^*, \bar{R}Z_j^*, \bar{\theta}\bar{\theta}_j^*, \bar{\theta}Z_j^*, \bar{Z}Z_j^*)$$

Le système (4.1) est intégré en utilisant la méthode de Runge-Kutta du quatrième ordre. Enfin, on a une relation entre la fréquence γ , le nombre de Taylor Ta et le nombre d'onde q et qui s'écrit formellement sous la forme,

$$\mathcal{R}(T_a, q, \gamma, E) = 0 \quad (4.2)$$

5. Résultats et conclusion

Les valeurs critiques des nombres de Taylor et d'onde, obtenues à partir des courbes de stabilité marginale, sont présentées, en fonction du nombre de fréquence lorsque $E = 0,01$, sur la figure Fig.2 Le diagramme de stabilité $Tac(\gamma)$ sépare les états stables des états instables dans le sens où lorsque $Ta < Tac$ l'écoulement est stable et il est bien décrit par l'écoulement de base purement azimuthal qui devient instable, avec l'apparition des tourbillons de Taylor, lorsque $Ta = Tac$. Comme on peut le voir sur la figure 2 (a), ce diagramme est composé de dix branches sécantes à une valeur spécifique du nombre de fréquence. Ces intersections correspondent aux états bi-critiques obtenus à partir des courbes de stabilité marginale. Ces dernières ne sont pas uni-modales comme le cas Newtonien mais elles se constituent de plusieurs minimums. La dynamique du système admet alors une compétition entre des modes d'instabilité de différentes natures. Chaque branche dans le diagramme de la figure 2 (a) est identifiée selon le mode dominant. En outre, l'écoulement est considéré potentiellement instable à la fréquence intermédiaire $\gamma_p = 2,4$ qui est le minimum de la courbe $Tac(\gamma)$. La valeur critique du nombre de Taylor à cette fréquence est $Tac = 95,03$. Un comportement similaire est observé dans le cas Newtonien avec une légère différence notée au niveau de γ_p et Tac où $\gamma_p = 2$ et $Tac = 123,07$ [4]. De plus, une augmentation du nombre de Taylor critique est observée, en faisant varier le nombre de fréquence γ à partir de γ_p , menant alors à une stabilisation de l'écoulement lorsque $\gamma \neq \gamma_p$. Il est à noter par ailleurs que lorsque $\gamma > 4$ le diagramme de stabilité est composé de quatre branches paraboliques. L'évolution du nombre d'onde critique, illustré sur la figure 2 (b), présente à son tour dix branches en fonction du mode dominant. La transition d'un mode à un autre est accompagnée d'un saut/chute. Les discontinuités dans l'évolution du nombre d'onde sont très fortes pour $\gamma > 4$. Selon la gamme du nombre de fréquence, l'instabilité peut passer d'un régime de nombres d'onde critiques modérés à un régime élevé ou vice-versa. Sur l'intervalle $8,25 \leq \gamma \leq 9$, les résultats numériques indiquent que le mode M_0 est dominant et l'instabilité se produit avec un nombre d'onde critique modéré qui évolue entre 5,9 et 7,6.

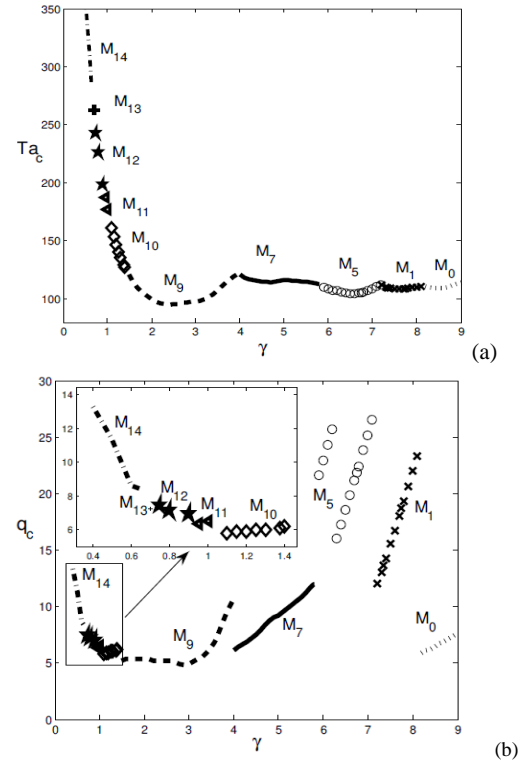


Figure 2. Diagrammes de stabilité décrivant l'évolution des nombres de Taylor et d'onde critiques en fonction du nombre de fréquences γ pour $E = 0,01$, a) $Tac(\gamma)$; b) $qc(\gamma)$

Références

- [1] H. RUBIN, C. ELATA, Stability of Couette flow of dilute polymer solutions. Phys. Fluids 9, 1926- 1933. 1969.
- [2] M. DENN, & J. J. ROISMAN, Rotational stability and measurement of normal stress functions in dilute polymer solutions. AIChEJ. 15, 454-459. 1969.
- [3] M. Riahi, S. Aniss, M. T. Ouazani and S. Skali Lami, "Stability of a Pulsed Taylor-Couette Flow in a Viscoelastic Fluid", Nihon Reoroji Gakkaishi Vol.42, No.5, 321~327(2014).
- [4] A. Aouidef, C. Normand, A. Stegner, J. E. Wesfreid, "Centrifugal instability of pulsed flow" Phys Fluids, 11, 3665 (1994).
- [5] R. G. Larson, Eric S. G. Shaqfeh And S. J. Muller, "A purely elastic instability in Taylor-Couette flow", J. Fluid Mech. (1990), vol. 218, p p . 573-800
- [6] J. A. C. Weideman. S. C. Reddy. A MATLAB differentiation matrix suite. ACM transaction on mathematical software, 26,(2000)

Rayleigh-Bénard flow for a shear-thinning fluid in a 3D cavity

D. HENRY, S. MILLET, S. DAGOIS-BOHY, V. BOTTON AND H. BEN HADID

Laboratoire de Mécanique des Fluides et d'Acoustique, CNRS/Université de Lyon,
École Centrale de Lyon/Université Lyon 1/INSA Lyon,
ECL, 36 avenue Guy de Collongue, 69134 Ecully Cedex, France.

Abstract:

A continuation method is used to study Rayleigh-Bénard convection in a shear-thinning fluid inside a 3D parallelepiped cavity. The focus is put on the two stable branches which exist in the Newtonian case, a stable primary branch of transverse rolls B_1 and a primary branch of longitudinal roll B_2 , stabilized beyond a secondary bifurcation point S_2 . Although the primary bifurcation points are unchanged, the shear-thinning properties strongly modify the bifurcation diagram. Indeed, the stable solutions can exist at much smaller Rayleigh numbers Ra , on subcritical branches beyond saddle-node points SN_1 and SN_2 . See details of this work in [1].

Keywords: *convection in cavities, bifurcation, non-Newtonian fluids*

1 Introduction

The convection obtained by heating from below, known as the Rayleigh-Bénard convection, has been widely studied, first for theoretical reasons as the analysis of pattern formation and then for its great interest in practical applications. In such situations, a critical Rayleigh number Ra_c has to be reached for the onset of flows, and subsequent flow bifurcations, either steady or oscillatory, can occur before a chaotic state is reached, which gives very interesting flow dynamics. The dynamics is particularly rich in confined situations where geometry effects and boundary conditions play an important role and where symmetry considerations are involved. Numerical linear stability analyses have first been carried out to determine the variation of Ra_c with the aspect ratio of the cavity [2]. For further non-linear studies, the numerical methods using parameter continuation and bifurcation methods proved to be very efficient [3, 4, 5].

All these aforementioned studies generally refer to Newtonian fluids, i.e. fluids in which the viscous shear stress is proportional to the shear rate, with a constant of proportionality which is the viscosity. However, the liquids involved in many engineering applications and in some geophysical phenomena very often present more complex rheological behaviors. We can mention the viscoplastic rheology often associated with geophys-

ical flows, where the fluids generally behave as shear-thinning liquids. Models, where the viscosity remains a scalar but varies with the local shear rate, decreasing for a shear-thinning fluid, are proposed, as the Carreau inelastic model.

Different works on Rayleigh-Bénard flows in shear-thinning fluids have used the Carreau model. They first concern numerical simulations in either 2D rigid cavities [6] or 2D cavities with stress-free lateral walls [7]. The Rayleigh-Bénard dynamics is also studied in extended layers with weakly nonlinear analyses [8]. All these studies point out the existence of flows appearing on subcritical solution branches when shear-thinning is strong enough.

We did not find any paper related to three-dimensional (3D) numerical simulations of such Rayleigh-Bénard flows in shear-thinning fluids, inside confined 3D cavities. This led us to adapt a 3D spectral finite element code using continuation and developed for Newtonian fluids to study the Rayleigh-Bénard convection in a shear-thinning fluid inside a parallelepiped enclosure.

2 Mathematical model and numerical techniques

The mathematical model consists of a parallelepiped cavity filled with a shear-thinning fluid and differentially heated. The cavity has aspect ratios $A_z = l/h = 2$ and $A_y = w_d/h = 1$, where l is the length of the cavity, h is its height and w_d is its width, as shown schematically in figure 1. The lower and upper walls are isothermal and held at different temperatures, T_H^* for the lower wall and T_C^* for the upper wall, with $T_H^* > T_C^*$, whereas the other walls are insulated. All the boundaries are rigid walls with no-slip conditions. The non-Newtonian fluid is assumed to follow the four-parameter Carreau inelastic model, i.e. its dynamic viscosity varies with the shear rate, from the value at zero shear rate to the asymptotic value at infinite shear rate, with an intermediate power-law variation. Such viscosity μ_c following the Carreau model can be expressed as

$$\frac{\mu_c - \mu_\infty}{\mu_0 - \mu_\infty} = [1 + (\delta|\dot{\gamma}^*|)^2]^{(n-1)/2}, \quad (1)$$

where μ_0 and μ_∞ are the limit Newtonian dynamic viscosities at zero and infinite shear rate, respectively, n is the power-law index, δ is a characteristic time of the fluid, and $\dot{\gamma}\mathbf{u}$ is the shear rate. Using μ_0 to normalize the viscosity and κ/h^2 (κ is the thermal diffusivity) to normalize the shear rate, and defining I as μ_∞/μ_0 , we can also write

$$\frac{\mu_c}{\mu_0} = I + (1 - I) [1 + (L|\dot{\gamma}\mathbf{u}|)^2]^{(n-1)/2}. \quad (2)$$

In this expression, $L = \delta\kappa/h^2$ is the dimensionless characteristic time and $\dot{\gamma}\mathbf{u}$ is the dimensionless shear rate. Finally the Carreau dynamic viscosity can also be expressed as $\mu_c = \mu_0(1 + \mu)$, where μ is the dimensionless departure from the dynamic viscosity at zero shear rate.

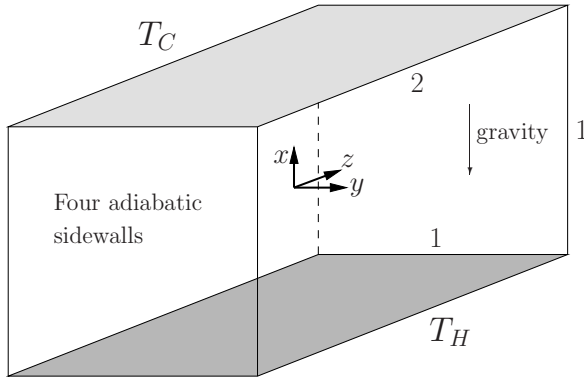


Figure 1: Geometry of the dimensionless heated cavity.

The other physical properties of the fluid (thermal diffusivity κ , density ρ) will be assumed constant, except that, according to the Boussinesq approximation, the fluid density is considered as temperature dependent in the buoyancy term with a linear law $\rho = \rho_m(1 - \beta(T^* - T_m^*))$, where β is the thermal expansion coefficient and T_m^* is a reference temperature taken as the mean temperature $(T_H^* + T_C^*)/2$.

The convective motions are then modeled by the Navier-Stokes equations coupled to an energy equation. Using h , h^2/κ , κ/h , $\rho_m\kappa^2/h^2$ and $\Delta T^* = (T_H^* - T_C^*)$ as scales for length, time, velocity, pressure and temperature, respectively, these equations take the following form:

$$\nabla \cdot \mathbf{u} = 0, \quad (3)$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + Pr \nabla \cdot ((1 + \mu(\mathbf{u})) \dot{\gamma} \mathbf{u}) + Pr Ra T e_x, \quad (4)$$

$$\frac{\partial T}{\partial t} + (\mathbf{u} \cdot \nabla) T = \nabla^2 T, \quad (5)$$

with the corresponding boundary conditions. The non-dimensional parameters are the Rayleigh number, $Ra =$

$\beta \Delta T^* g h^3 / (\kappa \nu_0)$ and the Prandtl number, $Pr = \nu_0 / \kappa$, where ν_0 is the kinematic viscosity defined as $\nu_0 = \mu_0 / \rho_m$ and g is the gravitational acceleration.

The governing equations of the model are solved in the three-dimensional domain using a spectral element method, with a spatial discretization obtained through Lagrange polynomials with Gauss-Lobatto-Legendre points distributions and a time discretization carried out using a semi-implicit splitting scheme. This code was then transformed in a continuation code allowing steady state solving, eigenvalue and eigenvector calculation, and determination of bifurcation points through Newton methods, as it is described in [4, 5]. Such methods, originally developed for Newtonian fluids, were recently adapted to deal with Carreau fluids.

3 Results

The bifurcation diagram obtained in the Newtonian case is given in figure 2.

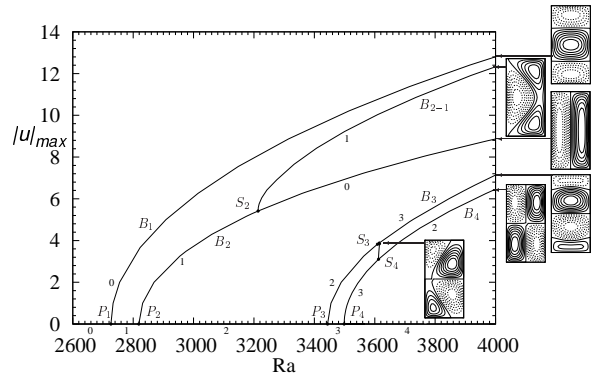


Figure 2: Bifurcation diagram in the case of a Newtonian fluid in a $1 \times 1 \times 2$ cavity heated from below ($Pr = 1$).

In this case, the four primary branches (denoted as B_1 to B_4), which emerge beyond a threshold (P_1 to P_4 , respectively) from the no-flow solution, are found to evolve supercritically. The first primary branch B_1 , which corresponds to two transverse rolls, is stable in the calculated range of Ra . The second primary branch B_2 , which corresponds to a single longitudinal roll, is unstable at its onset, but stabilized at a secondary bifurcation point S_2 , where an unstable two-oblique-roll structure appears. In contrast, the third and fourth primary branches B_3 and B_4 (corresponding to three transverse rolls and four-roll structure, respectively) remain unstable. Stable solutions will then only exist on the B_1 and B_2 branches, from the primary bifurcation point P_1 ($Ra_{P_1} = 2726.53$) for B_1

and from the secondary bifurcation point S_2 ($Ra_{S_2} = 3213.62$) for B_2 . It is why we will study the effect of shear-thinning on these two branches.

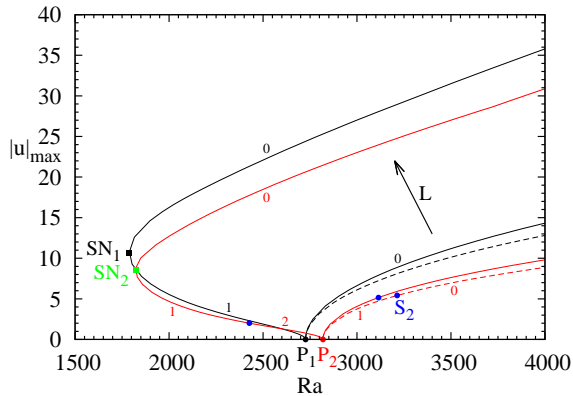


Figure 3: Bifurcation diagrams in the case of a shear-thinning fluid with $n = 0.5$ for three different values of L , 0, 0.01 and 0.1 ($I = 0$). First branch B_1 initiated at P_1 in black, second branch B_2 initiated at P_2 in red. The branches in the Newtonian case ($L = 0$) are given as dashed lines.

For a shear-thinning fluid with $n = 0.5$, the evolution of the first two branches for different values of the parameter L is shown in figure 3. We first see that the primary bifurcation points do not change with the non-Newtonian properties and are the same as in the Newtonian case. Compared to the Newtonian case ($L = 0$, dashed lines), the change of the value of L to 0.01 does not change much the bifurcation diagram: the flow intensities are only slightly increased, whereas the threshold for the bifurcation point S_2 is slightly decreased, which gives a quicker stabilization of the branch B_2 . In contrast, the bifurcation diagram for $L = 0.1$ has strongly evolved. The two primary bifurcations have changed from supercritical to subcritical, with an already important subcriticality. Both subcritical branches B_1 and B_2 turn towards larger Ra values at saddle-node points SN_1 (black solid square, $Ra_{SN_1} = 1788.96$) and SN_2 (green solid square, $Ra_{SN_2} = 1822.42$), respectively. For $L = 0.1$, as the first primary branch B_1 emerges subcritically, it is now one time unstable at onset and is stabilized beyond the saddle-node point SN_1 . Conversely, the second primary branch B_2 is now two time unstable at onset, becomes one time unstable at the secondary bifurcation point S_2 and is eventually stabilized beyond the saddle-node point SN_2 . For such sufficiently large values of L , the important bifurcation points are then the saddle-node points SN_1 and SN_2 as, for both primary branches, they determine the range of Ra where stable flow solutions can be obtained. This range, already important for $L = 0.1$, will

still increase for larger values of L .

4 Conclusion

The continuation method was successfully used to study Rayleigh-Bénard convection in a shear-thinning fluid inside a 3D cavity. It allowed to characterize the modifications induced by the shear-thinning properties on the flow dynamics and particularly the appearance of subcritical branches, thanks to the precise calculation of the stable and unstable solutions and of the saddle-node points.

References

- [1] D. Henry, S. Millet, S. Dagois-Bohy, V. Botton, H. Ben Hadid. Rayleigh-Bénard flow for a Carreau fluid in a parallelepiped cavity. *J. Fluid Mech.*, 936:A24, 2022.
- [2] I. Catton. Convection in a closed rectangular region: the onset of motion. *ASME J. Heat Transfer*, 92:186–188, 1970.
- [3] D. Puigjaner, J. Herrero, C. Simó, and F. Giralt. Bifurcation analysis of steady Rayleigh-Bénard convection in a cubical cavity with conducting sidewalls. *J. Fluid Mech.*, 598:393–427, 2008.
- [4] J.F. Torres, D. Henry, A. Komiya, S. Maruyama, and H. Ben Hadid. Three-dimensional continuation study of convection in a tilted rectangular enclosure. *Phys. Rev. E*, 88:043015, 2013.
- [5] J.F. Torres, D. Henry, A. Komiya, and S. Maruyama. Bifurcation analysis of steady natural convection in a tilted cubical cavity with adiabatic sidewalls. *J. Fluid Mech.*, 756:650–688, 2014.
- [6] O. Benouared, M. Mamou, and N. Ait Mes-saoudene. Numerical nonlinear analysis of subcritical Rayleigh-Bénard convection in a horizontal confined enclosure filled with non-Newtonian fluids. *Phys. Fluids*, 26:073101, 2014.
- [7] M. Jenny, E. Plaut, and A. Briard. Numerical study of subcritical Rayleigh-Bénard convection rolls in strongly shear-thinning Carreau fluids. *J. Non Newtonian Fluid Mech.*, 219:19–34, 2015.
- [8] M. Bouteraa, C. Nouar, E. Plaut, C. Metivier, and A. Kalck. Weakly nonlinear analysis of Rayleigh-Bénard convection in shear-thinning fluids: nature of the bifurcation and pattern selection. *J. Fluid Mech.*, 767:696–734, 2015.

Contrôle des instabilités d'écoulement subsonique sur plaque plane à l'aide d'un système de bosse-creux

Y. BANDADI^a, A. SBAIBI^b, A. LACHHEB^c, R.SAADANI^d

^a. FS Meknès : y.bandadi@edu.umi.ac.ma

^b. FST Mohammedia: ahmed.sbaibi@fstm.ac.ma

^c. EST Meknès : lachhebamine2013@gmail.com

^d. EST Meknès : r.saadani@umi.ac.ma

Résumé

On présente dans ce manuscrit des résultats de la simulation numérique d'un écoulement subsonique sur plaque plane avec des rugosités sous excitation acoustique. Les résultats montrent l'atténuation des ondes de Tollmien-Schlichting caractérisant les premières instabilités dans la couche limite, ce qui permet alors le contrôle des instabilités en vue d'éviter la transition à la turbulence. L'étude est menée à l'aide d'un code de simulation numérique directe des équations de Navier-Stokes dans leur forme pseudo-compressible.

Mots clefs : Acoustique, Instabilités, Contrôle, Tollmien-Schlichting.

1. Introduction

La réceptivité est le processus par lequel une perturbation dans l'environnement de la couche limite (onde acoustique, onde de vortécité, un obstacle géométrique, une courbure de la paroi, vibration de la plaque.) interagit avec l'écoulement pour générer des instabilités.

Si l'amplitude des perturbations est suffisamment faible l'instabilité obtenue est dite, primaire, elle se manifestent sous la forme d'ondes de Tollmien-Schlichting (T-S), lorsque cette instabilité atteint une amplitude suffisante elle déclenche une nouvelle instabilité, dite secondaire, qui déstructure ces ondes planes de (T-S) par l'émergence de perturbations tridimensionnelles qui s'amplifient très fortement et conduisent finalement à la transition de la couche limite laminaire vers la couche limite turbulente. La figure (Fig.1) présente les différents scénarios de transitions

pour les écoulements couches limites proposés par Morkovin [1]

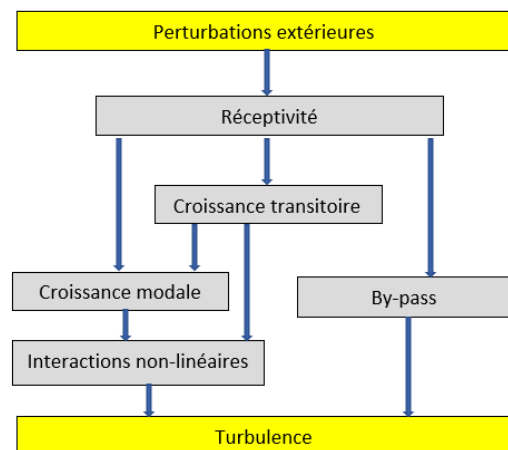


Fig.1 : Scenarios de transition pour les écoulements de couches limites

Tollmien (1929) et Schlichting (1933) étudièrent l'instabilité primaire en adoptant la théorie linéaire, qui consiste à introduire dans l'écoulement plan des perturbations périodiques à faible amplitude pour que les termes non linéaires soient négligeables et transformer ainsi les équations de Navier-Stokes perturbées en une équation différentielle ordinaire.

2. Méthode numérique

La méthode numérique utilisée prend en compte l'effet de la viscosité dans la résolution des équations de Navier-Stokes incompressible par la méthode de pseudo-compressibilité, dans cette méthode l'équation de continuité :

$$\nabla U = 0$$

Est remplacée par :

$$\nabla U = \frac{-1}{\beta} \times \frac{\partial P}{\partial t}$$

Où β la substitution de $\rho.c^2$ est appelée facteur de compressibilité. Avec un nombre d'itération

suffisamment grand en pseudo temps t on retrouve l'équation :

$$\nabla U = 0$$

Ainsi le système devient hyperbolique, cette propriété permet de bénéficier de techniques utilisées dans la résolution des équations compressibles.

Les équations de Navier-Stokes sont exprimées en termes de flux :

$$\frac{\partial q}{\partial t} + \frac{\partial f}{\partial x} + \frac{\partial g}{\partial y} = \frac{1}{Re} \left[\frac{\partial(f_v)}{\partial x} + \frac{\partial(g_v)}{\partial y} \right]$$

Le vecteur de variables q et les vecteurs de flux incompressibles sont les suivants :

$$q = \begin{bmatrix} p_\rho \\ u \\ v \end{bmatrix}; f = \begin{bmatrix} \beta u \\ p_\rho + u^2 \\ uv \end{bmatrix}; g = \begin{bmatrix} \beta v \\ uv \\ p_\rho + v^2 \end{bmatrix}$$

Et les termes visqueux sont donnés par :

$$f_v = \begin{bmatrix} 0 \\ \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial x} \end{bmatrix}; g_v = \begin{bmatrix} 0 \\ \frac{\partial u}{\partial y} \\ \frac{\partial v}{\partial y} \end{bmatrix}$$

Pour étudier la réceptivité acoustique de la couche limite, plusieurs simulations numériques sont à faire : on doit en premier établir l'écoulement stationnaire le long de la plaque à obstacle géométrique (Fig.2), puis introduire l'onde acoustique (perturbation de la pression de forme sinusoïdale), ensuite on retranche l'état initial de l'état final à condition qu'ils soient séparés d'un multiple de la période de l'excitation afin d'éliminer l'onde acoustique et le bruit dû au phénomène de Stokes, le résultat est ensuite mis à un filtrage.

Les ondes de Tollmien-Schlichting sont observées en aval de l'obstacle au niveau de la composante longitudinale de la vitesse $u(x)$ (Fig.3).

3. Résultats

On peut clairement voir que l'ampleur des instabilités qui risquent d'apparaître sur la couche limite pourraient bien s'atténuer voir disparaître, ce qui rend cette méthode de d'inversement de l'accident géométrique un moyen qui s'impose entre autres ([2], ... [6]) dans le contrôle des instabilités d'écoulements.

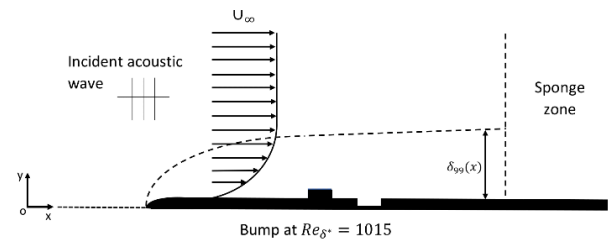


Fig.2 : Croquis du domaine de simulation

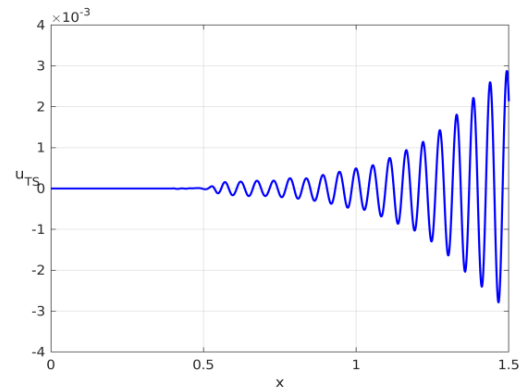


Fig.3 : Développement des ondes T-S pour une bosse

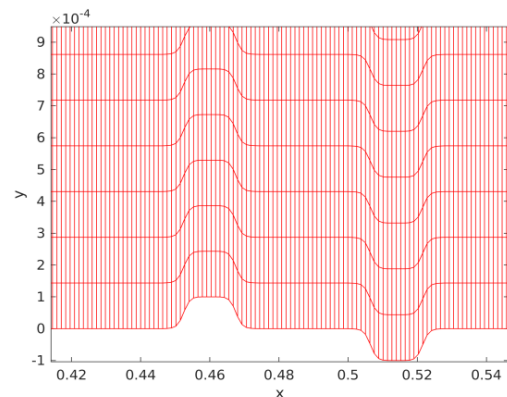


Fig.4 : Maillage au voisinage de système {bosse-creux}

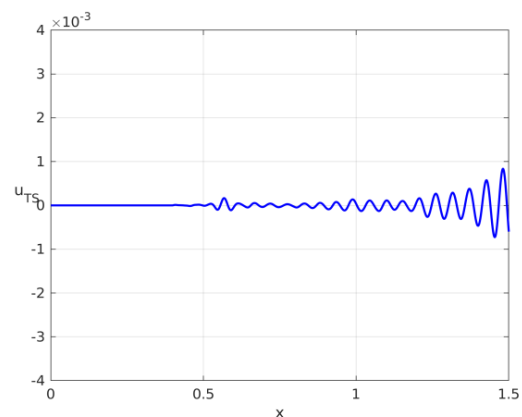


Fig.5 : Ondes T-S pour le système {bosse-creux}

Références

- [1] M. V. Morkovin, *On the Many Faces of Transition*, in *Viscous Drag Reduction*, Wells C.S., Springer, Boston, MA, 1969.
- [2] J. A. Franco, *Effect of humps and indentations on boundary-layer transition of compressible flows using the AHLNS methodology*, 6th European Conference on Computational Mechanics, Glasgow, 2018.
- [3] C. Martel, *Stabilization of Tollmien-Schlichting Waves by Mode Interaction*, Springer, 2018
- [4] L. Siconol, *Boundary layer stabilization using free-stream vortices*, JFM RAPIDS, 2014.
- [5] J. Unadkat, *Applications and Computation of Unsteady Boundary Layers over Finite Domains*, Thèse, University of Manchester, 2017
- [6] M. Serdar Genç, *Traditional and New Types of Passive Flow Control Techniques to Pave the Way for High Maneuverability and Low Structural Weight for UAVs and MAVs*, intechopen, 2020

Coupling Heat Transfer and Peristaltic Transport of a Newtonian Fluid Through an Elliptical Duct

Couplage entre le Transfert de Chaleur et le Transport Péristaltique d'un Fluide Newtonien dans une Conduite Elliptique

H. RACHID¹, M. T. OUZZANI², N. LAHLOU³

1,2,3. Laboratory of Mechanics, Department of Physics, Faculty of Sciences Ain Chock, Hassan II University-Casablanca, Morocco

Abstract :

The aim of this paper is to study a mathematical model coupling the peristaltic pumping with heat transfer phenomenon for an incompressible Newtonian fluid in an elliptical tube. The Navier-Stokes and energy equations have been solved for long wavelength, small Reynolds and Peclet numbers approximations. Exact expressions of velocity and temperature have been found in the wave frame analysis. The impacts of pertinent parameters on mechanical efficiency, entropy and Bejan number have been analyzed with the help of graphs.

Résumé :

Le but de cet travail est d'étudier un modèle mathématique couplant le phénomène de pompage péristaltique avec le transfert de chaleur pour un fluide Newtonien incompressible dans un tube elliptique. Les équations de Navier-Stokes et de l'énergie ont été résolues pour le cas des grandes longueurs d'onde et des faibles nombres de Reynolds et de Peclet. Les expressions exactes de la vitesse et de la température ont été trouvées dans le repère lié à l'onde. Les effets des pertinents paramètres sur l'efficacité mécanique, l'entropie et sur le nombre de Bejan ont été graphiquement analysés.

Keywords : *Peristaltic flow, Mechanical efficiency, Entropy, Bejan number, Elliptical duct*

Mots clés : *Écoulement péristaltique, Efficacité mécanique, Entropie, Nombre de Bejan, Conduite elliptique*

1 Introduction

Fluids flow through deformable tubes/channels when a sinusoidal wave propagates along the walls is called "peristalsis". This phenomenon has several applications as in medicine, human body (blood flow, chyme transport patterns) and industry. In the literature, the most of researchers studied this flow for the simple case when the duct having circular section. Heat transfer is a widespread natural phenomenon of nature. The coupling between this last phenomenon and peristaltic flow becomes an important subject of several researches and of great importance

for the practical design of heat exchangers. This coupling is also applicable in biological systems and in industry. Over the last decade, the importance of this coupling attract the scientific researchers to publish several theoretical articles. In addition to the first law of thermodynamics, the second law is also often related to heat transfer phenomenon in physical or chemical systems. The rectification of this law permits the construction of thermodynamic functions of state especially the entropy due to an irreversible process. Entropy is an extensive function which characterizes the state of "disorder" of each system. It increases when the system process is more irreversible [1]. The fluid flows through elliptical ducts are of increasing importance in micro-fluidics, biological flow (like blood sample preparation and glucose monitoring), heat exchangers, fuel cells. Generally, the elliptical channel heat sinks design can significantly reduce the size and weight of the apparatus [2]. Furthermore, it is worth noting that the elliptical shape is very common in current mini- and micro-devices since it can be easily manufactured with micro-drilling techniques or chemical etching. It is thus indispensable to determine the fluid flow and heat transfer performance in elliptical ducts. Although the importance of heat transfer phenomenon in engineering and industry, there is not much theoretical work has been done using heat exchangers with elliptical cross-section. The complexity of the mathematical model corresponding to this complex geometry versus circular duct is due to the fact that the condition of symmetry of the flow is not verified and the transformation to a cylindrical-polar coordinates system is very inefficient. In this case, it is necessary to choose the hydraulic diameter $D_h = \frac{4A}{\Gamma}$ as a characteristic length to evaluate the Reynolds number (A and Γ are the cross-sectional area and the perimeter of the duct, respectively). The originality of this work is to present a mathematical modeling of a Newtonian peristaltic pumping coupled with heat transfer through elliptical duct when the wall temperature is constant. For long wavelength and low Reynolds and Peclet numbers approximations, the exact solution of the problem has been

found in closed form. The effects of pertinent parameters on the physical quantities, especially the pressure drop, mechanical efficiency, entropy generation and on the Bejan number have been discussed through graphs.

2 Mathematical formulation

We consider the fully developed steady and laminar flow for an incompressible viscous fluid through an elliptical duct. The deformable walls are given by sinusoidal equations as follows:

$$\tilde{\alpha}(\tilde{Z}, \tilde{t}) = \alpha_0 + d \sin \frac{2\pi}{\lambda} (\tilde{Z} - c\tilde{t}) \quad (1)$$

$$\tilde{\beta}(\tilde{Z}, \tilde{t}) = \beta_0 + d \sin \frac{2\pi}{\lambda} (\tilde{Z} - c\tilde{t}) \quad (2)$$

α_0, β_0 are the semi-major and the semi-minor axis of an ellipse. d, c and λ are the amplitude, the speed and the wavelength, respectively (see Fig. (1)).

Using the following dimensionless parameters:

$$\begin{aligned} x &= \frac{\tilde{x}}{D_h}, y = \frac{\tilde{y}}{D_h}, z = \frac{\tilde{z}}{\lambda}, t = \frac{c\tilde{t}}{\lambda}, w = \frac{\tilde{w}}{c}, p = \frac{D_h^2 \tilde{p}}{\mu \lambda c}, \\ \theta &= \frac{\tilde{T} - \tilde{T}_w}{\tilde{T}_b - \tilde{T}_w}, Q = \frac{\tilde{Q}}{\pi c D_h^2}, \delta = \frac{\beta_0}{\alpha_0}, \phi = \frac{d}{\beta_0}, \varepsilon = \frac{D_h}{\lambda}, \\ Ec &= \frac{c^2}{c_p(\tilde{T}_b - \tilde{T}_w)}, \Omega = \frac{\tilde{T}_b - \tilde{T}_w}{\tilde{T}_0}, S = \frac{\tilde{S}}{\frac{k(\tilde{T}_b - \tilde{T}_w)^2}{D_h^2 \tilde{T}_0^2}}, \\ Re &= \frac{\rho D_h c}{\mu}, Pr = \frac{\mu c_p}{k}, Pe = Pr Re, Br = Pr Ec \end{aligned} \quad (3)$$

For $\varepsilon \ll 1, Re \ll 1$ and $Pe \ll 1$ we obtain the dimensionless equations which govern the problem in the wave frame analysis as follows:

$$\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} = \frac{\partial p}{\partial z} \quad (4)$$

$$\frac{\partial^2 \theta}{\partial x^2} + \frac{\partial^2 \theta}{\partial y^2} = -Br \left[\left(\frac{\partial w}{\partial x} \right)^2 + \left(\frac{\partial w}{\partial y} \right)^2 \right] \quad (5)$$

the dimensionless boundary conditions are:

$$w = -1 \quad \text{and} \quad \theta = 0 \quad \text{for} \quad \frac{x^2}{\alpha^2} + \frac{y^2}{\beta^2} = 1 \quad (6)$$

where $\alpha(z) = \frac{E(e)}{\pi} \left[\frac{1}{\delta} + \phi \sin(2\pi z) \right]$

and $\beta(z) = \frac{E(e)}{\pi} \left[1 + \phi \sin(2\pi z) \right]$.

with $e = \sqrt{1 - \delta^2}$ and $E(e) = \int_0^{\frac{\pi}{2}} \sqrt{1 - e^2 \sin^2 \alpha} d\alpha$.

The solution of (4) with the help of (6) we obtain the velocity as:

$$w = -1 + \frac{1}{2} \frac{\partial p}{\partial z} \frac{\alpha^2 \beta^2}{\alpha^2 + \beta^2} \left[\frac{x^2}{\alpha^2} + \frac{y^2}{\beta^2} - 1 \right] \quad (7)$$

The integration of (7) over the elliptical cross-section gives:

$$\frac{\partial p}{\partial z} = - \frac{4(\alpha^2 + \beta^2)(Q + \alpha \beta - \int_0^1 \alpha \beta dz)}{\alpha^3 \beta^3} \quad (8)$$

the pressure rise and the frictional force over a wavelength can be found through the following relations:

$$\Delta p = \int_0^1 \frac{\partial p}{\partial z} dz; \quad F = - \int_0^1 \alpha \beta \frac{\partial p}{\partial z} dz \quad (9)$$

We find the mechanical efficiency as follows [3, 4]:

$$E = \frac{Q \Delta p}{\Delta p A_0 + F} = \frac{\frac{Q}{A_0}}{1 + \frac{F}{\Delta p A_0}} \quad (10)$$

where $A_0 = \alpha_{(z=0)} \beta_{(z=0)} = \frac{E(e)^2}{\pi^2 \delta}$.

According to (7), the exact solution of (5) can be assumed in the following form:

$$\theta = C_1 x^4 + C_2 y^4 + C_3 x^2 y^2 + C_4 x^2 + C_5 y^2 + C_6 \quad (11)$$

By inserting (11) in (5), comparing the terms of polynomial coefficients and applying the boundary condition (6) on (11) we find the six unknown coefficients as:

$$C_1 = \frac{1}{12} Br \left(\frac{\partial p}{\partial z} \right)^2 \frac{\beta^4 (\alpha^4 - 6\alpha^2 \beta^2 - \beta^4)}{(\alpha^2 + \beta^2)^2 (\alpha^4 + 6\alpha^2 \beta^2 + \beta^4)} \quad (12)$$

$$C_2 = \frac{1}{12} Br \left(\frac{\partial p}{\partial z} \right)^2 \frac{\alpha^4 (\beta^4 - 6\alpha^2 \beta^2 - \alpha^4)}{(\alpha^2 + \beta^2)^2 (\alpha^4 + 6\alpha^2 \beta^2 + \beta^4)} \quad (13)$$

$$C_3 = -Br \left(\frac{\partial p}{\partial z} \right)^2 \frac{\alpha^4 \beta^4}{(\alpha^2 + \beta^2)^2 (\alpha^4 + 6\alpha^2 \beta^2 + \beta^4)} \quad (14)$$

$$C_4 = \frac{1}{3} Br \left(\frac{\partial p}{\partial z} \right)^2 \frac{\alpha^4 \beta^4 (\beta^2 - \alpha^2)}{(\alpha^2 + \beta^2)^2 (\alpha^4 + 6\alpha^2 \beta^2 + \beta^4)} \quad (15)$$

$$C_5 = \frac{1}{3} Br \left(\frac{\partial p}{\partial z} \right)^2 \frac{\alpha^4 \beta^4 (\alpha^2 - \beta^2)}{(\alpha^2 + \beta^2)^2 (\alpha^4 + 6\alpha^2 \beta^2 + \beta^4)} \quad (16)$$

$$C_6 = \frac{1}{12} Br \left(\frac{\partial p}{\partial z} \right)^2 \frac{\alpha^4 \beta^4 (\alpha^4 + 4\alpha^2 \beta^2 + \beta^4)}{(\alpha^2 + \beta^2)^2 (\alpha^4 + 6\alpha^2 \beta^2 + \beta^4)} \quad (17)$$

The entropy generation and the Bejan number [5] will be found in their dimensionless form as follows:

$$\begin{aligned}
 S &= S_H + S_F \\
 &= \left(\left(\frac{\partial \theta}{\partial x} \right)^2 + \left(\frac{\partial \theta}{\partial y} \right)^2 \right) + \frac{Br}{\Omega} \left(\left(\frac{\partial w}{\partial x} \right)^2 + \left(\frac{\partial w}{\partial y} \right)^2 \right) \quad (18) \\
 Be &= \frac{S_H}{S} = \frac{1}{1 + \chi}
 \end{aligned}$$

with $\chi = \frac{S_F}{S_H}$ is the irreversibility ratio.

3 Results and discussions

The impacts of aspect ratio of ellipsis δ , occlusion ϕ , Brinkman number Br and of volume flow rate Q on pumping and heat transfer phenomenon have been studied in this section. Before, and for $\delta = 1$ we obtain the same results found by Shapiro et al. [3] and Nadeem et al. [6]. Fig. (2) shows that the mechanical efficiency $E = 0$ when $\Delta p = 0$, or $Q = 0$ or both. It can also be seen that there is a critical value Q_c for which E is maximum. For $Q \in]0, Q_c[$ E increases while for $Q \in]Q_c, Q_{max}[$ it decreases to zero.

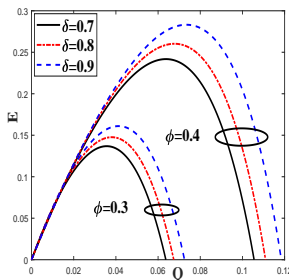


Figure 2: Mechanical efficiency E vs. flow rate Q for different values of occlusion ϕ and aspect ratio δ .

It is concluded from Fig. (3) that the maximum entropy generation S_m along the wavelength increases with δ , ϕ , Br and Q . Physically, entropy generation is a measure of the magnitudes of the irreversibilities present during the process. It is a measure of molecular disorder or randomness of a system. The results show that all parameters enhance the total irreversibility of the system.

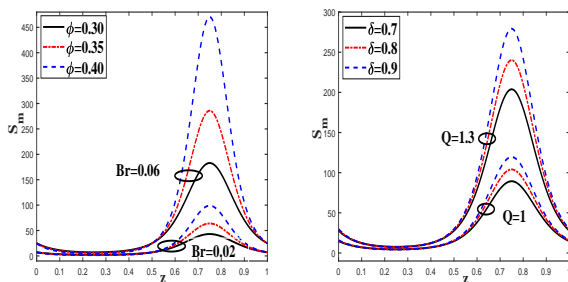


Figure 3: Effects of Brinkman number Br , occlusion ϕ , flow rate Q and aspect ratio δ on entropy generation S_m vs. z .

In order to more explain the variation of the entropy versus these parameters we plotted in Fig. (4) the Bejan number Be_m along the wavelength. This figure showed that Be_m increases with ϕ , Br and Q while it decreases with increasing δ . These remarks mean that the irreversibility ratio $\chi = \frac{S_F}{S_H}$ increases with increasing δ while it decreases with an increase in ϕ , Br and Q .

We concluded that the present work is a mechanical and thermal optimization problem. The optimization criteria consists of increasing the mechanical efficiency (E) and minimizing the entropy generation (S) to improve the performance of the peristaltic pump. The effects of geometrical and physical parameters on E and S have been used to study this optimization.

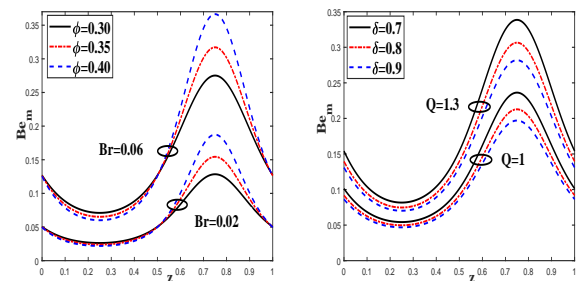


Figure 4: Influences of Brinkman number Br , occlusion ϕ , flow rate Q and aspect ratio δ on Bejan number Be_m vs. z .

References

- [1] J. Letessier, J. Rafelski, A. Tounsi, *In search of entropy*, Acta Phys Pol A (1994) 699-716.
- [2] L. Su, Z. Duan, B. He, H. Ma, Z. Xu, *Thermally developing flow and heat transfer in elliptical minichannels with constant wall temperature*, Micromachines (2019) 713.
- [3] A. H. Shapiro, M. Y. Jaffrin, S. L. Weinberg, *Peristaltic pumping with long wavelengths at low Reynolds number*, J Fluid Mech (1969), 799-825.
- [4] H. Rachid, M. T. Ouazzani, *Modeling coupled heat and mass transfer in peristaltic cylindrical flow of Robertson-Stiff fluid*, Int J Appl Mech (2019) 1950092.
- [5] A. Bejan, *Entropy Generation Optimization*, CRC Press Inc, 1996.
- [6] S. Nadeem, N. S. Akbar, N. Bibi, A. Sadaf, *Influence of heat and mass transfer on peristaltic flow of a third order fluid in a diverging tube*, Commun Nonlinear Sci (2010), 2916-2931.

Caractérisation du comportement rhéologique des solutions aqueuses à base d'argile grise et modélisation de leur transport en conduits péristaltiques

N. LAHLOU¹, M. OUZZANI TOUHAMI¹, S. ANISS¹, H. RACHID¹

1. Laboratoire de mécanique, faculté des sciences Ain Chock, université Hassan II de Casablanca
nouhalahlou10@gmail.com

Résumé

Ce travail est divisé en deux parties distinctes mais complémentaires. La première partie traite de l'étude de l'effet de la concentration sur la rhéologie des solutions aqueuses d'argile grise. Le cas où la concentration est de 50% est particulièrement intéressant, car il a été nécessaire d'insérer une approche spécifique pour la détermination des paramètres rhéologiques introduits par la loi de comportement du modèle de Robertson-Stiff. Dans la deuxième partie, nous étudions le cas général de l'écoulement d'un fluide décrit par ce modèle en conduite péristaltique et dont le but est de déterminer des relations de type "débit-chute de pression" et des rendements mécaniques qui pourraient permettre un meilleur contrôle de ces écoulements.

Mots clés:

Rhéologie/Argile grise/conduits péristaltiques/Robertson-Stiff/débit/chute de pression/rendement mécanique

1. Introduction

Comme cela est le cas des matériaux composites, les matériaux à base d'argile continuent de prendre une place de plus en plus importante dans divers domaines de l'industrie moderne ; ceci est notamment le cas en génie civil, en industrie cosmétique ou encore pour l'industrie pharmaceutique [1]. Dans une logique de contrôle qualité, des tests sont souvent menés à différents niveaux pour s'assurer de la prise en compte des conditions d'exploitation préalablement arrêtées ; celles-ci portant fréquemment sur des propriétés rhéologiques du produit à l'état frais ou encore sur des propriétés liées au transport de ces produits en milieu confiné.

Aussi et au vu de la diversité des applications qui sont aujourd'hui envisagées pour ces matériaux [1,2], la précision de cette double correspondance entre des paramètres de nature structurelle et des paramètres à connotation rhéologique d'une part et entre ces paramètres rhéologiques et le phénomène de transport d'autre part revêt sans nul doute une importance particulière. Dans la mesure où il devient de plus en plus opportun d'introduire, au niveau des lois de comportement, des modèles empiriques capables à eux seuls de faciliter l'identification des différentes corrélations recherchées et de déboucher en conséquence sur des applications industrielles importantes.

Notre travail s'inscrit dans cette démarche et se divise en deux parties complémentaires. Pour la première partie, nous montrons notamment que la variation de la concentration en poudre d'argile permet d'identifier plusieurs formes de lois de comportement avec ou sans seuil. Un intérêt particulier est porté au cas où la masse de la phase liquide est deux fois plus importante que celle de la phase solide dans la mesure où les modèles rhéologiques listés sur le logiciel Rheowin ne permettent pas d'aborder les lectures expérimentales avec une bonne régression. L'utilisation d'une méthode d'optimisation des erreurs nous a permis de vérifier pour ce cas particulier que le comportement rhéologique est très proche de celui observé par Robertson et Stiff en 1976 [2].

Dans la deuxième partie de ce travail, nous proposons, pour le contrôle du transport de ces solutions dans des conduites péristaltiques, un jeu de relations approchées permettant de relier le débit transporté à la perte de charge imposée. Dans ce contexte nous insistons sur le cas particulier où le fluide est décrit par le modèle de Robertson Stiff.

2. Impact de la concentration sur le comportement rhéologique des suspensions aqueuses d'argile grise :

2.1 Matières premières et élaboration

L'argile grise que nous attribuons ici provient du Moyen Atlas marocain. Cette argile est pauvre en carbonate ($\text{CaO} + \text{MgO}$), elle est composée majoritairement de silice SiO_2 (58,06%) et contient un pourcentage important de K_2O . Le spectre obtenu à partir d'un test de diffraction des rayons X (fig.1) montre la présence de silice libre sous forme de quartz SiO_2 , de muscovite $\text{KAl}_3\text{Si}_3\text{O}_{10}(\text{OH})_2$ et d'hallowite $\text{Al}_2\text{Si}_2\text{O}_5(\text{OH})_4 \cdot 2\text{H}_2\text{O}$ sous forme de phase minérale argileuse.

Pour l'élaboration des échantillons, nous procédons comme suit :

- Le tamisage de la poudre d'argile (l'échelle des orifices utilisés est de 500 μm)
- L'ajout d'eau distillée selon les concentrations souhaitées (les ratios [matière sèche (S) / matière liquide (L)] retenus sont de 25%, 50%, 75% et 100%)
- Mélange à travers un agitateur contenant un barreau magnétique avec une vitesse de 6 r/s pendant 5 min.

4.2 Formulation mathématique :

Le système des équations de Cauchy se réduit dans ces conditions à :

$$\begin{cases} \frac{1}{r} \frac{\partial}{\partial r} (rV_r) + \frac{\partial V_z}{\partial z} = 0 \\ \frac{\partial p}{\partial r} = 0 \\ \frac{\partial \tau_{rz}}{\partial r} + \frac{\tau_{rz}}{r} = \frac{\partial p}{\partial z} \end{cases} \quad (3)$$

Et les conditions initiales et aux limites sont données par :

$$\begin{cases} V_z(r=R(z)) = \frac{\partial R(z,t)}{\partial t} = -c \\ V_r(r=R(z)) = 0 \end{cases} \quad \begin{cases} V_z(r=R(z)) = \frac{\partial R(z,t)}{\partial t} = -c \\ V_r(r=R(z)) = 0 \end{cases} \quad (4)$$

Pour l'écoulement péristaltique des fluides à seuil, il est souhaitable de pouvoir corréler les différents paramètres (rhéologiques, géométriques et dynamiques) pour optimiser le transport du fluide. Le rendement mécanique de la pompe est défini alors comme le rapport de l'énergie récupérée par le fluide, utile pour son transport, au travail fourni par les parois par longueur d'onde. [5]

$$E = \frac{Q \Delta P}{\Delta P R^2 (z=0) + F} = \frac{Q/R^2 (z=0)}{1 + F / \Delta P R^2 (z=0)} \quad (5)$$

5. Résultats et discussion :

5.1 Effet des paramètres rhéologiques et géométriques sur la zone de pompage :

On observe (fig.4) qu'un gradient de pression faible est suffisant de transporter un débit donné. Par contre dans la partie étroite, un gradient de pression assez important est nécessaire pour transporter le même débit et spécialement si $z= 0.75$. On remarque l'augmentation des paramètres implique une augmentation de la valeur maximale du gradient de pression.

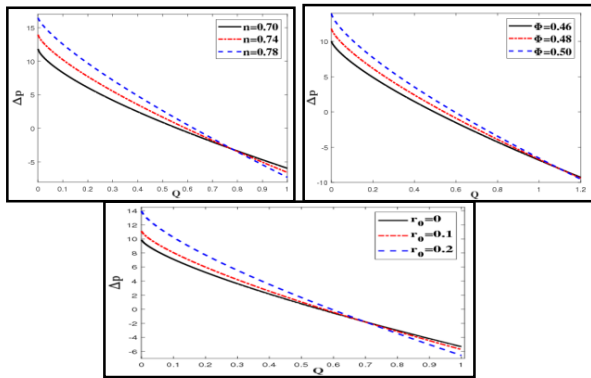


Fig.4 : détermination de la zone de pompage

5.2 Effet des paramètres rhéologiques et géométriques sur le rendement

On observe que la seule zone qui décrit l'évolution de la chute de pression en fonction du débit est celle de pompage. On remarque d'après la figure 5 que la zone du pompage péristaltique augmente avec l'augmentation des paramètres (n, r0 et φ). La chute de pression suit les mêmes variations que le gradient de pression. On observe également un point

d'intersection quand on fait varier l'un ou l'autre des paramètres considérés, ce point est situé dans la zone du pompage. A partir de l'expression du rendement mécanique, on observe qu'il dépend à la fois du débit moyen et le rapport de deux forces ; forces de frottement sur forces de pression. L'évolution de ce rendement débute par une valeur nulle du débit jusqu'à une valeur maximale critique et puis il diminue et tend vers 0. On remarque également que le maximum du rendement augmente avec l'augmentation des paramètres (n, r0 et φ).

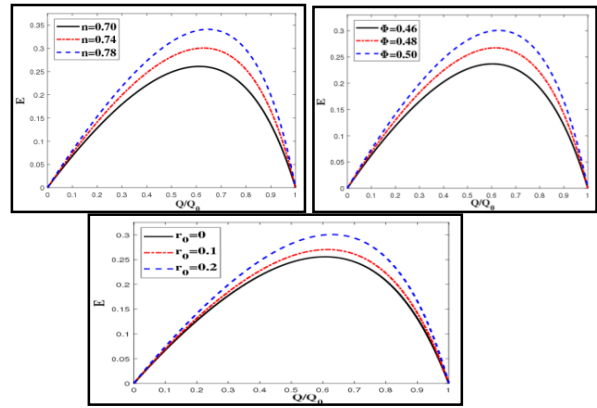


Fig.5 : Rendement mécanique en fonction du débit

6. Conclusion :

Les résultats obtenus dans le cadre de ce travail montrent qu'une connaissance de base relative à l'impact des paramètres rhéologiques et géométriques sur le rendement de ce type de transport est nécessaire pour élaborer la procédure qui permet d'accéder à la force ou la pression à appliquer pour imposer un débit voulu. C'est le cas par exemple pour le pompage (suspensions minérales, béton), l'injection dans des milieux poreux (forage, renforcement de sol), l'injection dans les coffrages (béton), l'enduction (mortier), etc.

Références

[1] M. Martin, Caractéristiques rhéologiques et composition des boues de forage, la houille blanche (1970) 463-470
 [2] R. E. Robertson, H. A. Stiff Jr., An improved mathematical model for relating shear stress to shear rate in drilling fluids and cement slurries, Society of petroleum engineers journal 16 (1976) 31-36.
 [3] N. Lahlou, M. Ouazzani Touhami, R. Moussa, R. Hattaf, Effect of the Formulation of Sodium Activation Solutions on the Setting Time of Metakaolin Based Geopolymers, Journal of Materials Science and Engineering B 9 (1-2) (2019) 6-12
 [4] R.M. Beirute, R.W. Flumerfelt, An evaluation of the Robertson-Stiff model describing rheological properties of drilling fluids and cement slurries, Society of petroleum engineers journal (1977) 97-100
 [5] N.Lahlou, R.Hassan, M. Ouazzani Touhami. Modeling coupled heat and mass transfer in peristaltic cylindrical flow of Robertson-stiff fluid. International Journal of Applied Mechanics (IJAM) <https://doi.org/10.1142/S1758825119500923>

New lubrication model based on ferrohydrodynamic equations of Jenkins model

S. LAGHRABLI, M. NABHANI, M. EL KHLIFI

*Laboratory of Mathematics, Cryptography, Mechanics and Numerical Analysis
Faculty of Sciences and Techniques Mohammeda, Hassan II University of Casablanca, Morocco
laghrablis@yahoo.fr*

Abstract

This study deals with a new lubrication model describing ferrofluid flow in finite journal bearings. The angular momentum of ferrofluid is taken into account by using the ferrohydrodynamic equations of Jenkins model [1, 2]. In the obtained modified Reynolds equation, the effect of two control parameters is considered, namely the Jenkins viscosity and the magnetic force coefficient parameter as given in [3]. This partial differential equation is numerically resolved by finite differences using Gauss-Seidel over-relaxation method. We conclude that for a given magnetic field and at low and medium eccentricity ratios, the pressure and load capacity are higher and the friction factor is lower for the Jenkins model compared with those of the Neuringer-Rosensweig model, but these trends are reversed for high eccentricity ratios.

Keywords: *Partial differential equations, Numerical methods, Reynolds equation*

1. Introduction

Ferrofluid or magnetic fluid is a stable colloidal suspension of small particles of ferromagnetic material in a carrier liquid. In order to avoid agglomeration from van der Waals and magnetic attractions, the ferro-magnetic particles have to be coated with a surfactant. When a magnetic field is applied on the ferrofluid, each particle experiences a magnetic body force depending on the particles magnetization and on the strength of the applied magnetic field. Magnetic fluids are useful in many applications like sensors, centrifugal switches, dampers, sealing, filtering, separation, ink-jet printing and heat transfer. They have also been used in the lubrication of journal bearings with some advantages compared to conventional lubricants by the increase of viscosity due to the effect of magnetic field.

In ferrohydrodynamic, to describe the flow of ferrofluid, the continuity equation, Navier-Stokes equations, Maxwell equations and constitutive equation for the magnetization are combined. Three models are used in literature to describe the flow of ferrofluid, namely the

Neuringer-Rosensweig model [4], the Shliomis model [5] and the Jenkins model [6]. Jenkins presented the isothermal static equilibrium theory for ferrofluid in some detail. In determining the system of equations governing the state, he obtained integrals of the linear momentum equations and identified the magnetic energy density function for ferrofluid. A continuum theory was thus proposed [6].

In the present analysis, our aim is to study ferrofluid lubrication of finite journal bearing using the Jenkins flow model taking into account the angular momentum of the ferrofluid. A magnetic field created by displaced finite wire is used. A modified Reynolds equation is derived and resolved by finite difference method. The effect of two control parameters is investigated: the magnetic force coefficient and the Jenkins viscosity.

2. Modified Reynolds equation

An axial cylindrical finite journal bearing is considered. The applied magnetic field involves the current passing through a finite wire located at a distance R_0 greater than the bearing radius, as given by Osman *et al* [7]. The wire is placed at an optimum angle $\psi = \pi/2$ [8]. The ferrofluid is assumed to be incompressible, non-electric conducting, to have a constant temperature and homogeneous distribution of ferromagnetic particles.

Using Jenking model [1, 2] and neglecting fluid inertia, under the usual assumption of hydrodynamic lubrication the momentum and the continuity equations are given by:

$$-\frac{\partial p}{\partial x} + \left(\eta - \frac{1}{2} \rho \alpha^2 \bar{\mu} H \right) \frac{\partial^2 v_x}{\partial y^2} + \frac{1}{2} \mu_0 \bar{\mu} \frac{\partial H^2}{\partial x} = 0 \quad (1)$$

$$\frac{\partial p}{\partial y} = 0 \quad (2)$$

$$-\frac{\partial p}{\partial z} + \left(\eta - \frac{1}{2} \rho \alpha^2 \bar{\mu} \frac{H y}{H} \right) \frac{\partial^2 v_z}{\partial y^2} + \frac{1}{2} \mu_0 \bar{\mu} \frac{\partial H^2}{\partial z} = 0 \quad (3)$$

$$\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} = 0 \quad (4)$$

where x, y, z are the Cartesian coordinates, p is the lubricant pressure, H is the magnetic field intensity,

H_y is the radial magnetic field component, η is the magnetic fluid viscosity, μ_0 is the free space permeability, ρ is the lubricant density, the symbol α^2 is the material constant of Jenkins Model, $\bar{\mu}$ is the magnetic susceptibility, v_x is the circumferential velocity component, v_y is the radial velocity component, v_z is the axial velocity component.

Integrating equations (1) and (3) and applying the appropriate boundary conditions on velocity, substituting the obtained velocity components v_x and v_z in the continuity equation (4) and integrating across the film thickness, using the boundary conditions on v_z , the dimensionless form of the modified Reynolds equation is obtained:

$$\frac{\partial}{\partial \theta} \left(f(\theta, \tilde{z}) \frac{\partial \tilde{p}}{\partial \theta} \right) + \gamma \frac{\partial}{\partial \tilde{z}} \left(g(\theta, \tilde{z}) \frac{\partial \tilde{p}}{\partial \tilde{z}} \right) = 6 \frac{\partial \tilde{h}}{\partial \theta} + \frac{\lambda_2}{\gamma} \frac{\partial}{\partial \theta} \left(F(\theta, \tilde{z}) \frac{\partial \tilde{H}}{\partial \theta} \right) \quad (5)$$

$$+ \lambda_2 \frac{\partial}{\partial \tilde{z}} \left(G(\theta, \tilde{z}) \frac{\partial \tilde{H}}{\partial \tilde{z}} \right)$$

where

$$\theta = \frac{x}{R}, \quad \tilde{z} = \frac{z}{L}, \quad \tilde{h} = \frac{h}{C}, \quad \tilde{p} = \frac{p}{\eta \omega \left(\frac{R}{C} \right)^2}, \quad \tilde{H} = \frac{H}{H_0}, \quad \tilde{H}_y = \frac{H_y}{H_0}$$

c is the bearing clearance, L is the bearing length,

$H_0 = \frac{I_e}{4\pi R}$ is the characteristic value of magnetic field intensity. The non-dimensional film thickness is then given by:

$$\tilde{h} = 1 + \varepsilon \cos \theta \quad (6)$$

Where $\varepsilon = e/c$ is the eccentricity ratio.

$$f(\theta, \tilde{z}) = \frac{\tilde{h}^3}{(1 - \lambda_1 \tilde{H})}, \quad g(\theta, \tilde{z}) = \frac{\tilde{h}^3}{(1 - \lambda_1 \frac{\tilde{H}_y}{\tilde{H}})}$$

$$F(\theta, \tilde{z}) = f(\theta, \tilde{z}) \tilde{H}, \quad G(\theta, \tilde{z}) = g(\theta, \tilde{z}) \tilde{H},$$

$$\gamma = \frac{R^2}{L^2}, \quad \lambda_1 = \frac{1}{2\eta} \rho \alpha^2 \bar{\mu} H_0, \quad \lambda_2 = \frac{C^2 \mu_0 \bar{\mu} H_0^2}{L^2 \eta \omega}$$

Boundary conditions on dimensionless pressure are:

$$\tilde{p}(\theta, \tilde{z} = 0.5) = 0, \quad \frac{\partial \tilde{p}}{\partial \tilde{z}}(\theta, \tilde{z} = 0) = 0, \quad \tilde{p}(\theta = 0, \tilde{z}) = 0,$$

$$\tilde{p}(\theta^*, \tilde{z}) = \frac{\partial \tilde{p}(\theta^*, \tilde{z})}{\partial \theta} = 0$$

θ^* is the film rupture or reformation angle; it is not a prescribed value and thus is iteratively determined during numerical calculations.

3. Numerical simulations

Equation (5) is discretized by finite difference method using the second-order centered scheme. The resulting set of algebraic equations is iteratively solved using the Gauss-Seidel method with relaxation:

$$\tilde{p}_{i,j}^{N_{ITER}+1} = (1 - \sigma) \tilde{p}_{i,j}^{N_{ITER}} + \sigma (a_{ij} \tilde{p}_{i+1,j}^{N_{ITER}} + b_{ij} \tilde{p}_{i-1,j}^{N_{ITER}} + c_{ij} \tilde{p}_{i,j+1}^{N_{ITER}} + d_{ij} \tilde{p}_{i,j-1}^{N_{ITER}} + e_{ij}) \quad (7)$$

where σ is relaxation coefficient and N_{ITER} is the number of iterations,

$$a_{ij} = \left(\frac{f_{i+1,j} - f_{i,j}}{(2\Delta\theta)^2} + \frac{f_{ij}}{\Delta\theta^2} \right) / a_0, \quad b_{ij} = \left(\frac{f_{i,j} - f_{i-1,j}}{(2\Delta\theta)^2} \right) / a_0, \quad c_{ij} = \left(\gamma \frac{g_{i,j} - g_{i,j+1}}{\Delta\tilde{z}^2} + \gamma \frac{g_{i,j+1} - g_{i,j+2}}{(2\Delta\tilde{z})^2} \right) / a_0$$

$$d_{ij} = \left(\gamma \frac{g_{i,j} - g_{i,j-1}}{\Delta\tilde{z}^2} - \gamma \frac{g_{i,j+1} - g_{i,j+2}}{(2\Delta\tilde{z})^2} \right) / a_0, \quad e_{ij} = \left(6\varepsilon \sin \theta_i - \gamma_2 \frac{F_{i+1,j} - F_{i,j}}{2\Delta\theta} \frac{(H_{i+1,j+1} - H_{i,j+1}) - (H_{i,j+1} - H_{i-1,j+1})}{2\Delta\theta} \right) / a_0$$

$$- \left(\gamma_1 F_y \frac{(H_{i+1,j+1} - 2(H_{i,j+1} + H_{i-1,j+1}))}{\Delta\theta^2} \right) / a_0, \quad a_0 = 2 \left(\frac{f_{ij}}{\Delta\theta^2} + \gamma \frac{g_{ij}}{\Delta\tilde{z}^2} \right)$$

where $\gamma_2 = \lambda_2 \left(\frac{\varphi}{1 + 2.5\varphi} \right)$, $\Delta\theta$ and $\Delta\tilde{z}$ are the mesh size in circumferential and axial directions, respectively.

Equation (7) is solved using a FORTRAN program. In this study, a grid of 100×50 points is used. Numerical value of the relaxation coefficient is $\sigma = 1.56$. The convergence is obtained when the criterion $\max \left| \frac{\tilde{p}_{i,j}^{N_{ITER}+1} - \tilde{p}_{i,j}^{N_{ITER}}}{\tilde{p}_{i,j}^{N_{ITER}}} \right| \leq 10^{-5}$ is satisfied. Once the convergence is achieved, the carrying capacity components, the friction force and the side leakage are numerally evaluated using Simpson's rule.

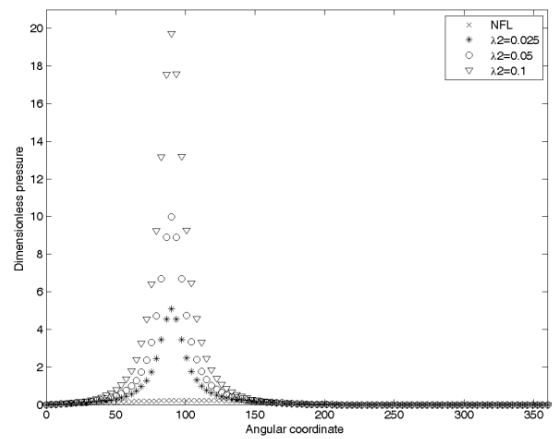


Figure 1: Effect of the magnetic force coefficient λ_2 on dimensionless pressure distribution \tilde{P} at $\lambda_1 = 0.01$ and eccentricity ratio $\varepsilon = 0.1$.

4. Results

The magnetic force coefficient λ_2 is derived from the term of the body force per unit volume in the momentum equation and represents the effect of the magnetic field on the momentum of the ferrofluid. Its effect is studied for a Jenkins viscosity parameter $\lambda_1 = 0.01$, which takes into account the angular momentum of the ferrofluid. It is derived from the body couple per unit volume.

Figure 1 shows the effect of the magnetic force coefficient λ_2 on the dimensionless film pressure distribution. It's observed that the magnetic force coefficient increases the dimensionless film pressure as compared to the non-magnetic lubricant (NFL). This is by the increase in ferrofluid viscosity in the presence of the magnetic field. Increasing the value of the magnetic force coefficient leads to a large increase of pressure

through a limited region around the angle $\theta = \frac{\pi}{2}$, which is the angle position of the wire

5. Conclusion

The present study investigates the ferrofluid finite journal bearings lubrication using the Jenking model which takes into account angular momentum of the ferrofluid. An applied magnetic field created by displaced finite wire is considered. A modified Reynolds equation is derived. Numerical solution is performed by finite difference method to obtain the bearing performance characteristics. It is found that the pressure and load capacity are higher and the friction factor is lower for the Jenkins model compared with those of the Neuringer-Rosensweig model, but these trends are reversed for high eccentricity ratios.

References

- [1] Maugin GA. *The method of virtual power in Continuum mechanics: application to coupled fields*. Acta Mech. (1980;35(1):1-70.
- [2] Jenkins JT, *A theory of magnetic fluids*. Archive for Rational Mechanics and Analysis. 1972;46:42-60.
- [3] Laghrabli S, El Khelifi M, Nabhani M, Bou-Saïd B. *Ferrofluid lubrication of finite journal bearings using Jenkin model*. Lubrication Science. 2017; 29:441–454.
- [4] Neuringer JL, Rosensweig RE. *Magnetic fluids*. *The Physics of Fluids*. 1964; 7(12):1927–1937.
- [5] Shliomis MI. *Effective viscosity of magnetic suspensions*. Soviet Physics—Journal of Experimental and Theoretical Physics. 1972; 34(6):1291–1294.
- [6] Jenkins JT. *A theory of magnetic fluids*. Archive for Rational Mechanics and Analysis. 1972; 46:42–60.

[7] Osman TA, Nada GS, Safar ZS. *Effect of using current-carrying-wire models in the design of hydrodynamic journal bearings lubricated with ferrofluid*. Tribology Letters 2001;11(1):61–70.

[8] Tarapov IE. *Movement of a magnetizable fluid in the lubricating layer of a cylindrical bearing*. Magnetohydrodynamics 1972; 8:444–448.

Instabilité de Faraday en cellule de Hele-Shaw avec deux fréquences commensurables

A. Rachik, M. Assoul et S. Aniss

University Hassan II of Casablanca, Faculty of Sciences Ain-Chock, Laboratory of Mechanics, Casablanca, Morocco
azeddinerachik@gmail.com , said.aniss@etu.univh2c.ma

Résumé :

Dans ce travail, nous présentons une étude de l'instabilité de Faraday de l'interface liquide-air en cellule de Hele-Shaw. Cette dernière est soumise à des oscillations verticales périodiques avec deux fréquences commensurables. Le problème de stabilité linéaire est réduit à une équation de Mathieu régissant l'évolution de l'amplitude de l'interface. En utilisant l'intégration numérique par la méthode de Runge-Kutta, pour la résolution numérique de cette équation, nous déterminons les courbes de stabilité marginale et par la suite l'amplitude réduite de forçage critique en examinant les effets des rapports des fréquences et des amplitudes d'excitations.

Mots clés : *Instabilité paramétrique; Equation de Mathieu; Cellule de Hele-Shaw.*

1 Introduction

Faraday (1831) [1] a été le premier à aborder expérimentalement les instabilités paramétriques. Il a noté qu'une instabilité à la surface libre se manifeste par l'apparition au centre de la couche liquide d'une onde stationnaire "standing wave" ayant une fréquence égale à la moitié de celle des vibrations. Par la suite, avec des expériences similaires à celle de Faraday (1831) [1], Matthiessen (1868, 1870) [2] a noté que l'onde peut aussi avoir la même fréquence que celle des vibrations. Plus tard, Rayleigh (1883) [3] a effectué une campagne expérimentale sur le problème de Faraday en proposant des explications qualitatives notamment que l'onde à la surface est de type paramétrique. Benjamin et Ursell (1959) [5] ont effectué une étude de stabilité linéaire et ils ont réduit l'équation relative à l'élévation de la surface libre, par rapport à l'équilibre, à un système d'équations de Mathieu. Une analyse de stabilité linéaire complète a été proposée par Kumar et Tuckerman (1994) [4]. Cette analyse traite le cas général de deux couches fluides visqueuses superposées de hauteur respectives h_1 et h_2 confinées entre deux parois rigides d'extension infinie dans les directions horizontales. Récemment, Li et al. (2018) [6] se sont intéressés aux ondes de Faraday dans une cellule de Hele-Shaw par des études expérimentales, numériques et théoriques. Inspirés de la théorie de Kelvin-Helmholtz-

Darcy, ils ont abouti aux ondes stationnaires stables à la moitié de la fréquence de la vibration forcée externe. En parallèle, le cas d'un système constitué de trois couches fluides avec deux interfaces a été étudié par Li et al. (2018) [7], dans ce travail, un système de deux ondes de Faraday couplées a été expérimentalement observé au niveau des deux interfaces des trois couches de fluides (air, éthanol pur et huile de silicone) dans une cellule Hele-Shaw couverte en présence d'une vibration verticale périodique. Edwards and Fauve (1994) [11] ont étudié expérimentalement l'instabilité de Faraday dans un système de petite profondeur avec un fluide visqueux et avec deux fréquences de forçage simultanées. Beyer and Friedrich (1995) [10] ont fait une étude de stabilité linéaire de l'instabilité de Faraday pour le cas de double fréquence d'excitation. Suite à ces travaux, nous nous intéressons dans ce travail à l'instabilité de Faraday en cellule de Hele-Shaw avec deux fréquences commensurables. Nous focalisons notre attention sur les résonances paramétriques et sur les effets des rapports des fréquences et des amplitudes d'excitations.

2 Formulation mathématique

Considérons une couche fluide de hauteur h et d'épaisseur e dans une cellule de Hele-Shaw de longueur L . La cellule de Hele-Shaw est soumise à des oscillations périodiques de la forme $Z(t) = \alpha_1 \cos(\Omega_1 t) + \alpha_2 \cos(\Omega_2 t)$.

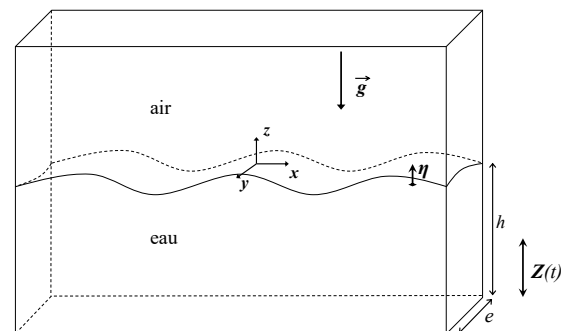


Figure 1: Cellule de Hele-Shaw remplie de liquide en oscillations périodiques dans la direction verticale.

2.1 Équations de conservation

Les équations de Navier-Stokes et l'équation de continuité s'écrivent :

$$\frac{\partial \mathbf{U}}{\partial t} + (\mathbf{U} \cdot \nabla) \mathbf{U} = -\frac{1}{\rho} \nabla P - \tilde{g} \mathbf{k} + \nu \nabla^2 \mathbf{U} \quad (1)$$

$$\nabla \cdot \mathbf{U} = 0 \quad (2)$$

où $\mathbf{U}(x, y, z, t)$ est le champ de vitesse, ρ est la masse volumique du fluide, P la pression, ν la viscosité de cinématique, \mathbf{k} le vecteur unitaire selon la direction verticale vers le haut. $\tilde{g} = g [1 - F_1 \cos(\Omega_1 t) - F_2 \cos(\Omega_2 t)]$ est l'accélération ressentie par le liquide dans le référentiel lié à la cuve, et $F_i = \alpha_i \Omega_i^2 / g$ sont les amplitudes de forçages adimensionnés où Ω_i sont les fréquences d'excitations et α_i les amplitudes d'excitations. En approximation de Hele-Shaw, l'équation de Navier-Stokes est intégrée entre les parois frontales (entre $-\frac{\epsilon}{2}$ et $\frac{\epsilon}{2}$) [9]. Le champ de vitesse moyenné est noté par: $\mathbf{u}(x, z, t) = (u, w)$. L'essentiel de la dissipation visqueuse se produit dans la direction normale aux parois (dans la direction y), et pour les longueurs d'onde étudiées, le temps d'atténuation est beaucoup plus faible que le temps de diffusion visqueuse, $\nu \nabla^2 \mathbf{u} \ll \gamma \mathbf{u}$. On obtient :

$$\frac{\partial \mathbf{u}}{\partial t} + \frac{6}{5} (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho} \nabla p + \tilde{g} \mathbf{k} - \gamma \mathbf{u} \quad (3)$$

$$\nabla \cdot \mathbf{u} = 0 \quad (4)$$

Avec $\nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial z^2}$, $\gamma = \frac{12\nu}{\epsilon^2}$ est la viscosité externe de Rayleigh.

2.2 Conditions aux limites

Nous désignons par $\eta(x, t)$, la déformation de l'interface par rapport à sa position d'équilibre, $z = 0$. Au fond de la cellule, en $z = -h$, nous imposons la condition d'imperméabilité et au niveau de l'interface libre, en $z = \eta$, nous imposons la condition cinématique. Ces deux conditions s'écrivent :

$$w(x, z, t) = 0, \quad z = -h \quad (5)$$

$$\frac{\partial \eta}{\partial t} + \frac{\partial \eta}{\partial x} u - v = 0 \quad (6)$$

En négligeant les termes visqueux, la continuité des contraintes normales à l'interface s'écrit :

$$p - p_\infty = \sigma \nabla \cdot \mathbf{n} \quad (7)$$

où $p_\infty = p_{atm}$ est la pression du deuxième fluide, l'air, σ est la tension superficielle et \mathbf{n} représente le vecteur normal pour l'interface.

2.3 Formulation du problème linéaire

L'écoulement est considéré permanent et irrotationnel $\mathbf{u} = \nabla \phi$. L'équation (3) est intégrée et réécrite sous la forme d'une équation de Bernoulli dynamique. Nous cherchons des solutions de petites amplitudes proches du repos, $\phi = 0$ et $\eta = 0$, pour lesquelles les termes non-linéaires sont négligeables. Dès lors, les conditions aux limites sont satisfaites non pas en $z = \eta$ mais en $z = 0$. On obtient, ainsi, le système linéarisé suivant :

$$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial z^2} = 0, \quad -h < z < \eta \quad (8)$$

$$\frac{\partial \phi}{\partial z} = 0, \quad z = -h \quad (9)$$

$$\frac{\partial \eta}{\partial t} - \frac{\partial \phi}{\partial z} = 0, \quad z = 0 \quad (10)$$

$$\frac{\partial \phi}{\partial t} - \frac{\sigma}{\rho} \frac{\partial^2 \eta}{\partial x^2} + \tilde{g} \eta + \gamma \phi = 0, \quad z = 0 \quad (11)$$

2.4 Analyse en mode normaux

Les solutions recherchées sont périodiques par rapport à la direction x , en conséquence nous posons :

$$\phi(x, z, t) = K(t) \tilde{\varphi}(z) \sin(kx) \quad (12)$$

$$\eta(x, t) = \zeta(t) \cos(kx) \quad (13)$$

En utilisant les équations (8), (9) et (10), nous obtenons une nouvelle expression de $\phi(x, z, t)$:

$$\phi(x, z, t) = \frac{\partial \zeta}{\partial t} \frac{\cosh(k(h+z))}{k \sinh(kh)} \sin(kx) \quad (14)$$

Les équations (13), (14) et (11) permettent d'aboutir à l'équation de Mathieu amortie :

$$\ddot{\zeta} + \gamma \dot{\zeta} + gk \tanh(kh) \left(1 + \frac{\sigma}{\rho g} k^2 - F_1 \cos(\Omega_1 t) - F_2 \cos(\Omega_2 t)\right) \zeta = 0, \quad (15)$$

nous faisons le changement de variable $\tau = \Omega_1 t$, l'équation (15) se réduit à l'équation de Mathieu quasi-périodique amortie suivante:

$$\ddot{\zeta} + 2\mu \dot{\zeta} + [\delta + \epsilon (\cos(\tau) + r \omega^2 \cos(\omega\tau))] \zeta, \quad (16)$$

avec $\delta = gk \tanh(kh) [1 + \sigma k^2 / (\rho g)] / \Omega_1^2$, $\omega = \Omega_2 / \Omega_1$, $r = \alpha_2 / \alpha_1$, $\epsilon = -gk \tanh(kh) F_1 / \Omega_1^2$ et $\mu = \gamma / (2\Omega_1)$.

3 Résultats et discussions

En adaptant l'intégration numérique par la méthode de Runge-Kutta du quatrième ordre de la même manière que

le travail de Zounes et Rand [8] à l'équation (16), nous avons déterminé les zones de stabilité marginale. La figure (2. a) montre les courbes de stabilité marginale.

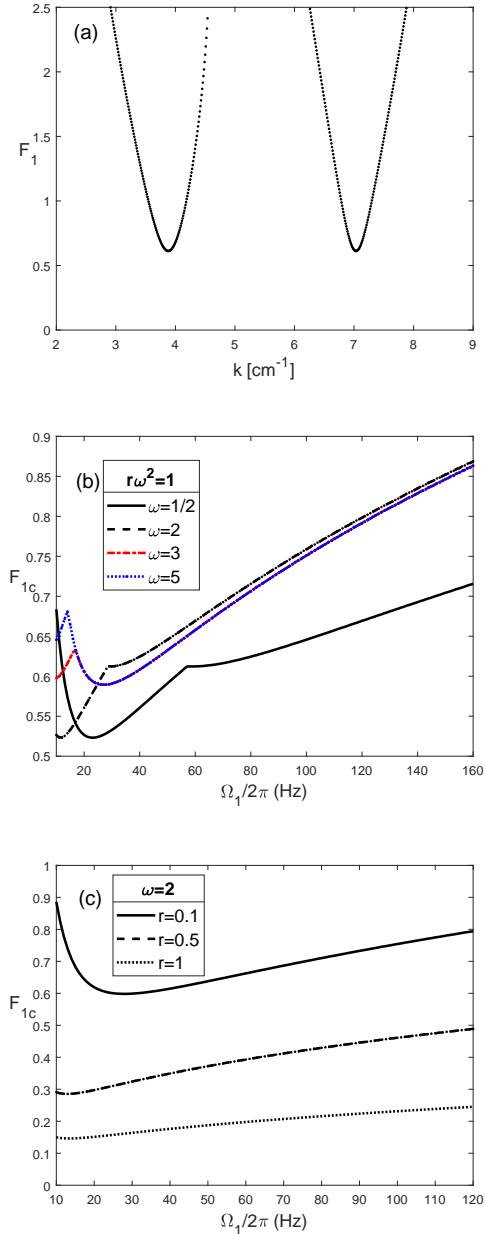


Figure 2: (a): Courbes de stabilité marginale (F_1, k) avec $\Omega_1 = 179 Hz$ et $\omega = 2$; (b) et (c): forçage critique en fonction de la fréquence d'excitation (F_{c1}, Ω_1); $h = 1 cm$ et $e = 1 mm$.

La figure (2. b) montre l'évolution de l'amplitude de forçage critique, F_{1c} , en fonction de la fréquence dimensionnelle, Ω_1 , pour différentes valeurs du rapport des fréquences ω . La modulation avec deux fréquences commensurables entraîne deux effets, stabilisant et déstabilisant. Pour les basses fréquences, pour Ω_1 , l'amplitude du forçage critique décroît à mesure que ω augmente.

En revanche, pour les hautes fréquences, l'amplitude critique, F_{1c} croît avec l'augmentation ω . Notons ici, pour les hautes fréquences, que lorsque ω est supérieure à 2, le rapport des fréquences n'a aucun effet sur le seuil de l'instabilité. Sur la figure (2. c), nous illustrons, pour différents rapports d'amplitudes, l'évolution du forçage critique en fonction la fréquence d'excitation Ω_1 . Nous notons que le seuil de forçage diminue quand le rapport d'amplitudes augmente.

4 Conclusion

Dans ce travail nous avons étudié l'instabilité de Faraday d'un liquide confiné dans une cellule de Hele-Shaw laquelle effectue un mouvement d'oscillation vertical avec deux fréquences. Notre étude a présenté une réduction des équations de conservation à l'équation de Mathieu. La résolution de cette équation a été effectuée par l'intégration numérique par la méthode de Runge-Kutta. Les résultats ont porté sur les courbes de stabilité marginale, puis sur l'amplitude de forçage critique réduite pour différents ratios de fréquences et d'amplitudes. Nous avons montré que le rapport des fréquences a un effet déstabilisant sur les basses fréquences et un effet stabilisateur sur les hautes fréquences. Mais le rapport des amplitudes entraîne toujours une déstabilisation.

References

- [1] M. Faraday. Phil. Trans. R. Soc. Lond. 1831 121, 299-340, published 1 January 1831
- [2] L. Matthiessen. Ann. Phys. (Leipzig) 134, 1868, 107, Matthiessen L., Ann. Phys. (Leipzig) 141, 1870, 375
- [3] L. Rayleigh. Philos. Mag. (5) 15, 1883, 235, Rayleigh L., Philos. Mag. (5) 16, 1883, 50
- [4] K. Kumar and L. S. Tuckerman. Parametric instability of the interface between two fluids. *J. Fluid Mech.* 279:49-68, 1994.
- [5] T. B. Benjamin and Ursell. The Stability of the Plane Free Surface of a Liquid in Vertical Periodic Motion. Published 22 September 1954. the Royal Society, 1958.
- [6] J. Li, X. Li, K. Chen, B. Xie and S. Lia. Faraday Waves in a Hele-Shaw cell, *Physics of Fluids*, Volume 30, Issue 4, N 042106 , 2018.
- [7] X. Chen Li, X. Ming Li and S. Liao. Observation of two coupled Faraday waves in a vertically vibrating Hele-Shaw cell with one of them oscillating horizontally, Volume 30, Issue 1, number 012108, 2018.
- [8] R. S. Zounes and R. Rand. Transition Curves for the Quasiperiodic Mathieu Equation, *SIAM J. Appl. Math.*, 58 (1998), 1094-1115
- [9] P. Gondret and M. Rabaud. Shear instability of two-fluid parallel flow in a Hele-Shaw cell (1997) *Physics of Fluids*, 9 (11), pp. 3267-3274.
- [10] J. Beyer and R. Friedrich. Faraday instability: Linear analysis for viscous fluids. *Phys. Rev. E*51, 1162 (1995).
- [11] W. S. Edwards and S. Fauve. Patterns and quasi-patterns in the Faraday experiment. Cambridge University Press: 1994

Instabilité d'une couche fluide Newtonienne horizontale en présence d'une modulation périodique de température imposée à la frontière supérieure

Mohamed Sakine¹, Mehdi Riahi^{2,3}, Mohamed Hayani choujaa³, Said Aniss³ et Abdelmajid Daya²

1. Laboratoire de Mécanique, Université Moulay Ismail, Faculté des Sciences et Techniques, Errachidia, Maroc.
2. Département de Mécanique, Ecole Royal de l'Air, Marrakech, Maroc
3. Laboratoire De Mécanique, Université Hassan Faculté des Sciences Ain Chock, Casablanca, Maroc

Abstract

We are interested in the linear stability analysis of a Newtonian fluid layer of infinite extension in the horizontal directions and confined between two parallel walls. We consider a periodic temperature modulation on the upper boundary of the fluid layer in the absence of a stationary component of the temperature gradient. Floquet theory and the Chebyshev spectral collocation method have been used to solve the linear stability problem in the case of rigid-rigid boundary conditions. Although the stationary version of this flow configuration is stable in the absence of modulation, it is shown that eventual destabilization is possible in the presence of a modulation imposed on the upper boundary of the fluid layer.

Keywords : *stability, convection, temperature modulation*

Résumé

Nous nous intéressons à l'analyse de stabilité linéaire d'une couche fluide Newtonienne, d'extension infinie dans les directions horizontales et confinée entre deux parois parallèles. Nous considérons une modulation de température périodique sur la frontière supérieure de la couche fluide en absence de composante stationnaire du gradient de température. La théorie de Floquet et la méthode de collocation spectrale de Chebychev ont été utilisées pour résoudre le problème de stabilité linéaire dans le cas de conditions aux limites rigide-rigide. Bien que la version stationnaire de cette configuration d'écoulement soit stable en absence de modulation, il est montré qu'une éventuelle déstabilisation est possible en présence d'une modulation imposée sur la frontière supérieure de la couche fluide.

Mots clés : *stabilité, convection, modulation de température.*

1. Introduction

Les écoulements induits par une différence de température au sein d'un système sont rencontrés dans bon nombre d'applications industrielles et requièrent une

compréhension physique fine des phénomènes en jeu pour optimiser de tels environnements thermiques. De plus, ces écoulements se produisent souvent en milieu confiné, en régime fortement instationnaire et proche ou au-delà de la région de transition laminaire-turbulent. La maîtrise de ces transferts thermiques complexes nécessite l'utilisation d'outils expérimentaux mais également numériques adaptés à l'étude et/ou la prévision de tels phénomènes.

Dans ce cadre, une étude de stabilité linéaire est menée dont le but est de déterminer le critère d'instabilité d'une couche fluide Newtonienne d'extension infinie dans les directions horizontales et confinée entre deux parois parallèles dont la frontière supérieure est soumise à une modulation périodique de la température. Le système d'équation aux dérivées partielles régissant la dynamique du problème est résolu en utilisant la théorie de Floquet et les méthodes spectrales et les résultats obtenus permettent d'aboutir aux différents diagrammes de stabilité séparant les zones d'écoulements stables des zones instables. Il est à noter que cette configuration d'écoulement a fait l'objet de plusieurs travaux de recherche à des conditions aux limites différentes [1,2,3]. Le cas d'une modulation en phase a été traité récemment par Souhar et al [3] dans le cadre de la théorie de Floquet.

2. Formulation mathématique

On considère une couche fluide Newtonienne de viscosité cinématique ν et de diffusivité thermique k confinée entre deux parois horizontales d'extension infinie dans les directions x^* et y^* . La température $T_m + T_s \cos(\omega^* t^*)$ est imposée sur la paroi supérieure située en $z^* = d$ (figure 1), où d représente la distance entre les deux parois. Nous considérons de plus que $T_s \ll T_m$ afin de garder constantes les caractéristiques physiques du fluide à la température ambiante T_m .

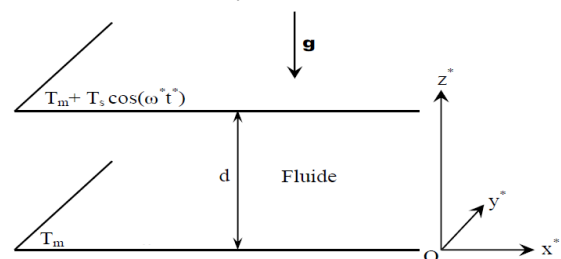


FIGURE1 : Schéma du système étudié.

L'équilibre de la couche fluide correspond à un champ de vitesse nul et un champ de température $T_e = \frac{T^* - T_m}{T_s}$ vérifiant l'équation adimensionnelle suivante :

$$\frac{\partial T_e}{\partial t} = \frac{\partial^2 T_e}{\partial z^2} \quad (1)$$

avec les conditions aux limites

$$T_e(z=0, t) = 0, T_e(z=1, t) = \cos(\omega t) \quad (2)$$

où $\omega = \omega^* d^2/k$, $z = z^*/d$ et $t = t^*k/d^2$.

Nous cherchons la solution de température de base sous la forme suivante :

$$T(z, t) = T_1(z) \cos(\omega t) + T_2(z) \cos(\omega t) \quad (3)$$

avec

$$T_1(z) = \frac{\cosh(\sigma z) \cos(\sigma(1-z)) \cosh(\sigma) - \cosh(\sigma(1-z)) \cos(\sigma z) \cosh(\sigma)}{\cosh^2(\sigma) - \cos^2(\sigma)}$$

$$T_2(z) = \frac{\sinh(\sigma z) \sin(\sigma(1-z)) \cosh(\sigma) - \sinh(\sigma(1-z)) \sin(\sigma z) \cosh(\sigma)}{\cosh^2(\sigma) - \cos^2(\sigma)}$$

Nous perturbons cet état d'équilibre en superposant à celui-ci des perturbations infinitésimales

$$p^* = p + p_e, T^* = T + T_e, V^* = V(x, y, z), \rho^* = \rho + \rho_e$$

Le système d'équations linéaires correspondant à ces perturbations est donnée par l'équation de continuité, les équations de mouvement de Navier-Stokes et l'équation de la chaleur en approximation de Bousinesq tels que :

$$\nabla \cdot \mathbf{V} = 0 \quad (4)$$

$$Pr^{-1} \frac{\partial \mathbf{V}}{\partial t} = -\nabla p + \Delta \mathbf{V} + Ra T e_z \quad (5)$$

$$\frac{\partial T}{\partial t} + w \frac{\partial T_e}{\partial z} = \Delta T \quad (6)$$

où $\Delta = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} + \frac{\partial^2}{\partial z^2}$, Pr est le nombre de Prandtl défini par $Pr = \frac{\nu}{\kappa}$. La grandeur Ra est le nombre de

Rayleigh défini par $Ra = \frac{\beta g T_s d^3}{\nu \kappa}$ où β désigne le coefficient de dilatation thermique et la viscosité dynamique du fluide. Ce système adimensionnel a été obtenu en considérant d , k/d^2 et d/k comme échelles de longueur, de temps et de vitesse respectivement.

La projection vertical du rotationnel du rotationnel de l'équation de mouvement (5), en utilisant l'équation de continuité, permet de réduire le système d'équations (4)-(6) au système suivant :

$$\left(Pr^{-1} \frac{\partial}{\partial t} - M \right) M w = q^2 Ra \Delta_2 T \quad (7)$$

$$\left(\frac{\partial}{\partial t} - M \right) T + w \frac{\partial T_e}{\partial z} = 0 \quad (8)$$

avec $\Delta_2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2}$. A ce système, on associe les conditions aux limites de type rigide-rigide :

$$T = w = \frac{\partial w}{\partial z} = 0 \text{ en } z = 0, 1.$$

Nous cherchons la solution du système, (7)-(8), associé aux conditions aux limites en mode normaux sous la forme :

$$(w(x, y, z, t), T(x, y, z, t)) = (\bar{w}(z, t) e^{i(q_x x + q_y y)}, \tilde{T}(z, t) e^{i(q_x x + q_y y)})$$

où les quantités q_x et q_y représentent respectivement les nombres d'onde dans les deux directions horizontales. En introduisant les expressions ci-dessus dans le système d'équations (7)-(8) nous obtenons :

$$\left(Pr^{-1} \frac{\partial}{\partial t} - M \right) M \bar{w} = q^2 Ra \Delta_2 \tilde{T} \quad (7)$$

$$\left(\frac{\partial}{\partial t} - M \right) \tilde{T} + \bar{w} \frac{\partial T_e}{\partial z} = 0 \quad (8)$$

M est défini par $M = \frac{\partial^2}{\partial z^2} - q^2$ avec $q^2 = q_x^2 + q_y^2$.

3. Résolution numérique

Nous adoptons la théorie des méthodes de collocation spectrale pour la résolution spatiale du problème de stabilité linéaire. Ces méthodes se révèlent d'un grand intérêt en raison de leur très grande précision [4]. Le système de stabilité s'écrit sous la forme d'un système de Floquet ayant la forme suivante :

$$\mathbf{B} \frac{\partial}{\partial t^*} \mathbf{X} = \mathbf{M}(t) \mathbf{X} \quad (9)$$

avec $\mathbf{M}(t)$ sont des matrices d'ordre $8N$, \mathbf{B} est une matrice singulière. Le système (9) est intégré en utilisant la méthode de Runge-Kutta du quatrième ordre. Enfin, on a une relation entre la fréquence γ , le nombre de Taylor T_a et le nombre d'onde q et qui s'écrit formellement sous la forme,

$$\mathcal{R}(Ra, k, \gamma) = 0 \quad (10)$$

4. Résultats et discussions

La résolution numérique par la combinaison des méthodes spectrales de collocation de Tchebychev et la théorie de Floquet permet de dégager les cartes de stabilité relative à ce système afin de déterminer les zones d'instabilités. Pour ce faire, on commence par fixer une valeur de la fréquence et tracer la courbe de stabilité marginale présentant l'évolution du nombre de Rayleigh en fonction du nombre d'onde. L'instabilité est caractérisée alors par le nombre de Rayleigh critique et le nombre d'onde critique qui correspondent aux coordonnées du point minimum de cette courbe.

On présente sur la figure 2 le diagramme de stabilité qui représente l'évolution du nombre de Rayleigh critique en fonction de la fréquence de modulation. Bien qu'une couche fluide chauffée par une température constante par le haut est linéairement stable, il s'avère d'après cette figure qu'elle devient instable en présence de la modulation de température. Lorsque la fréquence de modulation est suffisamment faible le nombre de Rayleigh critique relatif au seuil de convection est de l'ordre de 1707. Ce dernier correspond aussi au seuil de convection critique lorsque la couche fluide est chauffée par le bas.

Une stabilisation de l'écoulement, traduite par une augmentation du nombre de Rayleigh critique, est obtenue au fur et à mesure que la fréquence ω augmente. Il est à signaler en outre que la stabilité du système est caractérisée par la présence de brisures dans le diagramme de stabilité. Ce phénomène est dû à l'existence d'une compétition entre les modes d'instabilité observés au niveau des courbes de stabilité marginale. Ceci est accompagné par l'apparition des points de bifurcation de co-dimension deux ainsi que des états bi-critiques entre deux modes d'instabilité.

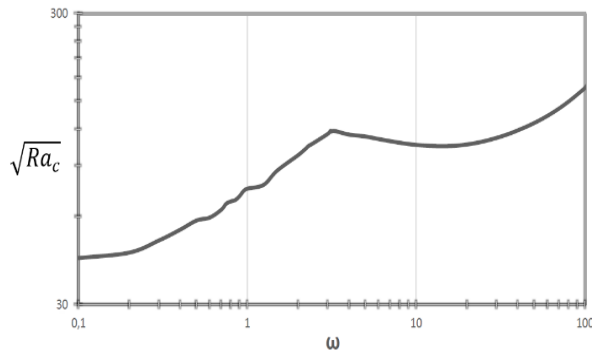


Figure 2 : Evolution de la racine carrée du nombre de Rayleigh critique en fonction de la fréquence adimensionnelle ω dans le cas Rigide-Rigide.

Dans la figure 3 nous illustrons un exemple de courbes de stabilité marginale pour une gamme de fréquence intermédiaire. D'après cette figure, on constate que les courbes de stabilité ne sont plus uni-modales, identiquement au cas des hautes fréquences, mais elles sont composées par deux paraboles ayant deux minimums et un état bi-critique est alors envisagé. Ce dernier correspond à un état caractérisé par deux nombres d'ondes critiques ayant le même nombre de Rayleigh critique. Le comportement de ces états est très intéressant dans l'optique où il altère fortement la réponse du système étudié (harmonique vs sous-harmonique) ainsi que le motif de convection qui en résulte.

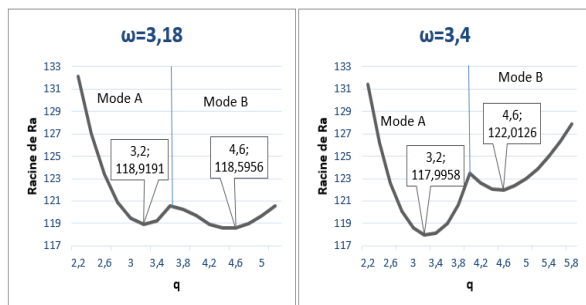


Figure 3: Exemples de courbes de stabilité marginale pour deux valeurs de fréquences de modulation au voisinage d'un état bi-critique.

L'évolution du nombre d'onde critique en fonction de la fréquence de modulation est présentée sur la figure 4. On

constate que dans les basses fréquences, le nombre d'onde critique tends vers $k_c = 3.1$ et augmente lorsque la fréquence de modulation augmente. De plus, cette évolution est caractérisée par la présence des discontinuités dans quelques valeurs de fréquences. Par exemple, en $\omega = 2.3$ le nombre d'onde critique saute de 3.9 jusqu'à 4.4 tandis qu'il chute brutalement de 4.6 à 3.3 lorsque $\omega = 3.17$.

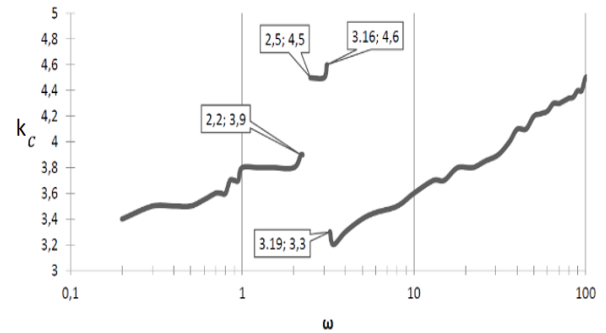


Figure 4 : Evolution numérique du nombre d'onde critique k_c en fonction de la fréquence adimensionnelle ω dans le cas Rigide-Rigide.

5. conclusion

Dans ce travail, nous avons étudié l'effet d'une modulation de température sur le seuil de convection d'une couche fluide horizontale d'extension infinie. Cette modulation est supposée périodiquement variable dans le temps avec une composante stationnaire nulle. L'objectif était de déterminer le critère de déclenchement du mouvement de convection en termes du nombre de Rayleigh critique et du nombre d'onde critique en fonction de la fréquence adimensionnelle de la modulation. Nous avons utilisé la théorie de Floquet et la méthode spectrale pour résoudre numériquement le problème de stabilité linéaire. Nous avons montré que la modulation permet de déstabiliser une configuration d'écoulement linéairement stable. Cette déstabilisation est maximale dans le domaine de basses fréquences de modulation et minimale

Références

- [1] G. Venezian, Effect of modulation on the onset of thermal convection, J. Fluid Mech .
- [2] S. Rosenblat and Herbert D. M., Low frequency modulation of thermal instability, J. Fluid Mech.pp. 385-398.
- [3] M. Souhar et S. Aniss, Effect of Phase Thermal Modulation Without Stationary Temperature Gradient on the Threshold of Convection, Journal of Heat Transfer · 2016.
- [4] J . A. C. Weideman. S. C. Reddy. A MATLAB differentiation matrix suite. ACM transaction on mathematical software, 26,(2000)

Thème 08 :

Transferts thermiques, transferts de masse, énergies renouvelables et environnement

Numerical Study of Latent Heat Discharge of a Phase Change Material Shell-and-Tube Thermal Energy Storage System

K. OUDAOUI^{1,*} & M. FARAJI¹

¹ Physics Department, ERDS Laboratory, Faculty of Sciences Ain chock, Hassan II University, Casablanca, Morocco,

(*) kenzaoudaoui@gmail.com

Abstract:

In this paper we study numerically the latent thermal energy discharging of a coaxial PCM water heat exchanger. PCMs technology is used to improve the heat capacity and heat transfer during melting and solidification processes. A numerical study of the thermal performance of multiple PCMs in the latent heat storage unit, based on the enthalpy formulation has been presented. A FORTRAN code, using the finite volume method has been developed for the solution of the corresponding mathematical model.

Keywords: *Multiple Phase Change Material, Solar heat exchanger, Solidification.*

1. Introduction

Thermal energy storage is an important and very practical way to conserve available energy and improve its use. Therefore, the most relevant solution is the use of thermal energy storage system; this solution can solve the mismatch between energy supply and demand [1]. Considering the consequences, phase change materials, used in latent heat storage units, are important elements for solar energy [2]. The use of PCMs as storage materials, especially in solid and liquid phase, has the disadvantage of low thermal conductivity [3], this latter limits the exchange between the heat transfer fluid and the storage medium. Therefore, the design of an efficient and cost-effective storage material requires the development of technologies to improve the thermal performance of heat storage units [4]. One of the most effective ways is to improve the performance of heat exchangers by applying multiple PCMs. The advantages of this latter lies to the simplicity and reliability of the PCM. However, the phase change process must be carefully analyzed to avoid any unmanageable situation in the actual operation of these devices [5]. Many studies, both experimental and numerical, have been conducted over the years to improve the entertainment of latent heat thermal energy storage systems (LHTES) and to enable their analysis [6]. The objective of this paper is to develop a mathematical model, in order to conduct a study to examine the effects of some parameters, on PCMs, such as heat transfer fluid, inlet and outlet temperature, mass flow rate. Also, the main study is to improve the discharge time of the PCMs, on the thermal performance of the exchanger.

2. Physical and mathematical model

The studied system (Figure 1) is a heat exchanger containing tubes, and an annular space filled with three PCMs with different melting temperatures. Along the entire length of the flow, water exchanges heat with the

PCMs, when the water is cold, it enters and the heat, stored in the PCMs, is transported by water, which exits through the end and extracts the thermal energy stored in the phase change materials. The heat transfer fluid enters at a constant temperature $T_i < T_{melt}$ and causes the solidification of the PCMs in the shell-and-tube system. In order to simplify the mathematical model, the following assumptions were made:

- Axial conduction and viscous dissipation in the fluid are negligible;
- The effect of natural convection during melting is taken into account by using an efficient thermal conductivity of the PCM liquid phase;
- The thermos-physical properties of the heat transfer fluid and the PCM will be independent of temperature;
- The thermal resistance of the inner tube is negligible.

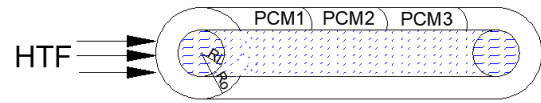


Figure 1: Schematic of the storage unit using 3 PCMs

The appropriate energy balance is of the heat transfer fluid and phase change materials which are expressed as follow:

For the HTF:

$$\frac{\partial \theta_f}{\partial t} = -\frac{m_f}{\rho_f \pi R_i^2} \frac{\partial \theta_f}{\partial x} - \frac{2U}{(\rho C_p)_f R_i} (\theta_f - \theta_i) \quad (1.1)$$

Where: $\theta_i = T(x, r = R_i, t) - T_{m3}$, $\theta_f = T_f - T_{m3}$

For the Phase change materials, the equation is defined by:

$$(\rho C_p)_p \frac{\partial \theta}{\partial t} = \frac{\partial}{\partial x} \left(k_p \frac{\partial \theta}{\partial x} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left(k_p \frac{\partial \theta}{\partial r} \right) - \rho_p \Delta H \frac{\partial f}{\partial t} \quad (1.2)$$

The method used to solve equation (1.2) is the enthalpy method [7]. The liquid fraction is estimated by:

$$f = 0, \theta < \theta_m \quad (1.3)$$

$$0 < f < 1, \theta = \theta_m, f = 1, \theta > \theta_m$$

The initial and boundary conditions are:

$$\theta_f(x, t = 0) = T_i - T_{m3}, \quad (1.4)$$

$$\theta_f(x, r, t = 0) = T_i - T_m \quad (1.5)$$

$$\frac{\partial \theta(x=0, r, t)}{\partial x} = \frac{\partial \theta(x=L, r, t)}{\partial x} = 0 \quad (1.6)$$

$$k_p \frac{\partial \theta(x=L_1, r, t)}{\partial x} |_{PCM_{1,i}} = k_p \frac{\partial \theta(x=L_2, r, t)}{\partial x} |_{PCM_{2,i}}$$

$$= k_p \frac{\partial \theta(x=L_3, r, t)}{\partial x} |_{PCM_{3,i}} \quad (1.7)$$

$$-k_p \frac{\partial \theta(x, r=R_i, t)}{\partial r} = U(\theta_f(x, t) - \theta(x, r=R_i, t) + T_{m_3} - T_m) \quad (1.8)$$

$$\theta(x=L_1, r, t)_{PCM_{1,i}} = \theta(x=L_2, r, t)_{PCM_{2,i}}$$

$$= \theta(x=L_3, r, t)_{PCM_{3,i}} \quad (1.9)$$

The thermal conductivity of the PCM liquid is defined by the following equation, at the interfaces:

$$k_m = fk_l + (1-f)k_s \quad (1.10)$$

Propriety	Paraffin 53	Rubitherm Paraffin	C ₂₀ H ₄₂
ρ [kg/m ³]	830	930	800
k_{liquid} [W/m ² K]	0,19	0,21	0,16
k_{solid} [W/m ² K]	0,28	0,2	0,212
C_p [J/kg.°C]	2385	812	2050
Q_{Lat} [kJ/Kg]	164	214	237,4
T_{fusion} [°C]	53	60	36,49

Table 1 : Thermo-Physical properties of PCM [8]

3. RESULT AND DISCUSSION:

Our objective is the control and stabilization of the temperature of the heat transfer fluid passing through the heat exchanger which absorbs the heat transfer fluid using solar energy, thus, store heat in the phase change materials, these are integrated into the exchanger containing 3 PCMs, with different thermo-physical properties [8], as shown previously. The results of the numerical simulation are illustrated and interpreted in the following section:

3.1 Temporal variation of the heat transfer fluid and temperature:

The heat transfer fluid enters to the exchanger at a temperature of 25°C, it transfers the heat to the three PCMs, when the temperature increases to 60°C, the PCM takes the heat and stores it as sensible heat in the water which is evacuated towards the exchanger outlet. The temperature of the fluid, $T_{f, out}$ remains constant and its equal to the initial temperature, for 40 min (see figure 2). After 46 min, the two first PCMs solidify and the PCM3 starts to transfer its latent heat, the heat exchange, at this time, between the heat transfer fluid and PCM3, occurs at an almost constant rate, until the heat is released after 80 min. After 46 minutes, the two first PCMs are in solidification phase and the 3rd PCM begins to transfer its latent heat. At this point, the heat exchange between the heat transfer fluid and the 3rd PCM, is about to be constant until the heat is released for almost 80 minutes. It can be clearly stated, that the necessary time for complete discharge decreases significantly with the

decrease of the inlet temperature of the heat transfer fluid and the liquid fraction increases.

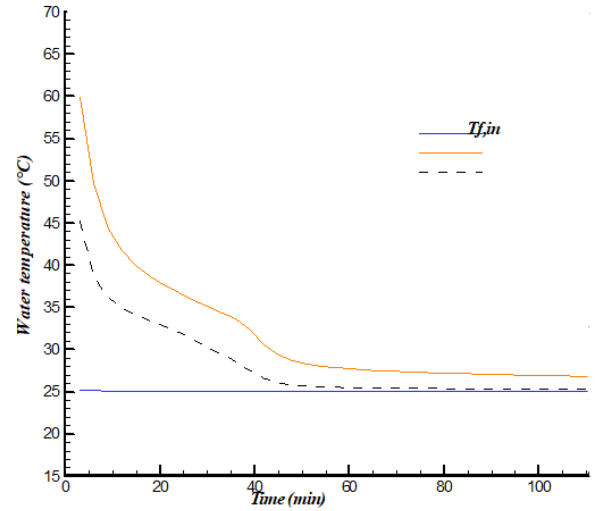


Figure 2: Temporal variation of the heat transfer fluid and temperature

3.2 Temporal variation of heat transfer rate:

The figure 3 indicates that there is a maximum heat transfer rate of PCM interface during the early stage of thermal storage ($t < 45$ min). The maximum heat transfer rate of the PCM interfaces increases as the heat transfer charge of the HTF increases from 15 kJ to 26 kJ, the reason can be attributed to the increase of the heat transfer surface area as the mass of PCM increasing, and hence the heat transfer fluid became constant after 50 min, also the heat transfer performance can be attributed to the weaker convection. Furthermore, the increase and further stabilization in the heat transfer from the tube is caused by the increase in the amount of cool liquid in the lower part of the shell and tube exchanger, which leads to enhance the heat transfer from the heat exchanger by natural convection. Once the solid PCM has melted completely the heat transfer rate of solid-liquid PCM interface closes to zero (see Figure 4). Therefore, the maximum heat transfer rate of solid-liquid interface shifts down.

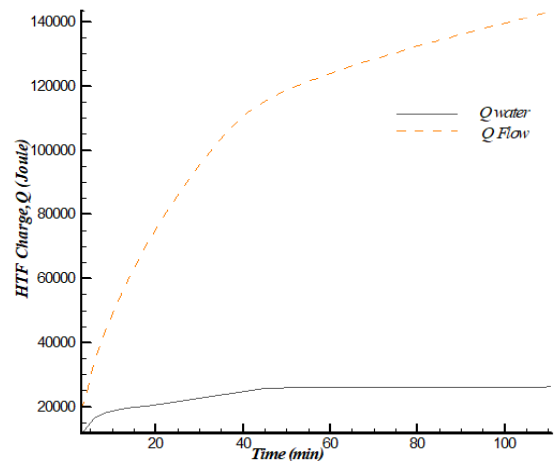


Figure 3: Temporal variation of heat transfer rate

3.3 Temporal variation of the liquid fraction for PCMs:

Figure 4 shows the variation of the liquid fraction for PCMs during the melting process. The analysis of the result allows to explain the melting process in the PCMs placed in the heat exchanger, we notice that the 1st PCM solidifies after 29 min, then the second one solidifies after 40 min, contrary to the 3rd PCM which remained in the crystallization phase for 38 min, at the time when the second PCM solidifies completely, the third PCM takes about 120 min to solidify. This behavior is due to the fact that sensible heat is needed to start the phase change process for a higher melting point of the PCM. The optimal value of the liquid fraction is slightly higher for PCM with a higher melting point. Indeed, the value of the liquid fraction, at the beginning of the discharge process is about 0.05 for the 3rd PCM. The discharge of the latter starts after 120 min. The storage of sensible heat in the first two PCMs slows down the heat extraction of the last PCM.

Conclusion:

The performance of multiple PCMs in the thermal energy storage system was studied and numerically simulated in this paper. The objective was to improve and enhance the discharge process of the PCMs associated with the heat exchanger. The results indicated that the storage of sensible heat in the first two PCMs slows down the heat extraction of the last PCM. In future work, improvements in the discharge time of each PCM will be sought. In addition, the design will be further improved.

Acknowledgements:

I would like to thanks the National Scientific Research and Technology Center (CNRST) for providing me a national Doctoral Research Scholarship.

Nomenclature:

c	Specific heat (J/Kg. °C)
f	Liquid fraction
k	Thermal conductivity (W/m.°C)
L	Length of the tube (m)

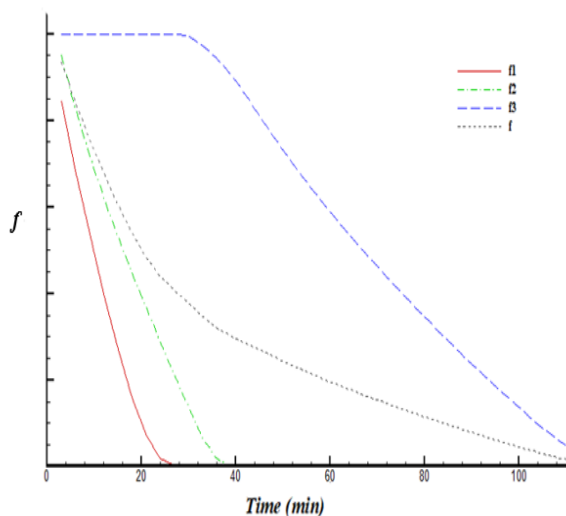


Figure 4 : Temporal variation of the liquid fraction for PCMs

m_f	Mass of the HTF (kg)
Q	Heat (J)
T	Temperature (°C)
t	Time (min)
U	Local convection heat transfer coefficient (W/m ² .°C)

Greek Symbols:

ΔH	Latent heat (J/Kg)
θ	T-T _m (°C)
ρ	Density (kg/m ³)

Subscripts:

1,2,3	PCM1, PCM2, PCM3
f	Heat transfer fluid (HTF)
i	Interface of PCMs
l, s	Liquid phase and solid phase

References

- [1] Ziqi Wang, Zhongqing Yang, Ruiming Fang, Yunfei Yan, Jingyu Ran, Li Zhang, A State-of-the-art review on action mechanism of photothermal catalytic reduction of CO₂ in full solar spectrum, Chemical Engineering Journal, Volume 429, 2022, 132322, ISSN 1385-8947, <https://doi.org/10.1016/j.cej.2021.132322>.
- [2] Farid, M.M.; Khudhair, A.M.; Razack, S.A.K.; Al-Hallaj, S. A review on phase change energy storage: Materials and applications. Energy Convers. Manag. 2004, 5, 1597–1615
- [3] F. Bruno, M. Belusko, M. Liu, and N. H. S. Tay, Using solid-liquid phase change materials (PCMs) in thermal energy storage systems. Woodhead Publishing Limited, 2015.
- [4] Sarbu, Ioan, and Calin Sebarchievici. 2018. "A Comprehensive Review of Thermal Energy Storage" Sustainability 10no.1:191.<https://doi.org/10.3390/su10010191>
- [5] Faraji H, Benkaddour A, Oudaoui K, El Alami M, Faraji M. Emerging applications of phase change materials: a concise review of recent advances. Heat Transfer. 2020. <https://doi.org/10.1002/HTJ.21938>.
- [6] A. Benkaddour, M Faraji, Numerical Investigation of a Phase Change Material Building Integrating Solar Thermal Collector PCM-BST, Journal of Thermal Science and Engineering Applications, 1-34
- [7] Voller, V. R.; Cross, M.; Markatos, N. C. (1987): An Enthalpy Method for Convection/ Diffusion Phase Change. International Journal of Numerical Methods in Engineering, vol. 24, no. 1, pp. 271 - 284.
- [8] Murat Kenisarin, Khamid Makhomov, Solar energy storage using phase change materials, Renewable and Sustainable Energy Reviews, 2007, doi: [10.1016/j.rser.2006.05.005](https://doi.org/10.1016/j.rser.2006.05.005).

Investigation by LBM of heat transfer within fins equipped channel heated by a heat source

Mustapha FARAJI*

Renewable Energy & Dynamic Systems Laboratory - Renewable Energy and Laser- Plasma Group, Physics Department, Faculty of Sciences Ain Chock, Hassan II University of Casablanca, Morocco

*E-mail: farajimustapha@yahoo.fr

Abstract

The present paper focus on the forced convection channel equipped with heat source and constrained with ribs. To analyze the impact of the obstacles on streamlines and isotherms the Lattice Boltzmann Method (LBM) is introduced. It was found that the fins influence deeply the dynamic and heat fields and the heat removal depends closely to the air flow pattern.

Keywords: LBM, Forced convection, Cooling

I) Introduction

Thermal management will play a vital role in the coming decade for all types of electronics products. The major cause of an electronic chip failure is due to temperature rise. So it is a great challenge for the packaging engineers to remove the heat from the electronics chips very effectively. A heat sink is an object that transfers thermal energy from a higher temperature device to a lower temperature fluid medium. Castelan et al. [1] studied an analytical model of plate fin heat sink on forced convection, their analytical model will be used in an optimization routine to reduce the weight of an existing heat sink in order to show that fast and precise optimization of cooling system is possible with analytical models. Paulo Canhoto et al. [2] analyzed a heat sink composed of parallel tubes in a solid matrix of fixed dimensions. In another paper Paulo Canhoto et al. [3] address the optimization of a heat sink formed by parallel circular or non-circular ducts in a finite volume. Results for optimum dimensionless thermal length, optimum hydraulic diameter and maximum heat transfer rate density are presented for five different duct shapes subjected to a fixed pumping power constraint. Patel et al. [4] stated that the combined conduction-convection heat transfer in the heat sink produces very complex three-dimensional heat flow pattern with large longitudinal upstream directed heat recirculation zones in the highly conducting aluminum materials. A detailed description of the average heat transfer coefficient, temperature, heat flux and Nusselt number was obtained. Drogenik et al. [5] investigate the performance forced convection cooling and optimization procedure for forced convection heat transfer heat sinks. In the present paper we investigate, using Lattice Boltzmann Method, the forced convection and heat transfer within a channel in presence of two fins. The streamlines, isotherms and local Nusselt number are analyzed.

II) Mathematical model

The considered configuration here is a 2D channel of height $H=0.25$ ($m=50$ nodes) and length $L=1$ ($n=200$ nodes), heated from below by heat source $ABCD$ (h_s, l_s) with temperature $\theta=1$ localized in the middle of the bottom wall. Two rectangular fins (h_f) are situated in entrance region at the north wall and in the exit region at the south wall. The heat source simulates an electronic component. The walls of the channel are adiabatic (Figure 1). The air assumed to be an incompressible Newtonian fluid with $Pr=0.71$ and Reynolds number value $Re = u_0 m / \alpha = 250$, enters at temperature $\theta=0$ and velocity, $u_0=0.08$.

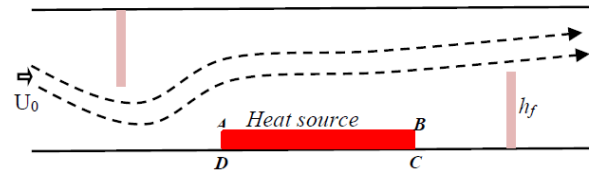


Figure 1: Physical model

The Lattice Boltzmann Method uses two distribution functions for the dynamic and thermal fields, $f_k(x,t)$ and $g_k(x,t)$ respectively. We consider a 2D model of the LBM on a square grid of geometric step $\Delta x = \Delta y = l$ at nine discrete velocities D2Q9 model [6,7].

✓ **For the dynamic field**

$$f(x + c_k \Delta t) = f_k(x, t) + W_m [f_k^{eq}(x, t) - f_k(x, t)] \quad (1)$$

$$W_m = \frac{\Delta t}{\tau_m}, \quad \tau_m \text{ is the relaxation time for the dynamic field.}$$

✓ **For the thermal field**

$$g(x + c_k \Delta t, t + \Delta t) = g_k(x, t) + W_s [g_k^{eq}(x, t) - g_k(x, t)] \quad (2)$$

$$W_s = \frac{\Delta t}{\tau_s}, \quad \tau_s \text{ is the relaxation time for the thermal field.}$$

The discrete c_k velocities of the D2Q9 model are defined as follows [7]:

$$c_k = \begin{cases} (0,0) & \text{for } k = 0 \\ c(\cos\theta_k, \sin\theta_k), & \theta_k = (k-1)\frac{\pi}{2} \quad \text{for } k = 1,2,3,4 \\ c\sqrt{2}(\cos\theta_k, \sin\theta_k), & \theta_k = (k-5)\frac{\pi}{2} + \frac{\pi}{4} \quad \text{for } k = 5,6,7,8 \end{cases} \quad (3)$$

Where $c = \frac{\Delta x}{\Delta t}$ with $\Delta t = l$ is the time step.

the link between the kinematic viscosity ν and the characteristic of relaxation time. and thermal diffusivity:

$$\nu = c_s^2 (\tau_m - \frac{1}{2}), \quad \alpha = c_s^2 (\tau_s - \frac{1}{2}) \quad (5)$$

$c_s = \frac{c}{\sqrt{3}}$ is the lattice speed of sound.

f_k^{eq} and g_k^{eq} are respectively the local equilibrium distribution functions for dynamic and thermal fields from where:

$$f_i^{eq}(x, t) = \omega_k \rho(x, t) \left[1 + 3 \frac{\vec{c}_k \vec{u}}{c^2} + \frac{9}{2} \frac{(\vec{c}_k \vec{u})^2}{c^4} - \frac{3}{2} \frac{\vec{u} \vec{u}}{c^2} \right] \quad (7)$$

$$g_k^{eq} = \omega_k \theta(x, t) \left[1 + 3 \frac{\vec{c}_k \vec{u}}{c^2} \right] \quad (7)$$

With ω_k called the nodal weights or the weights factors, for the D2Q9 model the nodal weights are presented as follows [6,7]:

$$\omega_k = \frac{4}{9} \text{ for } k = 0, \frac{1}{9} \text{ for } k = 1, 2, 3, 4, \frac{1}{36} \text{ for } k = 5, 6, 7, 8 \quad (8)$$

And finally, the macroscopic quantities such as density, velocities and temperature are calculated as follows:

$$\rho(x, t) = \sum_{k=0}^{k=8} f_k(x, t), \quad \rho \vec{u}(x, t) = \sum_{k=0}^{k=8} \vec{c}_k f_k(x, t) \quad (10)$$

$$\theta(x, t) = \sum_{k=0}^{k=8} g_k(x, t) \quad (12)$$

3) Boundary conditions

-Dynamic boundary conditions

For $x=0, x=1$ and $0 < y < 1$, $U=V=0$

For $y=0, y=1$ and $0 < x < 1$, $U=V=0$

-Bounce-back conditions

$$f_k(x, t) = f_{opp,k}(x, t) \quad (13)$$

where $k=1, 2, 3, 4, 5, 6, 7, 8$ and $(opp, k)=3, 4, 1, 2, 7, 8, 5, 6$

-Thermal boundary conditions

At the cold wall: $\theta = \theta_c = 0$

At the heat source: $\theta = \theta_H = 1$

-Bounce back conditions

$$g_k(i, j) = (\omega_k + \omega_{opp,k}) \theta(x, t) - g_{opp,k}(i, j) \quad (14)$$

where $k=1, 2, 3, 4, 5, 6, 7, 8$ and $(opp, k)=3, 4, 1, 2, 7, 8, 5, 6$

-At the heat source

$$\theta = \theta_H = 1$$

-At the adiabatic walls

$$\frac{\partial \theta}{\partial \eta} = 0 \quad (20)$$

- At the West boundary, flow in:

$$\rho = \frac{f_0 + f_2 + f_4 + 2(f_3 + f_6 + f_7)}{1 - u_o}$$

$$f_1 = f_3 + 2\rho * \frac{u_o}{3}, f_5 = f_7 + \rho * \frac{u_o}{6}, f_8 = f_6 + \rho * \frac{u_o}{6}$$

$$\theta = 0, g_1 = -g_3, g_5 = -g_7, g_8 = -g_6$$

- At the East boundary, fluid exit:

$$f_1(L) = 2f_1(n-1) - f_1(n-2)$$

$$f_5(L) = 2f_5(n-1) - f_5(n-2)$$

$$f_8(L) = 2f_8(n-1) - f_8(n-2)$$

$$g_k(n) = 2g_k(n-1) - g_k(n-2), k = 0..8$$

- heat source Local Nusselt number

$$Nu = -l \frac{\partial \theta}{\partial \eta} \Big|_{\eta} \quad (23)$$

- heat source Average Nusselt number

$$\overline{Nu} = \frac{1}{(2h_S + l_S) \Delta \theta} \int_0^{2h_S + l_S} Nu_h dl \quad (26)$$

with η is the normal of walls

dl is the element length of the exposed faces of heat source.

III) Results and discussions

Figure 2 shows the streamlines shape in the channel heated by the heat source in the presence of an upstream fin located to the north and a downstream fin located south of the canal. Figure 3 shows the corresponding isotherms.

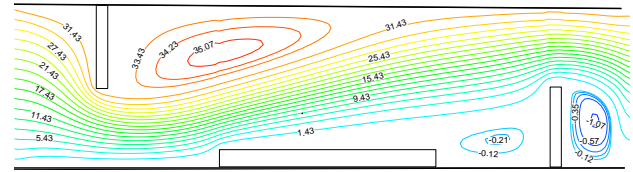


Figure 2: Streamlines

The cold fluid enters the canal from the left side is turned down because of the presence of the north obstacle. The current lines are condensed and form a narrow dynamic boundary layer impacting the left vertical face of the source. The fluid rebounds by stirring the horizontal upper surface of the source. A significant thermal gradient takes place at corner B of the source as shown in Figure 3.

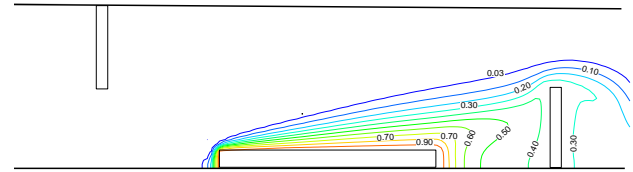


Figure 3: Isotherms

The isotherms are dense in this area and form a thermal boundary layer. A large amount of heat is extracted from the heat source and carried by the fluid that moves towards the outlet to the right of the channel passing over the source. We can see that the thermal boundary layer widens with a good distance from the isotherms. The flow is developed but the heat transfer is reduced. After this stage, the fluid impacts the clever fin and bounces above the fin to leave the canal. Three fluid recirculation vortices, of different structures, are observed in Figure 2. The first is located to the right of the left fin above corner A of the source. It is a positive cell that promotes the evacuation of heat from the source. The 2nd vortex is isolated between the CD face of the source before the south east right fin of negative sign, which shows that a fraction of the heat extracted from the source is returned to it, which disadvantages the proper transfer of heat in this place. Indeed, Figure 3 shows that the isotherms between the CD side and the fin are deformed and pushed back to the CD side by the negative vortex while outside this micro cavity the isotherms are released and distanced and reflect an ease of heat evacuation. The last vortex located after the fin on the right at the exit of the channel, it is also negative, and a fraction of heat is re-injected in the opposite direction of the flow. Indeed we notice on Figure 3 that the isotherms instead of leaving the channel they are jostled by the vortex towards the fin downstream.

Figure 4 illustrates the evolution of the local Nusselt number, Nu , calculated at the periphery η of the heat source. The analysis of this figure makes it possible to consolidate the observations made in the preceding paragraph. Indeed, on the portion AB of the heat source the local Nusselt number increases rapidly by going from A to B recording a maximum value at the corner B where $Nu_{max}=19.85$. This shows that the heat transfer at this location is very intense. This is due to the impact of the cold fluid redirected by the left fin and the presence of the narrow dynamic and thermal boundary layers in this area as explained in the section above. After point B, the fluid is released and the boundary layer widens which causes a drop in heat transfer and these results in the decay of the local Nusselt number along the upper face of the source in a parabolic way. Note that the average value of the Nusselt calculated on the heat source wetted periphery is equal to 7.04. A local Nusselt number minima is observed entering corners B and C with $Nu_{min}=1.64$. The Nusselt number then undergoes a significant increase when the fluid stirs the corner C and then, it undergoes a fall along the CD side. This is mainly due to the negative recirculation of the fluid between the CD side and the downstream fin.

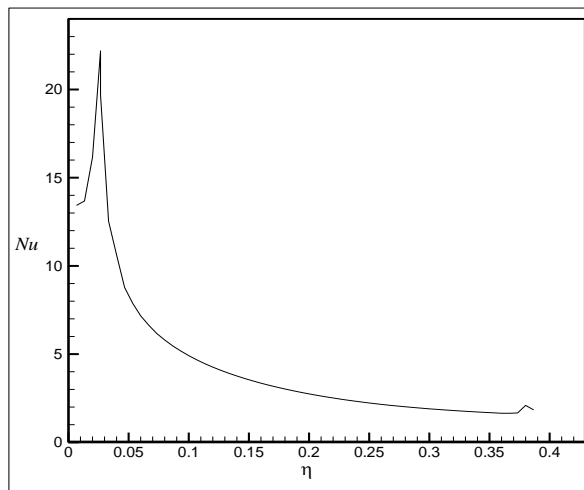


Figure 4: Evolution of the local Nusselt number, Nu , at the periphery η of the heat source.

NOMENCLATURE

c_k	discrete lattice speed, m/s
c_s	lattice sound speed, m/s
$f_k(x,t)$	particle distribution function for density
$g_k(x,t)$	particle distribution function for temperature
H	height of the cavity, m
L	width of the cavity, m
Nu	local Nusselt number
Pr	Prandtl number
T	temperature, K
t	time, s
$u = (u,v)$	macroscopic velocities, m/s
$U = (U,V)$	dimensionless velocity ($=u/(\alpha/H)$)
$x=(x,y)$	Cartesian coordinates

Greek symbols

α	thermal diffusivity, m^2/s
Δt	lattice time step
Δx	lattice space step
ν	kinematic viscosity, m^2/s
ω_k	weights for the particle equilibrium distribution function
ρ	fluid density, kg/m^3

θ	dimensionless temperature $(T-T_c)/(T_h-T_c)$
$\Delta\theta$	difference between the dimensionless temperature of the heat source and the cold wall $(\theta_h-\theta_c)$
τ_m	single particle relaxation times for density
τ_s	single particle relaxation times for temperature
η	heat source periphery

Subscripts

c	cold
h	heat
k	lattice link number
opp	opposite

Superscript

eq	equilibrium
------	-------------

IV) Conclusion

The exponential increase in thermal load in air cooling devices requires the thermal management system to be optimized to attain the highest performance in the given space. The analysis of the forced convection channel with fins state that the Left fin deviates the fluid flow and a significant dynamic gradient takes place at the downstream corner of the source recording a maximum value Nusselt number. This shows that the heat transfer at this location is very intense. Two negative vortexes are observed after the upstream heat source corner and at the channel exit in front of the right fin. They act unkindly in heat removal out of the channel.

References

- [1] A. CASTELAN, B. COUGO, S. DUTOUR, T. MEYNARD, 3D Analytical modelling of plate fin heat sink on forced convection
<https://www.sciencedirect.com/science/article/pii/S037847541830243X>
Manuscript_10de657a23130d84e20ecc1872d198d2
- [2] Paulo Canhoto, A. Heitor Reis, Optimization of fluid flow and internal geometric structure of volumes cooled by forced convection in an array of parallel tubes International Journal of Heat and Mass Transfer 54 (2011) 4288–4299
- [3] Paulo Canhoto, A. Heitor Reis, Optimization of forced convection heat sinks with pumping power requirements, International Journal of Heat and Mass Transfer 54 (2011) 1441–1447
- [4] Hardik Patel, V. K. Matawala, Performance Evaluation and parametric optimization of a Heat Sink for Cooling of Electronic Devices with Entropy Generation Minimization European Journal of Sustainable Development Research 2019, 3(4), em0100
- [5] U. Drogenik, A. Stupar, and J. W. Kolar, “Analysis of theoretical limits of forced-air cooling using advanced composite materials with high thermal conductivities,” IEEE Trans. on Components, Packaging and Manufacturing Technology, vol. 1, no. 4, pp. 528–535, 2011.
- [6] A. Mohamad, Lattice Boltzmann Method, vol. 70. Springer, 2011.
- [7] M. Faraji, El M. Berra, Lattice Boltzmann Computations of Natural Convection Heat Transfer of Nanofluid in a Square Cavity Heated by Protruding Heat Source, Journal of Thermal Science and Engineering Applications, ASME-TSEA,, AUGUST 2020, Vol. 12 / 041021-1

Analytical and numerical solutions of the Graetz problem in ducts of arbitrary cross-section

M. SHAIMI*, R. KHATYR and J. KHALID NACIRI

Laboratory of Mechanics, Faculty of Sciences Ain Chock, Hassan II University of Casablanca, Casablanca, Morocco

* Corresponding author: shaimimohamed8@gmail.com

Abstract

This paper presents a comparison between an analytical and numerical solution of the Graetz problem in ducts of arbitrary cross-section. The cross-sectional shape is given, in polar coordinates r, θ , by the solution of the equation $1 - r^2 + \varepsilon r^k \sin(k\theta) = 0$ where ε and k are geometrical parameters. The shape factor method is used to determine analytically a general solution of the temperature distribution. A domain transformation and a Fourier series decomposition are used to transform the energy equation into a system of parabolic partial differential equations which is solved numerically using a finite difference method. Analytical and numerical results are presented for the case of an elliptical cross-sectional shape ($\varepsilon = 0.22$) and they are in good agreement. It was found that the analytical solution can be used without limitations for $k > 2$.

Keywords: Graetz Problem; Arbitrary Cross-Section; Shape Factor Method; Finite Difference Method

1. Introduction

The Graetz problem is the study of the laminar forced convection in a thermal entrance region which is one of the basic thermal design theories for heat exchangers. Shah and London [1] have presented the solutions to the Graetz problem in ducts with different cross-sectional shapes by using various analytical and numerical methods. Letelier and Leuthesser [2] have presented an analytical solution technique to determine the developed velocity distribution for the flow in ducts of unconventional shapes. A known particular solution w_p of the Poisson equation is linearly combined with a harmonic function w_h as $w = w_p + \varepsilon w_h$ where ε is a parameter that defines the new shape of the duct. The cross-sectional shape and the range of possible values of the parameter ε are determined by using the no-slip condition. Barrera et al. [3] have presented an analytical solution of the Graetz problem in ducts of arbitrary cross-section given by the solution of the equation. $1 - r^2 + \varepsilon r^k \sin(k\theta) = 0$. They used the shape factor method to determine the temperature distribution analytically.

The objective of this paper is the comparison between the analytical and numerical solutions of the Graetz problem in ducts of arbitrary cross-section. The analytical solution is based on the shape factor method presented by Barrera et al. [3]. For the numerical solution, a domain transformation and a Fourier series decomposition are used to transform the energy equation into a system of equations that will be solved by using the finite difference method. The numerical method used in this paper is in part analytical to avoid the use of Computational Fluid Dynamics (CFD) software for three-dimensional simulations which are time-consuming

and while the arbitrary cross-sectional shapes are not explicitly given. The analytical and numerical results are going to be used in a case study of the elliptical cross-sectional shape to solve the energy equation and validate the correctness of the different techniques used in the analytical and numerical solutions along with their limits of validation.

2. Mathematical formulation

To compare the analytical and numerical solutions, the following assumptions are considered where both methods are valid. The flow is steady, laminar, incompressible, thermally developing, and hydrodynamically fully developed. The fluid is Newtonian with constant thermophysical properties. The body forces, viscous dissipation, and axial conduction are negligible and there is no heat source.

2.1 Dynamic problem

The governing equation for the velocity problem (the axial momentum equation) is [1]:

$$\frac{1}{r^*} \frac{\partial}{\partial r^*} \left(r^* \frac{\partial w^*}{\partial r^*} \right) + \frac{1}{r^{*2}} \frac{\partial^2 w^*}{\partial \theta^2} = \frac{1}{\mu} \frac{dp^*}{dz^*} = \text{constant} \quad (1)$$

Where (r^*, θ, z^*) are the cylindrical coordinates, μ is the viscosity of fluid, $w^*(r^*, \theta)$ and $\frac{dp^*}{dz^*}$ are respectively the axial velocity component and the pressure gradient along the streamwise direction.

For the fully developed flow, Eq. (1) shows that the pressure gradient is constant. The solution of Eq. (1) can be written as a particular solution linearly combined with a harmonic function. The particular solution can be chosen as any known solution such as the solution in the case of a circular cross-section. The harmonic function can be chosen as the real or imaginary part of any analytical function of the complex variable $re^{i\theta}$ such as the imaginary part of the function $(re^{i\theta})^k$. Thus, the dimensionless axial velocity is:

$$w(r, \theta) = 1 - r^2 + \varepsilon r^k \sin(k\theta) \quad (2)$$

where $w = \frac{w^*}{w_0}$, $r = \frac{r^*}{a}$, $w_0 = -\frac{a^2}{4\mu} \frac{dp^*}{dz^*}$ is a characteristic velocity and a is a characteristic radius.

The cross-sectional shape is not defined yet. The solution in Eq. (2) is only a mathematical solution of Eq. (1). The physical problem is defined by an equation with conditions at the boundaries. By using the boundary condition which is the no-slip condition $w = 0$ at the not yet defined cross-sectional shape, the following equation is obtained:

$$1 - r^2 + \varepsilon r^k \sin(k\theta) = 0 \quad (3)$$

The solution $R(\theta)$ of the Eq. (3) defines the cross-sectional shape. The new shape represents a deformation of the basic circular shape. k is a positive integer that indicates

the number of corners of the new shape and ε is a deformation parameter in which the range of possible real values is limited by the condition of closed contours [4]

$$\text{where } |\varepsilon| < \varepsilon_c(k) = \frac{2}{k} \left(\frac{k-2}{k} \right)^{\frac{k-2}{2}}.$$

2.2 Thermal problem

The energy equation under the assumptions used in this paper is:

$$w^*(r^*, \theta) \frac{\partial T^*}{\partial z^*} = \alpha \left(\frac{1}{r^*} \frac{\partial}{\partial r^*} \left(r^* \frac{\partial T^*}{\partial r^*} \right) + \frac{1}{r^{*2}} \frac{\partial^2 T^*}{\partial \theta^2} \right) \quad (4)$$

where $T^*(r^*, \theta, z^*)$ is the temperature distribution and α is the thermal diffusivity.

The fluid at the inlet ($z^* = 0$) has a constant temperature T_0 and the duct's wall is set to a fixed temperature T_w .

The dimensionless variables are:

$$r = \frac{r^*}{a}; z = \frac{z^*}{aPe}; w = \frac{w^*}{w_0}; T = \frac{T^* - T_w}{T_0 - T_w} \quad (5)$$

where $Pe = \frac{aw_0}{\alpha}$ is the Péclet number.

By inserting Eq. (5) in Eq. (4), the dimensionless energy equation is written as follows:

$$w(r, \theta) \frac{\partial T}{\partial z} = \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial T}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 T}{\partial \theta^2} \quad (6)$$

In addition to the periodicity in the tangential coordinate θ , the dimensionless boundary conditions are:

$$T(r, \theta, 0) = 1; T(R(\theta), \theta, z) = 0; T(r, \theta, z) = T(r, \theta + 2\pi, z) \quad (7)$$

3. Analytical solution

Following the procedure presented by Barrera et al. [3]. The temperature distribution is sought in the form of an asymptotic expansion in terms of the parameter ε :

$$T(r, \theta, z) = T_0(r, z) + \varepsilon T_1(r, z) \sin(k\theta) + \dots \quad (8)$$

The inlet condition in Eq. (7) becomes:

$$T_0(r, z=0) = 1; T_1(r, z=0) = 0 \quad (9)$$

To satisfy the boundary condition at the wall, the temperature distribution is modified as follows:

$$T(r, \theta, z) = w(r, \theta) \{ f_0(r, z) + \varepsilon f_1(r, z) \sin(k\theta) + \dots \} \quad (10)$$

The temperature in Eq. (10) satisfies the boundary condition at the wall for the general case of the cross-sectional shape $R(\theta)$ given by the solution of Eq. (3). Equating the Eqs. (8) and (10) and matching the powers of the parameter ε :

$$f_0(r, z) = \frac{T_0(r, z)}{1-r^2}; f_1(r, z) = \frac{T_1(r, z) - r^k f_0(r, z)}{1-r^2} \quad (11)$$

The temperature distribution is continuous and to have continuous functions at $r = 1$, the following regularity conditions are imposed:

$$T_0(r = 1, z) = 0; T_1(r = 1, z) = f_0(r = 1, z) \quad (12)$$

Substituting the Eq. (8) into the energy equation (6) yields the following equations at the orders respectively ε^0 and ε^1 :

$$(1-r^2) \frac{\partial T_0}{\partial z} = \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial T_0}{\partial r} \right) \quad (13)$$

$$(1-r^2) \frac{\partial T_1}{\partial z} + r^k \frac{\partial T_0}{\partial z} = \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial T_1}{\partial r} \right) - \frac{k^2}{r^2} T_1(r, z) \quad (14)$$

The forms of the solution used in Eqs. (8) and (10) can be justified by using the energy equation, the boundary conditions, and the periodicity in θ through the Fourier series decomposition. The only non-null terms will be the basic solution and the one in $\sin(k\theta)$ while the others will lead to homogenous equations with homogenous conditions in which solutions are zeros.

$T_0(r, z)$ is the solution of the basic circular cross-section which can be expressed, by using the separation of variables, as an expansion in terms of a complete set in the form:

$$T_0(r, z) = \sum_n C_{n0} \psi_{n0}(r) e^{-\lambda_{n0} z} \quad (15)$$

The eigenfunctions $\psi_{n0}(r)$ are given by [1,3]:

$$\psi_{n0}(r) = e^{-\frac{\lambda_{n0} r^2}{2}} M_{n0} \left(\frac{1}{2} - \frac{\lambda_{n0}}{4}, 1, \lambda_{n0} r^2 \right) \quad (16)$$

where M_{n0} are the Kummer functions, and the eigenvalues λ_{n0} are found by applying the regularity condition (12) then the constants C_{n0} can be found using the inlet condition (9).

Eq. (14) at the first order of the parameter ε includes explicitly the dependence of the non-circular cross-sectional shape through the parameter k . The regularity condition (12) is a non-homogenous z -dependent condition. To have a homogenous condition, a new function $\Theta_1 = T_1 - g_1$ is introduced where $g_1 = -\frac{r^{k-1}}{2} \frac{\partial T_0}{\partial r}$. The following non-homogenous equation is obtained:

$$\frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial \Theta_1}{\partial r} \right) - \frac{k^2}{r^2} \Theta_1(r, z) = (1-r^2) \frac{\partial \Theta_1}{\partial z} + Q_1(r, z) \quad (17)$$

where the non-homogenous term $Q_1(r, z)$ is given by:

$$Q_1(r, z) = r^k \frac{\partial T_0}{\partial z} + (1-r^2) \frac{\partial g_1}{\partial z} - \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial g_1}{\partial r} \right) + \frac{k^2}{r^2} g_1(r, z) \quad (18)$$

First, the homogenous equation ($Q_1 = 0$) is solved by using the separation of variables. So, its solution is written as an expansion in terms of a complete set in the form:

$$\Theta_1(r, z) = \sum_n C_{nk} \psi_{nk}(r) e^{-\lambda_{nk} z} \quad (19)$$

The eigenfunctions are:

$$\psi_{nk}(r) = \lambda_{nk}^{\frac{k}{2}} r^k e^{-\frac{\lambda_{nk} r^2}{2}} M_{nk} \left(\frac{2k+2-\lambda_{nk}}{4}, k+1, \lambda_{nk} r^2 \right) \quad (20)$$

The eigenvalues λ_{nk} are calculated by using the regularity condition ($\psi_{nk}(r=1) = 0$). The coefficients C_{nk} are considered to be z -dependent $C_{nk}(z)$ to take into consideration the non-homogenous term $Q_1(r, z)$.

Eq. (17) becomes:

$$(1-r^2) \sum_n \frac{dC_{nk}}{dz} \psi_{nk}(r) e^{-\lambda_{nk} z} = -Q_1(r, z) \quad (21)$$

Multiplying Eq. (21) by $r\psi_{lk}(r)$ then integrating both sides and using the orthogonality of the eigenfunctions and the inlet condition $C_{nk}(z=0) = 0$, the coefficients are:

$$C_{lk}(z) = -\frac{\int_0^z \int_0^1 Q_1(r, \zeta) e^{\lambda_{lk} \zeta} \psi_{lk}(r) r dr d\zeta}{\int_0^1 (1-r^2) (\psi_{lk}(r))^2 r dr} \quad (22)$$

Finally, the general solution is:

$$T(r, \theta, z) = w(r, \theta) \left\{ \frac{\sum_n C_{n0} \psi_{n0}(r) e^{-\lambda_{n0} z}}{1-r^2} - \varepsilon \left[\frac{r^{k-1} \sum_n C_{n0} \frac{d\psi_{n0}}{dr} e^{-\lambda_{n0} z}}{1-r^2} + \frac{\sum_l C_{lk}(z) \psi_{lk}(r) e^{-\lambda_{lk} z}}{1-r^2} + \frac{r^k \sum_n C_{nk} \psi_{nk}(r) e^{-\lambda_{nk} z}}{1-r^2} \right] \sin(k\theta) \right\} \quad (23)$$

4. Numerical solution

To investigate the limits of validity, a numerical solution without any restriction on the parameter ε is proposed and

will be compared with the results of the analytical solution. So, a new variable $\eta = \frac{r}{R(\theta)}$, where $0 \leq \eta \leq 1$, is introduced to transform the domain from the physical irregular cross-sectional shape given by $R = R(\theta)$ to a computational rectangular domain.

The energy equation (6) becomes:

$$W(\eta, \theta) \frac{\partial \theta}{\partial z} = \frac{1}{R^2} \left\{ \left[1 + \left(\frac{R'}{R} \right)^2 \right] \frac{\partial^2 \theta}{\partial \eta^2} + \frac{1}{\eta} \left[1 + \frac{4R'^2 - 2RR''}{R^2} \right] \frac{\partial \theta}{\partial \eta} - \frac{2R'}{\eta R} \frac{\partial^2 \theta}{\partial \eta \partial \theta} + \frac{1}{\eta^2} \frac{\partial^2 \theta}{\partial \theta^2} \right\} \quad (24)$$

where $W(\eta, \theta) = w(\eta R, \theta)$, $\Theta(\eta, \theta, z) = T(\eta R, \theta, z)$, $R' = \frac{dR}{d\theta}$, and $R'' = \frac{d^2 R}{d\theta^2}$.

The boundary conditions (7) become:

$$\Theta(\eta, \theta, z = 0) = 1; \Theta(\eta = 1, \theta, z) = 0 \quad (25)$$

An additional condition is needed so the solution is considered symmetrical about the axis ($\eta = 0$) which is valid for all the even values of the positive integer k . So, the following condition is imposed:

$$\left. \frac{\partial \Theta}{\partial \eta} \right|_{\eta=0} = 0 \quad (26)$$

The solution of the energy equation (24) is sought as a decomposition of the Fourier series in the tangential coordinate θ due to the periodicity. Thus, the temperature distribution is written as:

$$\Theta(\eta, \theta, z) = \sum_m \{ A_m(\eta, z) \cos(m\theta) + B_m(\eta, z) \sin(m\theta) \} \quad (27)$$

Substituting the Eq. (27) into the energy equation (24) and multiplying by $\cos(l\theta)$ or $\sin(l\theta)$ then integrating both sides and using the orthogonality of cosine and sine yield the following system of equations:

$$\begin{cases} \frac{\partial A_l}{\partial z} = \frac{1}{2\delta_l \pi} \int_0^{2\pi} S \cos(l\theta) d\theta \\ \frac{\partial B_l}{\partial z} = \frac{1}{\pi} \int_0^{2\pi} S \sin(l\theta) d\theta \end{cases} \quad (28)$$

with:

$$S = \frac{1}{W(\eta, z) R^2} \left(\sum_m \left\{ \left[1 + \left(\frac{R'}{R} \right)^2 \right] \frac{\partial^2 A_m}{\partial \eta^2} + \frac{1}{\eta} \left[1 + \frac{4R'^2 - 2RR''}{R^2} \right] \frac{\partial A_m}{\partial \eta} - \frac{2mR'}{\eta R} \frac{\partial A_m}{\partial \eta} - \frac{m^2}{\eta^2} A_m(\eta, z) \right\} \cos(m\theta) + \sum_m \left\{ \left[1 + \left(\frac{R'}{R} \right)^2 \right] \frac{\partial^2 B_m}{\partial \eta^2} + \frac{1}{\eta} \left[1 + \frac{4R'^2 - 2RR''}{R^2} \right] \frac{\partial B_m}{\partial \eta} + \frac{2mR'}{\eta R} \frac{\partial B_m}{\partial \eta} - \frac{m^2}{\eta^2} B_m(\eta, z) \right\} \sin(m\theta) \right) \quad (29)$$

and $\delta_l = \begin{cases} 1 & \text{if } l = 0 \\ 0 & \text{if } l \neq 0 \end{cases}$ is the Kronecker delta.

The boundary conditions are:

$$A_l(\eta, z = 0) = \delta_l; \quad B_l(\eta, z = 0) = 0; \quad A_l(\eta = 1, z) = 0; \quad B_l(\eta = 1, z) = 0; \quad \left. \frac{\partial A_l}{\partial \eta} \right|_{\eta=0} = 0; \quad \left. \frac{\partial B_l}{\partial \eta} \right|_{\eta=0} = 0 \quad (30)$$

The finite difference method is used to solve the system of equations (28) with the conditions (30).

5. Results and discussions

Figs. 1.a, 1.b, 1.c, and 1.d show the evolution of the analytical and numerical solutions of the temperature along the semimajor and semiminor axes of an ellipse for various cross-sections where $k = 2$ and respectively $\varepsilon = 0.22$ and

0.4. The analytical and numerical results for $\varepsilon = 0.22$ are in good agreement. For $\varepsilon = 0.4$, they start to disagree clearly. However, for $k > 2$ the critical value of the parameter $|\varepsilon|$ is $\varepsilon_c(k) < 0.385$ which means that the analytical solution is reliable.

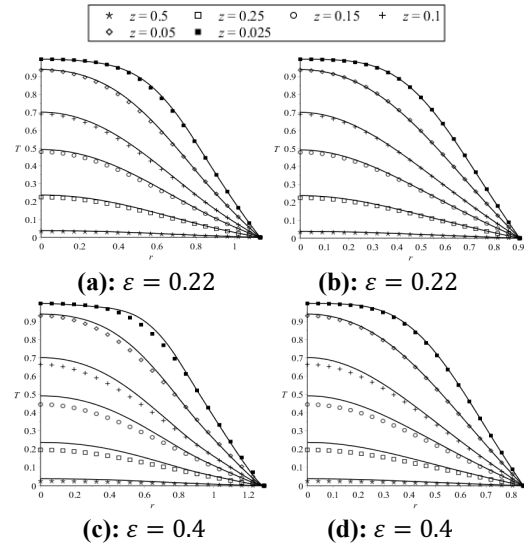


Figure 1. The evolution of the analytical (solid lines) and numerical (symbols) solutions of the temperature along the semimajor (a, c) and semiminor (b, d) axes of an ellipse ($\varepsilon = 0.22; 0.4$) for various cross-sections.

6. Conclusion

In this paper, analytical and numerical solutions of the Graetz problem in ducts of arbitrary cross-sections are presented. The analytical and numerical results in the case of the elliptical cross-section ($\varepsilon = 0.22$) are in good agreement. They start to disagree for $\varepsilon = 0.4$ however to get closed contours $|\varepsilon| < 0.385$ for $k > 2$ which means that the analytical solution can be used without problems. For small values of the variable z , a large number of eigenvalues (analytical solution) or a smaller z -step (numerical solution) might be needed to get precise results. For the numerical method and according to the finite difference technique used, the stability of the scheme must be taken into consideration (a smaller z -step must be needed for a smaller η -step).

Acknowledgment

This work is done with the financial support of the National Center for Scientific and Technical Research (CNRST).

References

- [1] R.K. Shah, A.L. London, *Laminar flow forced convection in ducts*, Academic Press, New York, 1978.
- [2] M.F. Letelier, H.J. Leutheusser, *Laminar flow in conduits of unconventional shape*, J. Eng. Mech. 111 (1985) 768-776.
- [3] C.H. Barrera, M.F. Letelier, D.A. Siginer, *The Graetz problem in tubes of arbitrary cross section*, Acta Mech. 227 (2016) 3239-3246.
- [4] D.A. Siginer, M.F. Letelier, *Laminar flow of non-linear viscoelastic fluids in straight tubes of arbitrary contour*, Int. J. Heat Mass Transfer 54 (2012) 2731-2745.

LBM simulation of the coupling of mixed convection and surface radiation in a lid-driven cavity cooled with a non-uniform temperature

Y. DAHANI¹, M. HASNAOUI², A. AMAHMID³, S. HASNAOUI⁴, A. MANSOUR⁵

LMFE, Department of Physics, Cadi Ayyad University, Faculty of Sciences Semlalia, B.P. 2390, Marrakesh, Morocco.

1. youssef.dahani@uca.ac.ma, 2. hasnaoui@uca.ac.ma, 3. amahmid@uca.ac.ma, 4. safae.hasnaoui@edu.uca.ac.ma, 5. a.mansour@uca.ac.ma

Abstract

The coupling between mixed convection and surface radiation in a lid-driven square cavity filled with air ($Pr = 0.71$) is studied. One of its two thermally active walls is heated isothermally and the temperature of the opposite one is characterized by sinusoidal spatial variation. The parameters governing the problem are the emissivity of its inner walls, ε ($0 \leq \varepsilon \leq 1$), the Richardson number, Ri ($0.01 \leq Ri \leq 100$), varied through the Grashof number, Gr , for a fixed Reynolds number Re ($Re = 100$). The results of the study show that the combined effects of the Richardson number and surface radiation on the overall structure of the flow and heat transfer characteristics are significant.

Résumé :

Le couplage entre convection mixte et le rayonnement de surface dans une cavité carrée à paroi entraînée remplie d'air ($Pr = 0.71$) est étudié. L'une de ses deux parois thermiquement actives est chauffée de manière isotherme tandis que la température de l'autre est caractérisée par une variation spatiale sinusoidale. Les paramètres régissant le problème sont l'émissivité de ses parois internes, ε ($0 \leq \varepsilon \leq 1$), le nombre de Richardson, Ri ($0.01 \leq Ri \leq 100$), varié à travers le nombre de Grashof, Gr , pour un nombre de Reynolds fixe Re ($Re = 100$). Les résultats de l'étude montrent que les effets combinés du nombre de Richardson et du rayonnement de surface sur la structure globale de l'écoulement et les caractéristiques de transfert de chaleur sont significatifs.

Mots clefs: *Étude numérique, Convection mixte, Cavité carrée à paroi entraînée, Refroidissement non uniforme, Rayonnement de surface, Méthode de Boltzmann sur réseau.*

Keywords: *Numerical study, Mixed convection, Lid-driven square cavity, Non-uniform cooling, Surface radiation, Lattice Boltzmann method.*

1. Introduction

The fundamental problem of combined forced and natural convection heat transfer in a cavity has received great interest through the decades. This interest is motivated by the importance of the problem in many engineering and industrial applications. Solar collectors, cooling of electronic devices, heat exchangers, materials processing, crystal growth, float glass production, metal coating and casting, etc. [1] are examples of applications where convection plays an important role. Combined forced convection and natural convection, also known as mixed convection, occurs when natural and forced convection mechanisms work together to transfer heat.

Earlier, Arpaci and Larsen [2] presented analytical results of mixed convection heat transfer in tall cavities, differentially heated with a movable vertical side and horizontal adiabatic boundaries. They showed that the forced and buoyancy-driven parts of the problem could be solved separately and combined to obtain the general mixed convection problem. Most of the previous works dealing with convection problems were performed assuming constant temperatures on the cold and hot walls of the cavity. However, the cooling and/or heating with non-uniform temperatures may be encountered in many industrial processes and furnaces [3–5]. In the case of lid-driven cavities, very few works have been devoted to situations where the boundary temperature has a sinusoidal spatial variation. The literature review shows that, in most of the studies addressing the problem of mixed convection in lid-driven cavities, the radiation effect was ignored although it may play a non-negligible role. In fact, in the study conducted by Balaji et al. [6], the effect of surface radiation on laminar mixed convection induced in a two-dimensional a differentially heated lid-driven cavity was found important. In fact, even in the full mixed convection regime, radiation contribution to the total heat transfer has reached 64%.

In the present study we focus on the combined effect of mixed convection and radiation and the sinusoidal cooling on heat transfer in a lid driven cavity using the lattice Boltzmann method.

2. Problem definition and mathematical formulation

2.1 Problem definition

The problem consists to determine steady-state laminar heat transfer characteristics in an lid-driven square cavity, of height H' , filled with air and heated from the upper wall with a constant temperature, T'_h . The heated wall is driven with a constant velocity u_0 as indicated in Fig. 1. The cavity is insulated from two parallel vertical walls and the temperature of the bottom wall is subject to a spatial variation of the form $T'(y) = T'_c - \Delta T' \sin\left(\frac{2\pi y}{H'}\right)$ [3]. The inner surfaces of the cavity, in contact with the fluid are assumed gray, diffuse emitters and reflectors of radiation.

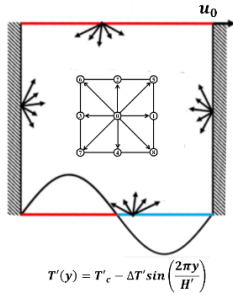


Figure 1. Schematic diagram of the studied configuration and lattices disposition within the cavity for the D2Q9 arrangements.

2.2 Governing equations

The governing equations for mixed convection flow using conservation of mass, momentum and energy can be written as follows:

$$\text{div}(\vec{u}) = 0 \quad (1)$$

$$\frac{\partial(\rho_0 \vec{u})}{\partial t} + \nabla \cdot (\rho_0 \vec{u} \vec{u}) \quad (2)$$

$$= -\nabla P + \rho_0 \nu \Delta \vec{u} + \rho \vec{g}$$

$$\rho_0 c_p \left(\frac{\partial T'}{\partial t} + \vec{u} \cdot \nabla T' \right) = \nabla \cdot (k \nabla T') \quad (3)$$

$$\rho = \rho_0 (1 - \beta(T' - T'_0)) \quad (4)$$

2.3 Numerical method

The numerical tool developed by our research team to model fluid flow and heat transfer in a lid-driven cavity under the effects of radiation and non-uniform cooling is based on the LBM approach. The robustness of this method in solving different kinds of convection problems was proved in previous studies while a detailed description of the numerical method is provided in Ref. [7].

2.4 Heat transfer

The average convective and radiative Nusselt numbers, calculated along the top active wall, are respectively calculated as:

$$\text{Nu}_{\text{Cv}} = - \int_{\text{wall}} \left. \frac{\partial T}{\partial X} \right|_{X=0} dY \quad (5)$$

and

$$\text{Nu}_{\text{Rd}} = \int_{\text{wall}} N_r Q_r |_{X=0} dY \quad (6)$$

The total average Nusselt number is obtained by summing the convective and radiative components:

$$\text{Nu} = \text{Nu}_{\text{Cv}} + \text{Nu}_{\text{Rd}} \quad (7)$$

2.5 Numerical validation and grid size effect

In the present study, the computations were carried out with a grid of 160×160 nodes. This choice was based on preliminary tests performed with different grids for $Ri = 1$ and $\varepsilon = 1$ as shown in Table 1. The results presented in this table in terms of Nu , show relative deviations less than 1% between the results obtained using 160×160 nodes and those corresponding to the finest grid of 320×320 nodes.

Table 1. Grid sensitivity analysis for $Ri = 1$ and $\varepsilon = 1$.

Grid size	80×80	160×160	320×320
Nu	4.2618	4.1782	4.1991

The validity of the present code based on the LBM was proved by comparing our results with those published by Ghia et al. [8] in the case of a lid-driven square cavity. Qualitative comparisons illustrated in Figures 2 in terms of velocity profiles show a good agreement between our results and those obtained by Ghia et al. [8].

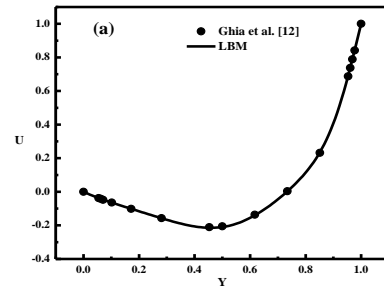


Figure 2. Validation in terms of velocity U profile along the Y axis at cavity mid-width for $Re = 100$.

3. Results and discussion

3.1 Fluid flow and temperature fields

For moderate Richardson numbers, illustrated by $Ri = 1$, the flow and temperature fields are exemplified in Fig. 3 for $\varepsilon = 0$ and 1. Fig. 3 shows that the flow is mainly unicellular with the formation of two small cells in the lower corners. In this case the system is a configuration heated from above and non-uniformly cooled from below. For such a configuration, the fluid heated and driven by the moving wall hits the side wall and moves then downward to meet the coldest corner of the cooled wall for the first time. This first contact between the hot fluid and the cold wall induces a small flow cell in the lower left corner of the cavity as it can be seen in Fig. 3.

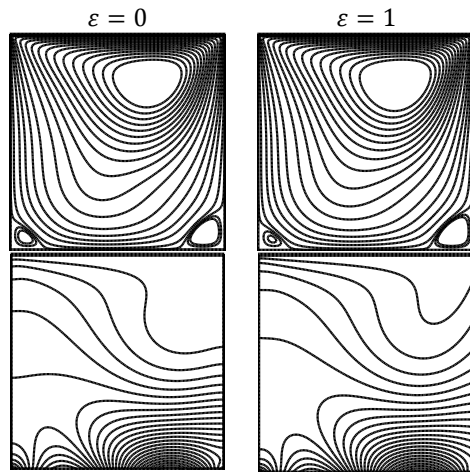


Figure 3. Streamlines (top) and isotherms (bottom) $Ri = 1$, and $\varepsilon = 0$ and 1. For $Ri = 100$, case illustrating the dominance of natural convection, the flow and temperature fields are illustrated in Fig. 4 for $\varepsilon = 0$ and 1. In this case the configuration is thermally stable in the absence of wall movement (when the cooling process is ensured by a constant cold temperature), but it is destabilized by the non-uniform cooling and the driving effect of the moving wall, which leads to the formation of a multicellular flow structure.

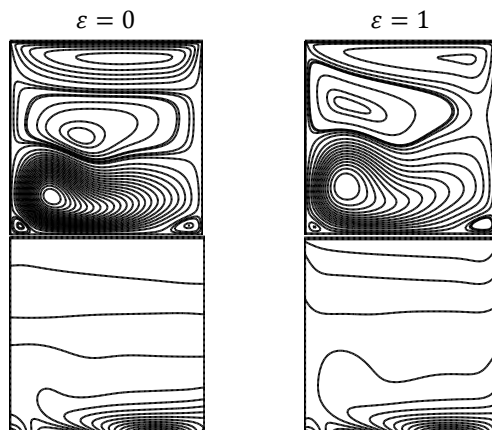


Figure 4. Streamlines (top) and isotherms (bottom) $Ri = 100$, and $\varepsilon = 0$ and 1.

3.2 Heat transfer

The effect of Richardson number on the convective, radiative, and total Nusselt numbers is illustrated in Figure 5. The increase of Ri from its lowest value (resulting from the decrease of the driven wall velocity) leads to a decrease of the convective Nusselt number because of the decrease in the overall intensity of the flow. While the radiative component increases with Ri due to the increase in the actual height H' of the cavity.

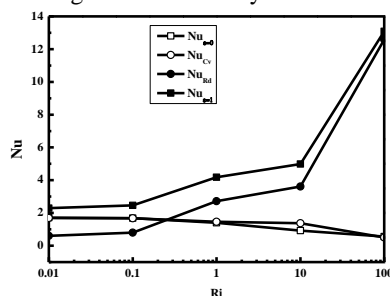


Figure 5. Variations of Nusselt numbers vs. Ri , and $\varepsilon = 0$ and 1.

4. Conclusions

Laminar mixed convection coupled with surface radiation in a lid-driven square cavity has been investigated numerically using the lattice Boltzmann method. The moving wall is heated using a constant temperature and the opposite one is cooled by a spatially varying temperature.

The main results obtained are summarized as follows :

- In mixed convection regime ($Ri \approx 1$), the flow structure is monocellular.
- In the dominant natural convection regime ($Ri \gg 1$), the flow changes from a structure dominated by a positive cell.
- The evolution of the flow with Ri is notably affected by the presence of the radiation.

References

- [1] D. Chatterjee, MHD Mixed Convection in a Lid-Driven Cavity Including a Heated Source, Numer. Heat Transf. Part A Appl. 64 (2013) 235–254. <https://doi.org/10.1080/10407782.2013.779191>.
- [2] V.S. Arpaci, P.S. Larsen, Convection Heat Transfer, Prentice Hall, New York, 1984.
- [3] E. Bilgen, R.B. Yedder, Natural convection in enclosure with heating and cooling by sinusoidal temperature profiles on one side, Int. J. Heat Mass Transf. 50 (2007) 139–150. <https://doi.org/10.1016/j.ijheatmasstransfer.2006.06.027>.
- [4] G. Mills, P. Mullinger, B. Jenkins, Industrial and Process Furnaces Principles, Design and Operation, Second Ed, Butterworth-Heinemann, Oxford, 2013. <https://doi.org/https://doi.org/10.1016/B978-0-08-099377-5.18001-3>.
- [5] I.E. Sarris, I. Lekakis, N.S. Vlachos, Natural Convection in a 2D enclosure with sinusoidal upper wall temperature, Numer. Heat Transf. Part A Appl. 42 (2002) 513–530. <https://doi.org/10.1080/10407780290059675>.
- [6] C. Balaji, M. Hölling, H. Herwig, Combined laminar mixed convection and surface radiation using asymptotic computational fluid dynamics (ACFD), Heat Mass Transf. Und Stoffuebertragung. 43 (2007) 567–577. <https://doi.org/10.1007/s00231-006-0145-3>.
- [7] Y. Dahani, M. Hasnaoui, A. Amahmid, A. El Mansouri, S. Hasnaoui, Lattice Boltzmann Simulation of Combined Effects of Radiation and Mixed Convection in a Lid-Driven Cavity with Cooling and Heating by Sinusoidal Temperature Profiles on One Side, Heat Transf. Eng. 41 (2020). <https://doi.org/10.1080/01457632.2018.1558009>.
- [8] U. Ghia, K.N. Ghia, C.T. Shin, High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method, J. Comput. Phys. 48 (1982) 387–411. [https://doi.org/10.1016/0021-9991\(82\)90058-4](https://doi.org/10.1016/0021-9991(82)90058-4).

Assessment of the magnetic field effect on thermal convection in a Bénard square porous cavity salted from its vertical walls

A. Mansour†, A. Amahmid, M. Hasnaoui, I. Filahi and Y. Dahani

FSSM, LMFE, Unit affiliated to CNRST (URA 27), University Cadi Ayyad, Marrakech, Morocco,

†Corresponding Author Email: a.mansour@uca.ac.ma

Abstract

The present work investigates numerically the thermal natural convection within a square porous cavity saturated by conductor binary mixture under effect of vertical magnetic field. Three main different flow modes are considered. Attention is focused on the changes in streamlines and the corresponding isotherms and isoconcentrations at various magnetic field magnitudes. Also, the heat and mass transfer transition presented by the average Nusselt and Sherwood numbers are examined. The results disclose that enhancing the magnetic field intensity causes significant changes of streamlines, isotherms and isoconcentrations lines. It is found that the interaction between flow modes and magnetic field may affect considerably the flow intensity and the heat and mass transfer.

Keywords: Numerical study; Porous media; Natural convection; Heat and mass transfer; Magnetic field.

1. Introduction

Natural convection of electrically conducting fluids in the presence of magnetic field has been one of the major interesting research subjects due to its widely engineering applications [1,2].

Nowadays, many researchers have given considerable attention to study analytically and numerically this phenomenon. In fact, Garandet et al. [3] derived analytical expressions to study the free convective flow within a 2-dimensional cavity under the transverse magnetic field and showed that the distributions of velocity and temperature in the core region followed the power law of Hartmann number. Rudraiah et al. [4] performed a numerical investigation of MHD free convection within the rectangular enclosure filled with electrically conducting fluid and subjected to magnetic field parallel to the gravity. A numerical study is presented by Sathiyamoorthy and Chamkha [5] for natural convection flow of electrically conducting liquid gallium in a square cavity. Natural convection flows of an electrically conducting fluid under a uniform magnetic field at different angles with respect to horizontal plane are investigated numerically in rectangular cavities by Yu et al. [6]. Magnetic nanofluid natural convection in the

porous enclosure considering Brownian motion is studied numerically by Dogonchi et al. [7].

The main aim of this work is to apply the magnetic field to the Bénard square porous cavity differentially salted from the vertical sides. Based on authors knowledge and above literature survey that the problem of combined effect of applied magnetic field and different flow modes is not studied in earlier works. Thus, the study is important to understand the effects of vertical applied magnetic field at different strength on heat and mass transfer and fluid flow structures and intensities.

2. Mathematical formulation

The physical model considered in this work is a square cavity filled with an isotropic and homogenous porous medium, saturated with an electrically conducting binary mixture. The bottom wall of the cavity is maintained at a hot temperature T_h , while the top one is maintained at cold a temperature T_c . The right vertical wall is maintained at a concentration (S'_1) higher than that of left one (S'_0). The top and bottom walls are assumed impermeable to mass transfer, while the right and left ones are assumed adiabatic. A magnetic field of strength \mathbf{B}_0 is applied in a direction normal to the horizontal walls of the porous cavity. The binary fluid mixture saturating the porous medium is modeled as a Boussinesq incompressible fluid whose density varies linearly with temperature and concentration.

Using the Darcy model and taking into account the magnetic field effect, the dimensionless equations governing the flow and heat and mass transfer in the saturated porous medium are written as follows [10]:

$$\frac{\partial^2 \psi}{\partial x^2} + (1 + Ha^2) \frac{\partial^2 \psi}{\partial y^2} = -R_T \frac{\partial T}{\partial x} \quad (1)$$

$$\frac{\partial T}{\partial t} + U \frac{\partial T}{\partial x} + V \frac{\partial T}{\partial y} = \nabla^2 T \quad (2)$$

$$\varepsilon \frac{\partial S}{\partial t} + U \frac{\partial S}{\partial x} + V \frac{\partial S}{\partial y} = \frac{1}{Le} \nabla^2 S \quad (3)$$

$$U = \frac{\partial \psi}{\partial y} \quad \text{and} \quad V = -\frac{\partial \psi}{\partial x} \quad (4)$$

where Ψ , T , and S are the dimensionless stream function, temperature and concentration, respectively. U and V (X and Y) are the dimensionless horizontal and vertical velocities (coordinates). The hydrodynamic, thermal and solutal boundary conditions associated with the present problem are [9]:

$$\left. \begin{aligned} \text{For } X = 0 \text{ and } 0 \leq Y \leq 1 : \Psi = 0, \frac{\partial T}{\partial X} = 0, S = 0 \\ \text{For } X = 1 \text{ and } 0 \leq Y \leq 1 : \Psi = 0, \frac{\partial T}{\partial X} = 0, S = 1 \\ \text{For } Y = 0 \text{ and } 0 \leq X \leq 1 : \Psi = 0, \frac{\partial S}{\partial Y} = 0, T = 1 \\ \text{For } Y = 1 \text{ and } 0 \leq X \leq 1 : \Psi = 0, \frac{\partial S}{\partial Y} = 0, T = 0 \end{aligned} \right\} (5)$$

The above dimensionless equations show that the solutions for the present configuration are governed by four dimensionless parameters, namely, the thermal Rayleigh number R_T , the Lewis number Le , the Hartmann number Ha and the normalized porosity ϵ . They are respectively defined as:

$$R_T = g\beta_T\Delta T'KL'/\alpha\nu, Le = \alpha/D, Ha = B_0\sqrt{K\epsilon/\mu} \text{ and } \epsilon = \epsilon'/\sigma \quad (6)$$

where $\Delta T'$ is the temperature differences respectively ($\Delta T' = T'_h - T'_c$), g is the gravitational acceleration, K the permeability of the porous medium, L' the length of the porous cavity, D the mass diffusivity, α the thermal diffusivity ($\alpha = \lambda/(\rho c)_f$ where λ is the thermal conductivity of the saturated porous medium and $(\rho c)_f$ the heat capacity of fluid mixture), B_0 the magnetic field intensity, ϵ the electric conductivity, ϵ' the porosity of the porous medium, σ the heat capacity ratio ($\sigma = (\rho c)_p/(\rho c)_f$ where $(\rho c)_p$ is the heat capacity of the saturated porous medium) and ν and μ are the kinematic and dynamic viscosities of the fluid respectively. The normalized porosity and the Lewis number are fixed at $\epsilon = 1$ and $Le = 10$ in the present study.

The heat and solute transfer across the cavity are given in terms of the Nusselt and Sherwood numbers defined as:

$$Nu = \int_0^1 \frac{\partial T}{\partial Y} \Big|_{Y=1} dX \quad \text{and} \quad Sh = \int_0^1 \frac{\partial S}{\partial X} \Big|_{X=0} dY \quad (7)$$

3. Numerical Method

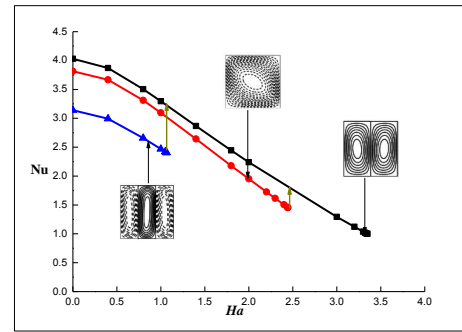
The numerical method used for solving the full governing equations is based on a second order finite-difference scheme. The temperature and concentration equations, Eqs. (2)–(3), are solved iteratively using the alternate direction implicit method. Nodal values of the stream function are obtained, from Eq. (1), via a point successive-over-relaxation method. Details concerning the validation of the present code were reported by Bourich et al. [8].

4. Results and discussion

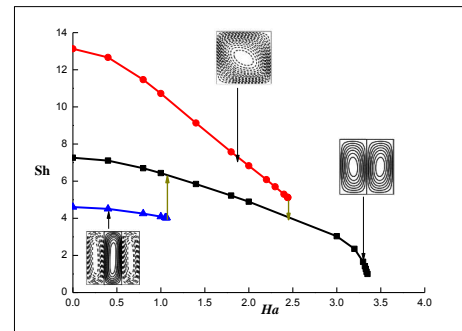
The steady solutions submitted to the effect of a magnetic field in the present work are characterized by monocellular flow, bicellular flow, and tricellular flow, denoted MF, BF, and TF, respectively. For other details about these solutions see Mansour et al. [9].

The effect of Hartmann number on the maintaining of the multiplicity of solutions and on the corresponding heat and mass transfer for $R_T = 200$, and $Le = 10$ is presented in Figs. 1 (a-b). It can be seen that the heat and mass transfer is weakened by the increase of the Hartmann parameter for all the flow modes considered. The arrows in these figures indicate the transitions obtained when Ha

is progressively increased from 0. The increasing of Ha from 0 leads to the transitions of TF and MF toward BF at $Ha_{cr} = 1.067$ and $Ha_{cr} = 2.445$, respectively. The first transition is accompanied with an enhancement of 31.6 % for Nu and 55.1% for Sh , while the second transition increases Nu by 23.7% but decreases Sh by 18.9%. However, the BF is maintained until the critical value $Ha_{cr} = 3.35$, beyond which the flow vanishes and the heat and mass transfer is controlled by pure diffusion ($Nu=Sh=1$).



(a)



(b)

Fig. 1. Variations of Nu (a) and Sh (b) vs. Ha for $R_T = 200$, $Le = 10$ and different flow modes.

The influence of Ha on the flow structures and the temperature and concentration fields, corresponding all flow modes considered in this study is illustrated on the Figs. 2-4. An examination of the evolution of the streamlines, isotherms and isosolutes, corresponding TF (Figs 3(a-b)), with the Hartmann parameter showed that all fields remain qualitatively similar to those obtained in pure convection ($Ha = 0$). While the flow intensity reduces from $\Psi_{max} = 3.472$ to $\Psi_{max} = 2.854$. For MF, the flow cells became more organized and the flow intensity became weaker, hence Ψ_{max} decreases from 8.939 to 2.117, the temperature lines became more spaced while the concentration field receives a remarkable modifications in the vicinity of the vertical walls. For BF (Fig. 3a-b), increasing Ha , the flow cells became parallel vertically and the flow intensity decreases drastically from $\Psi_{max} = 5.994$ to $\Psi_{max} = 0.435$, the temperature lines became more and more horizontal, tend toward the

conductive regime shape. The concentration field is considerably affected by the magnetic field effect.

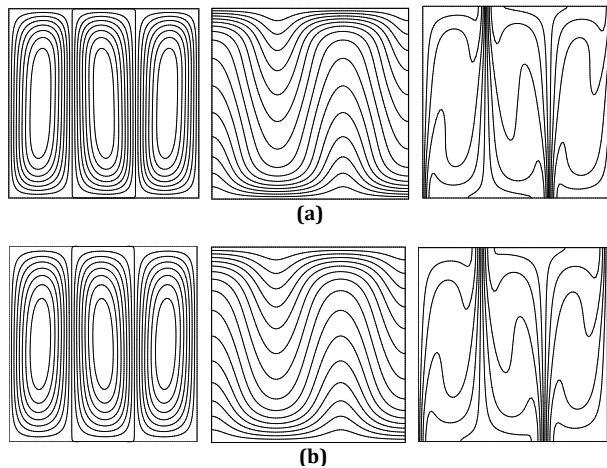


Fig. 2. Streamlines, isotherms, and isolines of bicellular flow obtained for $R_T = 200$, $Le = 10$, $N = 0$: (a) $Ha = 0$ $\Psi_{max} = 3.472$, $Nu = 3.139$ and $Sh = 4.610$ (b) $Ha = 1.06$ $\Psi_{max} = 2.854$, $Nu = 2.405$ and $Sh = 4.031$

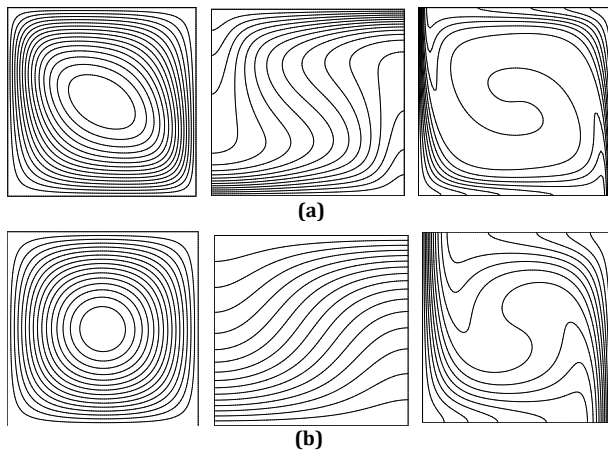


Fig. 3. Streamlines, isotherms, and isolines of monocellular flow obtained for $R_T = 200$, $Le = 10$: (a) $Ha = 0$, $\Psi_{max} = 8.939$, $Nu = 3.814$ and $Sh = 13.129$; (b) $Ha = 2.44$, $\Psi_{max} = 2.117$, $Nu = 1.459$ and $Sh = 5.132$

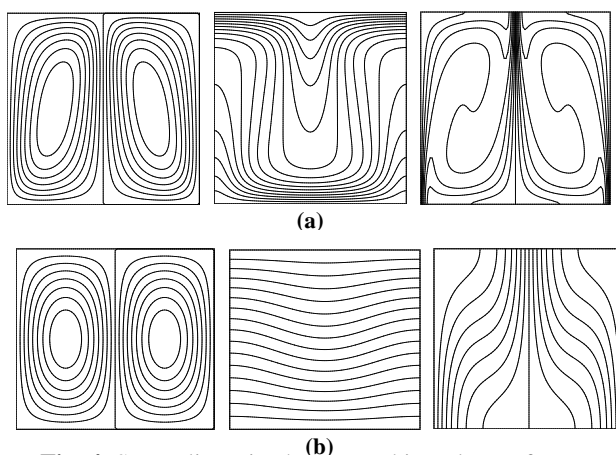


Fig. 4. Streamlines, isotherms, and isolines of bicellular flow obtained for $R_T = 200$, $Le = 10$, $N = 0$: (a) $Ha = 0$ $\Psi_{max} = 5.994$, $Nu = 4.030$ and $Sh = 7.266$ (b) $Ha = 3.3$ $\Psi_{max} = 0.435$, $Nu = 1.037$ and $Sh = 1.657$

Conclusion

The present work is to revisit the heat and mass transfer of thermally driven convection flow in the Bénard square porous cavity subjected to a solutal horizontal gradient under the vertical magnetic field.

Through in this study, it can be drawn that:

- ✓ With the enhancing magnetic field strength, remarkable changes in streamlines, isotherms and isoconcentrations contours have been found.
- ✓ The flow intensity and the heat and mass transfer are weakened by the increase of the Hartmann parameter for all the flow modes considered.
- ✓ The increase of Ha from 0 leads to the transitions of TF and MF toward BF.
- ✓ All obtained transitions are accompanied by an enhancement of Nu and Sh except that from MF to BF where we have decreasing of Sh .

References

- [1] C. Vives, C. Perry, *Effects of magnetically damped convection during the controlled solidification of metals and alloys*, Int. J. Heat Mass Transfer 30 (1987) 479–496.
- [2] H.P. Utech, M.C. Flemmings, *Elimination of solute banding in indium antimonide crystals by growth in a magnetic field*, J. Appl. Phys. 37 (1996) 2021–2024.
- [3] J.P. Garandet, T. Alboussiere, R. Moreau, *Buoyancy driven convection in a rectangular enclosure with a transverse magnetic field*, Int. J. Heat Mass Tran. 35 (4)(1992)741–748.
- [4] N. Rudraiah, R. Barron, M. Venkatachalappa, C. Subbaraya, *Effect of a magnetic field on free convection in a rectangular enclosure*, Int. J. Eng. Sci. 33 (8) (1995) 1075–1084
- [5] M. Sathiyamoorthy, A. Chamkha, *Effect of magnetic field on natural convection flow in a liquid gallium filled square cavity for linearly heated side wall (s)*, Int. J. Therm. Sci. 49 (9) (2010) 1856–1865.
- [6] P. Yu, J. Qiu, Q. Qin, Z.F. Tian, *Numerical investigation of natural convection in a rectangular cavity under different directions of uniform magnetic field*, Int. J. Heat Mass Tran. 67 (2013) 1131–1144.
- [7] A. Dogonchi, S.M. Seyyedi, M. Hashemi-Tilehnoee, A.J. Chamkha, D. Ganji, *Investigation of natural convection of magnetic nanofluid in an enclosure with a porous medium considering Brownian motion*, Case Stud. Therm. Eng. 14 (2019) 100502
- [8] Bourich, M., A. Amahmid and M. Hasnaoui, *Double diffusive convection in a porous enclosure submitted to cross gradients of temperature and concentration*, Energy Conversion and Management 45 (2004) 1655-1670.
- [9] Mansour, A., A. Amahmid, M. Hasnaoui and M. Bourich, *Multiplicity of solutions induced by thermosolutal convection in a square porous cavity heated from below and submitted to horizontal concentration gradient in the presence of Soret effect*, Numerical Heat Transfer Part A, 49 (2006) 69-94
- [10] Bourich, M., A. Amahmid, M. Hasnaoui, M. Er-raki and M. Mamou, *Analytical and numerical study of combined effects of a magnetic field and an external shear stress on Soret convection in a horizontal porous enclosure*, Numerical Heat Transfer Part A, 54 (2008) 1042-1060

Étude de la co-combustion du coke de pétrole et la biomasse dans les fours de cimenterie.

H. FETIAN¹, Z. NGADI¹, M.L. LAHLAOUTI¹

1. Laboratoire : Énergétique, Université Abdelmalek Essaadi, Faculté des sciences de Tétouan.

E-mail : zakia.ngadi@gmail.com.

Résumé

La substitution partielle du coke par des combustibles solides de substitution, tels que la biomasse, est reconnue comme une méthode avantageuse d'atténuation des gaz à effet de serre, et de consommation énergétique dans la production de ciment. L'objectif de cette étude est d'évaluer l'effet de l'utilisation de deux combustibles alternatifs de type biomasse, la poudre de grignon d'olives et coquille d'amande broyée, sur la performance de la combustion du four rotatif de ciment en comparant le comportement de co-combustion des deux particules au cas de combustion du coke seul à l'aide du logiciel Ansys. D'après les résultats les températures produites de la co-combustion par les grignons d'olives et par les coquilles d'amandes sont inférieures comparées à la température de la combustion du coke seul. Par contre une différence de comportement à la combustion en termes des particules du char a été remarquée.

Mots clés : *Co-combustion, modélisation CFD, biomasse, four rotatif.*

1. Introduction

La production du ciment est un processus énergivore avec une consommation d'énergie d'environ 3 MJ par kg de clinker produit [1]. En outre, la production du ciment génère une émission mondiale moyenne de carbone de 0,81 kg de CO₂/kg de ciment produit [2], [3], il est donc intéressant pour les producteurs de ciment d'envisager des combustibles de substitution disponibles, moins coûteux et neutres en CO₂. La biomasse représente une solution avantageuse comme combustible de substitution pour la production de ciment.

Plusieurs recherches sont établies ces dernières années concernant l'utilisation de la co-combustion charbon/biomasse[4]–[6]. L'étude présentée vise à évaluer l'effet de l'utilisation de deux combustibles alternatifs de type biomasse, à savoir grignon d'olives (GO) et coquille d'amande (CA), sur la performance de la combustion du four en comparant le comportement de co-combustion des deux particules au cas de combustion du coke de pétrole (CP) seul. L'approche adoptée est basée sur le modèle k-epsilon standard pour modéliser l'écoulement turbulent combiné au modèle FR/ED pour modéliser les réactions de combustion homogène. Le logiciel Ansys fluent basé sur les volumes finis est utilisé pour le calcul des résultats obtenus.

2. Caractéristiques des combustibles alternatifs étudiés

- Les grignons d'olives (GO) sont le principal sous-produits obtenus lors de la production d'huile d'olive.
- Les coquilles d'amande (CA) sont les principaux déchets générés par l'industrie des amandes.

Le tableau 1 représente une comparaison entre les différents combustibles utilisés dans cette étude.

Tableau 1 : Comparaison entre les AF et le coke de pétrole

Propriétés du carburant	CP	GO	CA
Analyse finale (massique)			
Carbone	88.6	51.6	49.68
Hydrogène	3.74	6	8.18
Oxygène	1.4	33.7	37.38
Soufre	3.98	0.35	1.04
Azote	1.62	1.89	3.75
Analyse approximative (massique)			
Volatile	10.6	57	66.93
Carbone fixe	87.02	16.4	20.01
Cendres	0.58	4.7	3 359
Humidité	1.58	21.9	9.7
Propriétés physiques			
PCI (KJ/Kg)	34805	15349	29860
DPS *(kg/m ³)	900	657	832.04
DMP* (µm)	15	200	20

*DPS : Densité des particules sèches, DMP : Diamètre moyen des particules.

3. Modélisation CFD de la co-combustion

3.1 Modèle k – ε standard

Le modèle k – ε standard a prouvé durant les années qu'il est un outil d'approche robuste pour la prédiction de la vitesse des écoulements. Ainsi, l'équation de continuité est donnée comme suit.

$$\vec{\nabla} \cdot \vec{v} = S_m \quad (1)$$

La composante de vitesse est donnée par l'équation (2) qui est l'équation de transport pour un régime établi. Ici, P est la pression statique. f est le torseur des contraintes.

$$(\vec{v} \cdot \vec{\nabla}) \vec{v} = -\frac{1}{\rho} \vec{\nabla} p + \nu \Delta \vec{v} + \vec{f} \quad (2)$$

Les équations de transport k et ε sont présentées par l'équation (3) et l'équation (4) [7], [8] :

$$\rho(\vec{U} \cdot \vec{\nabla}) K = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla K \right] + P_k - \rho \varepsilon \quad (3)$$

$$\rho(\vec{U} \cdot \vec{\nabla}) \varepsilon = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + C_{\varepsilon 1} \frac{\varepsilon}{K} P_k - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{K} \quad (4)$$

3.2 Modèle de dissipation tourbillonnaire modifié (FR/ED)

Le modèle FR/ED établit un rapport entre la vitesse de réaction et le taux de dissipation des structures tourbillonnaires présentes dans les produits et dans les réactifs [9],[10]. Alors, dans ce modèle trois vitesses de réaction sont calculées et la plus petite sera retenue. La première vitesse de réaction est calculée à l'aide de l'équation d'Arrhenius (Equation (5)),

$$R'_{i,r} = A_r T \exp\left(\frac{-E}{RT_g}\right) \quad (5)$$

Ar et E sont les paramètres du taux de réaction, R et T sont respectivement la constante des gaz parfaits et la température des gaz. La deuxième et la troisième vitesse de réaction représentent le taux de dissipation des structures tourbillonnaires des réactifs et des produits, ils sont calculés à partir de l'équation (6).

$$R_{i,r} = v'_{i,r} M_{w,i} A \rho \frac{\varepsilon}{k} \min\left(\min_R \frac{Y_R}{v'_{R,r} M_{w,R}}, B \frac{\sum_P Y_P}{\sum_j v'_{j,r} M_{w,j}}\right) \quad (6)$$

R et P représentent respectivement les réactifs et l'indice des produits. M est la masse moléculaire, v est le coefficient stœchiométrique et Y est la fraction de masse. A et B sont les taux de mélange. En cas de la co-combustion, A et B prennent respectivement une valeur de 0,4 et 5.

4. Conditions opératoires d'entrée

Le tableau 2 représente les conditions d'entrée pour le cas de co-combustion et de combustion. Plus de détails sur la géométrie, stratégie de calcul et condition aux limites sont disponibles sur le travail antérieur des auteurs [11].

Tableau 2 : Les conditions d'entrées pour les différents combustibles

	Co-combustion		Combustion	
	GO	CA	CP	CP
Débit (Kg/s)	0.5	0.5	0.7	1.2
T (K)	375			
Dmin (m)	1.10 ⁻⁴	5.10 ⁻⁶	5.10 ⁻⁶	
Dmax	9.10 ⁻⁴	3.10 ⁻⁴	1.10 ⁻⁴	
Dmoy	2.10 ⁻⁴	1,5.10 ⁻⁵	1,5.10 ⁻⁵	
ρ (Kg/m ³)	675	832	900	
Cp _s	Fonction polynomiale de température			
Cg	1.4	1.2	1.2	

Cg : Coefficient de gonflement, D : Diamètre.

5. Résultat et discussion

Le taux de substitution utilisée dans cette étude est à peu près 20% basé sur les thermies, en utilisant deux types de biomasse différents en co-combustion avec du coke de pétrole pulvérisé dans un four de production de ciment. Le but est d'évaluer l'apport thermique et les résultats obtenus, qui incluent la température et les concentrations en espèces.

Pour évaluer l'apport énergétique de l'utilisation de la biomasse, la prédiction de la température, pour les trois cas, est présentée sur la Figure 1.

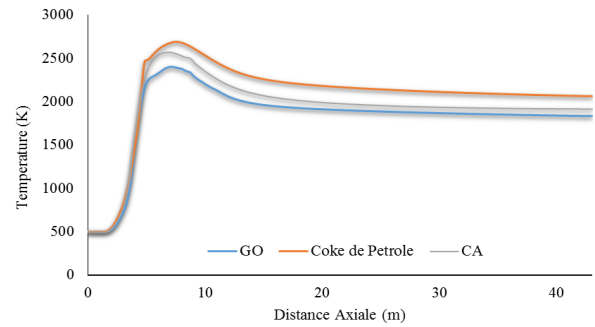


Figure 1 : profil de température suivant la direction axiale du four pour les trois types de combustibles étudié.

La température maximale atteinte dans le cas de substitution par coquille d'amande est de 200K élevée par rapport au cas de substitution par les grignons d'olives. Et est moins faible comparée au cas de combustion du coke seul. Une distribution de la fraction massique des substances volatiles des différents combustibles est donnée par la Figure 2.

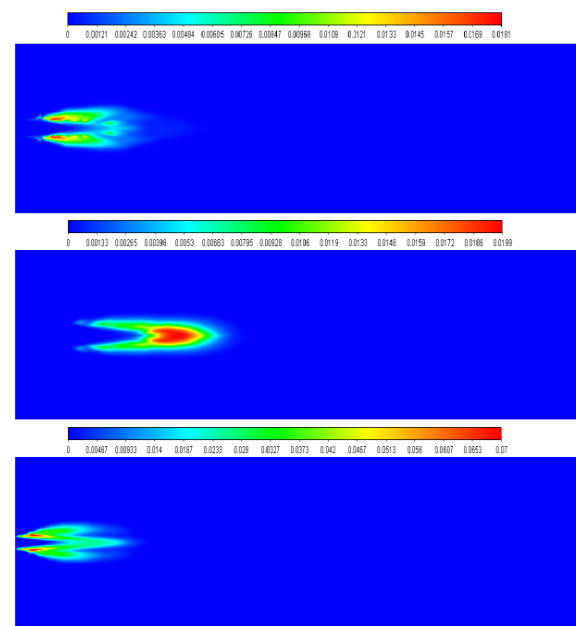


Figure 2 : distribution de la fraction massique des substances volatiles de haut en bas : CP, GO et CA

La figure 2 montre de haut en bas la distribution de la fraction massique des substances volatile du coke, GO et CA. Les particules de coquilles d'amande se dévolatiliser rapidement dès leur sortie du brûleur (Figure 2). Par contre les particules de grignons d'olives, se dévolatiliser plus loin de la sortie du brûleur.

Les figures 3 et 4 montrent la fraction en masse de char de cinq particules aléatoires des grignons d'olives et des coquilles d'amandes et du coke, respectivement.

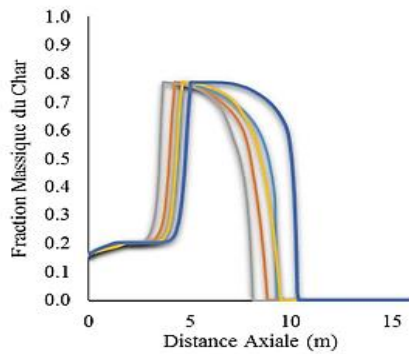


Figure 3 : Fraction massique de production du char des particules des grignons d'olives

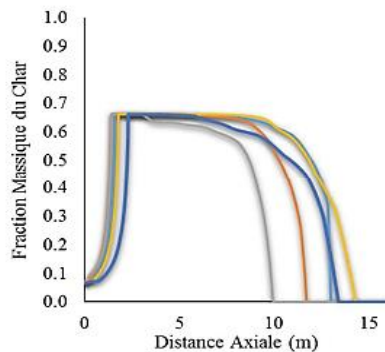


Figure 4 : Fraction massique de production du char des particules de coquille d'amandes

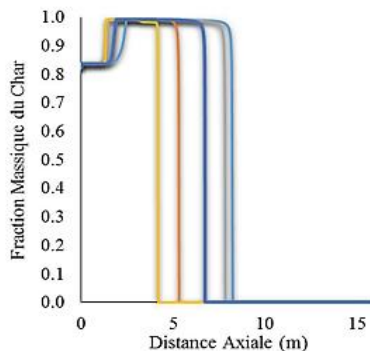


Figure 5 : Fraction massique de production du char des particules de coke de pétrole

Le trajet des particules de char pour le coke et les coquilles d'amandes se termine respectivement dans les alentours de 12m et 14m plus loin par rapport aux particules de char des grignons d'olives pour lesquels, on remarque que leurs trajectoires se terminent plus tôt à environ 9m (Figures 3, 4 et 5). Cela est dû à la forte teneur en carbone fixe du coke et des coquilles d'amandes par rapport aux grignons d'olives et aussi à la forme plus grande des particules de GO

6. Conclusion

Cette étude montre l'impact de l'utilisation des deux combustibles alternatifs, les grignons d'olives et les coquilles d'amande, en co-combustion avec le coke de pétrole sur la performance de la combustion dans les fours des cimenteries, pour ce but nous utilisons le modèle K- ϵ standard couplé au modèle FR/ED.

Les résultats de co-combustion ont été comparés avec les résultats de la combustion du coke seul. La température des produits de la co-combustion est inférieure à celle de

la température de la combustion du coke seul. La température plus basse est principalement due à la demande d'air en excès, aux teneurs en cendres et en humidité plus élevée des grignons d'olives et des coquilles d'amandes par rapport au Coke. Les particules de biomasse sont en général plus grandes et différent dans leur composition physique et chimique des particules du combustible fossile ce qui explique la différence de comportement présenté en char. La co-combustion de la biomasse reste avantageuse toutefois, une configuration détaillée doit correspondre à un bon compromis entre les fractions de masse des biomasses et les performances du brûleur.

7. Références

- [1] F. Schorcht, I. Kourti, B. M. Scalet, S. Roudier, and L. D. Sancho, "Best Available Techniques (BAT) Reference Document for the Production of Cement, Lime and Magnesium Oxide," European commission, 2013.
- [2] D. N. Huntzinger and T. D. Eatmon, "A life-cycle assessment of Portland cement manufacturing: comparing the traditional process with alternative technologies," *Journal of Cleaner Production*, vol. 17, no. 7, pp. 668–675, May 2009.
- [3] A. Aranda Usón, A. M. López-Sabirón, G. Ferreira, and E. Llera Sastresa, "Uses of alternative fuels and raw materials in the cement industry as sustainable waste management options," *Renewable and Sustainable Energy Reviews*, vol. 23, pp. 242–260, Jul. 2013.
- [4] K. G. Kolovos, G. Kyriakopoulos, and M. S. Chalikias, "Co-evaluation of basic woodfuel types used as alternative heating sources to existing energy network," *Journal of Environmental Protection and Ecology*, vol. 12, no. 2, pp. 733–742, 2011.
- [5] K. Papanikola *et al.*, "Food residue biomass product as an alternative fuel for the cement industry," *Environmental Science and Pollution Research*, vol. 26, no. 35, pp. 35555–35564, Dec. 2019.
- [6] P. E. Tsakiridis, M. Samouhos, and M. Perraki, "Valorization of Dried Olive Pomace as an alternative fuel resource in cement clinkerization," *Construction and Building Materials*, vol. 153, pp. 202–210, 2017.
- [7] B. E. Launder and D. B. Spalding, "The numerical computation of turbulent flows," *Computer Methods in Applied Mechanics and Engineering*, vol. 3, no. 2, pp. 269–289, 1974.
- [8] W. Jones and B. Launder, "The prediction of laminarization with a two-equation model of turbulence," *International Journal of Heat and Mass Transfer*, vol. 15, no. 2, pp. 301–314, Feb. 1972.
- [9] D. B. Spalding, "Mixing and chemical reaction in steady confined turbulent flames," *Symposium (International) on Combustion*, vol. 13, no. 1, pp. 649–657, Jan. 1971.
- [10] B. F. Magnussen and Bjørn F. Magnussen, "THE EDDY DISSIPATION CONCEPT A BRIDGE BETWEEN SCIENCE AND TECHNOLOGY," lisbone, 2005.
- [11] Z. Ngadi and M. L. Lahlaoui, "Coal and Biomass Co-Combustion: CFD Prediction of Velocity Field for Multi-Channel Burner in Cement Rotary Kiln," *Proceedings*, vol. 63, no. 1, p. 18, 2020.

L'impact de l'amélioration des performances des modules thermoélectriques sur son rendement

S. JENNAH¹, N. BELOUAGGADIA², R.LBIBB³, M.EZZINE⁴

1. Laboratory of Engineering and Materials (LIMAT),
Faculty of Sciences Ben M'Sik, Hassan II University of Casablanca
Casablanca, Morocco
jennahsara@gmail.com

2. Laboratory of Signals, Distributed Systems and Artificial Intelligence
ENSET, Hassan II University
Mohammedia, Morocco
belouaggadia@enset-media.ac.ma

3. Laboratory of Sustainable Development
Faculty of Sciences and Technics, University of Sultan Moulay Slimane
Beni mellal, Morocco
rachid.lbibb@gmail.com

4. Laboratory of Physico-chemistry of Applied Materials (LPCMA)
Faculty of Sciences Ben M'Sik, Hassan II University of Casablanca
Casablanca, Morocco
mohammed.ezzine@gmail.com

Résumé

L'amélioration du rendement des modules thermoélectriques nécessite l'amélioration de ces performances, alors une méthode d'optimisation sera étudiée dans cet article et cela par la gestion des conditions de fonctionnement. Notre objectif est la régulation de la température des soudures froides des thermocouples. Dans un premier cas les thermocouples seront parcourus par un fluide caloporteur chaud, qui est l'eau, circulé dans un canal en contact avec les soudures chaudes. Par la suite, une deuxième étude sera faite qui va être basée sur le refroidissement des soudures froides en utilisant le même fluide caloporteur intégré avec une température plus basse dans le canal qui est en contact avec les soudures froides. Les résultats obtenus révèlent qu'en ajoutant le fluide caloporteur froid la différence de température devient plus remarquable, ce qui va nous permettre d'avoir un rendement meilleur.

Mots clefs: *Thermoélectrique, Fluide caloporteur, Soudures chaudes, Soudures froides.*

1. Introduction

Les pompes à chaleur thermoélectriques sont des dispositifs qui exploitent la différence de température créée par l'effet Peltier entre les jonctions des semi-conducteurs. Les phénomènes thermoélectriques [1-3] ont

été observés et décrits au cours de la première moitié du 19ème siècle. En 1885, Rayleigh propose de créer un générateur thermoélectrique [4], mais la première application réelle n'apparaît qu'en 1947 avec le générateur thermoélectrique de Telkes [5]. Entre 1957 et 1965, de nombreux matériaux thermoélectriques ont été découverts, notamment le tellure de bismuth, qui présente le meilleur facteur de mérite à température ambiante [6]. Dans une pompe à chaleur thermoélectrique, les deux caractéristiques importantes sont la qualité du matériau utilisé et les résistances thermiques du dispositif. De nombreuses études ont été menées sur l'optimisation du dispositif thermoélectrique ; cependant, la plupart d'entre elles ne traitent qu'une partie du dispositif, comme les éléments thermoélectriques [7-9], la résistance thermique de contact [7], ou l'influence des échangeurs de chaleur [10,11].

2. Configuration

Le générateur thermoélectrique convertit l'énergie thermique stockée dans un fluide de transfert de chaleur en énergie électrique. Un fluide caloporteur chaud chauffe les soudures chaudes de la thermopile, et un fluide caloporteur froid refroidit les soudures froides par un écoulement supposé constant dans des tubes rectangulaires de largeur L , de longueur L_1 et de hauteur L_2 . L'interface cellule thermique/fluide de transfert de chaleur est supposée être une couche mince de résistance thermique négligeable et électriquement isolante.

Cas où la température des soudures froides est maintenue constante (Fig1 A). Dans ce cas, la température des soudures froides est maintenue constante, et un fluide caloporteur chaud réchauffe les soudures chaudes.

Et pour la deuxième configuration (Fig1 B), les soudures froides sont refroidies par un flux de fluide caloporteur froid.

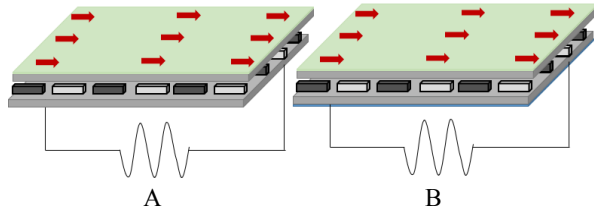


Fig 1. A) cas de température des soudures froides maintenues constante ; B) cas de refroidissement des soudures froides avec un fluide caloporteur.

3. Modèle mathématique

Le rendement d'un générateur thermoélectrique (TEG) est déterminé comme rapport entre la puissance utile et le flux de chaleur absorbé à l'extrémité chaude

$$\eta = \frac{P_u}{Q_h} \quad (1)$$

Tel que η , P_u , Q_h sont respectivement le rendement de conversion de la chaleur en électricité, la puissance utile et le flux de chaleur absorbé à l'extrémité chaude.

La puissance utile P_u délivrée par un générateur thermoélectrique (GTE) correspond à celle produite par l'effet Seebeck moins la puissance dissipée par effet Joule, elle est donnée par

$$P_u = n[I(S_P - S_N)(T_h - T_c) - RI^2] \quad (2)$$

Avec S_P et S_N se sont les coefficients Seebeck du matériau thermoélectrique pour la jonction P et la jonction N , T_h et T_c sont les températures de la soudure chaude et froide, I est le courant électrique, n est le nombre de thermocouples et R c'est la résistivité électrique du générateur thermoélectrique.

Les flux de chaleur sur les extrémités froide et chaude du thermocouple sont donnés par :

$$Q_h = n \left[I(S_P - S_N)T_h + K(T_h - T_c) - \frac{RI^2}{2} \right] \quad (3)$$

$$Q_c = n \left[I(S_P - S_N)T_c + K(T_h - T_c) + \frac{RI^2}{2} \right] \quad (4)$$

D'où Q_c est le flux de chaleur évacué à l'extrémité froide et K est la conductibilité thermique du générateur thermoélectrique.

Quand le générateur thermoélectrique alimente une charge, le courant délivré est :

$$I = \frac{n(S_P - S_N)(T_h - T_c)}{R_E + nR} \quad (5)$$

La puissance électrique atteint sa valeur maximale quand la résistance de la charge est égale à la résistance interne du générateur $R_E = nR$, le courant devient alors :

$$I = \frac{(S_P - S_N)(T_h - T_c)}{2R} \quad (6)$$

Tel que :

$$P_{max} = \frac{n(S_P - S_N)^2(T_h - T_c)^2}{4R} \quad (7)$$

La conductibilité thermique K du générateur thermoélectrique peut être exprimée comme suit :

$$K = \frac{A(k_P + k_N)}{2L_{th}} \quad (8)$$

Tel que, A et L_{th} se sont la surface et l'épaisseur de la thermopile, k_P est la conductibilité thermique de la jonction P et k_N est la conductibilité thermique de la jonction N .

La résistivité électrique R du générateur thermoélectrique, peut être décrite selon la géométrie et les propriétés électriques du générateur thermoélectrique :

$$R = \frac{2L_{th}(\rho_P + \rho_N)}{A} \quad (9)$$

Tel que, ρ_P est la résistivité électrique de la jonction P , et ρ_N est la résistivité électrique de la jonction N .

Le transfert de chaleur entre le fluide caloporteur et la thermopile se fait par convection avec un coefficient de convection h_f lié au nombre de Nusselt Nu_f par :

$$h_f = \frac{Nu_f K_f (L + L_2)}{2LL_2} \quad (10)$$

D'où, L et L_2 sont respectivement la largeur et la hauteur du tube circulaire du fluide, et λ_f est la conductibilité thermique du fluide caloporteur.

La corrélation entre les nombres de Nusselt Nu_f [11], Reynolds Re_f et Prandtl Pr_f est donnée par :

$$Nu_f = 4.4 + \frac{0.00398 \left(\frac{2\rho v L L_2}{\mu_f (L + L_2)} \times \frac{c_f \mu_f (L_2)}{K_f (L)} \right)^{1.66}}{1 + 0.0114 \left(\frac{2\rho v L L_2}{\mu_f (L + L_2)} \times \frac{c_f \mu_f (L_2)}{K_f (L)} \right)^{1.12}} \quad (11)$$

Tel que, μ_f , c_f , ρ et v sont respectivement la viscosité dynamique, la capacité, la densité et la vitesse du fluide caloporteur.

TABLEAU 1: PROPRIÉTÉS DES MATÉRIAUX UTILISÉS DANS LE TEG [12-16]:

Parameters	Values	Units
L	1	cm
L_2	1	cm
c_f	4185	J/Kg.k
K_f	0.636	W/m.K
μ_f	0.0005	Kg/m.s
A	2	mm ²
L_{th}	5	mm
ρ_N	E-5	Ωm
ρ_P	0.8E-5	Ωm
k_N	1.4	W/m.K
k_P	1.4	W/m.K
S_N	-200	$\mu V/K$
S_P	180	$\mu V/K$

4. Résultats et discussion

Pour étudier l'effet du débit sur les performances d'un GTE, nous avons calculé la puissance utile d'une thermopile constituée de thermocouples pour différents débits du fluide caloporteur. La figure 2 présente le résultat pour une thermopile avec 2500 thermocouples. Comme le montre la figure, la puissance utile augmente avec le débit. En effet, outre le fait que l'augmentation du débit entraîne un apport de chaleur important dans la thermopile, elle implique également un régime d'écoulement turbulent et une augmentation du coefficient de transfert, ce qui va favoriser le transfert de chaleur vers

les soudures chaudes et l'augmentation de leurs températures. Cependant, on remarque que la pente de variation de la puissance utile diminue en fonction du débit, ce qui nous amène à rechercher le débit optimal pour réduire les dépenses énergétiques liées à l'écoulement du fluide caloporteur. Par exemple, la puissance utile générée dans notre cas est en fonction du débit. Lorsque le débit est doublé, le gain de puissance n'est que de 26%. Dans ce cas, il est conseillé de régler le débit et d'utiliser une deuxième thermopile ; la puissance générée serait de deux fois la puissance obtenue.

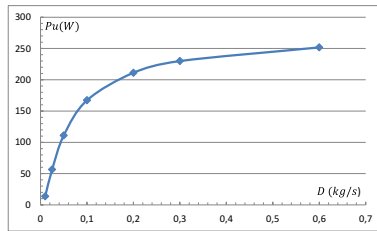


Fig 2. Variation de la puissance en fonction du débit

Fig 3 montrent l'évolution des températures des fluides caloporteurs et des soudures chaudes et froides des thermocouples avec un débit du fluide caloporteur chaud $\dot{Q}_h = 0,1 \text{ kg/s}$ d'une température de $T_h = 353 \text{ K}$ et avec un débit du fluide caloporteur froid $\dot{Q}_c = 0,2 \text{ kg/s}$ d'une température de $T_c = 300 \text{ K}$.

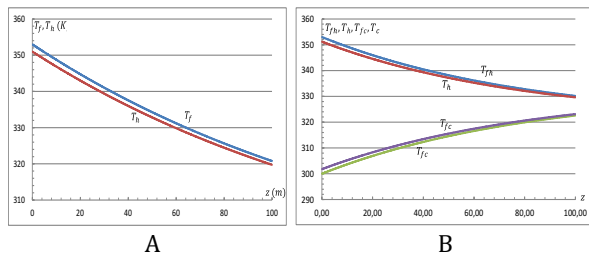


Fig 3. Variation de T_{f_h} et T_h en fonction de z ; A) : cas de température des soudures froides maintenue constante ; B) : Refroidissement des soudures froides avec un fluide caloporteur.

Nous pouvons voir que dans le deuxième cas, nous obtenons de meilleurs résultats que le premier, la température du fluide chaud dans le premier cas atteint 320 K contrairement au deuxième cas, nous avons une augmentation de 10 K, et elle atteint 330 K, et nous avons obtenu une puissance générée de 112,7 W avec une efficacité de 1,2% qui est plus élevée que le premier cas.

5. Conclusion

Bien que les phénomènes thermoélectriques ont été largement utilisés pour les applications de chauffage et de refroidissement, la production d'énergie a eu un marché minimal dans des applications spécialisées. Ces dernières années, l'intérêt s'est accru pour de nouvelles applications de production d'énergie par récolte thermoélectrique. L'utilisation généralisée des TEG dépend de leur optimisation.

Les résultats montrent que le refroidissement des soudures froides par un fluide froid permet d'avoir un niveau d'énergie important du fluide caloporteur chaud, qui peut

être exploité ou réinjecté dans une autre utilisation. On a trouvé aussi que le refroidissement des soudures froides permet d'avoir une amélioration de l'écart de température entre les soudures qui va permettre une génération d'énergie électrique plus importante, avec un rendement plus élevé qui est de 1,2%.

Références

- [1] T. J. Seebeck, "Magnetische Polarisation der Metalle und Erze durch Temperatur-Differenz. Abh. Akad. Wiss." (1822): 289-346.
- [2] W. Thomson. "4. on a mechanical theory of thermo-electric currents." Proceedings of the Royal society of Edinburgh 3 (1857): 91-98.
- [3] Peltier, J. C. A. Nouvelles expériences sur la calorificité des courans électriques. (1834).
- [4] Stockholm, J. G. Génération thermoélectrique. Actes des Journées Electrotechniques du Club EEA-Energie Portable : autonomie et intégration dans l'environnement humain. (2002).
- [5] Telkes, M. Solar thermoelectric generators. Journal of Applied Physics, (1954) : 25(6), 765-777.
- [6] Vining, C. B. Semiconductors are cool. Nature. (2001) : 413(6856), 577-578.
- [7] Min, G., & Rowe, D. M. Improved model for calculating the coefficient of performance of a Peltier module. Energy conversion and management. (2000) : 41(2), 163-171.
- [8] Lee, K. H., & Kim, O. J. Analysis on the cooling performance of the thermoelectric micro-cooler. International Journal of Heat and Mass Transfer, (2007) : 50(9-10), 1982-1992.
- [9] Omer, S. A., & Infield, D. G. Design optimization of thermoelectric devices for solar power generation. Solar Energy Materials and Solar Cells. (1998) : 53(1-2), 67-82.
- [10] Chein, R., & Chen, Y. Performances of thermoelectric cooler integrated with micro-channel heat sinks. International journal of refrigeration, (2005) : 28(6), 828-839.
- [11] Duffie, J. A., Beckman, W. A., & Blair, N. Solar engineering of thermal processes, photovoltaics and wind. John Wiley & Sons. (2020).
- [12] Rosengarten, G., Mutzenich, S., & Kalantar-Zadeh, K. Integrated microthermoelectric cooler for microfluidic channels. Experimental Thermal and Fluid Science. (2006) : 30(8), 821-828.
- [13] Kikas, N. P. Laminar flow distribution in solar systems. Solar Energy, (1995) : 54(4), 209-217.
- [14] Tondeur, D., Fan, Y., & Luo, L. Flow distribution and pressure drop in 2D meshed channel circuits. Chemical engineering science. (2011) : 66(1), 15-26.
- [15] Shah, R. K. London AL Laminar Flow Forced Convection in Ducts. Advances in Heat Transfer. (1978) : 168-169.
- [16] Ritzer, T. M., & Lau, P. G. Economic optimization of heat sink design. In AIP Conference Proceedings. American Institute of Physics. (1994, August) : (Vol. 316, No. 1, pp. 177-180).

Solution analytique et numérique de l'écoulement potentiel stationnaire autour d'un cylindre chauffé avec une densité de flux de chaleur uniforme

M. Darif*, K. Zniber, A. Oubarra, J. Lahjomri

Université Hassan II, Faculté des sciences Ain Chock, Laboratoire de Mécanique,
Km 8 route d'El Jadida BP 5366 Maarif, Casablanca, Maroc

*Email : musdraf82@gmail.com

Abstract

The problem of the steady state of forced convection of potential flow around a heated cylinder with uniform heat flux density is solved both analytically and numerically. The analytical solution is obtained by using the Boussinesq transformation $X = \phi/2$ and $Y = \Psi/2$ considering (X, Y) as coordinates, where ϕ is the potential velocity and Ψ the stream function. This transformation allows to remove the velocity from the energy equation and to transform the problem into the problem of forced convection over a finite flat plate of length 2 in the (X, Y) -plane. The temperature field in the real space is determined via the application of Green's formula and consequently the heat exchange law, characterized by the Nusselt number. To validate the analytical solution, the problem is also solved numerically by obtaining accurate solution with a finite difference method.

Keywords: *potential flow; forced convection; Neumann boundary; finite difference method.*

Résumé

Le problème de la convection forcée de l'écoulement potentiel stationnaire autour d'un cylindre chauffé avec une densité de flux de chaleur uniforme est résolu à la fois analytiquement et numériquement. La solution analytique est obtenue en utilisant la transformation de Boussinesq $X = \phi/2$ et $Y = \Psi/2$ en considérant (X, Y) comme coordonnées, où ϕ est le potentiel de vitesse et Ψ la fonction de courant. Cette transformation permet de supprimer la vitesse de l'équation de l'énergie et de transformer le problème en un problème de convection forcée sur une plaque plane finie de longueur 2 dans le plan (X, Y) . Le champ de température dans l'espace réel est déterminé à partir de l'application de la formulation de Green afin de déduire la loi d'échange thermique caractérisée par le nombre de Nusselt. Afin de valider la solution analytique, le problème est également résolu numériquement en obtenant une solution précise avec une méthode de différences finies.

Mots clefs : *écoulement potentiel ; convection forcée ; conditions de Neumann ; différences finies.*

I-Introduction :

Cette étude est consacrée à la détermination de la solution analytique et numérique du problème non trivial de la convection/diffusion de la chaleur en régime permanent

(ou de masse par analogie), par l'écoulement potentiel autour d'un cylindre chauffé avec un flux constant ou soumis un gradient de concentration. Ce problème classique en transfert de masse a été largement étudié dans la théorie de la solidification à partir d'une masse fondue par l'écoulement potentiel, et il se pose également dans l'agrégation limitée par advection-diffusion [2,3]. Le problème thermique a été étudié pour la première fois par King [1] dans le cas des conditions aux limites de Dirichlet (CLD) (température uniforme imposée autour d'un cylindre) dans le cas simplifié d'un champ de vitesse uniforme. Le but de l'étude de King [1] a été de mesurer la vitesse de l'écoulement à partir de l'étude de l'échange de chaleur entre un fil cylindrique chauffé et le fluide ambiant froid (anémomètre à fil chaud). La théorie considérant un écoulement potentiel avec (CLD) est plus compliquée et montre que le problème se ramène à la résolution de l'équation intégrale de Fredholm de première espèce [2] :

$$\int_{-1}^1 \exp[Pe(X - \xi)] K_0[Pe|X - \xi|] q_w(\xi) d\xi = \pi \quad (1)$$

Où Pe est le nombre de Péclet. K_0 est la fonction de Bessel modifiée de seconde espèce d'ordre zéro. Le problème fondamental consiste alors à déterminer la distribution du flux q_w inconnu sur le cylindre satisfaisant l'équation intégrale (1) et par conséquent celle de nombre de Nusselt local. Signalons qu'il n'y a pas de théorie générale pour résoudre l'équation (1) d'une façon explicite. Les solutions asymptotiques de l'équation (1) ont été trouvées avec une très grande précision par Choi et al. [2] pour les petites et grandes valeurs de Pe , en raccordant les deux solutions, pour déduire finalement la distribution du flux q_w en tout point du cylindre. Signalons que dans le cas des conditions aux limites de Neumann qui concerne l'objectif de ce travail, une solution analytique explicite en tout point de l'écoulement et pour toute valeur du nombre de Péclet est possible et peut être obtenue explicitement sous forme d'une intégrale, comme on peut le montrer ici dans cette étude.

Notre travail consiste à établir une solution analytique et numérique du problème de la convection forcée de l'écoulement potentiel stationnaire autour d'un cylindre chauffé avec une densité de flux de chaleur uniforme connue afin de déterminer la loi d'échange de chaleur pour différents régimes d'écoulement. On peut noter que la solution analytique asymptotique du champ de température indique

qu'il existe une singularité à l'infini qui cause beaucoup de difficultés pour la convergence de la solution numérique et ceci indépendamment de la méthode numérique utilisée. Nous avons pu résoudre efficacement cette singularité en procédant à un changement de fonction qui nous a permis d'éliminer ce comportement singulier pour la détermination de la solution numérique. Le système d'équations généré par la discrétisation est résolu par la méthode de Newton-Raphson et la décomposition LU pour calculer le Jacobien de la matrice du système.

II- Formulation mathématique :

II-1 Description du problème :

On considère l'écoulement potentiel bidimensionnel et stationnaire d'un fluide parfait autour d'un cylindre de rayon R avec une longueur suivant son axe x' , supposée très grande devant le rayon du cylindre. Le fluide très loin du cylindre est animé d'un mouvement avec une vitesse caractéristique U_∞ et une température T_∞ uniformes. Le cylindre est chauffé avec une densité de flux uniforme notée q_w .

II-2 Equation de l'énergie et conditions aux limites :

En introduisant les paramètres adimensionnels suivants :

$$\vec{u} = \frac{\vec{v}}{U_\infty} = \vec{\nabla}\phi, \theta = \frac{T - T_\infty}{\frac{q_w R}{k}}, r = \frac{r'}{R}, x = \frac{x'}{R}, y = \frac{y'}{R}$$

$$\phi = \frac{\psi}{U_\infty R} = (r + 1/r)\cos\theta, \Psi = \frac{\psi}{RU_\infty} = (r - 1/r)\sin\theta \quad (2)$$

L'équation de l'énergie devient :

$$(\vec{u} \cdot \vec{\nabla})\theta = \frac{1}{Pe} \nabla^2 \theta \quad (3)$$

Avec $Pe = U_\infty R / \alpha$ le nombre de Péclet (α la diffusivité thermique du fluide). Les conditions aux limites associées au problème sont données par :

$$\begin{cases} \theta = 0 & \begin{cases} r \rightarrow \infty \\ 0 < \theta < \pi \end{cases} \\ \frac{\partial \theta}{\partial r} = -1 & \begin{cases} r = 1 \\ 0 \leq \theta \leq \pi \end{cases} \\ \frac{\partial \theta}{\partial \theta} = 0 & \begin{cases} r > R \\ \theta = \pi \text{ et } \theta = 0 \end{cases} \end{cases} \quad (4)$$

II-3 Solution Analytique :

L'utilisation de la transformation de Boussinesq : $X = \phi/2$ et $Y = \Psi/2$, permet d'éliminer dans (3), le champ de vitesse du fluide en la transformant en une équation de type Oseen :

$$\frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2} - 2Pe \frac{\partial \theta}{\partial X} = 0 \quad \begin{cases} -\infty < X < +\infty \\ Y > 0 \end{cases} \quad (5)$$

En procédant à un changement de fonction

$$\Theta(X, Y) = e^{PeX} T(X, Y) \quad (6)$$

Nous pouvons éliminer aussi le terme de la dérivée première dans l'équation d'Oseen (5) qui peut être transformée en équation de Helmholtz dans le plan semi infini ($Y > 0$) :

$$\frac{\partial^2 T}{\partial X^2} + \frac{\partial^2 T}{\partial Y^2} - Pe^2 T = 0 \quad \begin{cases} -\infty < X < +\infty \\ Y > 0 \end{cases} \quad (7)$$

Les conditions aux limites associées à la fonction Ψ sont données par :

$$\begin{cases} T \rightarrow 0 & X^2 + Y^2 \rightarrow +\infty \\ \frac{\partial T}{\partial Y} = \frac{-e^{-PeX}}{\sqrt{1-X^2}} & \begin{cases} -1 < X < +1 \\ Y = 0 \end{cases} \\ \frac{\partial T}{\partial Y} = 0 & \begin{cases} |X| \geq +1 \\ Y = 0^+ \end{cases} \end{cases} \quad (8)$$

La fonction de Green [4] associée à l'équation de Helmholtz (7) dans le plan semi infini obéit à l'équation (3) :

$$\nabla^2 G - Pe^2 G = \delta(\vec{X} - \vec{X}_0) =$$

$$-\delta(X - \xi)\delta(Y - \eta) \quad \begin{cases} -\infty < X, \xi < +\infty \\ 0 < Y, \eta < +\infty \end{cases} \quad (9)$$

δ désigne la fonction de Dirac. L'application de la transformée de Fourier à (9) et par inversion et après quelques manipulations algébriques nous pouvons établir l'expression de la fonction de Green G comme :

$$G(X, Y; \xi, \eta) = \frac{1}{2\pi} \{K_0 [Pe\sqrt{(X - \xi)^2 + (\eta - Y)^2}] + K_0 [Pe\sqrt{(X - \xi)^2 + (\eta + Y)^2}]\} \quad (10)$$

Pour résoudre l'équation de Helmholtz (7), avec les conditions aux limites (8), nous appliquons la formule de Green sur un demi-cercle C qui est limité la surface fermée S :

$$\iint_S ((T\nabla^2 G - G\nabla^2 T) dS = \oint (T\vec{\nabla}G - G\vec{\nabla}T)\vec{n} dl =$$

$$\int_{-\infty}^{+\infty} [T \frac{\partial G}{\partial Y} - G \frac{\partial T}{\partial Y}]_{Y=0} dX \quad (11)$$

En substituant les équations (7) et (9) dans l'équation (11), et en utilisant la symétrie de la fonction de Green (10), et en inversant les rôles des variables (X, Y) et (ξ, η), on obtient la fonction $T(X, Y)$ sous forme d'une intégrale s'écrivant en fonction de la fonction de Green :

$$T(X, Y) = - \int_{-\infty}^{+\infty} G(X, Y; \xi, 0^+) \frac{\partial T}{\partial \eta} \Big|_{\eta=0^+} d\xi \quad (12)$$

En reportant l'expression de G donnée par (10) dans (12) et en utilisant les conditions aux limites (8) sur la fonction T , l'équation (12) devient :

$$T(X, Y) = - \frac{1}{\pi} \int_{-1}^{+1} e^{Pe\xi} q_w(\xi) K_0 [Pe\sqrt{(X - \xi)^2 + Y^2}] d\xi \quad (13)$$

$$q_w(\xi) = \frac{1}{\sqrt{1-\xi^2}} = - \frac{\partial \theta}{\partial \eta} \Big|_{\eta=0^+} = -e^{Pe\xi} \frac{\partial T}{\partial \eta} \Big|_{\eta=0^+} \quad (14)$$

Tenant compte des équations (6), (13) et (14), la solution du problème donnant le champ de température Θ en tout point

du fluide et quelle que soit la valeur du nombre de Péclet peut prendre la forme suivante sous forme d'une intégrale insoluble :

$$\theta(X, Y) = \frac{1}{\pi} \int_{-1}^{+1} e^{Pe(X-\xi)} \frac{1}{\sqrt{1-\xi^2}} K_0 \left[Pe \sqrt{(X-\xi)^2 + Y^2} \right] d\xi \tag{15}$$

III- Résultats et discussion :

En plus de la solution analytique, l'équation de l'énergie (3) est résolue également numériquement par une méthode de différence finie en utilisant un schéma centré d'ordre deux avec pas variables dans les deux directions. Les conditions aux limites sont aussi discrétisées à l'ordre deux. Le système d'équation généré par la discrétisation est résolu par la méthode de Newton-Raphson en utilisant la décomposition LU pour calculer le Jacobien de la matrice du système. Tous les résultats obtenus par la méthode numérique sont calculés avec un maillage (160 × 160). Sur les Figures 1 (a), (b) et (c) nous avons représenté les isothermes pour trois régimes de l'écoulement en fonction du nombre de Péclet. On observe qu'au fur et à mesure que Pe augmente, il apparaît un croisement et une compétition des effets de conduction et les effets de la convection, partant d'un régime de conduction dominant, i.e., Pe = 0.05, Fig1. (a) où la perturbation thermique se propage loin du cylindre sous forme de cercles concentriques dans toutes les directions, et d'une convection dominante Pe = 100, Fig1. (c) où les gradients de température sont confinés dans des couches thermiques minces, qui se forment à partir du point d'arrêt amont. Ces couches minces donnent naissance à un sillage thermique dans lequel les bouffées de chaleur sont convectées à l'aval du cylindre. Sur la Figure 2, nous avons illustré la variation du nombre de Nusselt local Nu(θ) en fonction de Pe en comparant les résultats de la solution analytique et numérique. Il est clair que les résultats obtenus par les deux méthodes sont en excellent accords, l'erreur relative entre les deux solutions est inférieure à 0.5%, ce qui constitue une validation des solutions obtenues par les deux méthodes.

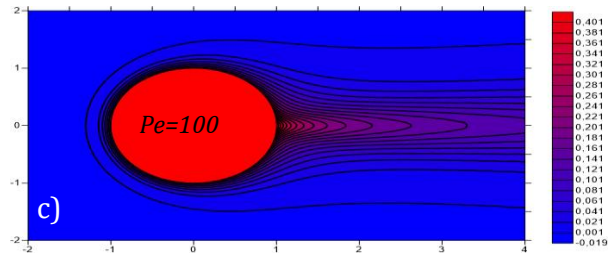
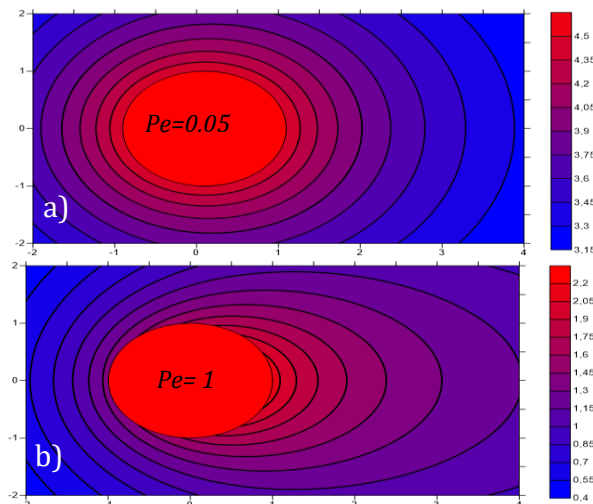


Figure 1 : Profils des isothermes.

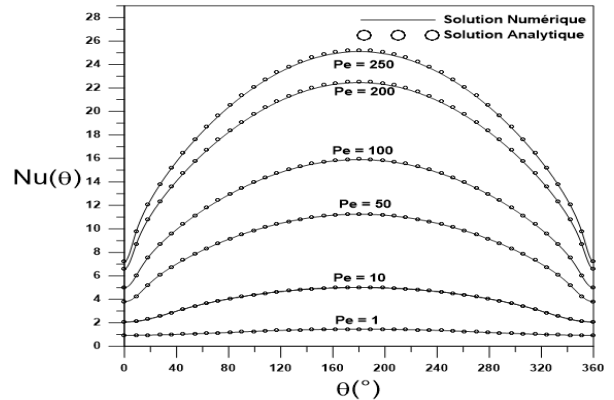


Figure 2 : Le nombre de Nusselt local pour différentes valeurs du nombre de Péclet.

IV -Conclusion

Le problème de la convection forcée de l'écoulement potentiel stationnaire autour d'un cylindre chauffé avec une densité de flux de chaleur uniforme est résolu à la fois analytiquement et numériquement. La loi d'échange de chaleur pour différents régimes d'écoulement a été déterminée en fonction du nombre de Péclet. Les résultats mettent clairement en évidence la compétition des effets de la conduction et la convection.

Les résultats obtenus par la solution analytique et numérique sont en excellent accords, l'erreur relative entre les deux solutions est inférieure à 0.5%, ce qui constitue une validation des solutions obtenues par les deux méthodes.

Références.

- [1] L.V. King, on the convection of heat from small cylinders in a stream of fluid: determination of the convection constants of small platinum wires with application to hot-wire anemometry, Philos. Trans. R. Soc. London A 214 (1914) 373–432.
- [2] J. Choi, D. Margetis T. M. Squires and M. Z. Bazant, "Steady advection-diffusion around finite absorbers in two-dimensional potential flows". Journal Fluid of Mechanic, vol. 536, pages 155-184, (2005).
- [3] A. haji-sheikh, E. Donald and J.V. beck "Axial heat conduction in moving semi-infinite fluid" International Journal of Heat and Mass transfer 2008
- [4] Andrei D. Polyanin. Handbook of liner partial differential equations for scientist vol.667, pages 387

Numerical simulation of Natural convection flow inside a Cavity filled with Alumina Al_2O_3 Nanofluid

Kamal BOUZID¹, Lahoucine BELARCHE¹, Btissam ABOURIDA², Ali SIADI¹,

¹MMGC, National School of Applied Sciences, Ibn Zohr University, BP 1136, Morocco

²LMPEE, Sciences Faculty, Cadi Ayyad University, B.P. 2390, Morocco

kamalbouzid9@gmail.com

Abstract

Natural convection of Nanofluid flow in two-dimensional cavity is numerically simulated. The configuration consists of a cavity that is filled with Al_2O_3 -water Nanofluid possessing a central heat-conducting solid block (T_h). An isothermal condition is established in the two vertical walls (T_c), while the remaining part remains in the adiabatic conditions. Empirical correlations are employed for the thermal conductivity and dynamic viscosity of the nanofluid. The Rayleigh number ($10^2 \leq Ra \leq 10^6$) and nanoparticles volume fraction ($0 \leq \phi \leq 4\%$) govern the parameters in this study. Based on the obtained results, it was found that the heat transfer inside the cavity is enhanced by introducing nanoparticles as well as increasing Rayleigh number.

Keywords: Numerical simulation; Nanofluid; 2D Natural convection.

1. Introduction

Convection heat transfer in cavities has been widely used in many industrial equipment such as electronic cooling system. The temperature control of the internal components is an important factor in the design, application and manufacture. The produced heat can be carried out in the form of free convection. Various electronic cooling systems includes microprocessors, integrated circuits (ICs), heat generating components of laptops, telecommunications equipments and many electronic devices that are trapped in the enclosure space.

M. Mahmoodi, S.M. Sebdani [1] have studied the Natural convection in a square cavity containing nanofluid and an adiabatic square block at the center. They detected that for all Rayleigh numbers with the exception of $Ra = 10^4$ the average Nusselt number increases with increase in the volume fraction of the nanoparticles. Sannad et al [2] have studied numerically the Effect of the heating block position on Natural Convection in a Three-Dimensional Cavity filled with nanofluids. The obtained results showed that: the increase of Rayleigh number leads to a large

improvement of heat transfer evacuated across the cavity. Similarly, the increase of the volume fraction causes an enhancement of the flow and thermal exchanges and the optimum configuration, from the thermal and dynamical point of view, is the one with the heat source located on the upper wall

To date, there are few studies that have investigated the natural convection flow of 2D cavity with central heat bloc filled with nanofluids. This study therefore set out to assess a numerical simulation of the effect of Rayleigh number and volume fractions on temperature distribution and heat transfer performance. The Navier-Stokes and energy equations are discretized using the finite volume method and solved numerically using the SIMPLEC Algorithm for different Rayleigh number.

2. Analysis And Modelling

2.1. Problem description

The natural convection in 2D cavity using Alumina (Al_2O_3) nanofluid are investigated in this study. The Fig.1 presents the configuration of the cavity, a heat bloc is located in the centre ($w = \frac{L}{3}$), the top and bottom wall are isolated (adiabatic wall). The East and West wall are at a Cold temperature. The relative's height of the cavity wall and the base fluid Prandtl number are respectively assumed to be $L/H = 1$ and 6.21 (Water).

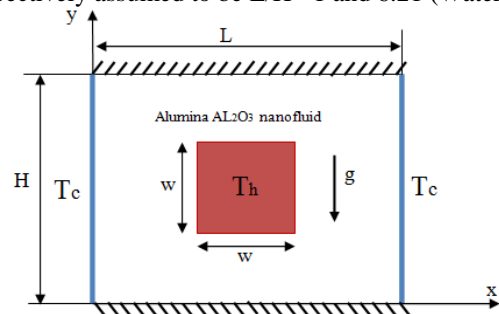


Figure 1. Schematic of the 2D cavity

The main assumptions adopted are: i) Laminar Newtonian and incompressible flow. ii) Alumina nanoparticles have the same spherical size and shape. iii) The model uses the Boussinesq approximation and

neglect The Viscous dissipation in the energy equation.

the continuity, momentum and energy equations for a laminar incompressible fluid can be expressed in dimensionless form as follows:

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0 \quad (1)$$

$$U \frac{\partial(U)}{\partial X} + U \frac{\partial(U)}{\partial Y} = -\frac{\partial P}{\partial X} + \left(\frac{\nu_{nf}}{\nu_f}\right) \left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2}\right) \quad (2)$$

$$V \frac{\partial(V)}{\partial X} + V \frac{\partial(V)}{\partial Y} = -\frac{\partial P}{\partial Y} + \left(\frac{\nu_{nf}}{\nu_f}\right) \left(\frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2}\right) + Ra * Pr * \frac{(\rho\beta)_{nf}}{\rho_{nf} \beta_f} \theta \quad (3)$$

$$U \frac{\partial \theta}{\partial X} + V \frac{\partial \theta}{\partial Y} = \left(\frac{\alpha_{nf}}{\alpha_f}\right) \left(\frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2}\right) \quad (4)$$

Where, the density ρ_{nf} , the calorific capacity $(\rho C_p)_{nf}$, the thermal expansion coefficient $(\rho\beta)_{nf}$, and the thermal diffusivity of the nanofluid α_{nf} are defined from the following expressions:

$$\rho_{nf} = (1 - \varphi) \rho_f + \varphi \rho_s \quad (5)$$

$$(\rho C_p)_{nf} = (1 - \varphi)(\rho C_p)_f + \varphi(\rho C_p)_s \quad (6)$$

$$(\rho\beta)_{nf} = (1 - \varphi)(\rho\beta)_f + \varphi(\rho\beta)_s \quad (7)$$

$$\alpha_{nf} = k_{nf} / (\rho C_p)_{nf} \quad (8)$$

The dynamic viscosity and the thermal conductivity are given respectively by :

$$\frac{k_{nf}}{k_f} = \frac{k_s + 2k_f - 2\varphi(k_f - k_s)}{k_s + 2k_f + \varphi(k_f - k_s)} \quad (9)$$

$$\mu_{nf} = \frac{\mu_f}{(1 - \varphi)^{2.5}} \quad (10)$$

The Nusselt number for the cold wall can be written as:

$$Nu(Y) = \left(\frac{k_{nf}}{k_f}\right) \frac{\partial \theta}{\partial X}$$

The average Nusselt number:

$$Nu_a = \frac{1}{H} \int_0^H Nu(Y) dY$$

2.2. Numerical method

The system of equations with the assumptions above were discretized and calculated using a **Fortran code based** on the finite volume method by adopting an 121x121 mesh along X, Y. The algebraic equations with pressure terms obtained after discretization are solved using the SIMPLEC algorithm. Then, the resulting discrete algebraic system is solved by the Alternating Direction Implicit scheme (ADI) and Thomas Algorithm.

The boundary conditions adopted in the study are: i) In the East and West wall: $U = V = 0$; $\theta = 0$ ii) On the Bloc: $U = V = 0$; $\theta = 1$ iii) On the remaining wall: $U = V = 0$; $\frac{\partial \theta}{\partial n} = 0$.

3. Code Validation

In order to ensure the reliability of the adopted code. Numerical results are checked through a comparison of average Nusselt number with natural convection's previous works [3]. the table 1 shows a good agreement with similar reference results.

	$Ra = 10^3$		$Ra = 10^5$	
	$\emptyset = 0\%$	$\emptyset = 2\%$	$\emptyset = 0\%$	$\emptyset = 2\%$
Reference [3]	1.05	1.65	4.0	4.15
Our code	1.08	1.61	3.87	4.3
Difference	2.8%	2.42%	3.25%	3.48%

Table 1. the average Nusselt number Validation for the present work versus Ref. [3]

4. Discussion and Results

In the present study, heat transfer characteristics inside a 2D cavity have been reported for two cases of Rayleigh number (10^4 and 10^6) which correspond to a laminar regime. The study is based on a comparison of temperature distribution and average Nusselt number for different volume fractions of Alumina nanofluid (0% and 4%).

4.1 Temperature distribution (Isotherms) and Isothermal lines:

To study the effect of nanofluid volume fractions on 2D cavity heat. The 2D isotherms and streamlines for different Rayleigh numbers 10^4 and 10^6 are presented in Fig.2. and Fig.4 For all cases the heated fluid ascends along the hot bloc, then moves horizontally, and then cooled and descends at the vicinity of the cold right and left wall. Also, a noticeable difference in the contour distribution is found near the Bloc with Rayleigh number and volume fraction rising. In terms of Rayleigh number rise, this effect leads to an increase in inertia forces compared to viscosity forces which strengthened the (Al_2O_3 + water) fluid. In terms of volume fraction increase, the introduction of Alumina particles in the base fluid (water) has strongly improved its thermal properties which results in a considerable rate of heat transfer.

4.2 Average Nusselt

To highlight the heat transfer performance, the Fig.3 illustrates variation of the average Nusselt number, calculated on the Cold wall, as a function of Rayleigh number and Al_2O_3 volume fractions. As can be seen from the figure, the average Nusselt number increases while the Rayleigh number increase and nanoparticles concentration. In fact, the increase of nanoparticles concentration arguments the effective thermal conductivity and the flow intensity. Meanwhile, the Rayleigh number rise from 10^2 to 10^6 motivates the inertia forces. Both of two factors enhance the heat transfer rate in our case of study.

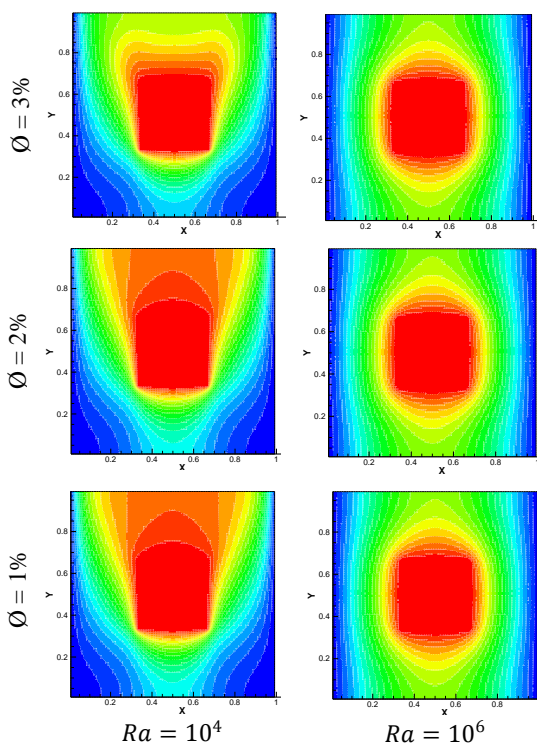


Figure 2. Isotherms at various combination of Ra and ϕ .

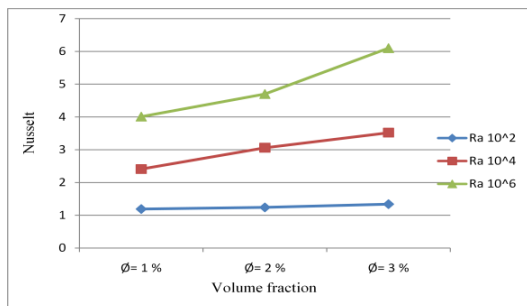


Figure 3. Variations of Nusselt with $Ra = 10^2$, 10^4 and 10^6 for three values of the nanoparticles volume fraction ($\phi = 1\%$, 2% and 3%).

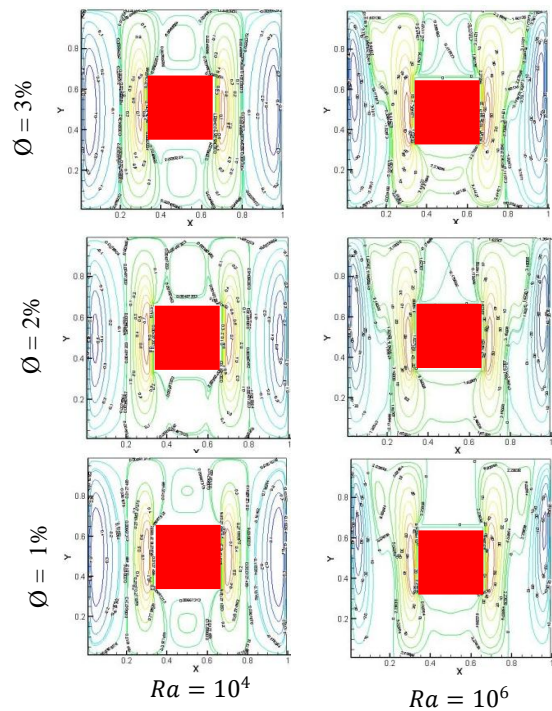


Figure 4. Isothermal lines at various combination of Ra and ϕ

5. Conclusion

The effects on fluid flow, heat transfer at various Rayleigh numbers and solid volume fractions are investigated in the present paper. From the above discussion following conclusions may be drawn:

- For higher values value of solid volume fraction and Rayleigh numbers heat transfer rate is increased.
- Average temperature decreases for increasing the values solid volume fraction as well as Rayleigh numbers.

References

- [1] M. Mahmoodi, S.M. Sebdani « Natural convection in a square cavity containing nanofluid and an adiabatic square block at the center », *Superlattices and Microstructures* 52 (2012) 261–275
- [2] Sannad et al « Effect of the heating block position on Natural Convection in a Three-Dimensional Cavity filled with nanofluid », *Journal of Applied Fluid Mechanics*, 12 (2019) 281–291.
- [3] H.F. Oztop, E. Abu-Nada, « Numerical study of natural convection in partially heated rectangular enclosures filled with nanofluids », *International Journal of Heat and Fluid Flow* 29 (2008) 1326–1336.

SIMULATION NUMERIQUE DE LA CONVECTION MIXTE COUPLEE AU RAYONNEMENT DANS UNE CAVITE CARREE A PAROI MOBILE PAR LA METHODE DE BOLTZMANN

A. DAIZ¹, A. BAHLAOUT¹, I. ARROUB¹, S. BELHOUIDEG¹, A. RAJI², M. HASNAOUI³

1. Laboratoire de Recherche en Physique et Sciences pour l'Ingénieur (LRPSI), Faculté Polydisciplinaire, Béni-Mellal, Maroc
 2. Laboratoire de Génie Energétique et Matériaux (LGEM), Faculté des Sciences et Techniques, Béni-Mellal, Maroc
 3. Laboratoire de Mécanique des Fluides et Energétique (LMFE), Faculté des Sciences Semlalia, Marrakech, Maroc
- Auteur correspondant : daizabelhak@gmail.com

Résumé

Le phénomène de la convection mixte couplée au rayonnement surfacique dans une cavité carrée à paroi mobile est étudié numériquement à l'aide de la méthode de Lattice Boltzmann (LBM). La cavité est localement chauffée sur ses parois, gauche et inférieure, et refroidie sur sa paroi droite alors que l'air y est confiné se comporte comme un fluide caloporteur parfaitement transparent ($Pr = 0.71$). Les paramètres de base régissant le problème physique sont tels que l'émissivité des parois, ϵ , le nombre de Reynolds, Re , et le nombre de Richardson, Ri . Les résultats obtenus indiquent que l'émissivité des parois augmente l'échange radiatif, réduit la composante convective et améliore le transfert de chaleur global.

Mots clés : *Convection mixte, rayonnement surfacique, Lattice Boltzmann, chauffage local*

1. Introduction

Le problème du transfert de chaleur par convection mixte couplée au rayonnement dans une cavité carrée à parois mobiles a suscité une attention considérable de la part des chercheurs. Ce problème est souvent rencontré dans de nombreuses applications d'ingénierie telles que les capteurs solaires, le refroidissement des appareils électroniques, la thermohydraulique des réacteurs nucléaires et bien d'autres.

Les effets combinés de la convection mixte et du rayonnement sur les caractéristiques de l'écoulement et le transfert de chaleur dans les cavités à parois mobiles sont très peu documentés dans la littérature. Antar et al. [1] ont étudié l'effet du rayonnement thermique sur le transfert de chaleur dans une cavité rectangulaire chauffée par le bas et entraînée par une paroi à vitesse variable. Les auteurs ont constaté que les valeurs de Nu calculées à vitesse variable se situent entre celles obtenues avec des profils de vitesse (constante) égale à

V_{min} et V_{max} . Belmiloud et Chemloul [2] ont étudié l'effet du rayonnement sur la convection mixte à l'intérieur d'une cavité différentiellement chauffée à paroi mobile. Leurs résultats montrent que le rayonnement affecte considérablement les deux composantes du transfert thermique. De plus, le nombre de Nusselt Nu diminue quand le nombre de Richardson augmente. Dahani et al. [3] ont analysé numériquement l'effet du rayonnement sur la convection mixte dans une cavité carrée à paroi mobile avec chauffage variable. Ils ont trouvé que la contribution du rayonnement au transfert de chaleur total n'est pas négligeable même aux faibles nombres de Richardson et cette contribution est améliorée en augmentant ce dernier paramètre. La contribution radiative dans le cas des cavités remplies de fluides gris à émission et absorption diffuses a été signalée et mise en évidence dans l'étude numérique de Addini et Nassab [4]. Ils ont obtenu que le nombre de Nusselt radiatif, évalué sur la paroi chaude, décroît progressivement avec l'augmentation de l'épaisseur optique du milieu.

Les travaux antérieurs montrent qu'il y a un manque des études traitant la convection mixte combinée au rayonnement dans des cavités discrètement chauffées à parois mobiles. Suite à cette constatation, on a proposé ce travail destiné à l'étude numérique d'un tel phénomène à l'aide de la méthode de Boltzmann sur réseau. Une étude paramétrique est réalisée en examinant l'effet des différentes combinaisons des paramètres gouvernants, à savoir le nombre de Richardson et l'émissivité des parois, sur les champs thermique et dynamique de l'écoulement et les performances thermiques au sein de la cavité.

2. Description du Problème et Formulation Mathématique

La figure 1 présente la configuration étudiée. Il s'agit d'une cavité carrée à paroi mobile et remplie d'air ($Pr = 0.71$). La cavité est chauffée de manière discrète sur ses parois gauche et inférieure et uniformément refroidie sur

sa paroi droite. La paroi supérieure est adiabatique et entraînée horizontalement vers la droite avec une vitesse constante u_0 . Les parois de la cavité sont supposées grises à émission et réflexion isotropes et de même émissivité. L'air est supposé Newtonien, incompressible et obéit à l'approximation de Boussinesq alors que son écoulement est considéré bidimensionnel et laminaire.

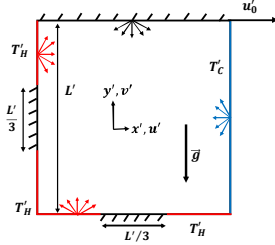


Figure 1. Configuration étudiée et conditions aux limites.

La méthode Lattice Boltzmann (LBM) est devenue un outil très populaire utilisé pour étudier la dynamique des fluides dans des géométries simples et complexes [5]. Dans cette méthode, les champs d'écoulement et de température sont résolus en utilisant deux fonctions de distribution différentes f et g .

Les équations régissant notre phénomène en utilisant l'approximation de Bhatnagar-Gross-Krook (BGK) peuvent être écrites comme suit :

$$f_k(r + c_k \Delta t, t + \Delta t) = f_k(r, \Delta t) - \frac{1}{\tau_f} (f_k(r, \Delta t) - f_k^{eq}(r, \Delta t)) + \Delta t F_k \quad (1)$$

$$g_k(r + c_k \Delta t, t + \Delta t) = g_k(r, \Delta t) - \frac{1}{\tau_g} (g_k(r, \Delta t) - g_k^{eq}(r, \Delta t)) \quad (2)$$

Où $c = (c_x, c_y)$ est la vitesse discrète de la particule située en $r = (x, y)$, $c_k \Delta t$ est la distance séparant deux nœuds voisins du réseau D_2Q_9 , Δt est le pas de temps, $F_k = 3w_k \rho g \beta c_{ky} (T - T_m)$ est la force externe (force de poussée) discrète en adoptant l'approximation de Boussinesq, $\tau_f = 3\nu + 0.5$ et $\tau_g = 3\alpha + 0.5$ sont respectivement les temps de relaxation pour les champs de vitesse et de température et f_k^{eq} et g_k^{eq} sont les fonctions d'équilibre local pour le champ d'écoulement et de température (leurs expressions et les détails de la technique de résolution sont disponibles dans la référence [5]).

Les grandeurs macroscopiques qui sont la densité, ρ , la vitesse, u , et la température, T , ont été calculées en utilisant les 3 expressions suivantes :

$$\rho = \sum f_k; \quad \rho u = \sum c_k f_k; \quad T = \sum g_k \quad (3)$$

Chaque paroi de la cavité est subdivisée en surfaces élémentaires ayant pour centres les nœuds. La détermination des facteurs de forme entre les différentes surfaces élémentaires est basée sur La méthode des

cordes croisées décrite par Hottel et Saroffim [6]. Le calcul des échanges thermiques par rayonnement entre les différents éléments surfaciques est basé sur la méthode des radiosités [7].

Les nombres de Nusselt moyens convectif et radiatif, calculés le long de la paroi verticale froide, sont respectivement définis par :

$$Nu_{cv} = - \int_0^1 \frac{\partial T}{\partial x} \Big|_{x=1} dy; \quad Nu_r = \int_0^1 N_r Q_r \Big|_{x=1} dy \quad (4)$$

Avec $N_r = \sigma T^4 L / k (T_H - T_C)$ est le nombre de rayonnement-convection et $Nu = Nu_{cv} + Nu_r$ le nombre de Nusselt moyen total.

3. Validation du Code Numérique

Notre code numérique a été validé en comparant les résultats obtenus à ceux obtenus par Wang et al. [8] dans le cas des effets combinés du rayonnement et de la convection naturelle dans une cavité différentiellement chauffée pleine d'air [tableau 1]. La comparaison qui porte sur Nu_{cv} , Nu_r et Nu , montre un excellent accord avec une différence maximale de l'ordre de 0.8 %.

Tableau 1. Résultats de validation du code de calcul obtenus pour $\varepsilon = 0.8$ et $Ra = 10^4$.

	Nu_{cv}	Nu_r	Nu
Présente étude	2.246	2.382	4.628
Wang et al. [8]	2.249	2.401	4.650
Erreur en %	0.1	0.8	0.5

4. Résultats et discussions

Cette étude est réalisée pour un nombre de Richardson fixé à $Ri = 1$ (valeur correspondant au régime de convection mixte) et une émissivité variant dans la gamme $0 \leq \varepsilon \leq 1$. Etant donné que la valeur de Rayleigh est fixée à $Ra = 10^5$, le nombre de Reynolds implique $Re = 375.29$. Les résultats obtenus sont illustrés en termes de lignes de courant et d'isothermes et du nombre de Nusselt moyen.

L'effet du rayonnement sur la structure dynamique et thermique de l'écoulement est illustré sur la figure 2 pour $Ri = 1$ (convection mixte dominante) en absence ($\varepsilon = 0$) et en présence du rayonnement ($\varepsilon = 1$). Pour $\varepsilon = 0$, les lignes de courant (figure 2a) montrent que l'écoulement est caractérisé par une grande cellule principale dont le centre est placé au milieu de la cavité et qui tourne dans le sens horaire. Sa formation est due aux effets de cisaillement entraînés par la paroi mobile et ceux d'Archimède provoqués par les différences de température entre les parois chaudes et celle froide. Aussi, il est bien vu que les isothermes correspondantes se condensent près des parois actives témoignant d'un

bon échange thermique convectif au voisinage de ces parois. En plus, la partie centrale de la cavité présente une zone froide qui est inactive thermiquement. Lors l'effet du rayonnement est pris en considération la figure 2b obtenue pour $\varepsilon = 1$ montre que le rayonnement n'affecte pas la structure dynamique de l'écoulement alors qu'il contribue à une légère déformation des isothermes.

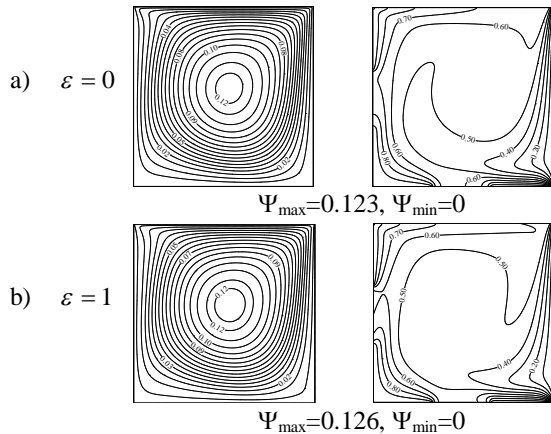


Figure 2. Lignes de courant et isothermes obtenues pour $Ri = 1$ et deux valeurs de ε : a) $\varepsilon = 0$ et b) $\varepsilon = 1$.

Les variations avec ε des nombres de Nusselt moyens, convectif, radiatif et total évalués sur la paroi froide sont illustrées sur la figure 3 pour $Ri = 1$. Il est remarqué que la composante convective reste pratiquement insensible ou subit une légère diminution avec ε . Quant à la composante radiative, son évolution en fonction de ε est caractérisée par une augmentation monotone linéaire. Il est à signaler que la composante radiative devient presque de même ordre de grandeur que celle convective pour une émissivité maximale. Suite à ces variations, Nu suit la même tendance que Nu_r et il est caractérisé par une évolution drastique avec ε .

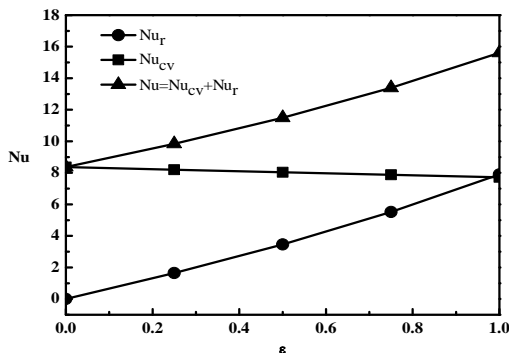


Figure 3. Variations avec l'émissivité ε de Nu_{cv} , Nu_r et Nu , pour $Ri = 1$.

5. Conclusions

Dans cette étude, nous avons effectué une étude numérique sur le phénomène de couplage entre la convection mixte et le rayonnement dans une cavité

carrée à paroi mobile. Les résultats obtenus indiquent que la structure d'écoulement est monocellulaire caractérisée par l'existence d'une grande cellule générée par les effets combinés de cisaillement et de poussée. Cette structure n'est pas influencée par l'émissivité tandis que le champ thermique est légèrement modifié par la présence du rayonnement. Aussi, il est de noter que l'émissivité des parois augmente l'échange radiatif, réduit la composante convective et améliore le transfert de chaleur global.

Références

- [1] M. A. Antar, R. Ben-mansour, S. A. Al-dini, *The effect of thermal radiation on the heat transfer characteristics of lid-driven cavity with a moving surface*, International Journal of Numerical Methods for Heat & Fluid Flow 24 (2014) 679-696.
- [2] M. A. Belmiloud, N. E. S. Chemloul, *Numerical study of mixed convection coupled to radiation in a square cavity with a lid-driven*, International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering International Scholarly and Scientific Research & Innovation 9 (2015) 1815-1821.
- [3] Y. Dahani, M. Hasnaoui, A. Amahmid, A. El Mansouri, S. Hasnaoui, *Lattice Boltzmann simulation of combined effects of radiation and mixed convection in a lid-driven cavity with cooling and heating by sinusoidal temperature profiles on one side*, Heat Transfer Engineering 41 (2020) 433-448.
- [4] M. M. Addini, A. G. Nassab, *Combined mixed convection and radiation simulation of inclined lid driven cavity*, Energy Equipment and Systems 6 (2018) 261-277.
- [5] A. A. Mohamad, *Applied Lattice Boltzmann method for transport phenomena*, Momentum Heat Mass Transfer, Sure Print, Calgary, 2007.
- [6] H. C. Hottel, A. F. Saroffim, *Radiative Heat Transfer*, McGraw-Hill, New York, 1967.
- [7] A. Bahlaoui, A. Raji, M. Hasnaoui, R. El Ayachi, M. Naimi, T. Makayssi, M. Lamsaadi, *Numerical study of mixed convection coupled with radiation in a vented partitioned enclosure*, International Scientific Journal for alternative energy and ecology 6 (2008) 62.
- [8] H. Wang, S. Xin, P. Le Quééré, *Étude numérique du couplage de la convection naturelle avec le rayonnement de surfaces en cavité carrée remplie d'air*, C. R. Mécanique, 344 (2006) 48-47.

Impact de l'inclinaison de l'orbite sur les transferts thermiques dans un CubeSat 1U en LEO

F. TRIBAK¹, F. BEN NEJMA², Z. ARAOUD², O. BENDAOU¹, A. ASSELMAN¹

1. Equipe d'optique et Photonique EOP, Faculté des sciences, Université Abdelmalek Essaadi, avenue Sebta, 93002, Tétouan, Maroc
ferdaous.tribak@etu.uae.ac.ma o.bendaou@uae.ac.ma aasselmann@uae.ac.ma
2. Laboratoire de recherche EMIR, Institut Préparatoire des Etudes d'Ingénieur de Monastir (IPEIM), Université de Monastir, 15 Avenue Ibn El Jazzar Monastir 5019, Tunisie
faycal.bennejma@ipeim.rnu.tn zouhour.ara@enim.rnu.tn

Résumé

L'estimation de la température est essentielle tant pour concevoir des CubeSats fiables que pour les maintenir dans des conditions d'efficacité maximale. Par conséquent, tous les paramètres qui pourraient avoir un impact sur les performances thermiques du satellite doivent être pris en compte dans l'analyse thermique. Cet article présente la simulation thermique d'un CubeSat 1U. L'objectif est de simuler, moyennant le logiciel commercial COMSOL MULTIPHYSICS, l'impact d'un paramètre important sur la distribution de la température du satellite : l'angle bêta. Pour étudier l'effet de ce paramètre sur le satellite un ensemble de simulations a été réalisé pour différents angles bêta.

Mots clefs : *Contrôle thermique, CubeSat, conduction, radiation, angle bêta.*

1. Introduction

Avant qu'un satellite ne quitte le sol, les scénarios qu'il vivra en orbite doivent être testés afin de minimiser l'apparition de pannes ou d'un fonctionnement inefficace. Cette pratique est également essentielle pour les projets CubeSat, une catégorie de satellite basée sur les dimensions de $10 \times 10 \times 10$ cm pour le modèle standard 1U. La majorité des CubeSats en LEO se situent à moins de 600 km, ont une orbite quasi-circulaire, avec une période d'environ 100 min. Le satellite en orbite est exposé au rayonnement externe de l'espace notamment solaire, albedo, terrestre, ainsi que la chaleur générée par ses composants internes. Cet environnement radiatif entraîne des variations extrêmes de température du satellite dont son maintien est essentiel pour le fonctionnement des composants électriques et par conséquent pour le succès de la mission. Définir la distribution de la température en orbite permet de choisir les bons dispositifs de contrôle thermique en évitant les gradients de températures élevés. Ainsi, l'objectif principal du contrôle thermique et d'analyser la température du satellite, de concevoir des mécanismes de contrôle thermique et de tester la solution proposée [1].

2. Analyse thermique du CubeSat

2.1 L'environnement du satellite en orbite

La température du CubeSat résulte des flux auxquels il est confronté en orbite. Elle est donc fonction des paramètres orbitaux (altitude, inclinaison, angle bêta qui représente l'angle entre le vecteur solaire et le plan de l'orbite..), de la géométrie du satellite, des matériaux et des propriétés de la surface.

Afin de déterminer la température du CubeSat, le bilan thermique doit être mis en œuvre pour tenir compte de toute l'énergie qui entre et sort du système. Le bilan énergétique global du Cubesat en régime transitoire est le suivant [2] :

$$Q_{solar} + Q_{albedo} + Q_{IR} + Q_{GEN} - Q_{rad} = m \cdot C_{avr} \cdot (dT_{avr}/dt) \quad (1)$$

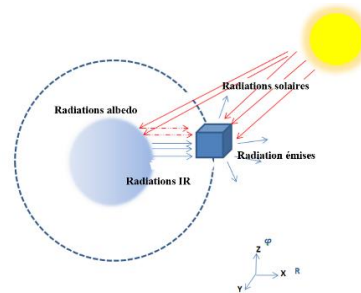


Figure 1 : Échange de chaleur entre le satellite et l'environnement spatial.

$$Q_{solar} = \alpha_s \cdot A_{prj} \cdot S \quad (2)$$

$$Q_{alb} = \alpha_s \cdot A_{prj} \cdot S \cdot a \cdot F_{sat-planet} \quad (3)$$

$$Q_{IR} = \alpha_{IR} \cdot A_{prj} \cdot Q_E \cdot F_{sat-planet} \quad (4)$$

$$Q_{rad} = \varepsilon \cdot \sigma \cdot T^4 \cdot A_{tot} \quad (5)$$

Avec :

α_s : l'absorptivité solaire de la surface.

α_{IR} : L'absorptivité du rayonnement infrarouge de la Terre.

- ϵ : Emissivité du rayonnement infrarouge.
- S : Constante solaire (W/m^2).
- Q_E : Energie infrarouge émise par la Terre (W/m^2).
- a : Facteur albédo.
- $F_{sat-planet}$: Facteur de forme entre la surface du CubeSat et la terre.
- σ : constante de Stefan-Boltzmann ($W/(m^2 K^4)$).
- A_{prj} : Surface du CubeSat exposée au Rayonnement (m^2).
- A_{tot} : Surface totale du CubeSat (m^2).
- T : Température du CubeSat.
- T_{avr} : Température moyenne du satellite.
- C : la chaleur spécifique moyenne.
- m : Masse du satellite.

2.2 Angle bêta

Une façon préliminaire de visualiser l'environnement global de rayonnement auquel est soumis un satellite en orbite est de se référer à l'angle β de l'orbite, qui est défini comme l'angle entre le plan de l'orbite et le vecteur soleil de tout objet en orbite autour de la Terre, comme le montre la figure 2. Il détermine la durée pendant laquelle le CubeSat est soumis à la lumière directe du soleil.

En changeant l'angle bêta, on obtient des orientations différentes du plan de l'orbite par rapport au Soleil et à la Terre, ce qui entraîne des plages de température différentes. L'importance du paramètre bêta avec les sources de rayonnement susmentionnées a été utilisée pour l'estimation de la température du CubeSat dans ce travail.

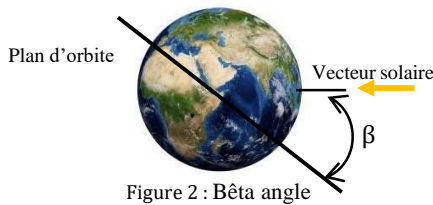


Figure 2 : Bêta angle

2.3 Modélisation et Simulation sur COMSOL Multiphysics

La géométrie représente un CubeSat avec six panneaux solaires, couvrant les côtés extérieurs du CubeSat, une batterie, quatre circuits imprimés, et des boulons qui fournissent un chemin thermique conducteur entre les côtés inférieur et supérieur du CubeSat. [3].

Tous les détails relatifs au câblage, aux connecteurs et à l'électronique n'ont pas été inclus dans ce travail car l'intérêt porte sur une simulation générique d'un CubeSat 1U.

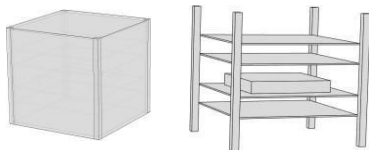


Figure 2: Structure externe et interne du CubeSat

L'avancement temporel a été réglé sur automatique, avec un pas de 30s et sur une durée de cinq cycles. Les propriétés physiques et orbitales sont résumées dans les tableaux suivants [1]. :

- Propriétés thermiques :

	ρ [kg/m^3]	c [$J/kg K$]	κ [$W/m K$]
Panneaux solaires	2325	1103	1.03
Boulons	2810	948	140
PCB	2120	975	0.64
Batterie	2247	1110	23

- Altitude et orbite :

Le satellite décrit une orbite circulaire, à une altitude de 431 km, ce qui correspond à une période de 5583s.

- Conditions aux limites :

Flux solaire	Flux terrestre	Coefficient albédo
1367 W/m^2	237 W/m^2	0.3

- Facteurs de Forme :

Dans ce travail, le rayonnement entre les surfaces internes du cube n'est pas pris en considération. Les facteurs de formes calculés analytiquement sont entre les surfaces externes du cube et la terre [3].

Le CubeSat est orienté de manière à ce que la face (X+) du cube soit en face du soleil et que la face (X-) soit en face de la terre.

Rayonnement IR	Fx(face directe terre)	0.8773
	Fyz (faces latérales terre)	0.2816
Rayonnement albedo	F y-	Max(0 , Fyz *cos β)
	Fx-	Max(0 , Fx *cos β)
	Fz-	Max(0 , Fyz *cos β)
	Fz+	Max(0 , Fyz *cos β)
	F y-	Max(0 , Fyz *cos β)

3. Résultats et discussion

Dans ce travail, le satellite étudié a été analysé dans différents angles bêta ($0^\circ, 40^\circ, 80^\circ$) afin d'évaluer son impact sur le transfert de chaleur et la température du satellite.

La simulation est effectuée sur 5 périodes pour atteindre le régime stationnaire, en 27900 s.

L'avancement temporel a été réglé sur automatique, avec un pas de 30s et une durée de cinq cycles.

Ci-dessous, les résultats de la simulation pour l'angle bêta égal à 0.

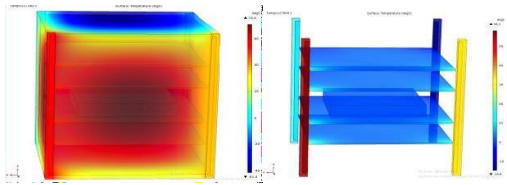


Figure 3: Variation de température sur la structure externe et interne du satellite pour $\beta = 0^\circ$ à l'instant 27780s

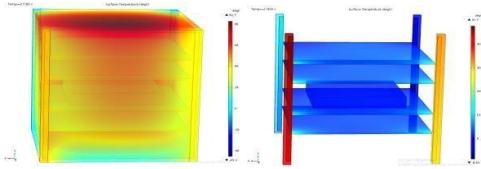


Figure 4: Variation de température sur la structure externe et interne du satellite pour $\beta = 40^\circ$ à l'instant 27780s

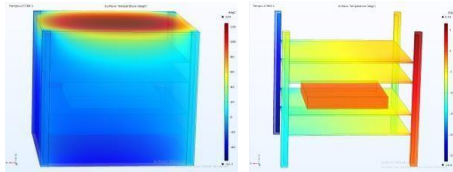


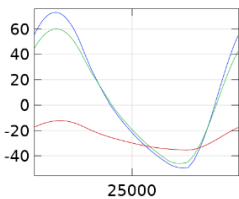
Figure 5: Variation de température sur la structure externe et interne du satellite pour $\beta = 80^\circ$ à l'instant 27780s

Bêta	Structure externe		Structure interne	
	Tmax (°C)	Tmin (°C)	Tmax (°C)	Tmin (°C)
0°	70.4	-41.4	58.1	-14.6
40°	82.7	-45.3	47.7	-8.65
80°	125	-57.7	6.54	-26.4

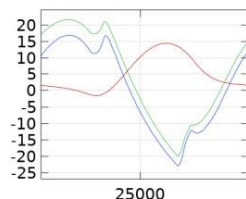
Après les 5 périodes de simulation, les résultats obtenus lors du dernier cycle sont présentés sur les graphiques ci-dessous où les courbes bleue, verte et rouge représentent la variation de la température du satellite pour les angles respectifs $\beta = 0^\circ$, $\beta = 40^\circ$ et $\beta = 80^\circ$.

— $\beta = 0^\circ$ — $\beta = 40^\circ$ — $\beta = 80^\circ$

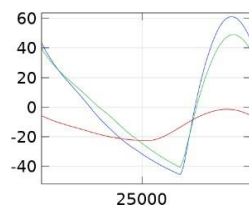
Face +X :



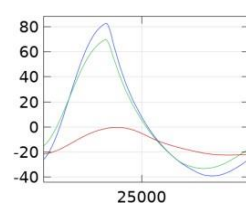
Face -X :



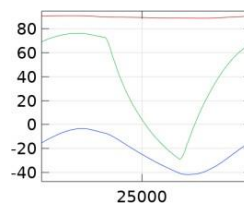
Face +Y :



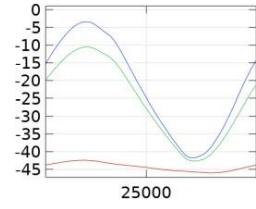
Face -Y :



Face +Z :



Face -Z :



Les figures 3, 4 et 5 montrent la variation de la température du satellite pour $\beta = 0$ après 5 périodes, $t = 27900s$ et le tableau qui les suit résume la variation de la température interne et externe du CubeSat.

Les graphiques ci-dessus montrent l'évolution de la température avec le temps en fonction de différents angles bêta pour chaque côté du CubeSat.

Pour $\beta = 0^\circ$, la face +X est entièrement exposée au soleil alors que pour $\beta = 80^\circ$, la face +Z du cube est la plus exposée au soleil.

Pour les valeurs bêta de 0° et 40° , la température des côtés du satellite a changé au fil du temps. Cela s'explique par le fait qu'une partie de son temps est passée en éclipse, tandis que le reste du temps, il fait face à la lumière du soleil. Cependant, pour une valeur bêta égale à 80° , la température du satellite est restée relativement constante tout au long de la période, car sa position et son orientation par rapport au soleil n'ont pas changé.

En effet, une orbite avec β égal à 0 aura le temps d'éclipse le plus long car elle est ombragée par le diamètre complet de la Terre, au fur et à mesure que β augmente, le temps d'éclipse diminue jusqu'à ce que β soit égal à 90 degrés, où le temps d'ensoleillement est maximal.

5. Conclusion

Le travail a simulé le transfert de chaleur d'un CubeSat 1U, avec des panneaux solaires couvrant les faces externes, une structure interne en aluminium, quatre PCBs, et une batterie sans génération de chaleur. La solution a été obtenue par la méthode des éléments finis (MEF) sur COMSOL Multiphysics sans prendre en compte le transfert de chaleur interne par rayonnement.

Après avoir effectué une série de simulations pour différents angles bêta, les résultats ont mis en évidence l'importance de ce paramètre et son impact sur la variation de température de chaque face du CubeSat, ce qui affecte les performances thermiques du satellite.

Références

- [1] R.D. Karam, Satellite Thermal Control Systems Engineer. Reston, VA: AIAA, 1998, Vol.181
- [2] D.Gilmore, M. Donabedian, *Spacecraft Thermal Control Handbook: Fundamental technologies*, in: *Spacecraft Thermal Control Handbook*, Aerospace Press, 2002.
- [3] E.Filho, L. Seman, V. Nicolau Simulation of a CubeSat with internal heat transfer using Finite Volume Method, Applied Thermal Engineering 2021.

Analytical study of heat conduction inside orthotropic media

Mohammed Nikchi, Rabiâa Idmoussa, Khalid Zniber, Jawad Lahjomri and Abdelaziz Oubarra

Laboratory of Mechanics, Faculty of Science Ain Chock, University Hassan II,
Casablanca 20100, Morocco
mohammed.nikchi-etu@etu.univh2c.ma

Abstract

The work we are going to present concerns the analytical study of steady-state heat conduction in orthotropic media in a two-dimensional relative to a cylindrical geometry. The objective of the present work is to solve analytically the steady state heat conduction equation in orthotropic media in two-dimensional cylindrical geometry with Dirichlet and Newman type boundary conditions. Main thermal conductivities ratio for each medium and the ratio of the radial thermal conductivities of mediums are investigated.

Keywords: Radial temperature, Radial flux, Orthotropic media

1. Introduction

Heat transfer is one of the most common modes of energy exchange. It is a phenomenon found in many sectors of industry and in our daily lives. Engineers and technicians are confronted with this kind of problem and try to maximize or minimize this phenomenon according to the needs of the industry and in order to save this expensive energy. Therefore, heat transfer has, both in the field of pure science and in that of technological applications, an often-essential role. These applications often require thermal characteristics that isotropic materials cannot satisfy. As a result, the study of heat transfer in non-conventional media is becoming more and more frequent.

The heat transfer within a multilayer cylinder is a problem of great technical interest [1]. The use of materials with varying thermal/mechanical properties often helps to provide desired performance characteristics for engineered structures and systems. Examples include nuclear fuel tubes in a nuclear reactor, compressed hydrogen storage [2], superconducting cables, piezoelectric transducers [3] and civil engineering structures. Analytical modeling is important to understand the nature of thermal transport, separation of variables method is used to model thermal conduction in cylindrical and spherical composite laminates [4,5]. This paper presents an analytical solution for the two-dimensional temperature and radial flow distribution in two cylinders subjected to the same constant temperature on the sections and sinusoidal flux on the lateral surface. The study parameters for this work are the ratio of the

main thermal conductivities for each medium and the ratio of the radial thermal conductivities of the two mediums that were studied.

2. Problem formulation

Let us consider cylindrical orthotropic media without heat source, of length L and inner and outer radii are respectively a and b , whose bases $z=0$ and $z=L$ are maintained respectively at constant temperatures (T_i , T_r) while a radial flux is applied on the lateral surface $r=b$.

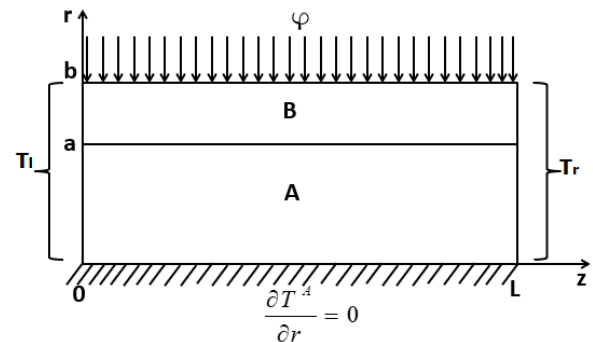


Figure 1. Bidirectional representation of the two orthotropic media.

The heat transfer equations for orthotropic cylinders are written:

$$\text{div}(\overline{K^i \text{grad} T^i}) = 0 \quad (1)$$

the index i represents the medium in which the study is made.

By introducing the dimensionless variables which are defined by:

$$Z = \frac{z}{L}, R = \frac{r}{b}, \Theta = \frac{T^i - T_g}{T_d - T_g}, \Phi_R^i = \frac{\phi_r^i}{\phi_{Ref}} \quad (2)$$

The heat diffusion equations in orthotropic media are written in the following form:

$$\frac{\partial^2 \Theta^i}{\partial R^2} + \frac{1}{R} \frac{\partial \Theta^i}{\partial R} + \nu^2 \frac{\partial^2 \Theta^i}{\partial Z^2} = 0 \quad (3)$$

With: $\nu^2 = K_{zz} / K_{rr}$

3. Problem Solving

Solving the thermal problem requires the integration of the equation system and therefore the knowledge of the different boundary conditions.

The resolution of the equation (2) allows access to the temperature profile in cylindrical coordinates.

The system of equations, with associated boundary conditions, governing the conductive heat transfer in orthotropic mediums in bidirectional steady state, is solved analytically by the variable separation technique.

$$\Theta^A(R,Z)=1-Z/G+\sum_{n=1}^{\infty} \frac{\nu}{n\pi} \cdot \frac{1}{I_1(\lambda\nu)} \left[1 + \frac{\left(\frac{1}{\delta}-1\right) \left(\frac{K_1(\lambda\nu)}{I_1(\lambda\nu)} + \frac{K_0(\lambda\nu\xi)}{I_0(\lambda\nu\xi)}\right)}{\frac{K_1(\lambda\nu)}{I_1(\lambda\nu)} + \frac{K_0(\lambda\nu\xi)}{I_0(\lambda\nu\xi)} - \frac{1}{\delta} \left(\frac{K_1(\lambda\nu)}{I_1(\lambda\nu)} - \frac{K_1(\lambda\nu\xi)}{I_1(\lambda\nu\xi)}\right)} \right] \cdot \sin(\lambda Z) I_0(\lambda\nu R) \tag{4}$$

$$\Theta^B(R,Z)=1-Z/G+\sum_{n=1}^{\infty} \frac{\nu}{n\pi} \cdot \frac{1}{I_1(\lambda\nu)} \left[I_0(\lambda\nu R) + \frac{\left(\frac{1}{\delta}-1\right) \left(\frac{K_1(\lambda\nu)}{I_1(\lambda\nu)} I_0(\lambda\nu R) + K_0(\lambda\nu R)\right)}{\frac{K_1(\lambda\nu)}{I_1(\lambda\nu)} + \frac{K_0(\lambda\nu\xi)}{I_0(\lambda\nu\xi)} - \frac{1}{\delta} \left(\frac{K_1(\lambda\nu)}{I_1(\lambda\nu)} - \frac{K_1(\lambda\nu\xi)}{I_1(\lambda\nu\xi)}\right)} \right] \cdot \sin(\lambda Z) \tag{5}$$

Where, $\nu = \sqrt{\frac{k_{zz}}{k_{rr}}}$ and $\lambda = \frac{n\pi}{G}$.

I_0, I_1 : Are Bessel functions of the first kind respectively of order 0 and 1.

K_0, K_1 : Are Bessel functions of the second kind respectively of order 0 and 1.

The Temperatures in each media are established by an iterative calculation, comparing the successive difference of all internal temperatures between two series orders and with a criterion of convergence, according to :

$$|\Theta^A(R,Z,n+1) - \Theta^A(R,Z,n)| + |\Theta^B(R,Z,n+1) - \Theta^B(R,Z,n)| < \varepsilon \tag{6}$$

4. Results and Discussion

The radial temperature profiles and the radial flux will be represented. The results are relative to the case of orthotropic mediums in cylindrical geometry, of constant form factor G (G = 2), for values of the ratio of thermal conductivities $\nu=0.5, 1, 2$ and values of the ratio of radial thermal conductivities $\delta=0.5, 1, 2$ and of sinusoidal imposed flux. Interface location between the mediums is taking equal to 0.8.

For all the following we take the black curve for $\nu=0.5$, the red curve for $\nu=1$ and the blue curve for $\nu=2$.

Figures [1-3] present the radial temperature profile in the two mediums middle according to a given value of radial thermal conductivities ratio associated to various values of the main thermal conductivities. Thus, for a fixed value of δ , the radial temperature shape rises from the center to the exterior lateral surface cylinder. Rise of main thermal conductivities ratio decreases thermal level in first cylinder and increase that in second cylinder. In addition, we notice that for a given value of main thermal conductivities ratio, the increase of δ leads to the same conclusions with an overall reduction in the thermal level. Overall, it appears that second medium is more sensitive to the effect of ν . In addition, we notice a particular radius corresponding to inversion of the

second medium temperature with respect to ν . This particular radius decreases by increasing δ .

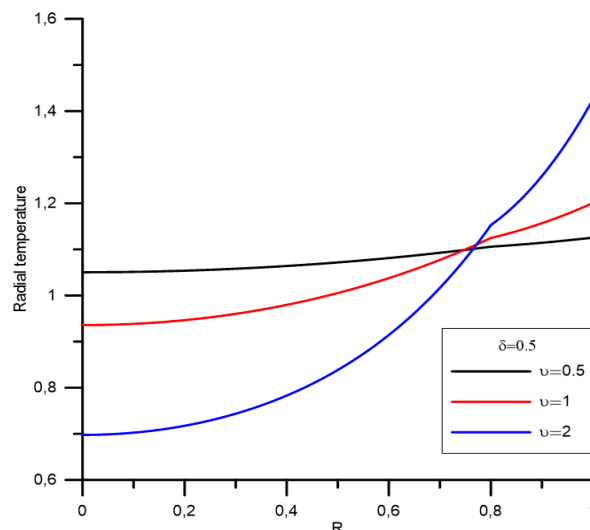


Figure 2. Middle radial temperature for $\delta=0.5$

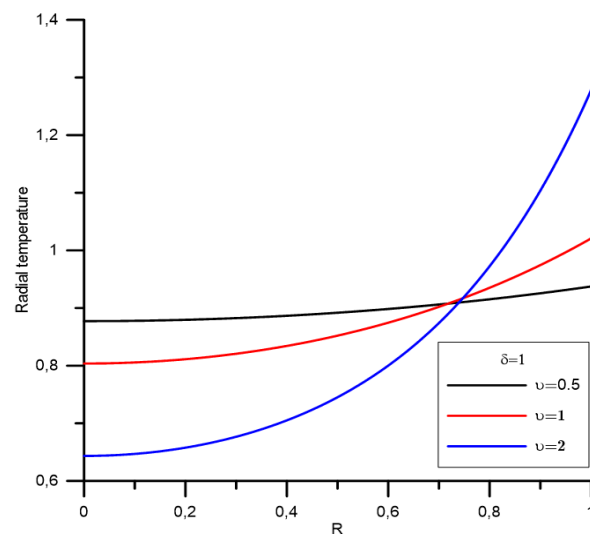


Figure 3. Middle radial temperature for $\delta=1$

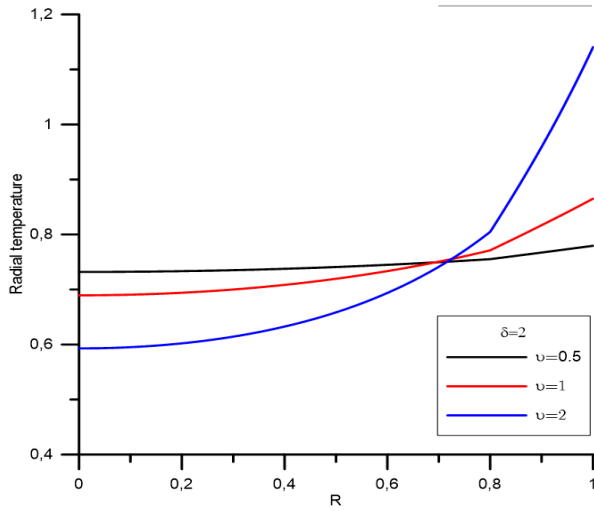


Figure 4. Middle radial temperature for $\delta=2$ curves [4-6] presents the radial flux of different curvatures associated with the mediums. It can be seen that the radial flux increases from the center to the outer end of the outer cylinder. The radial flux profile increases in the mediums according to the ratio of radial thermal conductivities δ . It is noted that when the main thermal conductivity ratios ν decreases, the radial flow profile becomes higher.

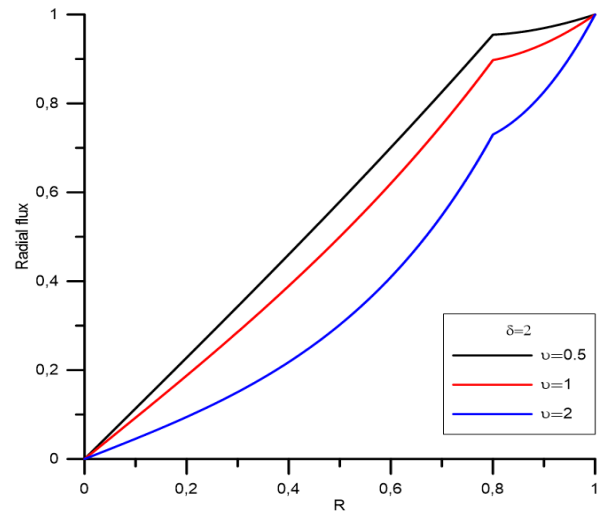


Figure 7. Middle radial flux for $\delta=2$

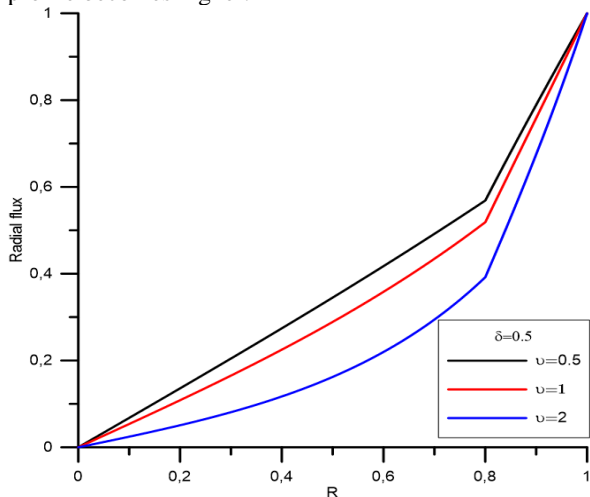


Figure 5. Middle radial flux for $\delta=0.5$

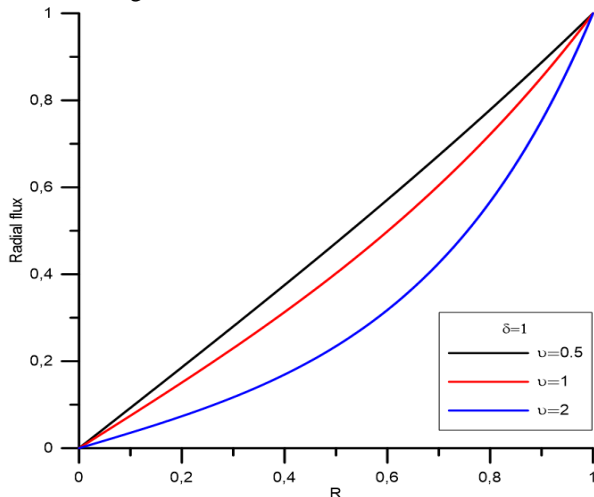


Figure 6. Middle radial flux for $\delta=1$

5. Conclusion

The work carried out within the framework of this article is related to the analytical determination of the temperature field and heat fluxes within orthotropic mediums in a two-dimensional steady state and for a cylindrical geometry, such that the lateral surface is subjected to a flux density and the lateral sections are isothermal. This study aims at understanding the orthotropic character and in particular the effect of the ratio of the two radial thermal conductivities, and main thermal conductivities ratio of each medium, on the heat transfer mechanism within the mediums. The main findings of this study are, the temperature and flux profile are smooth with no change in curvature at the interface for the case of radial thermal conductivity ratio equal to 1, increase of main thermal conductivities ratio reduces thermal level in first medium and increase that in second medium, and the radial flux increases from the center to the outer end of the outer cylinder.

References

- [1] AD.W. Hahn, M.N. Ozisik, Heat Conduction, 3rd. Ed., John Wiley & Sons, 2012.
- [2] A. S. Manahan, Environmental Chemistry, ninth ed., CRC Press, 2010.
- [3] J. Wang, L. Qin, W. Li, W. Song, Parametric analysis and optimization of radially layered cylindrical piezoceramic/epoxy composite transducers, Micromachines 9 (2018), 585:1-16.
- [4] A. M.H. Kayhani, M. Shariati, M. Norouzi, M. Karimi Demneh, Exact solution of conductive heat transfer in cylindrical composite laminate, Heat Mass Tran. 46 (2009) 83–94.
- [5] M. Norouzi, A. Amiri Delouei, M. Seilsepour, A general exact solution for heat conduction in multilayer spherical composite laminates, Compos. Struct. 106 (2013) 288–295.

Double-Diffusive Mixed Convection in a Shallow Rectangular Cavity with Single Lid-Driven Boundary Filled with Newtonian Fluid

Y. TIZAKAST¹, M. KADDIRI², M. LAMSAADI³

1. Sultan Moulay Slimane University, LGI, BP 523, Beni-Mellal 23000, Morocco (youssef.tizakast@usmba.ac.ma)
2. Sultan Moulay Slimane University, LGI, BP 523, Beni-Mellal 23000, Morocco (mouradkadiri@usms.ma)
3. Sultan Moulay Slimane University, LRPSI, BP 523, Beni-Mellal 23000, Morocco (lamsaadima@yahoo.fr)

Abstract

This study investigates double-diffusive mixed convection within a rectangular cavity filled with a Newtonian fluid and subjected to uniform temperature and concentration fluxes along the short vertical walls, the top wall slides in the same direction as the imposed fluxes. The investigation is carried out both numerically using finite difference method and analytically based on the parallel flow assumption. The fluid flow is governed by: Peclet number Pe , thermal Rayleigh number Ra_T , Lewis number Le , and buoyancy ratio N . Both adopted approaches showed perfect agreement for an inclusive range of governing parameters. The parameter $\frac{Ra_T}{Pe^{3.0}}$ accurately delimits the three convective regimes, namely natural, mixed, and forced convection. Moreover, increasing thermal Rayleigh number or buoyancy ratio promotes natural convection contribution in overall convection while Peclet number strengthens forced convection contribution.

Keywords: *Double-diffusive mixed convection, Heat and mass transfer, Lid-driven cavities, Parallel flow, Finite difference method.*

1. Introduction

Double-diffusive mixed convection refers to fluid flows driven Simultaneously by both shear force caused by the moving walls and buoyancy forces generated by applied thermal and solutal gradients with different diffusion rates. Such driven cavities are encountered in many engineering applications such as flow and heat transfer in solar ponds, crystal growing, glass production, food processing, galvanizing, and so on. Teamah and El-Maghlany [1] investigated the phenomenon in a rectangular enclosure for both assisting and opposing flow cases. They reported that for both cases, decreasing Richardson number or increasing buoyancy ratio absolute value enhances heat and mass transfer. Kumar et al. [2] used the velocity-vorticity form of Navier Stokes's equations to examine double-diffusive mixed convection inside a lid-driven square cavity. The obtained results

show that when temperature and concentration gradients are applied in opposite directions, negative buoyancy ratios produce aiding flows. More studies from the literature investigated the same phenomenon for single lid-driven cavities [3,4] and double lid-driven cavities [5,6].

However, double-diffusive mixed convection within rectangular cavities with associated thermal and solutal boundary conditions of Neumann type (i.e., imposed heat and mass fluxes to the boundaries) has not been considered enough, especially the combined effect of governing parameters on the dominance regions of convective regimes, namely natural, mixed, and forced convection. To fill in the gap, the present study investigates both numerically and analytically double-diffusive mixed convection fluid flow and heat and mass transfer inside a single lid-driven horizontal rectangular cavity filled with a Newtonian fluid, the results illustrate the effects of governing parameters on the dominance regions of the three convective regimes.

2. Physical Configuration and Mathematical Model

Figure 1 shows the configuration considered in the present work, a shallow horizontal rectangular cavity of height H' and length L' subjected to uniform density of heat and mass fluxes q' and j' , respectively, along the short vertical walls while the horizontal ones are insulated and impermeable. The top wall slides from left to right (i.e., in the same direction as the imposed heat and mass fluxes) with constant velocity u'_0 , while the remaining walls are motionless.

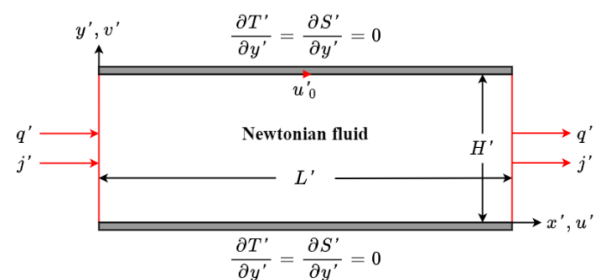


FIGURE 1. Geometry of the cavity along with the associated boundary conditions and coordinates system.

This study adopts the following commonly used main assumptions, i.e.,

- The flow is laminar given the small gradients applied to the cavity [7].
- The fluid is incompressible.
- The viscous dissipation is negligible.
- Physical properties are considered independent of temperature and concentration except for viscosity and density in the buoyancy term. The density obeys the Boussinesq approximation [8].
- The third dimension of the cavity is large enough to consider the problem as two dimensional [7].

For dimensionless variables, we use the characteristic scales: H' , $\rho(\alpha^2/H'^2)$, H'^2/α , α/H' , $q'H'/\lambda$, and $j'H'/D$ corresponding to length, pressure, time, velocity, characteristic temperature, and characteristic concentration, respectively, where ρ , α , λ and D are fluid density, thermal diffusivity, thermal conductivity, and mass diffusivity, respectively. Accordingly, and by introducing the stream function Ψ and the vorticity Ω which can be written as follows:

$$u = \frac{\partial \Psi}{\partial y}; \quad v = -\frac{\partial \Psi}{\partial x} \quad (1)$$

$$\Omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \quad (2)$$

the dimensionless equations governing the convection become:

$$\frac{\partial \Omega}{\partial t} + \frac{\partial(u\Omega)}{\partial x} + \frac{\partial(v\Omega)}{\partial y} = Pr \left[\left(\frac{\partial^2 \Omega}{\partial x^2} + \frac{\partial^2 \Omega}{\partial y^2} \right) + Ra_T \left(\frac{\partial T}{\partial x} + N \frac{\partial S}{\partial x} \right) \right] \quad (3)$$

$$\frac{\partial T}{\partial t} + \frac{\partial(uT)}{\partial x} + \frac{\partial(vT)}{\partial y} = \left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right] \quad (4)$$

$$\frac{\partial S}{\partial t} + \frac{\partial(uS)}{\partial x} + \frac{\partial(vS)}{\partial y} = \frac{1}{Le} \left[\frac{\partial^2 S}{\partial x^2} + \frac{\partial^2 S}{\partial y^2} \right] \quad (5)$$

$$\frac{\partial^2 \Psi}{\partial x^2} + \frac{\partial^2 \Psi}{\partial y^2} = -\Omega \quad (6)$$

associated with the following boundary conditions:

$$u = v = 0; \quad \frac{\partial T}{\partial x} + 1 = \frac{\partial S}{\partial x} + 1 = 0 \quad \text{for } x = 0 \text{ and } x = A; \quad (7)$$

$$u = v = 0 \text{ and } \frac{\partial T}{\partial y} = \frac{\partial S}{\partial y} = 0 \text{ for } y = 0; \quad (8)$$

$$u - Pe = v = 0 \text{ and } \frac{\partial T}{\partial y} = \frac{\partial S}{\partial y} = 0 \text{ for } y = 1 \quad (9)$$

Consequently, the following dimensionless parameters appear:

$$A = \frac{L'}{H'}; \quad Pr = \frac{\nu}{\alpha}; \quad Pe = \frac{u_0 H'}{\alpha}; \quad Ra_T = \frac{g \beta_t H'^4 q'}{\nu \alpha \lambda}; \quad (10)$$

$$Le = \frac{\alpha}{D}; \quad N = \frac{\beta_S \Delta S^*}{\beta_T \Delta T^*}$$

corresponding to: aspect ratio of the enclosure A , Prandtl number Pr , Peclet number Pe , thermal Rayleigh number Ra_T , Lewis number Le , and buoyancy ratio N , respectively.

Given that numerical and analytical results will be compared, and to avoid problems related to complex flows due to edge effects, local Nusselt and Sherwood numbers Nu and Sh , respectively, are evaluated far from the cavity end sides to quantify heat and mass transfer rates.

$$Nu(y) = -1/(\partial T / \partial x)_{x=A/2} \quad (11)$$

$$Sh(y) = -1/(\partial S / \partial x)_{x=A/2} \quad (12)$$

The average horizontal Nusselt and Sherwood numbers representing the overall horizontal heat and mass transfer, respectively, can thus be deduced:

$$\overline{Nu} = \int_0^1 Nu(y) dy \quad (13)$$

$$\overline{Sh} = \int_0^1 Sh(y) dy \quad (14)$$

3. Numerical and Analytical Approach

To Numerically solve the 2-D governing equations, we adopt the second-order central finite-difference scheme with a uniform mesh. The integrations of vorticity, energy, and concentration equations (3)-(5) are performed using the alternating-direction implicit method (ADI) commonly used for Newtonian fluids. To satisfy the conservation of mass, the Poisson equation (6) is solved by a point successive over-relaxation method (PSOR). For $A = 24$ (found as the smallest value of A beyond which mixed convection heat and mass transfer does not change), trial-and-error tests are conducted to find the best trade-off between computation time and solution accuracy. For our case, a uniform grid size of 341×81 is deemed sufficient to model accurately the fluid flow, temperature, and concentration distributions.

The flow is found to be parallel with respect to the horizontal boundaries of the cavity; while temperature and concentration fields are linearly stratified in the x -direction. These observations confirm the existence of an analytical solution based on the parallel flow approximation. For more details on both approaches, numerical and analytical, refer to Tizakast et al. [9].

4. Results and Discussion

Modified Richardson number Gr/Re^n , known also as mixed convection parameter, compare buoyancy forces to shear force due to moving walls. Thus, it is appropriate for delineating the dominance regions of natural, mixed, and forced convection by comparing natural and forced regime contributions in overall convection.

The majority of the works reported on the subject only present results for pure convection (natural or forced). However, separating the cases where the convective regime may be considered as either natural or forced from those where it may be classified as mixed is essential for understanding the different convective regimes. Here, the convective regime will be considered as dominant natural or forced convection if heat transfer and mass transfer rates deviate by no more than 5% from the values computed for the completely pure natural or forced convection, respectively. Furthermore, and in order to highlight the implications of mixed convection parameter alone, we set Lewis number to unity ($Le = 1$), which results in the same diffusion rate for both heat and mass transfer. Thus, to separate the three convective regimes, the following relative differences are introduced:

$$\varepsilon_{Nu_n} = \frac{|\overline{Nu} - \overline{Nu}_n|}{\overline{Nu}_n}; \quad \varepsilon_{Nu_f} = \frac{|\overline{Nu} - \overline{Nu}_f|}{\overline{Nu}_f} \quad (15)$$

where \overline{Nu}_n and \overline{Nu}_f are the mean Nusselt numbers corresponding to pure natural and pure forced convection, respectively. Natural convection dominates for $\varepsilon_{Nu_n} < 5\%$ while forced convection is predominant for $\varepsilon_{Nu_f} < 5\%$. Otherwise, the regime is classified as mixed. Relying on Eq. (15), the diagram presented in Fig.2 is constructed with Pe , Ra_T , and N varying in their respective ranges. Points ($\log(Ra_T)$, $\log(Pe)$) found analytically (solid lines) and numerically (symbols) and corresponding approximately to ($\varepsilon_{Nu_n} = 5\%$) and ($\varepsilon_{Nu_f} = 5\%$) are aligned in parallel straight lines. Such

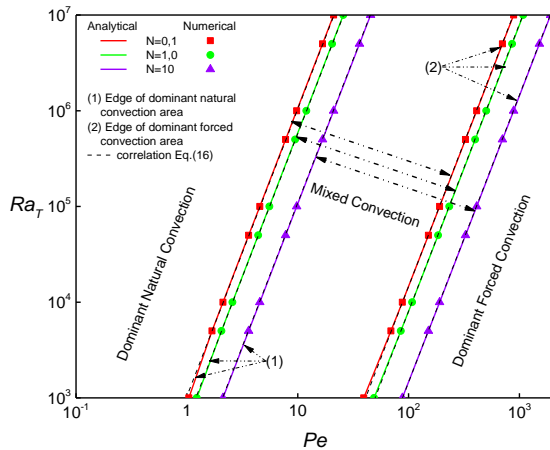


FIGURE 2. Diagram characterizing the dominance regions of natural, mixed, and forced convective regimes for different values of N .

results can be correlated in the form of the mixed convection parameter as follows:

$$\frac{Ra_T}{Pe^{3.0}} = \eta_n \text{ and } \frac{Ra_T}{Pe^{3.0}} = \eta_f \quad (16)$$

for natural and forced convection, respectively, where η_n and η_f values are given in Eqs. (17)–(19) below and shown in Fig. 2 with dashed lines.

The perfect agreement between numerical and analytical results, as shown in the figure, validates both numerical code and analytical approach. The first zone located above line (1) exhibits a dominating natural convection regime. The second zone, located below line (2) is characterized by a dominating forced convection regime. Finally, the third zone between the two straight lines is associated with a regime where buoyancy forces and shear force are of comparable magnitudes (mixed convection regime).

As for the influence of governing parameters on the convective regimes' dominant areas, it is observed that as Ra_T increases, the shift from dominant natural to dominant forced convection while crossing the mixed convection region requires higher values of Pe associated with higher shear force. This due to Ra_T effect on enhancing buoyancy forces, which results in stronger natural convection; thus, in order to make the transition, higher shear force is needed. As for the buoyancy ratio, it is clearly delaying the transition from natural to forced regime as it increases, where for the same value of Ra_T and as N rises, the associated value of Pe when making the transition increases. Such result can be explained by the fact that increasing buoyancy ratio enhances solutal volume force which dominates the moving walls effect. Overall, for single lid-driven rectangular cavity filled with Newtonian fluid, Ra_T and N enhances natural convection contribution in the overall convective regime while Pe strengthens forced regime contribution.

To recap, mixed convection regime in a single lid-driven rectangular cavity filled with a Newtonian fluid and submitted to uniform density of heat and mass fluxes from its short vertical sides is delineated as follows:

$$0.014598 < \frac{Ra_T}{Pe^{3.0}} < 1037.856 \text{ for } N = 0.1 \quad (17)$$

$$0.008051 < \frac{Ra_T}{Pe^{3.0}} < 575.5498 \text{ for } N = 1.0 \quad (18)$$

$$0.001459 < \frac{Ra_T}{Pe^{3.0}} < 108.6276 \text{ for } N = 10 \quad (19)$$

5. Conclusions

This paper investigates double-diffusive mixed convection inside a horizontal rectangular cavity filled with Newtonian fluid. The two established separate solutions, numerical and analytical, showed perfect agreement for an inclusive range of the governing parameters, validating the numerical code, the parallel flow approximation, and the choice of $A = 24$ as the large aspect ratio approximation value. Thus, the main findings can be listed as follows:

- The mixed convection parameter $\frac{Ra_T}{Pe^{3.0}}$ perfectly outlines the three separate convective regimes, namely natural, mixed, and forced convection.
- Thermal Rayleigh number and buoyancy ratio both enhance natural convection input in the overall convection, where the former intensifies thermal and solutal buoyancy forces while the latter enhances solutal volume force.
- Peclet number enhances forced regime input in overall convection as it strengthens shear force generated by the moving walls.

References

- [1] M. A. Teamah et al., *Numerical simulation of double-diffusive mixed convective flow in rectangular enclosure with insulated moving lid*, *Int J Therm Sci* 49 (2010), pp. 1625–38.
- [2] S. D. Kumar et al., *Numerical Simulation of Double Diffusive Mixed Convection in a Lid-Driven Square Cavity Using Velocity-Vorticity Formulation*, *Numerical Heat Transfer, Part A: Applications* 54 (2008), pp. 837–65.
- [3] S. Hussain et al., *Numerical simulation of double diffusive mixed convective nanofluid flow and entropy generation in a square porous enclosure*, *Int J Heat Mass Transfer* 122 (2018).
- [4] A. Arani et al., *Numerical simulation of double-diffusive mixed convection in an enclosure filled with nanofluid using Bejan's heatlines and masslines*, *Alexandria Eng J* (2017).
- [5] N. Nithyadevi et al., *Non-Darcy double diffusive mixed convection for nanofluid with Soret and Dufour effects in a lid-driven cavity*, *Int J Nanoparticles* 8 (2015), pp. 218–40.
- [6] M. A. Sheremet et al., *Mixed convection in a lid driven square cavity filled by a nanofluid: Buongiorno's mathematical model*, *Appl Math Comput* 266 (2015), pp. 792–808.
- [7] D. A. Siginer et al., *On the laminar free convection and stability of grade fluids in enclosures*, *Int J Heat Mass Transfer* 43 (2000), pp. 3391–405.
- [8] D. D. Gray et al., *The validity of the Boussinesq approximation for liquids and gases*, *Int J Heat Mass Transfer* 19 (1976), pp. 545–51.
- [9] Y. Tizakast et al., *Thermosolutal Mixed Convection in Shallow Rectangular Cavity with Imposed Uniform Heat and Mass Fluxes and Filled with Newtonian Power-Law Fluid*, *Inter J Heat and Technology*, 39 (2021), pp. 669–680.

Effet de l'insertion des Matériaux à Changement de Phase sur l'amélioration du confort thermique dans les bâtiments

O. Imghoure¹, S. Ouhaibi², N. Belouaggadia², M. Ezzine³, R. Lbibb⁴, Z. Younsi⁵

1. Laboratory of Engineering and Materials (LIMAT), Faculty of Sciences Ben M'Sik, Hassan II University of Casablanca, B.P 7955 Sidi Othmane, Casablanca, Morocco. Email: imghoure.oumaïma@gmail.com
2. Laboratory of signals, distributed systems and artificial Intelligence, ENSET, Hassan II University, Mohammedia, Morocco
3. Laboratory of Physico-chemistry of Applied Materials (LPCMA), Faculty of Sciences Ben M'Sik, Hassan II University of Casablanca, B.P 7955 Sidi Othmane, Casablanca, Morocco
4. Laboratory of Nanostructure and advanced materials, Mechanics and thermofluid, Faculty of Sciences and Technics, Hassan II University, Mohammedia, Morocco.
5. Univ. Lille, IMT Lille Douai, Univ. Artois, Yncrea Hauts-de-France, ULR 4515 – LGCgE, Laboratoire de Génie Civil et géo-Environnement, F-59000 Lille, France

Résumé

L'objectif visé est de disposer d'un modèle numérique permettant de réaliser une étude comparative de différents matériaux à changement de phase. Ce travail expose les principaux résultats d'une étude numérique du comportement thermique d'un mur multicouche soumis à l'irradiation solaire et aux conditions climatiques caractéristiques de la ville de Marrakech. Les résultats montrent que l'amplitude des variations de la température diminue de façon significative en intégrant le MCP dans les murs. Les MCP Bio-PCM23 et Climsel24 présentent une meilleure amélioration de la performance énergétique du mur. Ils permettent de réguler la température de la face interne autour de leurs températures de fusion respectives (23 °C et 24 °C).

Mots clés : *Matériaux à changement de phase ; Performance thermique ; Fusion.*

1. Introduction

La prise de conscience de l'importance de l'efficacité énergétique et de l'évidence des liens entre environnement et développement dans le monde est de plus en plus grandissante.

Dans le cas du Maroc, l'adoption de mesures drastiques pour le développement de l'efficacité énergétique, des énergies renouvelables et la réduction des émissions de gaz à effet de serre en est l'illustration. Au niveau politique, l'adoption d'une stratégie d'efficacité énergétique constitue un signal réel et vise à réaliser une économie d'énergie d'environ 20% à l'horizon 2030 à travers une meilleure utilisation de l'énergie dans tous les domaines d'activité économique et sociale [1].

Le bâtiment est l'un des grands secteurs consommateurs d'énergie au Maroc, il représente 33% de la consommation énergétique finale et enregistre une forte croissance de la consommation énergétique annuelle.

La conception de constructions bioclimatiques et énergétiquement efficaces apparaît comme l'une des actions les plus urgentes à mener pour mettre en œuvre la stratégie nationale en matière d'énergies renouvelables et d'efficacité énergétique. Une diminution significative des besoins de chauffage et de refroidissement du bâtiment ainsi qu'une amélioration du confort thermique peuvent être obtenues grâce à une isolation thermique performante et l'utilisation de certains matériaux de

construction ainsi qu'une conception architecturale appropriée.

Pour toutes ces raisons, les recherches menées aujourd'hui dans le domaine de l'efficacité énergétique des bâtiments vise à élaborer un ensemble de matériaux capables d'assurer une meilleure enveloppe en termes d'efficacité et de durabilité [2-4]. En outre, l'amélioration de l'enveloppe du bâtiment passe également par le recours aux énergies renouvelables à travers l'intégration du stockage thermique aux composants de l'enveloppe ; c'est le cas par exemple, du stockage thermique via des matériaux à changement de phase (MCP). Ces matériaux présentent l'avantage et la particularité de stocker et de libérer la chaleur grâce au changement d'état du matériau qui les constituent. Ces recherches ont montré également un grand intérêt pour les MCP en termes de gain énergétique durant toutes les périodes de l'année, étant donné le critère adaptatif aux changements de climat.

La présente étude vise donc à considérer les paramètres caractéristiques du mur contenant des MCP afin d'établir leurs influences sur sa performance thermique et réaliser un choix très judicieux permettant la réussite de l'intégration du MCP utilisé.

Une modélisation mathématique basée sur la méthode enthalpique a été utilisée pour calculer l'évolution de la température à travers un mur sandwich qui contient un MCP. Il est soumis à des modulations thermiques variables sur la face externe représentative d'un flux thermique solaire afin de voir son effet sur l'ambiance interne de l'habitat. Le modèle proposé est résolu numériquement en utilisant la méthode des volumes finis.

2. Modélisation et intégration d'un MCP dans un mur d'un bâtiment

2.1 Position du problème

On considère un mur appartenant à l'enveloppe externe d'un bâtiment dans la région de Marrakech. Il s'agit d'une structure multicouche d'épaisseur $L=24\text{cm}$, composée de trois couches (fig.1), une couche centrale en Brique, d'épaisseur e_B en interface des deux côtés avec une couche de mortier d'épaisseur $e_m = 2\text{cm}$. Dans la deuxième configuration (fig.2), une couche de MCP est introduite sur sa face extérieure de mur d'une épaisseur de 2 cm.

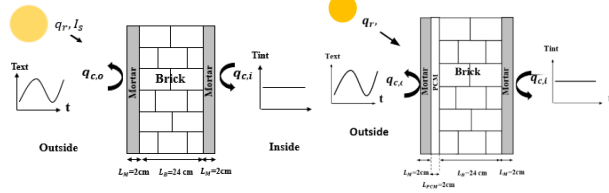


Fig.1 Mur de référence

Fig.2 Mur avec MCP

Matériaux	Température de fusion T_m (°C)	Chaleur latente L (kJ/Kg)	Chaleur spécifique solide C_{ps} (kJ.kg ⁻¹ .K ⁻¹)	Conductivité thermique (solide/liquide) λ_c (W.m ⁻¹ .K ⁻¹)	Densité volumique (solide) ρ_s (kg.m ⁻³)
MCP1 (CaCl ₂ .6H ₂ O)	29.9	187	1.4	0.53/1.09	1710
MCP 2 (Bio-PCM-Q23)	22-24	245.5	1.822	0.19/0.21	235
MCP 3 (Polyéthylène Ghilol 900)	34	150.5	2.26	0.19/0.19	1200
MCP 4 (RT27)	28	179	1.8	0.2	870
MCP 5 (Clim-sel C24)	24	108	3.6	0.7/0.5	1380

Tableau 1. Propriétés thermo physiques des matériaux [5].

3. Formulation mathématique

Nous proposons ici, un modèle mathématique qui décrit le comportement thermique du mur intégrant le MCP. L'étude est réalisée pour un mur exposé plein sud, le modèle de Perez est utilisé pour le calcul du flux de rayonnement solaire intercepté sur le mur extérieur.

$$\rho_j C_{pj} \frac{\partial T_j}{\partial t} = \lambda_j \frac{\partial^2 T_j}{\partial x^2} \quad (1)$$

L'équation (1) est valable pour les cas de matériaux entièrement solides ou liquides. Cependant, pour tenir entièrement du processus de changement de phase (fusion ou solidification) de la couche MCP, il faut inclure un terme supplémentaire (Eq2).

$$\rho_j C_{pj} \frac{\partial T_j}{\partial t} = \lambda_j \frac{\partial^2 T_j}{\partial x^2} - \rho_j L_f \frac{\partial f}{\partial t} \quad (2)$$

Où L_f est la chaleur latente et f_l correspond à la fraction liquide du MCP que l'on définit comme suit :

$$f_l = \begin{cases} 0 & \text{si } T < T_{fusion} \\ [0 - 1] & \text{si } T = T_{fusion} \\ 1 & \text{si } T > T_{fusion} \end{cases} \quad (3)$$

Les conditions aux limites incluent le transfert de chaleur radiatif et convectif combinés à l'extérieur et le transfert de chaleur par convection à l'intérieur, ils sont donnés par les équations (4) et (6) suivantes :

$$-k_1 \left. \frac{\partial T}{\partial x} \right|_{x=x_p} = h_{ext} (T_a - T_1) + \alpha I_s + q_r \quad (4)$$

Les échanges radiatifs et convectifs se définissent de la manière suivante :

$$q_r = \sigma \varepsilon (T_{sky}^4 - T_1^4) \quad (5)$$

$$-k_n \left. \frac{\partial T}{\partial x} \right|_{x=x_l} = h_{int} (T_n - T_2) \quad (6)$$

Où h_{int} et h_{ext} représentent des coefficients de transfert de chaleur sur les surfaces intérieurs et extérieurs de la paroi. T_{in} est la température intérieure.

T_a est la température de l'air extérieur, α_s est le coefficient d'absorption solaire de la surface externe, I_s est le rayonnement solaire sur la surface du mur extérieur, ε est l'émissivité de la surface du mur et σ est la constante Stephan-Boltzmann (5.65×10^{-8} W/m².K⁴).

L'équation directrice ainsi que les conditions aux limites sont discrétisées par la Méthode de Volume finis.

Le système d'équations est résolu itérativement en utilisant l'algorithme TDMA complété par une élimination directe et une substitution en arrière.

4. Effet de l'intégration d'un MCP sur la performance thermique d'un mur

Les résultats de calcul, représentés sur la fig. 3, montrent que l'intégration d'un MCP dans la structure d'un mur affecte considérablement le profil de la température à la face interne du mur. Suivant le type de MCP utilisé, elle varie entre 23°C et 25,5 °C avec une amplitude d'oscillation plus accentuée pour CaCl₂-6H₂O et RT 27. Les MCP Bio-PCM-Q23 et Climsel24 présentent une meilleure amélioration de la performance énergétique du mur. Ils permettent de réguler la température de la face interne autour de leurs températures de fusion.

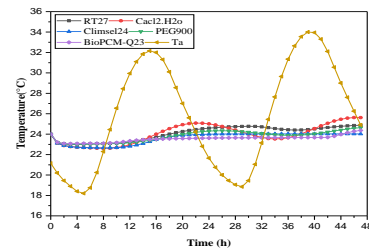


Fig. 3 Variation de la température à la surface intérieure pour différents types de MCP

Le profil de température à la face interne du mur dépend de celui à l'interface interne du MCP. La variation de la température, observée à la face interne du mur pour les MCP PEG900, CaCl₂-6H₂O et RT 27 montre que la température de ces derniers varie en fonction du temps, ce qui n'est pas compatible avec la coexistence des deux états physiques dans ces MCP lors du transfert de chaleur entre le milieu extérieur et l'intérieur du bâtiment. Pour comprendre les résultats obtenus, nous avons représenté sur les figures ci-dessous les courbes d'évolution temporelle de la température des différentes couches relative à chaque MCP étudié ainsi que celle de la fraction liquide.

Pour le MCP Climsel 24, Le MCP commence à fondre quand la température de l'interface externe atteint la température de fusion. Le changement de phase du MCP a lieu (fig. 5). Lors de phase descendante de la température extérieure, le changement d'état inverse va avoir lieu dès que la température de l'interface externe atteint la température de fusion du MCP, d'abord dans les couches limitrophes à l'interface externe puis de proche en proche les autres couches. Pendant le deuxième cycle d'évolution de la température extérieure, le même phénomène a lieu mais avec une vitesse de transformation d'état plus faible que le premier. Les courbes de variation de la fraction du liquide dans le MCP représentées sur la fig. 5, montrent qu'à partir de 17h, les deux états physiques coexistent dans le MCP, ce qui impose une température constante à l'interface interne du MCP, égale à sa température de fusion (fig.4 10), et explique l'évolution temporelle de la température de la face interne du mur (fig.3).

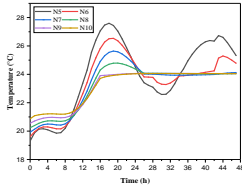


Fig. 4 Température de la face interne relative au climsel24

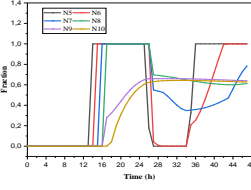


Fig. 5 Variation de la fraction liquide du Climsel24

Le MCP BioPCM-Q23 présente le même comportement que le Climsel24 avec une avance de fusion due à sa faible capacité calorifique volumique par rapport à celle de Climsel24 (fig. 6). Entre 12h et 42h, il y a coexistence des deux états physiques du MCP (fig. 7) et la température de l'interface interne reste constante, égale à la température de fusion (fig. 6). A partir de 42h, le MCP est transformé complètement à l'état liquide et on observe une augmentation de la température qui atteint la valeur de 25 °C. La température de la face interne du mur varie de la même manière que l'interface interne du MCP avec un écart dû à la résistance thermique des couches de brique et de mortier.

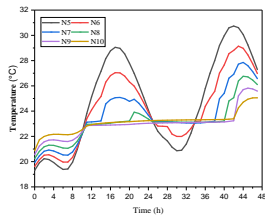


Fig. 6 Température de la face interne relative au BioPCM-Q23

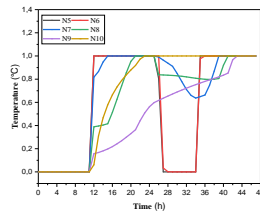


Fig. 7 Variation de la fraction liquide du BioPCM-Q23

Pour le RT 27 seulement les deux premières couches qui se transforment, totalement ou en partie, en liquide pendant la phase ascendante de la température extérieure (fig. 9), expliqué par une grande capacité calorifique volumique et une grande densité énergétique (fig. 8). La température présente un palier puis suit la même évolution que la température extérieure quand tout le MCP se transforme en solide. A partir de 40h, une partie de la couche 7 se transforme à l'état liquide et un deuxième palier est observé sur les courbes d'évolution de la température, à la température de fusion pour la couche 7 et à une température inférieure pour les autres couches.

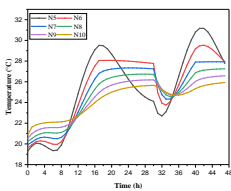


Fig. 8 Température de la face interne relative au RT27

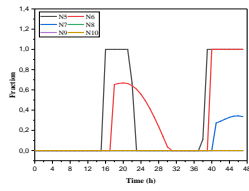


Fig. 9 Variation de la fraction liquide du RT27

Au cours du transfert de chaleur du milieu extérieur vers l'intérieur, les températures des MCP PEG 900 et CaCl2-

6H2O n'atteignent pas leurs températures de fusion respectives (34°C et 29,9 °C). Le MCP reste à l'état solide (fig. 10) et se comporte, alors, comme une simple couche de solide. L'allure de la courbe de variation de température est similaire que celle du milieu extérieur avec une amplitude plus faible due à un stockage par chaleur sensible (figures 8 et 9). L'amplitude d'évolution de la température de l'interface interne du MCP PEG 900 est plus faible que celle de CaCl2-6H2O du fait de la grande capacité calorifique volumique du premier par rapport au deuxième.

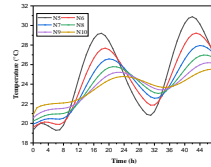


Fig. 8 Température de la face interne relative au PEG900

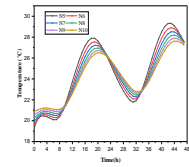


Fig. 9 Température de la face interne relative au CaCl2.H2o

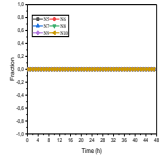


Fig. 10 Variation de la fraction liquide du PEG900 et CaCl2.H2o

Conclusion

Dans cette étude, l'effet de l'intégration des matériaux à changement de phase en tant qu'un isolant thermique, sur l'évaluation de la température de la surface intérieure ont été étudié numériquement. Les configurations étudiées ont été simulées sous les conditions climatiques de Marrakech.

Les résultats montrent que l'amplitude des variations de la température diminue de façon significative en intégrant le MCP dans les murs. Le BioPCM-Q23 joue un rôle déterminant dans l'amélioration du rendement énergétique du mur et serait considéré dans ce cas comme un écran thermique entre un milieu à température constante donnée par la température de fusion du MCP et l'intérieur bâtiment.

Références

- [1] <http://www.mhpb.gov.ma/>
- [2] Shukla, N., Fallahi, A., & Kosny, J. (2012). Performance characterization of PCM impregnated gypsum board for building applications. *Energy Procedia*, 30, 370-379.
- [3] Tyagi, V. V., Kaushik, S. C., Tyagi, S. K., & Akiyama, T. (2011). Development of phase change materials based microencapsulated technology for buildings : A review. *Renewable and Sustainable Energy Reviews*, 15(2), 1373-1391.
- [4] Liu, Z., Yu, Z. J., Yang, T., Qin, D., Li, S., Zhang, G., ... Joybari, M. M. (2018). A review on macro-encapsulated phase change material for building envelope applications. *Building and Environment*.
- [5] IMGHOUR, O., BELOUAGGADIA, N., EZZINE, M., et al. Performance evaluation of phase change materials for thermal comfort in a hot climate region. *Applied Thermal Engineering*, 2021, vol. 186, p. 116509.

Stockage de chaleur dans les matériaux à changement de phase application CSP

F. Redoine¹, N. Belouaggadia², K. Mansouri³

1. LIMAT, Faculté des sciences Ben Msik, université HASSAN II, Maroc, redoine.fatima@gmail.com
2. LSDSAI, ENSET, université HASSAN II, Mohammedia, Maroc, n.belouaggadia@gmail.com
3. LSDSAI, ENSET, université HASSAN II, Mohammedia, Maroc, khmansouri@hotmail.com

Résumé

Cet article rapporte l'étude comparative des différents matériaux à changement de phase (MCP) en intégrant dans un système de stockage par chaleur latente, dans le but d'améliorer les performances de stockage. Un modèle bidimensionnel pour un système de stockage a été développé. La sélection de six MCP différents ayant une chaleur latente de fusion élevée et une gamme de températures de fusion a été étudiée. Nous avons constaté que le MCP $Mg_{84}Cu_{16}$ l'un des matériaux les plus promoteurs pour les applications de stockage thermique en raison de la haute conductivité thermique rapportée jusqu'à présent et son temps de chargement est plus rapide que d'autre matériaux.

Mots clefs : Matériaux à changement de phase (MCP) ; Centrale solaire à concentration thermodynamique (CSP) ; magnésium et le cuivre ($Mg_{84}Cu_{16}$) ;

1. Introduction

L'un des principaux leviers technologiques permettant le développement industriel des procédés de production d'énergie renouvelable et à haut rendement consiste à développer une solution innovante de stockage de l'énergie. Ce système de stockage doit permettre de lisser la période de production et ainsi de suivre au plus près les besoins des consommateurs.

Parmi les solutions existantes, le stockage thermique par chaleur latente présente de nombreux avantages, ce qui en fait l'objet de plusieurs travaux de recherche aujourd'hui et en développement. Cette technologie repose sur le principe que certaines classes de matériaux, appelés matériaux à changement de phase (MCP), libèrent (transition liquide/solide) ou accumulent (transition solide/liquide) de l'énergie lorsqu'ils sont soumis à un changement de phase [1]. L'utilisation d'un système de stockage de la chaleur latente utilisant des matériaux à changement de phase (MCP) est une méthode efficace de stockage de l'énergie thermique.

Néanmoins, ces dernières années, un effort de recherche important a été fait pour trouver un système TES approprié en utilisant des matériaux à changement de phase (MCP) [2]. Il n'existe pas encore de centrale CSP commerciale fonctionnant avec un tel système. Toutes ces approches [3] utilisent des sels fondus comme MCP. Il est bien connu que le principal inconvénient des sels fondus est leur faible conductivité thermique qui conduit à l'utilisation d'échangeurs de chaleur sophistiqués. Pour

surmonter cet inconvénient, l'idée d'utiliser des alliages métalliques eutectiques a été reprise [4].

2. Modélisation et intégration du MCP dans un système de stockage sous forme cylindrique

2.1 Position du problème

Dans ce travail on va présenter le modèle mathématique et l'implantation numérique d'un réservoir de stockage thermique utilisant les matériaux à changement de phase. La géométrie simulée est présentée à la Figure 1. Il s'agit d'un conteneur cylindrique rempli de MCP. La géométrie se compose de deux cylindres concentriques, Le tube externe qui englobe le système et contenant le MCP et la conduite interne dans laquelle circule le fluide caloporteur HTF. Ce fluide entre à une extrémité à une température T_{in} , récupère une partie de la chaleur stockée dans les MCP et ressort à l'autre extrémité avec une température T_{out} plus élevée. Le réservoir est isolé pour limiter les pertes/gains de chaleur.

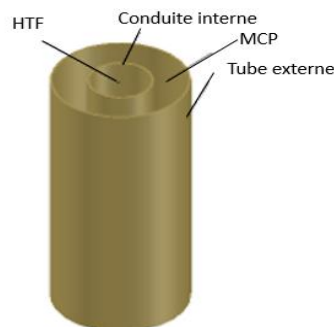


Figure 1 : Géométrie du conteneur de MCP

Les MCP utilisés dans cette étude sont le lithium nitrate trihydrate ($LiNO_3 \cdot H_2O$), le sodium carbonate decahydrate $Na_2CO_3 \cdot 10H_2O$, le sodium sulfate decahydrate ($Na_2SO_4 \cdot 10H_2O$), la cire Paraffin RT 28, le calcium chloride hexahydrate ($CaCl_2 \cdot 6H_2O$) et l'alliage métallique magnésium cuivre ($Mg_{84}Cu_{16}$). Leurs propriétés physiques sont données dans le tableau 1.

MCPs	T_m [K]	L [KJ/Kg]	C_{ps} [J/Kg/K]	C_{pl} [J/Kg/K]	λ_s [W/m/K]	λ_l [W/m/K]	ρ_s [Kg/m ³]	ρ_l [Kg/m ³]	Ref.
$Mg_{84}Cu_{16}$	761.15	232	788.39	905.34	106.99	39.23	2459.90	2178.85	[1]
$LiNO_3 \cdot H_2O$	303	296	1730	2770	1.32	0.58	2140	1780	[2]
$Na_2CO_3 \cdot 10H_2O$	305	267	1880	3350	1.25	0.56	1460	1442	[2]
$Na_2SO_4 \cdot 10H_2O$	305.5	251	1440	2570	1.23	0.544	1485	1420	[2]
paraffin RT28	301	245	1800	2400	0.2	0.2	880	760	[2]
$CaCl_2 \cdot 6H_2O$	302.6	192	1430	2310	0.54	0.54	1802	1562	[2]

Tableau 1 : Les propriétés physique des MCP

2.2 Formulation mathématique des équations gouvernantes

Le problème du changement de phase des MCP est un problème dit de « frontière mobile (moving boundary problem) ». Lors du changement de phase, les deux phases solide et liquide existent et sont séparées par une interface mobile. Plusieurs méthodes numériques ont été développées pour résoudre ce genre de problème (Bansal et al., 1992; Muray et al., 1959; Soma et al., 1993). Toutefois, la plus attrayante de ces méthodes et la plus communément utilisée et appelé la méthode « enthalpie » (Voller, 1990) [5]. Plusieurs approches ont été utilisées pour modéliser les unités de stockage thermique utilisant les matériaux à changement de phase. Dans la présente étude, le modèle mathématique pour résoudre le problème de changement de phase utilise les hypothèses suivantes :

- (1) Les effets de la convection naturelle dans la partie liquide sont considérés négligeables et sont ignorés.
- (2) Le MCP se comporte idéalement : les phénomènes comme la dégradation ou le sous refroidissement ne sont pas pris en compte.
- (3) On considère que le MCP a une température de fusion bien définie (changement de phase isotherme).
- (4) Le MCP est homogène et isotrope.
- (5) Les propriétés du MCP sont différentes pour les phases solide et liquide mais sont indépendantes de la température.
- (6) La surface externe de paroi est supposée adiabatique.
- (7) La paroi du tube intérieur est supposée mince et le matériau qui la constitue est supposé un bon conducteur thermique.

Il est possible de supposer que la variation de température et de fraction de liquide dans les directions y et z est négligeable. Donc, seule la conduction selon x est considérée. Il s'agit donc d'un problème de conduction à une dimension avec changement de phase. Comme l'a décrit Voller (1990), l'équation du transfert de chaleur pour le changement de phase contrôlé par la conduction s'écrit comme suit :

$$\rho \frac{\delta H}{\delta t} = \text{div}(\lambda \text{grad}T) \quad (1)$$

Une version modifiée de l'équation (1) est obtenue en séparant l'enthalpie H en deux composantes, soit en enthalpie sensible, h , et en enthalpie latente, Lf_1 :

$$H = h + Lf_1 \quad (2)$$

$$\text{avec } h = \int_{T_m}^T c_p dT \quad (3)$$

Où T_m est la température de fusion du MCP. En considérant un changement de phase isotherme, la fraction liquide f_1 est définie comme suit :

$$f_1(T) = \begin{cases} 0, & T < T_m \\ 1, & T > T_m \end{cases} \quad (4)$$

L'équation a une dimension obtenue directement à partir de l'équation (1), est donc :

$$\rho C_p \frac{\partial T}{\partial t} = \lambda \left(\frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial T}{\partial r} \right) \right) - L \frac{\partial f_1}{\partial t} \quad (5)$$

avec ρ la densité volumique, C_p est la capacité thermique et λ la conductivité thermique du MCP.

2.3 Modèle Numérique

Pour résoudre le problème de transfert de chaleur à une dimension dans un MCP encapsulé dans un conteneur (voir Figure 1), une solution par volumes finis avec un maillage uniforme de type B (notation de Patankar (1980)) [6] est utilisée. La particularité de ce type de maillage est que tous les volumes de contrôle (VC) ont un volume égal. Les frontières des VC sont uniformément positionnées et ensuite les nœuds sont placés aux extrémités ainsi qu'au centre de chaque VC ainsi créé. Il en résulte que les nœuds aux frontières ont des VC à volume nul et que chacun des autres a un volume égal. La Figure 2 illustre ce type de maillage.

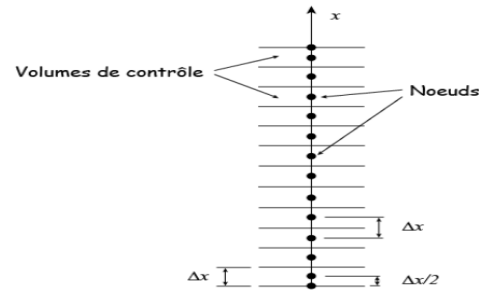


Figure 2 : Illustration d'un maillage de type B selon la notation de Patankar (1980)

Le cas d'un volume de contrôle i qui est complètement solide ou liquide est considéré. Dans ce cas, d'après la définition de l'enthalpie sensible h_i , équation (3), et de la fraction liquide f_1 , équation (5), il s'en suit que :

$$\frac{\partial h_i}{\partial t} = C_p \frac{\partial T_i}{\partial t} \quad (6)$$

$$\frac{\partial f_{1i}}{\partial t} = 0 \quad (7)$$

Avec C_p qui représente la chaleur spécifique de la phase solide ou liquide, en fonction de la phase du volume de contrôle considéré. En introduisant les équations (6) et (7) dans l'équation (5) et en utilisant la formulation implicite, l'équation de conduction transitoire devient :

$$a_i T_i = a_{i-1} T_{i-1} + a_{i+1} T_{i+1} + a_i^0 T_i^0 + b \quad (8)$$

Avec

$$a_{i+1} = -\frac{\Delta t}{\Delta h^2} - \frac{\Delta t}{2r\Delta h}$$

$$a_{i-1} = -\frac{\Delta t}{\Delta h^2} + \frac{\Delta t}{2r\Delta h}$$

$$a_i = 1 - a_{i+1} - a_{i-1}$$

$$b = T^{old} + \frac{L}{C_p(T_i - T_s)} T^{old}$$

On a résolu ce système algébrique par la méthode itérative (TDMA) [7] sous MATLAB.

2.4 Résultats et discussion

L'objectif de ce travail est de faire une étude comparative des différents matériaux à changement de phase dans les processus de fusion et de solidification, à titre d'exemple, la Figure 3 montre le profil de température des six MCP : le lithium nitrate trihydrate ($\text{LiNO}_3\text{H}_2\text{O}$), le sodium carbonate decahydrate $\text{Na}_2\text{CO}_3\cdot 10\text{H}_2\text{O}$, le sodium sulfate decahydrate ($\text{Na}_2\text{SO}_4\cdot 10\text{H}_2\text{O}$), la cire Paraffin RT 28, le calcium chloride, hexahydrate ($\text{CaCl}_2\cdot 6\text{H}_2\text{O}$) et l'alliage métallique magnésium-cuivre ($\text{Mg}_{84}\text{Cu}_{16}$).

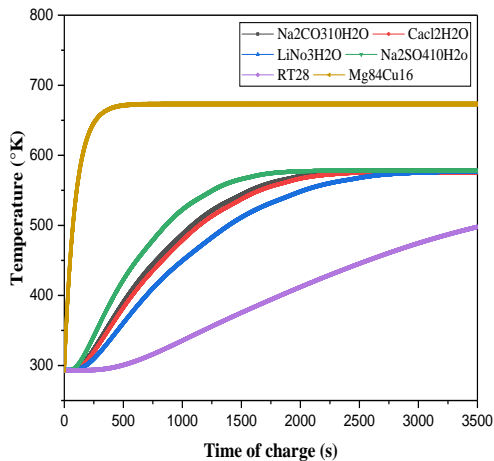


Figure 3 : Evolution de température pendant le processus de fusion des MCPs

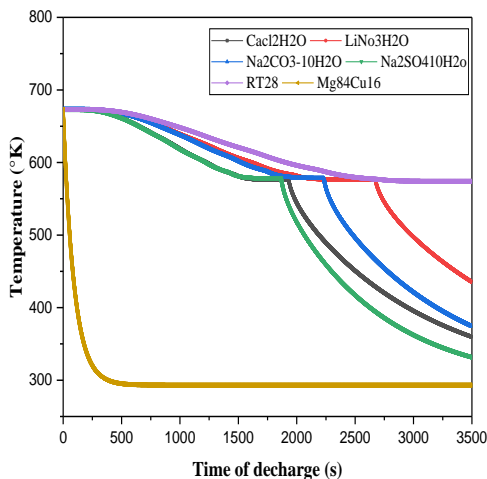


Figure 4 : Evolution de température pendant le processus de solidification des MCPs

Pour le test de fusion, la température d'entrée de l'huile a été fixée à 400 °C avec une vitesse d'huile de 1 m/s) [8]. Pour le test de solidification, la température d'entrée du HTF a été réglée à 20 °C avec la même vitesse d'huile. Dans le chauffage, on peut facilement distinguer trois régions différentes, La première correspond au préchauffage des MCPs où la température de ces MCPs

est progressivement augmentée jusqu'au point de fusion. La seconde correspond principalement au processus de fusion. Et la dernière correspond à la surchauffe du liquide de fusion jusqu'à la valeur fixée pour chaque MCP. Il faut souligner que pendant la fusion, il y a également un certain préchauffage et un certain surchauffage pour l'alliage métallique.

Un comportement similaire est observé dans le processus de refroidissement (Figure 4) D'abord, le refroidissement de la masse fondue liquide. Ensuite, la solidification à une température quasi-constante. Enfin, le refroidissement de la phase solide jusqu'à la température HTF d'entrée.

On remarque d'après les courbes de figure 3, pour l'alliage métallique ($\text{Mg}_{84}\text{Cu}_{16}$) le taux de chauffage s'accélère plus que d'autres matériaux. Pendant la fusion, cette différence de taux de chauffage montre la bonne capacité de ce matériau comme un matériau à haute conductivité thermique pour les applications CSP.

En raison de la plus haute conductivité thermique rapportée jusqu'à présent dans la plage de température de 400 °C-550°C et leur stabilité thermique.

Références

- [1] PERNOT, Eric. Modélisation du changement d'état solide-liquide. Application au stockage thermique par chaleur latente Adapté aux centrales solaires thermodynamiques. 2015. Thèse de doctorat. Pau.
- [2] D. Laing, C. Bahl, T. Bauer, and D. Lehmann, "Thermal energy storage for direct steam generation," *Sol. Energy*, vol. 85, no. 4, pp. 627-633, 2011, doi: 10.1016/j.solener.2010.08.015.
- [3] V. Zipf, A. Neuhäuser, D. Willert, P. Nitz, S. Gschwander, and W. Platzer, "High temperature latent heat storage with a screw heat exchanger : Design of prototype," *Appl. Energy*, vol. 109, pp. 462-469, 2013, doi: 10.1016/j.apenergy.2012.11.044.
- [4] Rodríguez-Aseguinolaza, J., Blanco-Rodríguez, P., Risueño, E., Tello, M. J., & Doppiu, S. (2014). Thermodynamic study of the eutectic Mg49-Zn51 alloy used for thermal energy storage. *Journal of Thermal Analysis and Calorimetry*, 117(1), 93-99.
- [5] V. R. Voller and C. Prakash, "A fixed grid numerical modeling methodology for convection-diffusion mushy region phase-change problems," vol. 30, 1987.
- [6] Baliga, B. R., & Patankar, S. V. (1983). A control volume finite-element method for two-dimensional fluid flow and heat transfer. *Numerical Heat Transfer*, 6(3), 245-261.
- [7] VERSTEEG, H. K. et MALALASEKERA, W. An introduction to computational fluid dynamics. The finite volume method, 1995.
- [8] A. Gil, E. Risueño, B. D. Aguanno, I. Loroño, and L. Martín, "Experiments on a lab scale TES unit using eutectic metal alloy as PCM," *Energy Procedia*, vol. 69, pp. 769-778, 2015, doi: 10.1016/j.egypro.2015.03.087.

Lattice Boltzmann analysis of mixed convection flow in a water based photovoltaic thermal collector

M. Boufkri¹, A. El Mansouri², M. Hasnaoui^{1,*} and A. Amahmid¹

¹UCA, Faculty of Sciences Semlalia, LMFE, BP 2390, Marrakesh, Morocco

²IZU, Faculty of Applied Sciences-Ait Melloul, BP 6146, Agadir, Morocco

*Corresponding author: hasnaoui@uca.ma

Abstract

This paper aims to investigate the effect of the water channel location, vis-à-vis of the PV module in a flat-plate photovoltaic thermal system (PV/T), on heat evacuation by means of mixed convection. Numerical simulations were performed for a two-dimensional inclined channel at an angle of 35°. The cold water enters with a constant temperature and flows below the PV module (Case A) or above it (Case B). Mixed convection inside the tilted channel was solved using the mesoscopic Lattice Boltzmann method (LBM). The Reynolds number was varied in a laminar range ($25 \leq Re \leq 200$) to examine its effect on heat transfer. Results indicate that water flowing above solar cells considerably enhances heat transfer in comparison with the conventional configuration (i.e. Case A).

Keywords: Numerical simulation; photovoltaic thermal system (PV/T); 2D mixed convection; Lattice Boltzmann method (LBM).

1. Introduction

Photovoltaic cells have been a topic of research since the end of the 1830s the Becquerels. However, the maturity has not been completely achieved since it only generates 1.3% of the world's capacity [1]. The worldwide installed power capacity reached up to 227 GW in 2015. Solar cell's overheating is one of the main factors that influences electrical efficiency of the PV panels. In order to mitigate such drawback, hybrid photovoltaic and thermal systems (PVT) were developed to control the temperature of the solar cells. Therefore, PVT collectors provides both thermal and electrical energy.

Massive efforts have been deployed throughout the last decades to investigate heat transfer mechanisms in the PVTs. Kaiser et al. [2] analyzed the impact of the gap size, air velocity and ross coefficient on PV

cells cooling for natural and forced convection cases. They concluded that an aspect ratio of 0.11 can inhibit the overheating of a flat plate PVT system. Also, the electrical power increases by 19.13% in forced convection mode compared to natural convection one.

In the same framework, Babajani et al. [3] studied numerically the mixed convection of nanofluid flow in an inclined PVT panel. The configuration consists of a two-dimensional channel where the left wall is adiabatic and the right wall is heated by four solar cells. Their results showed that heat transfer is enhanced by reducing the inclination angle while choosing a certain relative distance between solar cells.

So far, weak attention has been paid to the use of the lattice Boltzmann method to investigate mixed convection flows taking place in a water based photovoltaic thermal system. Also, most of the previous studies have mainly focused on PVT configurations where the heat transfer fluid (HTF) flows below the PV module. This study seeks to investigate the influence of the PV panel location vis-à-vis of the fluid channel on the mean outlet temperature and heat transfer characteristics.

2. Mathematical Modelling

1.1 Problem description

As shown in fig.1, the configurations under study consists of a two-dimensional inclined channels heated from one side by the hot PV panel. The latter is located above the flow in case A and below the flow in case B. The opposite plate is considered adiabatic (and transparent in case B) while the incoming water and the PV module are maintained, respectively, at a cold and hot temperature.

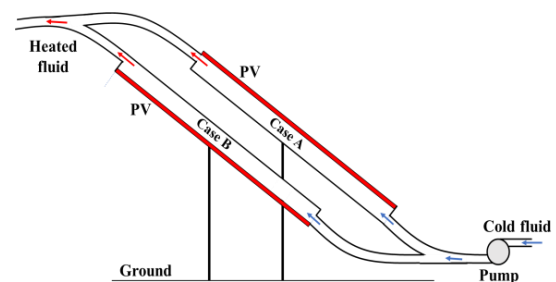


Figure 1. Schematic of the studied PVT solar collectors

Physical assumptions were adopted to simplify the numerical calculations. The flow was considered to be laminar; the fluid is Newtonian and incompressible. In addition, the viscous dissipation

was neglected and the fluid density varies according to the Boussinesq approximation.

1.2 2.2 Numerical method

For the two-dimensional incompressible flow, the lattice Boltzmann tool satisfies the conservative form of N-S equations [4]. According to BGK approximation, the discrete Boltzmann equations are given for the distribution functions “ f ” and “ g ” that are related respectively to the flow and thermal fields as follows:

$$f_k(r + c_k \Delta t, t + \Delta t) = f_k(r, t) - \frac{1}{\tau_v} \cdot (f_k(r, t) - f_{keq}(r, t)) + F_k \Delta t \quad (1)$$

$$g_k(r + c_k \Delta t, t + \Delta t) = g_k(r, t) - \frac{1}{\tau_T} \cdot (g_k(r, t) - g_{keq}(r, t)) \quad (2)$$

where k , Δt , τ_v , τ_T , F_k stand for the lattice discrete directions, the lattice time step, the flow relaxation time, the temperature relaxation time and the force term, respectively. The parameters c_k are called the discret lattice velocities of the D₂Q₉ model:

$$c_0 = (0, 0) \quad (4)$$

$$c_1 = (1, 0); c_2 = (0, 1); c_3 = (-1, 0); c_4 = (0, -1) \quad (5)$$

$$c_5 = (1, 1); c_6 = (-1, 1); c_7 = (-1, -1); c_8 = (1, -1) \quad (6)$$

The buoyancy force term was calculated according the Boussinesq approximation as follows:

$$F_k = 3\omega_k \rho g \beta \Delta T \times (c_{ky} \cos \gamma + c_{kx} \sin \gamma) \quad (7)$$

Moreover, for D₂Q₉ configuration, the local equilibrium distribution functions g_{keq} and f_{keq} are expressed as given bellow:

$$f_{keq}(r, t) = \omega_k \rho \left[1 + 3 \frac{\overline{c_k \cdot \bar{u}}}{c_s^2} + \frac{9}{2} \frac{(\overline{c_k \cdot \bar{u}})^2}{c_s^4} - \frac{3 \bar{u} \cdot \bar{u}}{2 c_s^2} \right] \quad (8)$$

$$g_{keq}(r, t) = \omega_k T \left[1 + 3 \frac{\overline{c_k \cdot \bar{u}}}{c_s^2} \right] \quad (9)$$

where ρ , u , T and c_s ($=\frac{1}{\sqrt{3}}$) are the density, the velocity, the temperature and the lattice speed of sound, respectively. The weighting factors ω_k are expressed for each direction as follows:

$$\omega_k = \begin{cases} \frac{4}{9} & \text{for } k=0 \\ \frac{1}{9} & \text{for } k=1,2,3,4 \\ \frac{1}{36} & \text{for } k=5,6,7,8 \end{cases} \quad (10)$$

The fluid density, velocities and temperature are deduced at the macroscopic scale by:

$$\rho = \sum f_k \quad (11)$$

$$\rho u = \sum f_k c_k \quad (12)$$

$$T = \sum g_k \quad (13)$$

To assess the reliability of our numerical code, a validation test was conducted in the case of mixed convection with credible results reported in previous works. The Nusselt number results obtained with our code were compared with those published by Aminossadati et al [5]. Table 1 shows a good agreement with the reference results.

Table 1. Comparison of the obtained average Nusselt number validation vs. from Ref. [5]

	Ri=0.1	Ri=1
Reference [5]	30.555	5.206
Our code	30.189	5.1522
Difference	1.2%	1%

3. Results and Discussion

As mentioned in the previous sections, the present study focuses on heat transfer when the PV panel changes its position. Accordingly, isotherms (Fig. 2) and average Nusselt number (Fig. 3) are illustrated vs the Reynolds number ($25 \leq Re \leq 200$) for both positions: Case A (PV cells on the top of the channel) and case B (PV cells on the bottom of the channel).

For a given value of the Richardson number ($Ri = 1$), the 2D temperature distribution exemplified in Fig. 2 shows that the exchange area between the hot PV is narrowed by increasing Re . In fact, in mixed convection regime, corresponding to $Re = 25$, the warm front due to the exchange between the hot PV and water spread out to cover a large part of the cavity for both cases; the cold zone being restricted to a small area at the entry into contact with the adiabatic wall. The cold zone expands by increasing the Reynolds number forcing a retraction of the heated fluid into a thin area along the active PV. This means that, an increase of the Reynolds number in its range leads to an increase in the rate of heat transfer for both cases, supported by the inertia effects.

Fig. 3 highlights a considerable difference between the configurations adopted in terms of the average Nusselt number and the average outlet temperature. It's obvious that higher Reynolds numbers lead to higher Nusselt numbers and lower outlet temperatures. In fact, high Reynolds numbers lead to limited heating time of the fluid, which

engenders lower temperatures at the exit. Moreover, the PV/T system in case B performs better in terms of heat transfer but leads to lower average outlet temperatures in comparison with those obtained in case A. This effect can be explained by combined effects of the channel tilt and the gravity force that facilitates heat transfer between the incoming water flow and PV modules located in the bottom (case B).

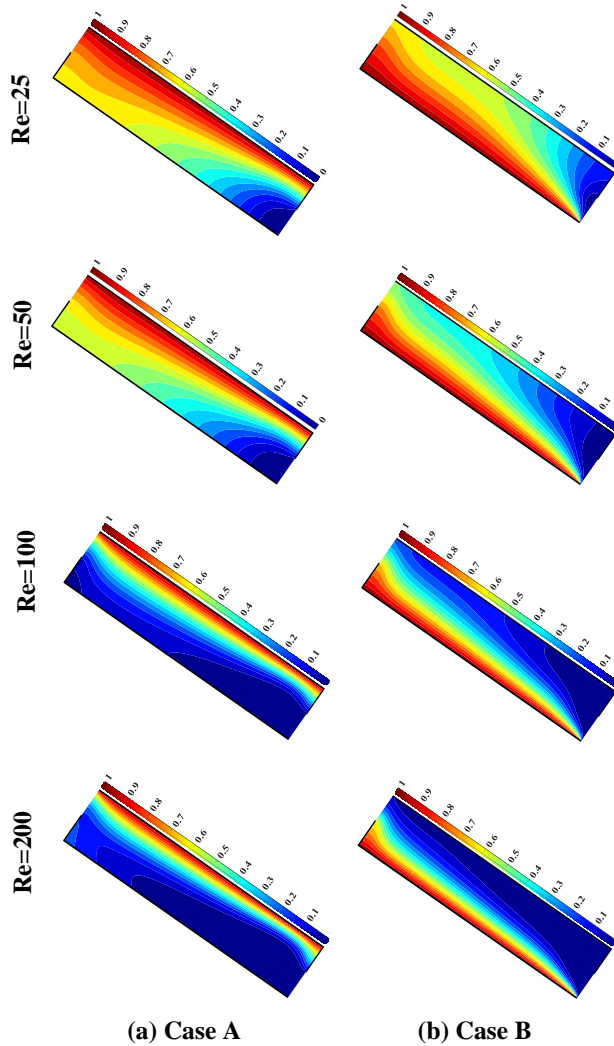


Figure 2. Isotherms obtained at various of Re for (a) case A and (b) Case B.

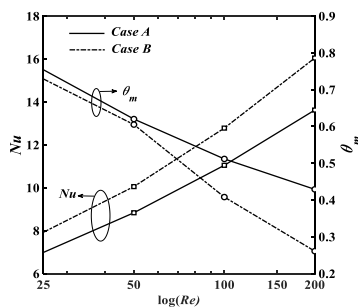


Figure 3. Variations of Nu and T_m with Re for case A and case B

4. Conclusion

Mixed convection in a water based photovoltaic thermal system has been studied numerically for two relative positions of the hot PV with respect to the working fluid and various values of the Reynolds number. The following conclusions are enumerated as follows:

- For both cases: similar temperature distributions are conducted and both affected by Reynolds number's rising.
- The increase in Reynolds number decreases the average outlet temperature, while it increases the average Nusselt number.
- Solar cells position in case B enhances the heat transfer's rate while the average outlet temperature becomes lower compared to case A.

References

[1] R. A. Agathokleous, S. A. Kalogirou, et S. Karellas, « Exergy analysis of a naturally ventilated Building Integrated Photovoltaic/Thermal (BIPV/T) system », *Renew. Energy*, vol. 128, p. 541-552, déc. 2018, doi: 10.1016/j.renene.2017.06.085.

[2] A. S. Kaiser, B. Zamora, R. Mazón, J. R. García, et F. Vera, « Experimental study of cooling BIPV modules by forced convection in the air channel », *Appl. Energy*, vol. 135, p. 88-97, déc. 2014, doi: 10.1016/j.apenergy.2014.08.079.

[3] M. Babajani, B. Ghasemi, et A. Raisi, « Numerical study on mixed convection cooling of solar cells with nanofluid », *Alex. Eng. J.*, vol. 56, n° 1, p. 93-103, mars 2017, doi: 10.1016/j.aej.2016.09.008.

[4] A. A. Mohamad, *Lattice Boltzmann Method: Fundamentals and Engineering Applications with Computer Codes*. London: Springer London, 2019. doi: 10.1007/978-1-4471-7423-3.

[5] S. M. Aminossadati, A. Kargar, et B. Ghasemi, « Adaptive network-based fuzzy inference system analysis of mixed convection in a two-sided lid-driven cavity filled with a nanofluid », *Int. J. Therm. Sci.*, vol. 52, p. 102-111, févr. 2012, doi: 10.1016/j.ijthermalsci.2011.09.004.

Modelling and simulation of a direct evaporative cooling system with purge zone

A. BOUCHAALA^{1,*}, O. MERROUN¹, A. MIKDAM², A. SAKIM¹

^{1,*} Laboratory of Artificial Intelligence & Complex Systems Engineering, ENSAM-Casablanca, Hassan II University of Casablanca, Morocco, *E-mail: ah.bouchaala14@gmail.com*

² Laboratory of Innovation in Sciences, Technologies and Modeling, Faculty of Science, Chouaïb Doukkali University

Abstract

Conventional compressor air conditioning systems consume plenty of power and handle HFCs that causes adverse effects on the environment. Thus, it is time to find eco-friendly alternatives. In this context, several cooling production processes have been developed. Direct Evaporative Cooling Systems (DECS) based on solid desiccant dehumidification is one of the most promising technologies that can provide comfortable indoor air conditions. The present paper presents a comprehensive study on predicting the thermal performance parameters of DECS such as the COP, cooling capacity and cooling energy. A straightforward optimization study was carried out by investigating the effect of using a desiccant wheel purge zone and its effect on the thermodynamic efficiency of the system.

Key words: *Desiccant wheel, Purge zone, Thermal COP, Cooling capacity, Cooling energy.*

1. Introduction

Currently, conventional compressor air conditioning systems are widely used for cooling and heating. However, they consume plenty of power and handle HFCs that causes adverse effects on the environment. Solar air conditioning systems constitute an interesting alternative to these traditional systems. They could be useful in mitigating CO₂ emissions and saving energy. This technology could be attractive mostly for counties enjoying a huge solar potential. Direct Evaporative Cooling Systems (DECS) based on solid desiccant dehumidification is one of the most promising technologies that can provide comfortable indoor air conditions [1]. The principal components of such cooling system consist of a Desiccant Wheel (DW), a rotative Sensible Heat Exchanger (SHE) and two water Flash Evaporative Coolers (FEC). The DW is the main component of the system. It is a rotative wheel filled with a desiccant material that adsorb the water vapor of air. The wheel rotates continuously between two separated air flow streams. The first one is called the process air that is humidified when flowing through the DW. The second one is a regeneration air maintained at low or medium temperature (60 °C - 120 °C) to catch the moister already adsorbed by the wheel. Hence, the system needs a thermal heat driven force to produce a cooling effect. However, the COP of these systems is small compared to conventional compressor air

conditioning systems and can only be useful and valued if a moderate temperature heat source (solar energy, biomass, industrial waste) is used as a driving force. In what follows, we present a short description of the system modelling and optimization by using a purge zone that will minimize the thermal heat consumption and hence the thermal COP of the system.

2. Thermodynamic modelling

Generally, the system can be considered as a three-temperature cooling closed cycle exchanging tow useful thermal energies with the surrounding environments; indoor building (Q_{in}) at temperature T_5 and outdoor building (Q_{out}) at temperature T_1 . The thermodynamic analysis of open adsorption cycles is developed by Pons and Kodama [2]. They introduced two virtual systems C and E in order to locate these entropy productions in the cycle. Therefore, the COP of the system can be compared to that of a Carnot cycle operating with a condenser at T_1 and an evaporator at T_5 . On the other hand, an energy and entropy balance are established. Although the heat losses are taken into account in this analysis, consequently, after recombination, the COP, the cooling energy per unit of mass (CE) and the cooling capacity of the system (CC) are given by the following relations: $COP = (h_5 - h_4)/(h_8 - h_7)$; $CE = h_5 - h_4$
 $CC = \dot{m} (h_5 - h_4)$.

Where h and \dot{m} are the enthalpy and air flow rate, respectively. However, some modifications are necessary when introducing the DW purge zone (Fig. 1). The air stream leaving the purge zone is mixed, in a mixer, with the regeneration air coming from the heat exchanger, as shown in Fig. 2.

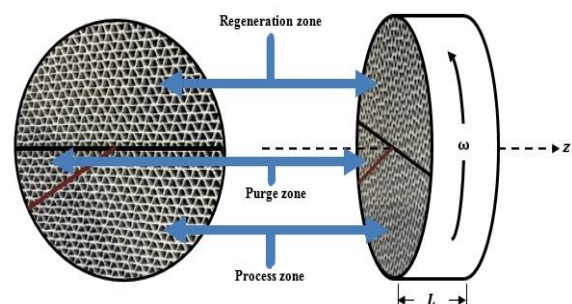


Fig. 1. A desiccant wheel with a purge zone

2.1 Mathematical modeling of the desiccant wheel

The desiccant wheel (DW) that we have chosen for this study consists of a matrix made of several sinusoidal and parallel channels through which air flows. This matrix is made of a composite material with a silica gel fraction between 0.7 and 0.8. This desiccant has a high affinity with water vapor. The DW length is 0.2 m.

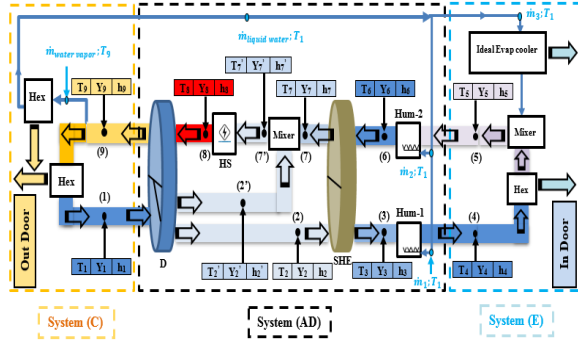


Fig. 2. A schematic view of a desiccant evaporative cooling system with purge zone

Before detailing the mathematical model, the following assumptions are made [3]:

- All the channels of the wheel are identical and adiabatic;
- One-dimensional airflow channels are considered;
- Radial heat and mass transfer inside the desiccant are negligible;
- Axial thermal conduction and mass diffusion of air along channels are neglected;
- Thermodynamic properties of air at the entries of wheel are uniform;

In order to determine the unknowns of the problem which are the water content of the desiccant, W , the temperature and the absolute humidity of air (T_a, Y_a) and desiccant material (T_d, Y_d), heat and mass balances are applied to a control volume of air and desiccant material and yield to the following governing equations:

- Moisture conservation and energy conservation of air are expressed as follows:

$$\frac{\partial Y_a}{\partial t} + u_a \frac{\partial Y_a}{\partial z} = \frac{4h_m}{\rho_a D_h} (Y_d - Y_a) \quad (1)$$

$$\frac{\partial T_a}{\partial t} + u_a \frac{\partial T_a}{\partial z} = \frac{4h}{\rho_a D_h (C_{pa} + Y_a C_{pv})} (T_d - T_a) \quad (2)$$

- Moisture conservation and energy conservation in the desiccant material are given by:

$$\varepsilon \rho_a \frac{\partial Y_d}{\partial t} + \rho_d \frac{\partial W}{\partial t} = \frac{2h_m}{\delta f} (Y_a - Y_d) \quad (3)$$

$$\rho_d f (C_{pd} + WC_{pl}) \frac{\partial T_d}{\partial t} = \frac{h}{\delta} (T_a - T_d) + \frac{H_{sor} h_m}{\delta} (Y_a - Y_d) \quad (4)$$

u_a is the air velocity; h and h_m are the convective heat transfer coefficient and the convective mass transfer coefficient, respectively. H_{sor} is the heat of adsorption; C_{pa} , C_{pv} , C_{pl} and C_{pd} are the specific heat of dry air, water vapor, water liquid and silica gel, respectively. δ , f , ε are the half wall thickness, the fraction of desiccant in the wheel and the porosity respectively; ρ_a and ρ_d are the density of air and desiccant respectively; D_h is the hydraulic diameter.

These equations are subject to boundary and initial conditions, which are obtained considering the periodic nature of the desiccant wheel. Equations 1-4 involve 5 unknowns. Therefore, we require an auxiliary equation to close the system. Here, the empirical correlation between the water content adsorbed by silica gel, W , and the corresponding equilibrium relative humidity ϕ over the desiccant with water content W , is used [4]:

$$\phi_d = 0,0078 - 0,05759W + 24,16554W^2 - 124,78W^3 + 204,226W^4 \quad (5)$$

The relationship between the absolute humidity, Y , and the equilibrium relative humidity, ϕ , is given by the following equation:

$$Y_d = \frac{0,622\phi_d P_{vs}}{P_{am} - \phi_d P_{vs}} \quad (6)$$

where P_{am} and P_{vs} are the atmospheric pressure and the saturating vapor pressure, respectively. The thermodynamic properties used in this simulation are cited in [5].

3. Validation of the model

Equations 1-4 are discretized by implicit forward finite difference method with a space grid size of $dz=0.004m$ and a time step of 0.1 s. Gauss Seidel iterative method is used to solve the obtained linear system equations. The numerical model was implemented in MATLAB. The convergence test is verified when the mass and energy balance reach a value lower than 0.5%.

The working conditions used in this cycle are the air velocity, the regeneration temperature T_{reg} and the rotation speed of the DW. Kodama et al. [6] have carried out an experimental optimization study to find the optimal working conditions. The results obtained are $u_a = 2$ m/s, $T_{reg} = 80.5$ °C and $\omega = 19.5$ rpm.

To validate our mathematical model, numerical simulations were performed firstly to predict the temperature, humidity, enthalpy and entropy of the different state of air (see points of the cycle). Then, the heat exchanged between the system and the external environment as well as the different sources of irreversibility were calculated. Finally, the thermal performance parameters of the system are calculated. The results obtained (Table1) are compared with experimental measurements [6]. As can be seen, the results are in good agreement with the experimental measurements.

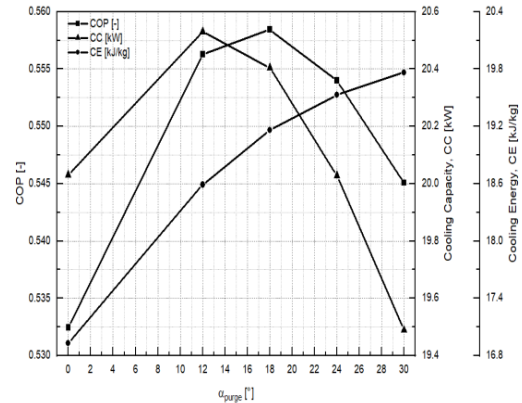
Table 1. Comparison between simulation results and experimental ones of Kodama et al. [6]

Cycle point	$T_{a, Exp}$ [°C]	$T_{a, Num}$ [°C]	$Y_{a, Exp}$ [g/kg _d air]	$Y_{a, Num}$ [g/kg _d air]
1	31,00	31,00	10,3	10,3
2	55,50	55,16	4,7	4,3
3	24,40	25,35	5	4,3
4	14,00	14,08	9,2	8,9
5	27,80	27,80	10	10
6	20,00	20,08	13	13,1
7	51,50	49,90	12,7	13,1
8	80,50	80,50	12,5	13,1
9	56	55,46	18,3	19,2

	Exp	Num	Er [%]
COP [-]	0,54	0,5324	1,4
CE [kJ/kg]	15,7	16,9273	7,82
CC [kW]	19,5	20,0306	2,72

4. System optimization by using purge zone

When the process air flows through the DW, its moisture decreases and then leaves the DW at a relatively high temperature. This is due, on the one hand, to the adsorption of water vapor by the desiccant material and, on the other hand, to the residual heat transferred from the regeneration zone to the process zone. This directly affects the thermal performance of the system. To overcome this problem, an air purge zone is introduced. It consists of removing a part of the outlet hot process air during the first degrees of rotation of the DW. Then, it is mixed with the regeneration air coming from the rotative heat exchanger towards the hot battery. To find the optimal purge zone, we performed simulations keeping the same operating conditions used previously. Then, we have changed the angle of the purge zone. The results are shown in Fig. 3. As can be easily seen, the introduction of a purge zone improves the COP. Furthermore, the optimal purge angle obtained is about 18° which corresponds to a maximum of COP. In fact, the increase of the purge area leads to withdraw a large amount of hot and humid air. This contributes to the cooling of the process air and reducing its moisture which improves the thermal performance parameters of the system. Recycling the withdrawn air from the purge zone contributes efficiently to reduce the thermal consumption of the hot battery. Consequently, the COP is further improved. We also notice that after reaching an optimum, the COP and CC decrease while the CE increases successively up to a limit value because it only depends on the dehumidification performance of the DW and the temperature difference between the air supplied to the building and the indoor air. Indeed, widening the purge area contributes to increase the purge air flow rate but at the same time the process air flow rate decreases. Hence, taking into account that the CC depends on the flow rate, this explains the decrease of cooling power produced and consequently the decrease in the COP. Therefore, the use of a purge zone becomes insignificant.

**Fig. 3. Variation of COP, CC and CE as a function of DW purge angle**

5. Conclusion

In this work, we have presented a mathematical model and a numerical simulation of a direct evaporative desiccant cooling system. The DW model is based on the governing equations obtained from the conservation of mass and energy over a control volume of DW channel. The established model was validated by comparing our simulation results with experimental results reported in the literature. In order to optimize the system, a desiccant wheel purge zone is introduced and its effects on thermal performances of the system are investigated. A purge zone of 18° contributes to enhance the COP and the thermodynamic efficiency of the system.

References

- [1] N. Enteria, H. Awbi, H. Yoshino, Desiccant Heating, Ventilating, and Air-Conditioning Systems, Springer, (2017).
- [2] Pons M, Kodama A. Entropic analysis of adsorption open cycles for air conditioning. Part 1: first and second law analyses. International Journal of Energy Research 24 (2000) 251-262.
- [3] W. Zheng, W.M. Worek, Numerical simulation of combined heat and mass transfer processes in a rotary dehumidifier, Numerical Heat Transfer Part A: Applications 23 (2) (1993) 211-232.
- [4] A.A. Pesaran, A.F. Mills, Moisture transport in silica gel packed beds: I –Theoretical study, International Journal of Heat and Mass Transfer 30 (6) (1987) 1037-1049.
- [5] L.A. Sphaier, W.M. Worek, Analysis of heat and mass transfer in porous sorbents used in rotary regenerators, International Journal of Heat and Mass Transfer 47 (14-16) (2004) 3415-3430.
- [6] Kodama A, Jin W, Goto M, Hirose T, Pons M. Entropic analysis of adsorption open cycles for air conditioning. Part 2: interpretation of experimental data. International Journal of Energy Research 24 (2000) 263-278.

ASSESSMENT OF HYDROKINETIC ENERGY POTENTIAL IN THE SEBOU RIVER ESTUARY

Z. AITCHFI¹, A. TOUAZIT¹, M. IGOUZAL^{1*}

1. Laboratory of Electronic Systems, Information Processing, Mechanics and Energy, Faculty of Sciences, Department of Physics, Ibn Tofail University, B.P 242, 14000 Kenitra, Morocco, emails: zineb.aitchfi@uit.ac.ma (Z. AITCHFI), azzeddin.touazit@uit.ac.ma (A. TOUAZIT).

*Corresponding author: mohammed.igouzal@uit.ac.ma (M. IGOUZAL).

Abstract

Evaluating hydropower in estuaries has been the subject of various studies around the world. These hydropower related studies, however, have not reached the Moroccan rivers estuaries yet. The Sebou river estuary has only been the subject of morphological and quality related studies, as such, the aim of the present work is to assess the hydrokinetic energy potential of this estuary. The hydraulic regime was simulated with MASCARET 1D model using a database from 2019. Simulating the estuary's regime through 2019 has allowed us to determine the areas with high hydropower potential (flow velocity exceeding 1 m/s).

Keywords: *Sebou river estuary, Hydraulic energy potential, Mascaret 1D, ArcGIS.*

1. Introduction

Evaluating hydropower has been the subject of various recent studies particularly the offshore energy which has been estimated along the Moroccan Atlantic Coast (Bouhrim and El Marjani, 2018). Presumably, studies concerning hydrokinetic energy have not quite reached Moroccan rivers estuaries yet. The estuary of the Sebou river, our study area, has been the subject of several hydraulic studies mainly morphological and quality related ones, therefore, the aim of the present work is to assess the hydrokinetic energy potential of the Sebou river estuary.

2. Study area and Hydrodynamic model

2.1 Overview of the Sebou river estuary

The Sebou river is about 600 km long. Its estuary is located between the Lalla Aicha guard dam (situated 62 km from the mouth) and the mouth of the river (Fig. 1), with waters used by many industries and the agricultural pumping stations situated along its way,

in addition to it providing an important seaway through the existence of the river port of Kenitra.

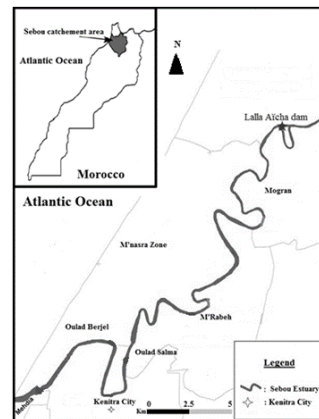


Fig. 1. Study area.

The hydraulic regime of the estuary is under the preponderance of the tidal cycle (Fig. 2) as well as the discharges from the Lalla Aicha dam upstream. Consequently, the flow regime knows major seasonal and inter-annual variations.

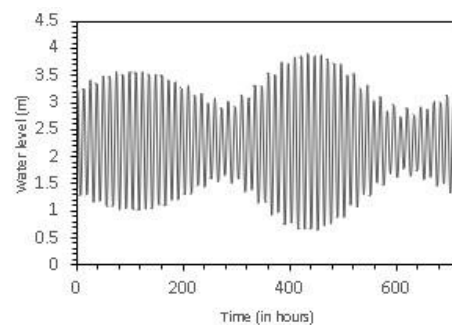


Fig. 2. Tidal oscillations at the mouth of Sebou estuary (April 2019)

2.2 MASCARET 1D

MASCARET is a one-dimensional hydraulic modelling software. The calculations carried out during the simulations are based on the one-dimensional Saint Venant equations of continuity (1) and momentum (2): (Goutal et al., 2012)

$$L_t \frac{\partial Z}{\partial t} + \frac{\partial Q}{\partial x} = q_1 \quad (1)$$

$$\frac{\partial Q}{\partial t} + \frac{\partial(\beta QV)}{\partial x} + gS \left(\frac{\partial Z}{\partial x} + J + J_s \right) = 0 \quad (2)$$

Q: flow (m^3/s), x: the distance along the channel (m), t: time (s), Z: water level (m), q_1 : water inflow or outflow (m^2/s), L_t : top width (m), S: cross-sectional area (m^2), g: gravitational acceleration (m/s^2), V: velocity (m/s), J is the average rate of energy dissipation and J_s represents losses that can occur in the presence of obstacles or abrupt changes in cross-sections. The coefficient β (Dimensionless) results from the velocity variations.

3. Results

3.1 Modelling the hydraulic regime of the estuary

The hydraulic regime of the Sebou estuary was modelled using MASCARET 1D. The condition at the upstream boundary (at the Lalla Aicha dam) is the discharge as a function of time and the condition at the downstream boundary (at the mouth) is the water level as a function of time. The modelling occurred during the months of June and July of the year 2019.

3.2 Calibration and validation of the model

The model was calibrated using the data of January 2019. During the calibration, the Manning's coefficient was adjusted through multiple simulations. The results obtained are then compared with the water level measurements carried out at the Kenitra station located 17 km from the mouth. The calibration resulted in a very good correlation between the water levels simulated by the model and those measured (Fig. 3).

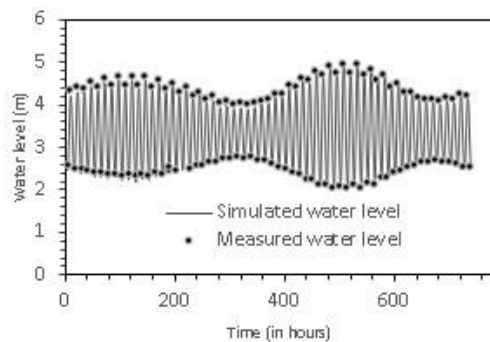


Fig. 3. Calibration of MASCARET model

To confirm the calibration test results, we performed a validation test of the model using data from February 2019 and maintaining the same value for the Manning's coefficient. The validation test also showed that the simulated water levels and observed ones were close (Fig. 4).

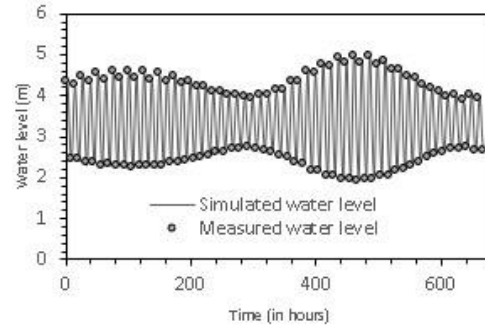


Fig. 4. Validation of MASCARET model

3.3 Model performance testing

To evaluate the performance of the hydraulic model, three statistical errors are used:

$$RMSE = \left[\frac{1}{n} \sum_{i=1}^n (h_{obs,i} - h_{model,i})^2 \right]^{\frac{1}{2}}$$

$$NOF = \left[\frac{RMSE}{h_{obs}} \right]; NSC = 1 - \left[\frac{\sum_{i=1}^n (h_{obs,i} - h_{model,i})^2}{\sum_{i=1}^n (h_{obs,i} - \bar{h}_{obs})^2} \right]$$

Where h_{obs} : observed water level, h_{model} : simulated water level, n : number of measurements.

Table 1 statistical analysis of model performance

Statistical Errors	RMSE	NOF	NSC
Calibration	0.06	0.017	0.99
Validation	0.062	0.018	0.99

The low RMSE and NOF values and the high NSC values confirm the performance of the model and the results of the calibration and validation tests.

4. Assessment of the hydraulic energy potential

To evaluate the hydraulic energy potential in the Sebou estuary, a simulation of the hydraulic regime of the site was carried out for the whole year 2019. The created model allowed us to determine the temporal evolution of hydraulic variables at each site of the study reach, in particular: the flow velocity since it is the main parameter on which the hydraulic energy directly depends. The minimum value of the flow velocity required to operate a hydrokinetic

turbine is 1 m/s, hence the need to determine the number of hours at each location with flow velocities greater than or equal to 1 m/s.

The assessment of the suitability of a site for hydrokinetic turbines installation is based not only on the spatio-temporal distribution of flow velocity but also on the length of the site. This is mainly useful for the installation of hydrokinetic parks, taking advantage of the amplitude of the flow velocity while respecting the distances required between these turbines. Outputs of the MASCARET model were integrated into the ArcGIS software. Fig. 5 shows a map of the annual average velocity calculated, at each site, where the velocity exceeds the exploitable value of 1 m/s.

The results of our study led to ten potentially exploitable areas. The shortest sites being located at 32 and 53 km (from the mouth). The sites which are located at 3, 8 and 12 km are ideal for individual tidal turbines. In contrast, the 15 to 19 km range (4km of length) is the most suitable for the installation of hydrokinetic farms.

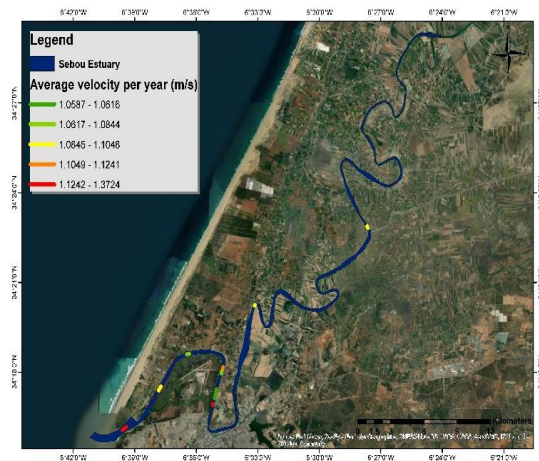


Fig. 5. Sites with high hydropower potential.

5. Conclusion

In this study, the hydraulic regime of the Sebou River estuary was modelled using a one-dimensional model (MASCARET) and a database of the year 2019. The calibration and validation of this model on real water level measurements enabled the simulation of the regime over the whole year 2019 which in turn allowed us to determine the zones with a high potential in hydraulic energy.

References

- [1] H. Bouhrim and A. El Marjani, Wave Energy Assessment Along the Moroccan Atlantic Coast, *Journal of Marine Science and Application* (2018).
- [2] S. Haddout, A. Maslouhi, B. Magrane, M. Igouzal, Study of salinity variation in the Sebou River Estuary (Morocco), *Desalination and Water Treatment* (2015).
- [3] S. Haddout, M. Igouzal, A. Maslouhi, B. Magrane, Modélisation du régime hydraulique de l'estuaire de la rivière Sebou avec étude de sensibilité, 12th Congress of Mechanics, Casablanca, Morocco, 2015.
- [4] N. Goutal, J.-M. Lacombe, F. Zaoui, K. El-Kadi-Abderrezzak, MASCARET: a 1-D Open-Source Software for Flow Hydrodynamic and Water Quality in Open Channel Networks; *River Flow 2012 – Murillo (Ed.)*, pp 1169-1174, 2012.

Amélioration du transfert thermique par utilisation d'une conduite à parois ondulées

S. ELALAOUI¹, A. OUBARRA², J. LAHJOMRI³, K. ZNIBER⁴

Université Hassan II de Casablanca, Faculté des Sciences Ain-Chock, Laboratoire de Mécanique,
Km 8 route d'El Jadida BP 5366 Maarif, Casablanca, Maroc.

elalaoui.soaad@gmail.com

Abstract

The work consists of a numerical study of the forced convection of an incompressible, laminar and transient flow of a Newtonian fluid within a channel with wavy walls of infinite length.

The temperature profile and the distribution of the local Nusselt number are determined numerically using the Alternating Direction Implicit method.

We present the effect of certain parameters on the radial temperature profile and on the Nusselt number.

Keywords: ADI, flow, laminar, transient, forced convection, heat transfer, corrugated channel

Résumé :

Dans ce travail, nous avons mené une étude numérique de la convection forcée d'un écoulement incompressible laminaire d'un fluide Newtonien à l'intérieur d'un canal à parois ondulées, de longueur infinie.

Le profil de température est déterminé numériquement grâce à la méthode ADI (Alternating Direction Implicit).

Nous présentons l'effet de certains paramètres sur le profil radial de la température.

Mots-clés : ADI, écoulement, laminaire, convection forcée, transfert de chaleur, canal à paroi ondulée.

1. Introduction

Il existe plusieurs travaux sur les écoulements dans des conduites à section variables dans la littérature ; notamment les conduites ondulées vu leur importance dans le domaine industriel. Parmi ces travaux on trouve :

Une étude numérique sur l'effet des paramètres géométriques été menée par H.M et al [1]. Les parois, supérieure et inférieure, du canal ondulé sont soumises à un flux de chaleur constant. Les équations gouvernantes sont résolues par la méthode des volumes finis. Ils ont conclu que l'utilisation d'un canal ondulé est une méthode appropriée pour augmenter le rendement thermique et obtenir une compacité plus élevée de l'échangeur de chaleur.

Une étude numérique détaillée de transfert de chaleur d'un fluide s'écoulant dans un canal ondulé avec un nombre de

Reynolds ($Re = 2000-5500$), et les angles de plaques ondulées V ($\theta = 20^\circ, 40^\circ, 60^\circ$), avec un flux de chaleur constant ($q'' = 580, 830, 1090 \text{ w/m}^2$), été

présentée par Yang et al [2], Les résultats numériques ont été validés avec les données de Naphon [3]. Ils ont conclu que l'augmentation des angles de plaques ondulées peut améliorer le transfert de chaleur.

Une étude numérique d'écoulement en régime transitoire avec un nombre de Reynolds variant de 200 à 3000 en trois dimensions, pour examiner l'effet géométrique sur les caractéristiques thermo hydrauliques des canaux ondulés-croisés périodiques, été présentée par J.L et al [4]. Ils ont eu comme conclusion que l'angle au sommet impacte fortement sur le transfert de chaleur et la perte de pression dans le canal ondulé.

Dans ce présent travail, nous étudions numériquement le comportement thermique pour un fluide newtonien, dans un canal à parois ondulées.

2. Configuration géométrique et formulation du problème :

Le système étudié est schématisé par un canal à parois sinusoïdales de longueur infinie, d'axe de symétrie Δ , de rayon $R(x)$, et désignant la distance radiale d'un point de la surface à l'axe de symétrie, (figure 1), dans lequel circule un fluide newtonien dont le profil de vitesse est celui de Poiseuille pour $x < 0$.

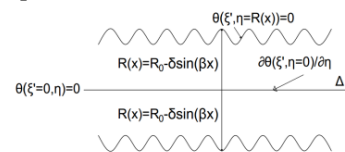


Figure 1 : Configuration étudiée et conditions aux limites.

Pour simplifier le problème étudié, on suppose que la conduction axiale et la dissipation visqueuse sont négligeables au sein du fluide.

Les différentes équations nécessaires à la résolution du problème considéré dans notre étude, se formulent comme suit :

$$\left[\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} \right] = \alpha \left(\frac{\partial^2 T}{\partial y^2} \right) \quad (1)$$

Où:
$$\alpha = \frac{k}{\rho c_p}$$

D'après l'article de Castellões et al. [5], pour un écoulement bidimensionnel dans une conduite ondulée, incompressible, laminaire d'un fluide Newtonien et pour une amplitude inférieure ou égale à 0.1 des parois ondulées, on a pour $x > 0$:

$$u(x, y) = \frac{3R_0 u_m}{2[\delta \sin(\beta x) - R_0]^3} [y^2 - (R_0 - \delta \sin(\beta x))^2] \quad (2)$$

$$v(x, y) = \frac{3}{2} \left[\frac{R_0 u_m \beta \delta \cos(\beta x) y [y^2 - (R_0 - \delta \sin(\beta x))^2]}{[-R_0 + \delta \sin(\beta x)]^4} \right] \quad (3)$$

Il est à signaler que l'on vérifie bien :

$$\begin{aligned} \text{div } \vec{u} &= \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0, \quad \left. \frac{\partial u}{\partial y} \right|_{y=0} = 0, \quad v(x, 0) = 0, \\ u(x, R(x)) &= 0, \quad v(x, R(x)) = 0 \end{aligned} \quad (4)$$

Et que pour $\delta = 0$ on a :

$$u(x, y) = \frac{3}{2} u_m \left(1 - \frac{y^2}{R_0^2} \right), \quad v(x, y) = 0 \quad (5)$$

Le nombre de Nusselt local est défini par :

$$Nu(x, t) = \frac{-D_n k \left. \frac{\partial T}{\partial n} \right|_{x, R(x), t}}{k(T_m(x, t) - T_p(x, t))} \quad (6)$$

Nous définissons de nouvelles variables pour transférer l'effet de la paroi ondulée des conditions aux limites vers les équations. Les coordonnées transformées sont :

$$\begin{cases} \eta = \frac{y}{R(x)} \\ \xi = \frac{x}{PeR_0} \end{cases} \quad (7)$$

A l'aide de ces nouvelles variables, on remarque que η varie entre 0 et 1 et le domaine devient rectangulaire. Cette transformation est introduite en vue d'une résolution numérique de l'équation de l'énergie par la méthode ADI sur un domaine rectangulaire.

Après avoir effectué les transformations de domaine, on fait une analyse adimensionnelle.

Nous définissons les quantités adimensionnelles suivantes :

$$\theta = \frac{T - T_e}{T_p - T_e}, \quad U(\xi, \eta) = \frac{u(x, y)}{u_m}, \quad V(\xi, \eta) = \frac{v(x, y)}{u_m}, \quad \tau = \frac{\alpha t}{R_0^2} \quad (8)$$

Avec u_m et R_0 représentent respectivement la vitesse maximale et la demi-distance moyenne.

En tenant compte les quantités adimensionnelles définis ci-dessus, et que $\omega = R_0 Pe \beta$ et $\varepsilon = \frac{\delta}{R_0}$ La pulsation adimensionnelle et le paramètre d'occlusion respectivement, les équations précédentes (1) et (6) deviennent :

$$\frac{\alpha}{R_0^2} \frac{\partial \theta}{\partial \tau} + \frac{3}{2} \frac{R_0 u_m}{(R_0 - \delta \sin(\omega \xi))^3} (1 - \eta^2) \left[\frac{\eta \delta \beta \cos(\omega \xi)}{R_0 - \delta \sin(\omega \xi)} \frac{\partial \theta}{\partial \eta} + \frac{1}{R_0 Pe} \frac{\partial \theta}{\partial \xi} \right] - \frac{3}{2} \left[\frac{R_0 u_m \beta \delta \cos(\omega \xi) \eta (1 - \eta^2)}{(R_0 - \delta \sin(\omega \xi))^4} \right] \frac{\partial \theta}{\partial \eta} = \alpha \frac{1}{(R_0 - \delta \sin(\omega \xi))^2} \frac{\partial^2 \theta}{\partial \eta^2} \quad (9)$$

$$Nu(\xi, \tau) = \frac{4 \sqrt{1 + \frac{\varepsilon^2 \omega^2}{Pe^2} \cos^2(\omega \xi)} \left. \frac{\partial \theta}{\partial \eta} \right|_{\xi, 1, \tau}}{[1 - \theta_m(\xi, \tau)]} \quad (10)$$

Et afin de confiner le domaine pour la résolution numérique, nous utilisons la transformation suivante pour transformer les valeurs infinies à des valeurs finies.

$$\xi = \frac{\xi'}{1 - \xi'} \Rightarrow \xi' = \frac{\xi}{\xi + 1} \quad (11)$$

En admettant la transformation traduite par l'Équation (11), les équations précédentes (9) et (10) deviennent :

$$\begin{aligned} \frac{\partial \theta}{\partial \tau} + \frac{3}{2} \left[\frac{(1 - \eta^2)(1 - \xi')}{1 - \varepsilon \sin\left(\omega \frac{\xi'}{1 - \xi'}\right)} \right] \frac{\partial \theta}{\partial \xi'} &= \frac{1}{\left(1 - \varepsilon \sin\left(\omega \frac{\xi'}{1 - \xi'}\right)\right)^2} \frac{\partial^2 \theta}{\partial \eta^2} \\ Nu(\xi', \tau) &= \frac{4 \sqrt{1 + \frac{\varepsilon^2 \omega^2}{Pe^2} \cos^2\left(\omega \frac{\xi'}{1 - \xi'}\right)} \left. \frac{\partial \theta}{\partial \eta} \right|_{\xi', 1, \tau}}{[1 - \theta_m(\xi', \tau)]} \end{aligned} \quad (13)$$

3. Résultats et discussion

La figure 2 présente la courbe de validation du code numérique en régime permanent utilisé en prenant les valeurs des paramètres indiqués, et qui permettent de faire la comparaison nécessaire. Ainsi, on remarque une concordance avec le code Maple et que les conditions aux limites sont bien respectées. De plus, on remarque plus clairement que si on augmente ξ' les niveaux de température augmentent également, et le domaine pour lequel la température demeure constante se réduit, ce qui montre que le transfert est de plus en plus efficace sur une section donnée au fur et à mesure que l'on progresse le long du canal, et loin de l'entrée de ce dernier, la température varie de moins en moins sur toute une section, ce qui montre que le régime développé est proche celui dont la température adimensionnelle vaut 1.

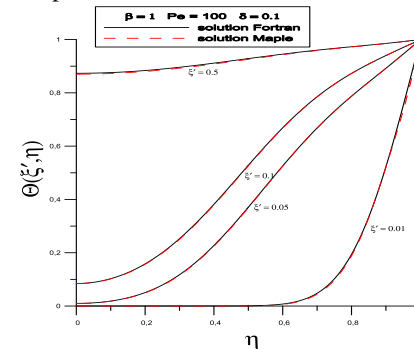


Figure 2 : Comparaison en régime permanent entre la solution numérique (Fortran) et la solution numérique (Maple) pour différents valeurs de la position axiale adimensionnelle

Dans la figure 3, on présente l'effet de l'amplitude δ de la paroi ondulée sur le transfert de chaleur. En effet, on voit qu'en augmentant l'amplitude la température radiale continue à augmenter globalement et fortement avec η . Ainsi, ceci montre que le transfert de chaleur progresse avec ce paramètre, ce qui explique entre autres l'importance du choix d'une paroi ondulée et son amplitude.

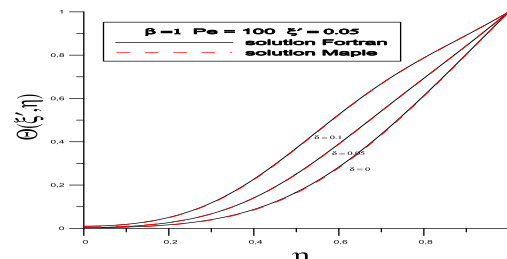


Figure 3 : Evolution de la température radiale: Influence de l'amplitude de la paroi du canal ondulé.

$\xi' = 0.05$, pour $\omega = 100$

Comme nous travaillons avec une conduite ondulée avec une pulsation β (proportionnelle au nombre de périodes spatiales par mètre de conduite), il est légitime d'étudier son influence sur le comportement de la température adimensionnelle radiale. Pour le faire, on présente la courbe dans la figure 4 qui montre clairement une augmentation substantielle de la température pour les petites valeurs de ξ' et beaucoup plus faible au delà de certaines valeurs de la distance axiale.

De plus on peut dire que cette influence commence "tardivement" pour les petites valeurs de ξ' .

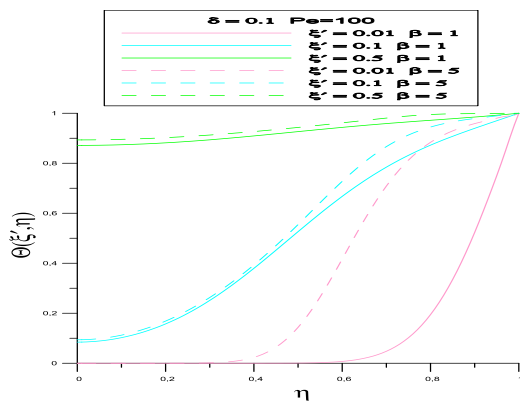


Figure 4 : Evolution de la température radiale: Influence de Pulsation ($\omega = 100, \omega = 500$)

La figure 5 représente les évolutions du nombre de Nusselt local pour $\beta = 1$ et $\delta = 0; \delta = 0,05; \delta = 0,1$. On remarque que l'amplitude d'oscillation de Nu augmente avec δ et que la valeur moyenne semble être proche de celle asymptotique pour $\delta = 0$ et de valeur proche de 7,5, ce qui est en accord avec l'article de référence [5].

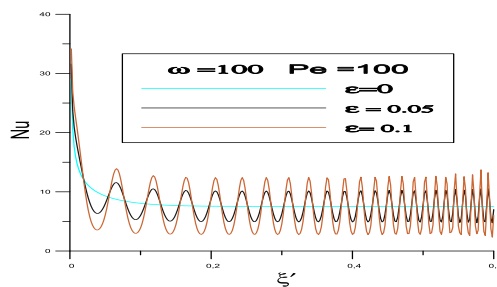


Figure 5: Evolution du nombre de Nusselt local pour $\omega = 100$: Influence de l'amplitude

A partir de la figure 6, on peut observer que la valeur moyenne de Nu_{as} semble être indépendante de β (ou ω), et que la longueur d'entrée thermique augmente (de $\approx 0,1$ à $\approx 0,4$) lorsque β (ou ω) diminue ce qui montre que même β (ou ω) permet d'améliorer

l'échange thermique, et ceci est en accord avec les observations précédentes faites pour $\theta(\xi', \eta)$.

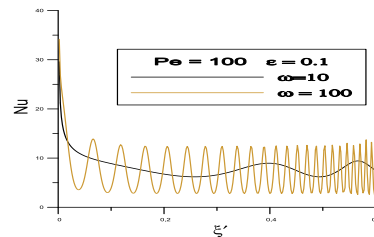


Figure 6: Evolution du nombre de Nusselt local pour $\omega = 100$ et pour $\omega = 10$

4. Conclusion :

Le travail présenté a pour objectif principal de vérifier si la géométrie de canal peut avoir un effet amplificateur du transfert thermique.

Les résultats obtenus montrent que le canal ondulé favorise le transfert de chaleur par convection, mais d'après la littérature scientifique dans ce domaine, il cause inévitablement de pertes de charge. Malgré les minimes pertes de charges dans le cas du tube lisse, ce dernier n'est pas forcément appréciable en ce qui concerne les applications pour transfert de chaleur.

La présence des ondulations donnera sous certaines conditions d'une meilleure performance du point de vue du transfert de chaleur et le système peut être optimisé.

Les performances du transfert de chaleur augmentent avec l'augmentation de l'amplitude de la paroi et la pulsation en comparant avec une paroi droite.

Références :

- [1] H.A. Mohammed, A.K. Abbas, J.M. Sheriff, « Influence of geometrical parameters and forced convective heat transfer in transversely corrugated circular tubes, Heat and Mass Transfer », Vol 44, p.p 116–126, 2013.
- [2] Yue-Tzu Yang, Peng-Jen Chen, « Numerical simulation of fluid flow and heat transfer characteristics in channel with V corrugated plates, Heat and Mass Transfer », Vol 46, p.p 437-445, 2010.
- [3] Naphon P, « Heat transfer characteristics and pressure drop in channel with V corrugated upper and lower plates, Energy Convers Manag », Vol 48, p.p 1516–1524, 2007.
- [4] Liu XP, J.L.Niu, « effects of geometrical parameters on the thermo hydraulic characteristics of periodic cross-corrugated channels, Heat and Mass Transfer », Vol 84, p.p 542–549, 2015.
- [5] Castellões et al., João N.N. Quaresma, Renato M. Cotta, « Convective heat transfer enhancement in low Reynolds number flows with wavy walls, Heat and Mass Transfer », Vol 53, p.p 2022–2034, 2010

Analysis of the thermodiffusion phenomenon by Optical Digital Interferometry

A. ERRARTE¹, A. MIALDUN², V. SHEVTSOVA^{1,3} and M. M. BOU-ALI¹

1. Mechanical and Manufacturing Department, Faculty of Engineering, Mondragon University, Loramendi Kalea, 4, 20500 Arrasate-Mondragon, Spain. aerrarte@mondragon.edu
2. Microgravity Research Centre, CP-165/62, Université libre de Bruxelles (ULB), 50, av. F.D. Roosevelt, B-1050 Brussels, Belgium.
3. Ikerbasque, Basque Foundation for Science, Bilbao, Spain.

Abstract

Optical Digital Interferometry is attempted in different systems to determine the transport properties of fluid mixtures. During thermodiffusion experiments, the concentration of a mixture species is tracked both in a thermogravitational micro-column (installation of Mondragon University) and in the Selectable Optical Diagnostic Instrument on-board the International Space Station. This work is to show the abilities and limitations of the systems, both in the analysis of binary and ternary mixtures from an optical point of view.

Keywords: *Soret, Optical Digital Interferometry, Thermodiffusion, Molecular diffusion, Mach-Zehnder.*

1. Introduction

The exposure of a ternary mixture to a thermal gradient results in a gradient of concentrations due to the phenomenon of thermodiffusion. Such phenomenon is characterised by the Soret coefficient, described by Eq. 1 for a component of a ternary mixture [1].

$$S'_{T,i} = \frac{(D'_{T,i}D_{jj} - D'_{T,j}D_{ij})}{(D_{ii}D_{jj} - D_{ij}D_{ji})} \quad (1)$$

As can be seen, the Soret coefficient is the result of the relationship between the thermodiffusion coefficients ($D'_{T,i}$ and $D'_{T,j}$) and the molecular diffusion coefficients (D_{ii} , D_{jj} , D_{ij} and D_{ji}). Due to the several fields where thermodiffusion plays a key role, the determination of those properties, both in binary and ternary mixtures, has been one of the great challenges during the last years, where several techniques have been developed.

Thermogravitational columns where samples are extracted from different column heights, for example, are powerful to investigate the thermodiffusion coefficients of multicomponent systems [1], but even if the theory allows [2], it is unfeasible to determine the diffusion coefficients since each period necessary to analyse the transition is an entirely new experiment. For this reason, manual techniques have been replaced by optical systems that allow analysing the continuous temporal evolution.

Notwithstanding optical methods are uniquely limited to the study of binary mixtures unless special multicomponent systems are studied.

In this work, we analyse the use of Optical Digital Interferometry to track concentration changes in two different devices: a thermogravitational micro-column (μ TGC) used on ground conditions and a Soret cell, part of the Selectable Optical Diagnostic Instrument, which operates in microgravity conditions inside the ISS. Experiments on the latter were made in the framework of the DCMIX4 project in cooperation with the European Space Agency [3].

2. Optical installations

There are three main optical techniques used to determine the diffusion coefficients on liquid mixtures: the Thermal Diffusion Forced Rayleigh Scattering (TDFRS), the Optical Beam Deflection (OBD) and the Optical Digital Interferometry (ODI). Although the three techniques are based on optical systems, the measurement methods are completely different. In TDFRS, the interference of two beams writes a grating in the sample. In the classical method, a small amount of dye added to the mixture is used to covert the intensity grating into a temperature grating, which in turn causes a concentration gradient due to thermodiffusion [4]. OBD, however, uses the deflection of a laser beam to detect the temporal evolution of the concentration gradient inside a Soret cell and ODI relies on the pattern generated by the interference of two laser beams to track concentration variations [5], [6]. Due to the versatility of the technique, the ability to analyse concentration changes at any point in the cell under study, and the success of the first DCMIX campaigns, it was decided to adapt the thermogravitational technique to the ODI analysis system, thus designing a new μ TGC [7].

2.1. The thermogravitational micro-column

The TG technique consists of placing a mixture in a rectangular cavity (in the case of μ TGC $L_x = 0.51$ mm, $L_y = 3$ mm and $L_z = 30$ mm) and applying a horizontal

temperature gradient. The temperature gradient generates a horizontal species separation, which in turn, gives rise to another counteracting flow due to the molecular diffusion. In addition, the effect of gravity generates convective flows amplify the separation along the vertical direction the column. Based on the Furry-Jones-Onsager theory, the components separation in the steady state are related to the thermodiffusion coefficient by Eq.2, while the unsteady-state analysis allows determining the diffusion coefficient in binary mixtures by Eq.3.

$$D'_{T,i} = -\frac{\alpha g L_x^4}{504\nu} \frac{\partial c_i}{\partial z} \quad (2)$$

$$D = \frac{(\alpha\pi\Delta Tg)^2 L_x^6}{\nu^2 9! L_y^2} t_r \quad (3)$$

The heart of the system is a two-laser (473 nm, and 633 nm) Mach-Zehnder interferometer (Figure 1), which allows studying both binary and ternary mixtures.

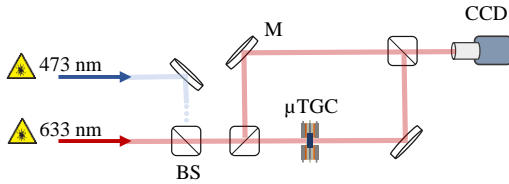


Figure 1: Schematic of the three-laser Mach-Zehnder interferometer used in the μ TGC.

The two-laser beams enter the interferogram (BS1) and are divided into two beams of same intensity (BS2). One of the beam passes through the columns sapphire windows while the other acts as reference. When both beams confront each other once again (BS3), an interference pattern is generated and captured by the CCD camera.

2.2. The Selectable Optical Diagnostic Instrument

SODI's system is similar to the one described in section 2.1, consists of a two-laser Mach-Zehnder interferometer, where two lasers of 670 nm and 935 nm wavelength are used (Figure 2).

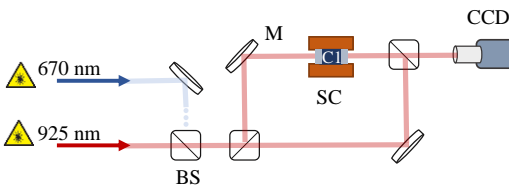


Figure 2: Schematic of the two-laser Mach-Zehnder interferometer used in SODI.

In this case, the analysis cell is a Soret cell of dimensions $L_x = 10$ mm, $L_y = 10$ mm and $L_z = 5$ mm and a thermal gradient is applied in the z direction. The system has the capacity to study six mixtures, which in the case of DCMIX project are five ternary and a reference binary mixture (Figure 3).

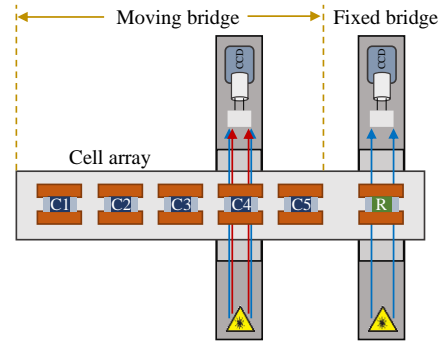


Figure 3: Sketch of the SODI cell array.

For the ternary mixtures, the bridge that holds the interferometer moves from cell to cell while the reference cell where a binary mixture is analysed has another interferometer with a single laser of 925 nm. This technique allows measuring the effective diffusion coefficient by fitting data to Eq. 4 and then to calculate the Soret coefficients by Eq. 5 and Eq. 6 [8].

$$n_i(z, t) = \Delta n_i^{st} \left(\frac{1}{2} - \frac{z}{H} \right) \quad (4)$$

$$- \frac{4}{\pi^2} \sum_{k, \text{odd}} \frac{1}{k^2} \cos\left(\frac{k\pi z}{H}\right) \exp\left(-\frac{k^2 \pi^2}{H^2} Dt\right)$$

$$\begin{pmatrix} \Delta n_1^{st} \\ \Delta n_2^{st} \end{pmatrix} = \begin{pmatrix} \left(\frac{\partial n_1}{\partial c_1}\right) & \left(\frac{\partial n_1}{\partial c_2}\right) \\ \left(\frac{\partial n_2}{\partial c_1}\right) & \left(\frac{\partial n_2}{\partial c_2}\right) \end{pmatrix} \begin{pmatrix} \Delta c_1^{st} \\ \Delta c_2^{st} \end{pmatrix} \quad (5)$$

$$S'_{T,i} = \Delta c_i^{st} / \Delta T \quad (6)$$

$(\partial n_j / \partial c_i)$ correspond to the solutal contrast factor of a component i at a laser wavelength of j .

3. Data post-processing

Although there are differences, the data processing in both techniques follows the same pipeline [7], [8]:

1. Extract the vertical profiles of the unwrapped phase $\phi(x = L/2, z, t)$. In the μ TGC 2D Fourier transform and Itoh's algorithm is used while in SODI the phase stepping and Itho's algorithm.
2. Select the proper reference image, $\phi(z, t_0)$.
3. Subtract the reference phase from the rest of the phase profiles $\phi' = \phi(z, t_i) - \phi(z, t_0)$.
4. Convert the calculated optical phase into refractive index $n'(z, t) = (\lambda/2\pi L) \phi'(z, t)$.
5. Fit the experimental profiles to obtain the analytical solution describing separation.
6. Transform the refractive index steady state separation into concentration separation.

4. Results extraction

This work is not linked to the analysis of a specific mixture but to demonstrate the abilities and limitations of each technique to determine the transport properties. Thus, representative results of each of the techniques are shown. Figure 4 shows the processing steps in the micro-column during the analysis of a binary mixture.

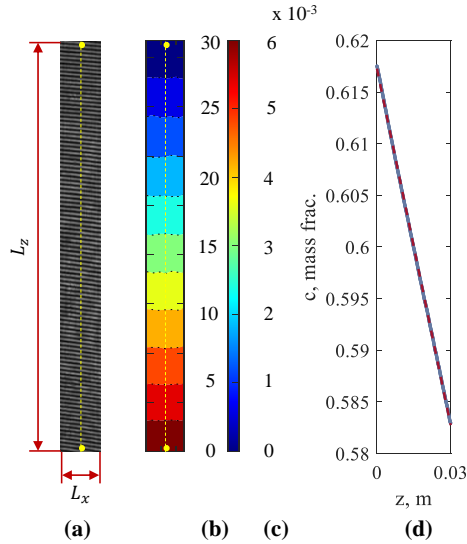


Figure 4: Interferogram (a), phase (b), refractive index (c) and steady state separation in function of the column height in the centreline (d).

As Figure 4 (d) shows the steady state separation $\partial c / \partial z$, which allows determining the DT coefficient by Eq. 2. The Figure 5, on the other hand, shows the refractive index analysis of a single laser in SODI.

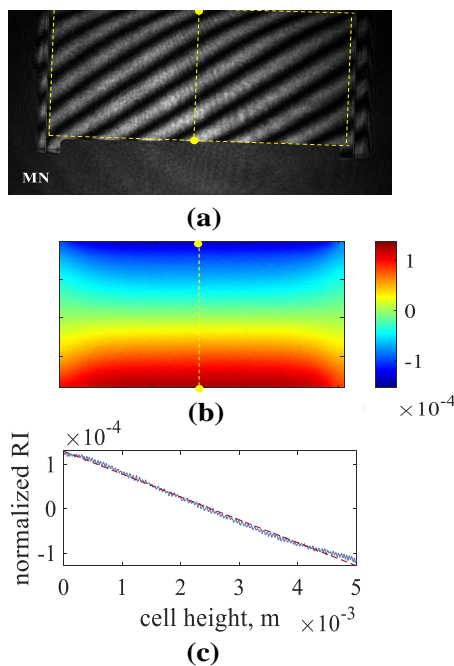


Figure 5: RI analysis of an experiment performed in SODI: interferogram (a), refractive index map (b) and the RI variation in the centreline (c).

Fig. 4 and Fig. 5 show the analysis of a single laser. However, the signals captured by the other sources are processed in the same manner. Final data fitting allows determining the transport properties.

5. Conclusions

Although optical systems allow analysing the time evolution of the components of ternary mixtures, determining the six diffusion coefficients describing the flows of ternary mixtures by a single optical technique involves solving systems of high complexity and low stability. Therefore, to describe the behaviour of these systems, it is necessary to combine different techniques. Combining the μ TGC and SODI, thermodiffusion and Soret coefficients can be obtained, and the experiments of both techniques can be compared by analysing the effective diffusion coefficient. Moreover, the analysis of such mixtures is directly conditioned by the quality of the contrast factor matrix, which is used to transform refractive index data into concentration. This fact limits the number of ternary mixtures that can be studied by optical systems.

References

- [1] M. M. Bou-Ali and J. K. Platten, "Metrology of the thermodiffusion coefficients in a ternary system," *J. Non-Equilibrium Thermodyn.*, vol. 30, no. 4, pp. 385–399, 2005.
- [2] J. Valencia, *et al*, "Fickian Diffusion Coefficient of Binary Liquid Mixtures in a Thermogravitational Column," *J. Non-Equilibrium Thermodyn.*, vol. 32, no. 3, 2007.
- [3] A. Mialdun *et al.*, "Data quality assessment of Diffusion Coefficient Measurements in ternary mixtures 4 (DCMIX4) experiment," *Acta Astronaut.*, vol. 176, pp. 204–215, 2020.
- [4] W. Koehler and P. Rossmanith, "Aspects of Thermal Diffusion Forced Rayleigh Scattering: Heterodyne Detection, Active Phase Tracking, and Experimental Constraints," *J. Phys. Chem.*, vol. 99, no. 16, pp. 5838–5847, 1995.
- [5] A. Königer *et al*, "Measurement of diffusion and thermal diffusion in ternary fluid mixtures using a two-color optical beam deflection technique," *J. Chem. Phys.*, vol. 132, no. 17, pp. 174506, 2010.
- [6] A. Mialdun and V. M. Shevtsova, "Development of optical digital interferometry technique for measurement of thermodiffusion coefficients," *Int. J. Heat Mass Transf.*, vol. 51, no. 11–12, pp. 3164–3178, 2008.
- [7] E. Lapeira, *et al*, "Digital Interferometry Applied to Thermogravitational Technique," *Microgravity Sci. Technol.*, vol. 30, no. 5, pp. 635–641, 2018.
- [8] A. Mialdun, *et al*, "Contribution to the benchmark for ternary mixtures: Measurement of the Soret, diffusion and thermodiffusion coefficients in the ternary mixture THN/IBB/nC12 with 0.8/0.1/0.1 mass fractions in ground and orbital laboratories," *Eur. Phys. J. E*, vol. 38, no. 4, pp. 27, 2015.

Validity limits of the FJO theory – thermogravitational microcolumn: Numerical analysis and experimental validation

A. Sanjuan¹, A. Errarte¹, M. Mounir Bou-Ali¹, C. Santamaria²

¹Mechanical and Manufacturing Department, Mondragon University, Loramendi 4, Apdo. 23, 20500 Mondragon, Spain

²Department of Applied Physics II, University of Basque Country, Apdo. 644, 48080 Bilbao, Spain

Corresponding author: M. Mounir Bou-Ali, mbouali@mondragon.edu

Abstract

In this work, we analysed both numerically and experimentally the validity limits of the Furry, Jones and Onsager theory (FJO). For model validation, we followed previously published numerical studies using ANSYS Fluent 2022 R1 software and the three Benchmark Fontainebleau binary mixtures. We studied the relation between the Grashof (Gr) and Schmidt (Sc) values together with the aspect ratio (A) of the microcolumn also for other mixtures such as polymeric solutions. In this case, we demonstrated the necessity of dimensioning a new column that satisfies the FJO theory.

Keywords: *FJO theory, thermogravitational column and transport properties.*

1. Introduction

Over the last few years, the study and interest in the phenomenon of thermodiffusion has grown significantly [1]. This effect was discovered by C. Ludwig [2] and Ch. Soret [3] in the 19th century describing the separation of species under a temperature gradient. The Soret coefficient (S_T) is the magnitude that represents this phenomenon and in the case of a binary mixture, it is the fraction between the thermodiffusion (D_T) and the molecular diffusion (D) coefficient [4].

Regarding the experimental technique, one of the earliest was the use of thermogravitational columns as in [5]. Previously, this technique and the theoretical description behind were explained by W. H. Furry, R. Clark Jones and L. Onsager known as FJO theory [6]. This theory includes some simplifications of the general hydrodynamic equations such as the forgotten effect.

By tradition, thermogravitational columns are used to obtain the thermodiffusion coefficient [7] when the mixtures reaches the steady state equilibrium by determining at this moment the variation of the concentration along the height of the column. The dimensions of the cavity, both total length (L) and width-gap (W), are limited by the FJO theory [6] in which the following equation is to be satisfied Eqn. (1):

$$Gr Sc < 1000 A \quad (1)$$

The Grashof number is defined as $Gr = (g \alpha W^3 \Delta T) / \nu^2$ and the Schmidt value is given by $Sc = \nu / D$ where g is the gravity force, α is the thermal expansion coefficient, ΔT represent the temperature difference and ν is the kinematic viscosity of the mixture. In addition, the aspect ratio is the fraction between the total length and the gap of the thermogravitational column.

Theoretically, Eqn. (1) assumption only fulfils the steady state analysis. For the study of the transient regime, and therefore also determination of the molecular diffusion coefficient (D), Eqn. (2) must be satisfied [9]. In this context, it is noted that the transitory analysis is even more restrictive.

$$Gr Sc < 200 A \quad (2)$$

The shortcoming of the traditional thermogravitational technique is that large sample volumes are needed and depending on the viscosity of the mixture (for example polymer solutions) the relaxation time to arrive at equilibrium state is much longer. In addition, it is not feasible to perform the transitory study so that it is indispensable to use other experimental methods (for example sliding symmetric tubes [10]) to determine D and thus the S_T magnitude. For this reason, new analysis methods have been developed, among which is the use of optical digital interferometry to obtain D_T and D directly [9] and calculate S_T .

The motivation of this work is to study and understand the physical theory of the thermogravitational effect, in view of the future determination of the transport properties of different complex solutions. The hypotheses of FJO have been analysed and validated numerically using ANSYS Fluent software and the dimensions of the microcolumn presented in [9]. The validity of the limitation of the equations (1) and (2) has been analysed for the three Benchmark Fontainebleau mixtures and a polymer solution. Preliminary numerical results, both in the stationary and transitory regimes showed that it is essential to respect Eqn. (1) and Eqn. (2) playing with the dimensions of the column or/and working conditions.

2. Theoretical approach – FJO theory

According to the FJO theory, the following Eqn. (3) is used to calculate the concentration difference (Δc_{sc}) between two points of the thermogravitational column in the steady state scenario [11].

$$\Delta c_{sc} = 504 \frac{\nu}{\alpha g} D_T c_0 (1 - c_0) \frac{L_z}{L_x^4} \quad (3)$$

where L_z is the vertical length of the analysed column part and L_x the gap distance. Once the concentration difference between two points is known, D_T is determined. For the transitory analysis, the following Eqn. (4) is defined [12]:

$$\Delta c_{tr} = \Delta c_{sc} \left[1 + 4 \frac{L}{L_z} \sum_{k=0}^{\infty} \frac{e^{-\frac{q_k^2 t}{4D}}}{q_k^2} \left(\cos \frac{q_k Z_2}{L} - \cos \frac{q_k Z_1}{L} \right) \right] \quad (4)$$

Δc_{tr} is the concentration variation over time, t_r represent the relaxation time, Z_1 and Z_2 are the vertical distances from the extremities of the microcolumn to the points to be analysed and $q_k = (2K + 1) \pi$ for $K = 0 \dots \infty$.

Fitting the experimental data with the transitory Eqn. (4) the relaxation time (t_r) is determined and therefore, it is possible to obtain D [9] by using the following Eqn. (5):

$$D = \left(\frac{\alpha}{\nu} \right)^2 \frac{(\pi \Delta T g)^2 L_x^6}{9 L^2} t_r \quad (5)$$

In order to get the D value, we need prior knowledge of thermophysical properties such as the thermal expansion coefficient (α) and the kinematic viscosity (ν).

3. Methodology

We analysed the FJO theory both numerically and experimentally using the mixtures presented in Benchmark Fontainebleau (THN-C₁₂, THN-IBB and IBB-C₁₂) [13] and the polymeric solution PS-Tol (polystyrene 4880 g/mol).

We used ANSYS Fluent 2022 R1 software to perform the numerical analysis. First, we defined the cavity of the microcolumn ($L = 30$ mm and $W = 0.51$ mm). To solve the governing equations, we applied the species transport model. Moreover, we defined all the thermophysical properties of the different mixtures using data from literature [9], [14]. Unlike the FJO theory, we defined that the density of the mixture changes both due to the temperature and concentration variation [15]. For that, we considered the Boussinesq approximation Eqn. (6) by means of a user defined function (UDF).

$$\rho = \rho_0 (1 - \alpha(T - T_0) + \beta(c - c_0)) \quad (6)$$

ρ is the mixture's density (ρ_0 initial density), T represent the temperature (T_0 working temperature), β is the mass expansion coefficient and c the concentration of the species (c_0 initial concentration).

We followed the work presented by Nauman et al. [11] for the model validation and the difference in concentration obtained by means of Eqn. (3) (FJO theory). We considered $L_z = 22$ mm. Table 1 shows the results obtained applying the same working conditions as in [11].

Mixture	Δc_{sc} Eqn. (2)	Δc_{sc} [11]	This work
THN-C ₁₂	0.0451	0.0453	0.0452
THN-IBB	0.0180	0.0177	0.0179
IBB-C ₁₂	0.0207	0.0206	0.0206

Table 1: Obtained numerical results and comparison for the model validation.

Regarding the experimental analysis, we focused on the technique presented in [16]. We used the same thermogravitational microcolumn and optical installation. This is a method based on two lasers of different wavelengths (473 nm and 633 nm) passing through the mixture to be analysed and interfering in the point of image acquisition camera. In the case of binary mixtures, it is sufficient to use one of the two available lasers. Then, by processing the obtained images using the Matlab software, it is possible to determine the transport properties relating the phase change with the variation of the refractive index and thus by means of the optical contrast factors acquire the concentration difference along the microcolumn.

4. Preliminary results

Preliminary numerical results of the concentration variation mixtures THN-C₁₂ and PS-Tol (at different ΔT) are shown below ($L_z = 28$ mm) (**¡Error! No se encuentra el origen de la referencia.**).

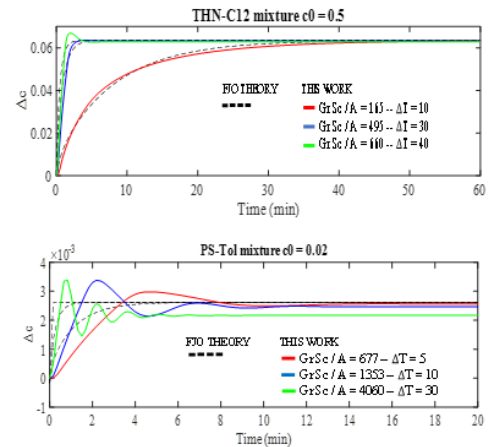


Figure 1: THN-C₁₂ and PS-Tol mixture concentration variation over time. FJO theory & Numerical results.

In the case of the THN-C₁₂ mixture, there is no problem in determining both D_T and D if the condition $Gr Sc < 200 A$ is fulfilled. On the contrary, due to the dimensions of the microcolumn and applied temperature difference, we observed that for the polymeric mixture it

is not reliable to analysis the transport properties in the transitory regime. Furthermore, it is showed that a higher value of $Gr Sc / A$ increases the error between the calculated Δc_{sc} Eqn. (3) and the numerical one. For this reason, we showed the necessity of dimensioning a new thermogravitational column that satisfies the FJO theory according to the characteristics of the mixtures to be analyzed.

5. Conclusions

In this work, the FJO theory has been studied to carry out analysing the transitory-stationary concentration variation of different mixtures. We showed the need to dimension a new thermogravitational column to determine polymeric solution transport properties, since the numerically analysed one ($L = 30$ mm and $W = 0.51$ mm) does not satisfy $Gr Sc < 200 A$ condition for the PS-Tol mixture.

6. Bibliography

- [1] W. Köhler and K. I. Morozov, "The Soret Effect in Liquid Mixtures - A Review," *J. Non-Equilibrium Thermodyn.*, 2016, doi: 10.1515/jnet-2016-0024.
- [2] C. Ludwig, "Diffusion zwischen ungleich erwärmten Orten gleich zusammengesetzter Lösungen," *Sitz. Ber. Akad. Wiss. Wien Math-Naturw.*, vol. 20, p. 539, 1856.
- [3] C. Soret, "Au Point De Vue De Sa Concentration Une Dissolution Saline Primitivement Homogène Dont Deux Parties Sont Portées a Des Températures Différentes.," *Arch. des Sci. Phys. Nat. Genève*, vol. 3, pp. 48–61, 1879.
- [4] J. K. Platten, "The Soret effect: A review of recent experimental results," *J. Appl. Mech. Trans. ASME*, vol. 73, no. 1, pp. 5–15, 2006, doi: 10.1115/1.1992517.
- [5] G. Müller and G. Vasaru, "The Clusius-Dickel Thermal Diffusion Column – 50 Years after its Invention," *Isot. Environ. Heal. Stud.*, vol. 24, no. 11–12, pp. 455–464, Jan. 1988, doi: 10.1080/10256018808624027.
- [6] W. H. Furry, R. C. Jones, and L. Onsager, "On the Theory of Isotope Separation by Thermal Diffusion," *Phys. Rev.*, vol. 55, no. 11, pp. 1083–1095, Jun. 1939, doi: 10.1103/PhysRev.55.1083.
- [7] P. Blanco, M. M. Bou-Ali, J. K. Platten, D. A. de Mezquia, J. A. Madariaga, and C. Santamaría, "Thermodiffusion coefficients of binary and ternary hydrocarbon mixtures," *J. Chem. Phys.*, vol. 132, no. 11, p. 114506, Mar. 2010, doi: 10.1063/1.3354114.
- [8] M. Larrañaga *et al.*, "Contribution to the benchmark for ternary mixtures: Determination of Soret coefficients by the thermogravitational and the sliding symmetric tubes techniques," *Eur. Phys. J. E*, vol. 38, no. 4, p. 28, Apr. 2015, doi: 10.1140/epje/i2015-15028-1.
- [9] B. Šeta, J. Gavalda, M. M. Bou-Ali, X. Ruiz, and C. Santamaria, "Determining diffusion, thermodiffusion and Soret coefficients by the thermogravitational technique in binary mixtures with optical digital interferometry analysis," *Int. J. Heat Mass Transf.*, vol. 147, p. 118935, Feb. 2020, doi: 10.1016/j.ijheatmasstransfer.2019.118935.
- [10] M. Larrañaga, D. A. S. Rees, and M. M. Bou-Ali, "Determination of the molecular diffusion coefficients in ternary mixtures by the sliding symmetric tubes technique," *J. Chem. Phys.*, vol. 140, no. 5, p. 054201, Feb. 2014, doi: 10.1063/1.4864189.
- [11] P. Naumann, A. Martin, H. Krieger, M. Larrañaga, M. M. Bou-Ali, and S. Wiegand, "Development of a Thermogravitational Microcolumn with an Interferometric Contactless Detection System," *J. Phys. Chem. B*, vol. 116, no. 47, pp. 13889–13897, Nov. 2012, doi: 10.1021/jp3098473.
- [12] J. . Valencia, M. . Bou-Ali, J. . Platten, O. Ecenarro, J. . Madariaga, and C. . Santamaría, "Fickian Diffusion Coefficient of Binary Liquid Mixtures in a Thermogravitational Column," *J. Non-Equilibrium Thermodyn.*, vol. 32, no. 3, Jan. 2007, doi: 10.1515/JNETDY.2007.022.
- [13] J. K. Platten *et al.*, "Benchmark values for the Soret, thermal diffusion and diffusion coefficients of three binary organic liquid mixtures," *Philos. Mag.*, vol. 83, no. 17–18, pp. 1965–1971, Jan. 2003, doi: 10.1080/0141861031000108204.
- [14] D. Zapf and W. Köhler, "Thermal and solutal non-equilibrium fluctuations in a polymer solution," *J. Chem. Phys.*, vol. 153, no. 22, p. 224902, Dec. 2020, doi: 10.1063/5.0030854.
- [15] F. H. Horne and R. J. Bearman, "Thermogravitational Thermal Diffusion in Liquids. I. The Formal Theory," *J. Chem. Phys.*, vol. 37, no. 12, pp. 2842–2857, Dec. 1962, doi: 10.1063/1.1733111.
- [16] E. Lapeira, A. Mialdun, V. Yasnou, P. Aristimuño, V. Shevtsova, and M. M. Bou-Ali, "Digital Interferometry Applied to Thermogravitational Technique," *Microgravity Sci. Technol.*, vol. 30, no. 5, pp. 635–641, Oct. 2018, doi: 10.1007/s12217-018-9632-7.

Effet Soret et convection mixte: séparation des espèces d'un fluide binaire de façon continue.

K. Sioud^{1,2}, A. Abdennadher², A. Bergeon¹, M.-C. Charrier-Mojtabi¹, A. Mojtabi¹

Corresponding author A. Mojtabi

1. Institut de Mécanique des Fluides de Toulouse, UMR CNRS INPT/UPS N° 5502, Université de Toulouse, France.

1 mojtabi@imft.fr, marie-catherine.mojtabi@univ-tlse3.fr, sioud.khairi@gmail.com, alain.bergeon@imft.fr

2. Université de Carthage, EPT, LIM (LR 01-ES-13), 2078, La Marsa, Tunisie

Abstract

By introducing a binary mixture, with a positive thermodiffusion coefficient, initially homogeneous, continuously, at the entrance of an open parallelepiped pipe, differentially heated, a vertical stratification of the mass fraction is found at the outlet of this column: the densest compound is at the bottom of the cell and the lightest at the top. Forced convection is associated with thermogravitation to produce continuously separated species. The theoretical and numerical studies developed, show a good agreement between the analytical solution obtained and the results of numerical simulations.

Résumé

En introduisant, de façon continue, un mélange binaire avec un coefficient de thermodiffusion positif, initialement homogène, à l'entrée d'une conduite parallélépipédique ouverte, différentiellement chauffée, on observe au niveau de la sortie de cette conduite une stratification verticale de la fraction massique : le composé le plus dense se retrouve en bas de la cellule et le plus léger en haut. On associe la convection forcée à la thermogravitation pour obtenir une production d'espèces séparées de façon continue. Les études théorique et numérique développées, montrent un bon accord entre la solution analytique obtenue et les résultats de simulations numériques.

Mots clefs : thermogravitation, effet Soret, convection mixte, séparation des espèces

Keywords: thermogravitation, Soret effect, mixed convection, species separation

1. Introduction

Lorsqu'un mélange binaire, initialement homogène, est soumis à un gradient thermique, un transfert de masse se

produit conduisant à l'hétérogénéité du mélange. Cette séparation est connue sous le nom de thermodiffusion ou effet Soret. En plus de l'expression usuelle du flux de masse J donnée par la loi de Fick, une partie due au gradient de température est ajoutée de sorte que :

$$J = -\rho D \nabla C - \rho C(1-C) D_T \nabla T \quad (1)$$

où ρ , D , D_T , représentent respectivement la masse volumique du mélange binaire, le coefficient de diffusion de Fick et le coefficient de thermodiffusion du constituant le plus dense de fraction massique C . T est la température du mélange.

La diffusion thermo-gravitationnelle a été largement étudiée en raison de ses nombreuses applications fondamentales et industrielles. Les auteurs suivants Nield et Bejan [1], Vafai [2] et Legros et al. [3] ont fait une synthèse de ces multiples applications. Après la découverte de l'effet Soret, les études marquantes ont été réalisées en 1939, par Furry, Jones et Onsager [4]. Ces auteurs ont obtenu une solution analytique du problème en considérant que la masse volumique du mélange ne dépend que de la température, au niveau du terme générateur de la convection, dans l'équation de Navier-Stokes (effet oublié). En 1959, Lorenz et Emery [5] ont utilisé des colonnes thermogravitationnelles, TGC, constituées d'un milieu poreux saturé par un mélange binaire, ce qui permet de travailler avec des cellules de plus grande épaisseur et donc d'obtenir une plus grande quantité de produit séparé. Au cours des vingt dernières années, de nombreux travaux ont été réalisés pour mesurer les coefficients de thermodiffusion et augmenter la séparation des espèces dans différentes configurations, colonnes verticales ou inclinées, Dutrieux et al. [6] et Platten et al. [7]. En 2007, Charrier-Mojtabi et al. [8] ont réalisé une étude analytique et numérique de la stabilité de l'écoulement dû à l'effet Soret dans une cavité poreuse horizontale.

Plus récemment, Mojtabi et al. [9] ont étudié l'influence des plaques délimitant la colonne TGC sur la séparation des espèces, en considérant leur épaisseur et leurs propriétés physiques. Mojtabi et al. [10] ont étudié le couplage entre la convection mixte et la thermogravitation dans une cellule horizontale dont les parois sont en

mouvement l'une par rapport à l'autre. Ils ont déterminé l'optimum de séparation en fonction du nombre de Peclet. Dans le présent travail, on considère une configuration différente de celles déjà étudiées. La cellule de thermogravitation est parallélépipédique, et différentiellement chauffée. Le fluide binaire entre avec un débit constant et une composition homogène en $x=0$ et ressort en $x=L$, une fois la séparation des espèces complètement effectuée. On peut, l'état stationnaire étant atteint, séparer le fluide binaire de fraction massique moyenne C_1 de celui de fraction massique moyenne C_2 et recommencer le processus.

2. Formulation mathématique

2.1 Figure de définition

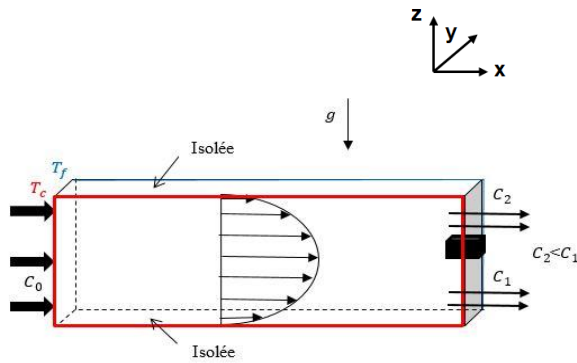


Figure 1 : figure de définition

On considère une cellule verticale, parallélépipédique, de longueur L , d'épaisseur e et de hauteur H avec $e \ll H \ll L$. La cellule ouverte à ses deux extrémités est placée dans le champ de pesanteur. Les deux parois verticales latérales sont isothermes, maintenues respectivement aux températures différentes T_c et T_f et les parois horizontales sont parfaitement isolées. Le fluide binaire entre avec un débit constant et avec une composition homogène C_0 . En sortie une lame séparatrice sépare le fluide de fraction massique C_1 en bas de la couche et C_2 en haut de la couche avec $C_1 > C_2$. Pour l'étude on a pris : $H=30$ mm, $e=0,5$ mm et $L=350$ mm. On justifiera, dans la suite de cette étude, la valeur prise pour L en fonction des données de H et de e .

2-2 Formulation mathématique

Les équations régissant le problème sont : l'équation de continuité, l'équation de Navier-Stokes, l'équation de l'énergie et l'équation de conservation des espèces :

$$\left\{ \begin{array}{l} \nabla \cdot \mathbf{V} = 0 \\ \left[\frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V} \cdot \nabla) \mathbf{V} \right] = -\frac{\nabla P}{\rho_0} - [1 - \beta_T(T - T_0) - \beta_C(C - C_0)]gz + \nu \nabla^2 \mathbf{V} \\ \frac{\partial T}{\partial t} + \mathbf{V} \cdot \nabla T = \nabla \cdot (a \nabla T) \\ \frac{\partial C}{\partial t} + \mathbf{V} \cdot \nabla C = \nabla \cdot [D \nabla C + D_T C_0 (1 - C_0) \nabla T] \end{array} \right. \quad (1)$$

Dans ces équations \mathbf{V} de composantes (u, v, w) désigne le vecteur vitesse, P la pression, β_T et β_C les coefficients d'expansion thermique et massique du fluide binaire, ρ_0 , sa masse volumique à T_0 et C_0 . La viscosité cinématique et la diffusivité thermique du fluide sont respectivement ν et a . Le fluide est supposé incompressible et obéit à l'approximation de Boussinesq.

On suppose que les régimes dynamique et thermique sont établis à l'entrée et que la longueur de la cellule est suffisante pour que le régime massique en sortie de la cellule soit établi.

Les conditions aux limites du problème s'écrivent ainsi : pour la vitesse (V)

$$\left\{ \begin{array}{l} y = 0, e; \mathbf{V} = 0 \\ z = 0, H; \mathbf{V} = 0 \\ x = 0; u = 36 u_{deb} \frac{z}{H} \left(\frac{z}{H} - 1 \right) \frac{y}{e} \left(\frac{y}{e} - 1 \right) \\ x = L; \frac{\partial \mathbf{V}}{\partial x} = 0 \end{array} \right. \quad (2a)$$

Pour la température (T)

$$\left\{ \begin{array}{l} y = 0; T = T_c \\ y = e; T = T_f \\ x = 0, T = (T_f - T_c) y/e + T_c \\ z = 0, H; \frac{\partial T}{\partial z} = 0 \end{array} \right. \quad (2b)$$

pour la fraction massique (C)

$$\left\{ \begin{array}{l} x = 0; C = C_0 \\ x = L; \frac{\partial C}{\partial x} = 0 \\ y = 0, e; D \frac{\partial C}{\partial y} + D_T C_0 (1 - C_0) \frac{\partial T}{\partial y} = 0 \\ z = 0, H; \frac{\partial C}{\partial z} = 0 \end{array} \right. \quad (2c)$$

3. Solution analytique $e \ll H \ll L$

Pour déterminer la solution analytique stationnaire du problème, on remarque que lorsque l'écoulement est établi, toutes les variables sont indépendantes de l'abscisse x d'une section droite perpendiculaire aux plaques chauffées.

On cherche donc une solution sous la forme :

$$\left\{ \begin{array}{l} \mathbf{V} = u(y, z)\mathbf{x} + w(y) \mathbf{z} \\ T = h(y) \\ C = mz + g(y) \end{array} \right. \quad (3)$$

Où m désigne le gradient de fraction massique selon z .

L'expression de la composante $u(z)$ de la vitesse \mathbf{V} suivant x est donnée par la relation classique où la vitesse serait induite par la seule convection forcée. Les équations simplifiées permettant la détermination de la solution analytique sont données par :

$$\left\{ \begin{array}{l} \nu \frac{\partial^3 w}{\partial y^3} + g \left(\beta_T \frac{\partial T}{\partial y} + \beta_C \frac{\partial C}{\partial y} \right) = 0, \quad \frac{\partial^2 T}{\partial y^2} = 0 \\ m w - D \frac{\partial^2 C}{\partial y^2} - D_T C_0 (1 - C_0) \frac{\partial^2 T}{\partial y^2} = 0 \end{array} \right. \quad (4)$$

On obtient, compte tenu des conditions aux limites et des conditions de conservation du débit massique et du flux massique à travers toute section droite $z = cst$ le système (5) ci-dessous.

$$\left\{ \begin{array}{l} w(y) = \frac{-g\beta_T\Delta T}{12\nu e} y(y-e)(2y-e)=0 \\ h(y) = \frac{T_f - T_c}{e} y + T_c \\ g(y) = -\frac{\Delta T}{1440\nu e D} [(e-y)(G(y) + 720C_0(1-C_0)\nu D_T \\ - 720\nu e(2C_0 - Hm)] \\ G(y) = \beta_T g m (6y^4 - 12ey^3 + 4e^2y^2 + 2e^3y + e) \end{array} \right. \quad (5)$$

avec $m = \frac{504vg\beta_T\Delta T^2 e^2 C_0(1-C_0)D_T}{g^2\beta_T^2\Delta T^2 e^6 - 362880D^2v^2}$

4- Comparaison solution analytique- solution numérique

Les résultats analytiques sont alors comparés aux résultats de simulations numériques directes (résolution du système (1) avec les conditions aux limites (2)). La simulation numérique est réalisée par éléments finis à l'aide du logiciel COMSOL Multiphysics en 2D et 3D.

Le mélange binaire considéré est un mélange 50 % eau- 50 % éthanol (Seta et al [11]).

On vérifie que la composante de la vitesse selon y est bien nulle-

On reporte sur la Fig2. la composante verticale de la vitesse w(y) dans tout plan x=Cst une fois l'écoulement établi. Les résultats analytiques sont en très bon accord avec les résultats numériques

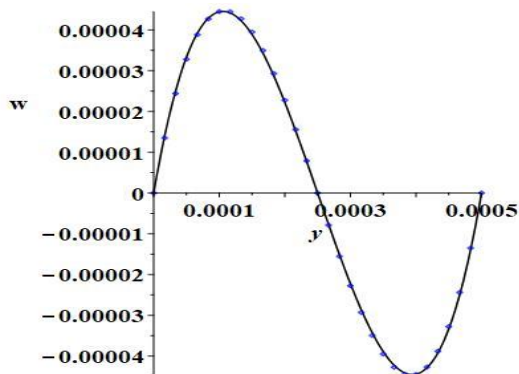


Fig.2 : comparaison de la composante verticale de la vitesse, w(y): analytique, trait continu, numérique, points noirs, w en m/s et y en m .

L'approximation de type écoulement parallèle dans le plan de section droite (y, z) est bien confirmée (e<<H).

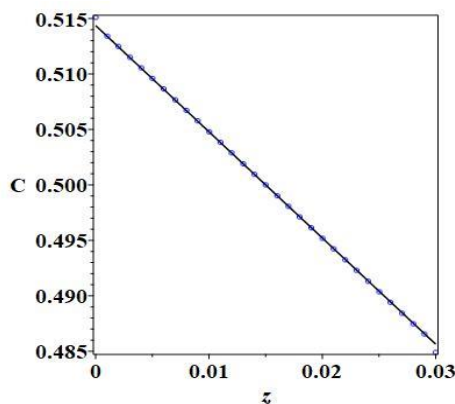


Fig 3 : Evolution de la concentration C=f(z) à y fixé : analytique, trait continu, numérique, points noirs.

Sur la Fig. 3. on a représenté l'évolution de la fraction massique en fonction de z, en x=320mm qui est la longueur d'établissement du régime massique (Fig.4). On retrouve bien la variation rectiligne de C en fonction de z ainsi qu'un bon accord entre résultats analytiques et numériques.

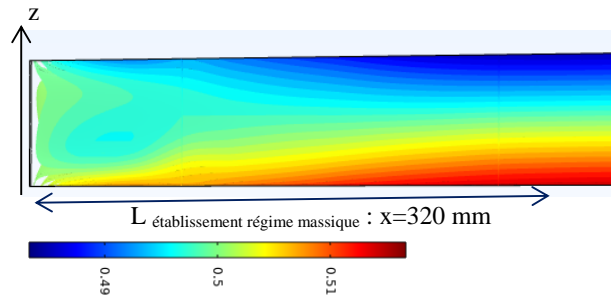


Fig.4. : Longueur d'établissement du régime massique

On peut noter que compte tenu des valeurs de la viscosité cinématique, de la diffusivité thermique et massique du fluide binaire, le temps d'établissement du régime dynamique est inférieur à celui du régime thermique, lui-même très inférieur à celui du régime massique

Références

- [1] D.A. Nield, A. Bejan, Convection in Porous Media (Springer, 1998).
- [2] K. Vafai, Handbook of Porous Media (CRC Press, 2015).
- [3] J.C. Legros, Y. Gaponenko, A. Mialdun, T. Triller, A. Hammon, C. Bauer, W. Khler, V. Shevtsova, Phys. Chem. Chem. Phys. 17, 27713 (2015).
- [4] W.H. Furry, R.C. Jones, L. Onsager, On the theory of isotope separation by thermal diffusion, Phys. Rev. 55 (1939) 1083–1095
- [5] M. Lorenz and A. H. Emery, The packed thermal diffusion column. Chem. Engng Sci. 11, (1959), 16-23
- [6] J.F. Dutrieux, J.K. Platten, G. Chavpeyer, M.M. Bou-Ali, On the measurement of positive Soret coefficients J. Phys. Chem. B, 106 (2002), pp. 6104-6114.
- [7] J.K. Platten, M.M. Bou-Ali, J.F. Dutrieux, Enhanced molecular separation in inclined thermogravitational columns, J. Phys. Chem. B, V.107 I.42 (2003) 11763-11767
- [8] M.C. Charrier-Mojtabi, B. Elhajjar and A. Mojtabi, Analytical and numerical stability analysis of Soret-driven convection in a horizontal porous layer, Physics of Fluids, V.19, I.12 (2007), A.N.124104.
- [9] A. Mojtabi, B. Ouattara, D.A. Rees, M-C. Charrier-Mojtabi, The effect of conducting bounding horizontal plates on species separation in porous cavity saturated by a binary mixture, Int. J. of Heat and Mass transfer, V.126, (2018), 479-488.
- [10] A. Mojtabi, A. Khouzam, Y. Loujaine, M-C. Charrier-Mojtabi, Analytical and numerical study of Soret mixed convection in two sided lid-driven horizontal cavity : Optimal species separation, Int. J. of Heat and Mass transfer, V.139, (2019), 1037-1046
- [11] B. Seta, E. Lapeira, D. Dubert, F. Galvada M.M.Bou-Ali, X.Ruiz, Separation under thermogravitational effects in binary mixture, Eur. Phys. J.E. (2019), pp.42-58.

Double-Diffusive Mixed Convection in a Shallow Rectangular Cavity with Double Lid-Driven Boundaries Filled with Newtonian Fluid: Opposing Flow Case

Y. TIZAKAST¹, M. KADDIRI², M. LAMSAADI³

1. Sultan Moulay Slimane University, LGI, BP 523, Beni-Mellal 23000, Morocco (youssef.tizakast@usmba.ac.ma)
2. Sultan Moulay Slimane University, LGI, BP 523, Beni-Mellal 23000, Morocco (mouradkadiri@usms.ma)
3. Sultan Moulay Slimane University, LRPSI, BP 523, Beni-Mellal 23000, Morocco (lamsaadima@yahoo.fr)

Abstract

The present work investigates double-diffusive mixed convection inside a rectangular cavity filled with a Newtonian fluid, both horizontal boundaries slide in opposite directions and the short vertical walls are subjected to uniform heat and mass fluxes. The sliding walls and applied gradients act in opposite directions leading to opposing flow case. The study establishes two approaches to solve the problem, a numerical solution using the finite difference method and an analytical solution derived based on the parallel flow assumption. The governing parameters are: Peclet number Pe , thermal Rayleigh number Ra_T , Lewis number Le , and buoyancy ratio N . Both implemented approaches illustrate perfect agreement for a wide range of governing parameters. The results show that Ra_T strengthens natural convection while Pe enhances the contribution of forced regime in overall convection. Furthermore, in the mixed convection regime, fluid circulation and heat and mass transfer rates decreased given the opposing flow nature of the problem.

Keywords: *Double-diffusive mixed convection, Opposing flow, Heat and mass transfer, Lid-driven cavities, Parallel flow, Finite difference method.*

1. Introduction

Mixed convection fluid flows occur due to combined shear force generated by sliding walls (or other external source) and buoyancy forces generated by applied temperature and concentration gradients. The majority of published papers in mixed convection field cover flow and heat transfer, Gangawane and Gupta [1] investigated mixed convection in a rectangular cavity with a moving vertical wall. They reported that the direction of the sliding wall strongly influences heat transfer characteristics. Lamarti et al. [2] considered a two-dimensional square cavity with a periodically oscillating lid and filled with a Newtonian fluid. The results showed that varying Reynolds and Grashof numbers affects strongly flow structure and heat transfer depending on the

wall moving direction. Moreover, varying Rayleigh number and the period of the heated portion influences heat transfer on convective structures. Other studies investigated the same phenomenon considering different combinations of cavity configurations and imposed temperature gradients [3-5].

The driven cavity configuration can be found in several engineering and industrial applications such as solar ponds, crystal growing, materials processing, glass production, food processing, and many others. In such applications, when there is heat transfer there is mass transfer; thus, the importance of double-diffusive mixed convection studies. Double-diffusive convection refers to fluid flows generated by buoyancy effects due to imposed thermal and solutal gradients with different diffusion rates. Sivasankaran et al. [6] examined the influence of sinusoidal heating on double-diffusive mixed convection in a lid-driven square cavity. They reported that the phase deviation and the amplitude ratio affect heat and mass transfer rates. Ababaei et al. [7] considered a right-angled trapezoidal cavity filled with a Newtonian fluid and partially heated and salted from the bottom wall. The results showed that for Lewis number $Le = 0.1$, the conduction mass transfer dominates mass transfer. The authors also reported that increasing Le decreases average Nusselt number.

The literature review shows a lack of studies considering double-diffusive mixed convection within rectangular cavities. What is more, the opposing flow case of double-diffusive mixed convection (natural and forced convection acts in opposite directions) inside rectangular lid-driven cavities with applied thermal and solutal boundary conditions of Neumann type (i.e., imposed heat and mass fluxes to the boundaries) has not been studied thus far. To cover the gap, the present study investigates both numerically and analytically double-diffusive mixed convection opposing flow case inside a double lid-driven horizontal rectangular enclosure filled with a Newtonian fluid.

2. Physical Configuration and Mathematical Model

The present study investigates the configuration in Figure 1, a shallow horizontal rectangular cavity of height H' and length L' with the horizontal long walls considered insulated and impermeable while uniform density of heat

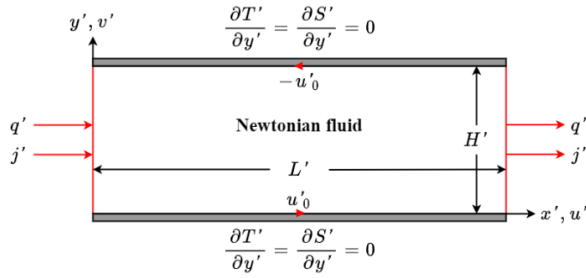


FIGURE 1. Geometry of the cavity along with the associated boundary conditions and coordinates system. and mass fluxes q' and j' , respectively, are applied to the short vertical boundaries. As for the kinematic boundary conditions, the bottom wall slides from left to right with constant velocity u'_0 , while the top wall moves in the opposite direction with the same uniform velocity u'_0 (i.e., the moving walls act in the opposite direction of the imposed heat and mass fluxes); whereas the vertical ones are motionless.

This work assumes the assumptions commonly used, i.e.,

- Flow is laminar given the small gradients [8].
- The fluid is incompressible.
- The viscous dissipation is negligible.
- Physical properties are considered independent of temperature and concentration except for viscosity and density in the buoyancy term. The density obeys the Boussinesq approximation.
- The third dimension of the cavity is large enough to consider the problem two dimensional [8].

For dimensionless variables, we use the characteristic scales: H' , $\rho(\alpha^2/H'^2)$, H'^2/α , α/H' , $q'H'/\lambda$, and $j'H'/D$ corresponding to length, pressure, time, velocity, characteristic temperature, and characteristic concentration, respectively, where ρ , α , λ and D are fluid density, thermal diffusivity, thermal conductivity, and mass diffusivity, respectively. Consequently, considering the stream function Ψ and the vorticity Ω given by:

$$u = \frac{\partial \Psi}{\partial y}; \quad v = -\frac{\partial \Psi}{\partial x} \quad (1)$$

$$\Omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \quad (2)$$

the dimensionless governing equations reduces to:

$$\frac{\partial \Omega}{\partial t} + \frac{\partial(u\Omega)}{\partial x} + \frac{\partial(v\Omega)}{\partial y} = Pr \left[\left(\frac{\partial^2 \Omega}{\partial x^2} + \frac{\partial^2 \Omega}{\partial y^2} \right) + Ra_T \left(\frac{\partial T}{\partial x} + N \frac{\partial S}{\partial x} \right) \right] \quad (3)$$

$$\frac{\partial T}{\partial t} + \frac{\partial(uT)}{\partial x} + \frac{\partial(vT)}{\partial y} = \left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right] \quad (4)$$

$$\frac{\partial S}{\partial t} + \frac{\partial(uS)}{\partial x} + \frac{\partial(vS)}{\partial y} = \frac{1}{Le} \left[\frac{\partial^2 S}{\partial x^2} + \frac{\partial^2 S}{\partial y^2} \right] \quad (5)$$

$$\frac{\partial^2 \Psi}{\partial x^2} + \frac{\partial^2 \Psi}{\partial y^2} = -\Omega \quad (6)$$

associated with the following boundary conditions:

$$u = v = 0; \quad \frac{\partial T}{\partial x} + 1 = \frac{\partial S}{\partial x} + 1 = 0 \quad \text{for } x = 0 \text{ and } x = A; \quad (7)$$

$$u - Pe = v = 0 \text{ and } \frac{\partial T}{\partial y} = \frac{\partial S}{\partial y} = 0 \quad \text{for } y = 0; \quad (8)$$

$$u + Pe = v = 0 \text{ and } \frac{\partial T}{\partial y} = \frac{\partial S}{\partial y} = 0 \quad \text{for } y = 1 \quad (9)$$

As a result, the following dimensionless parameters appear:

$$A = \frac{L'}{H'}; \quad Pr = \frac{\nu}{\alpha}; \quad Pe = \frac{u'_0 H'}{\alpha}; \quad Ra_T = \frac{g \beta_t H'^4 q'}{\nu \alpha \lambda};$$

$$Le = \frac{\alpha}{D}; \quad N = \frac{\beta_S \Delta S^*}{\beta_T \Delta T^*} \quad (10)$$

corresponding to: aspect ratio of the enclosure A , Prandtl number Pr , Peclet number Pe , thermal Rayleigh number

Ra_T , Lewis number Le , and buoyancy ratio N , respectively.

The average horizontal Nusselt number, describing the overall horizontal heat transfer, and the average horizontal Sherwood number, describing the overall horizontal mass transfer, are given respectively as follows:

$$\overline{Nu} = \int_0^1 Nu(y) dy \quad (11)$$

$$\overline{Sh} = \int_0^1 Sh(y) dy \quad (12)$$

with the local Nusselt and Sherwood numbers Nu and Sh , respectively, are assessed far from the cavity end sides:

$$Nu(y) = -1/(\partial T / \partial x)_{x=A/2} \quad (13)$$

$$Sh(y) = -1/(\partial S / \partial x)_{x=A/2} \quad (14)$$

This is done to avoid problems related to edge effects which result in complex flows near the vertical walls, allowing us to accurately compare numerical and analytical solutions.

3. Numerical and Analytical Approach

The well-known second-order central finite-difference scheme with a uniform mesh is used to numerically solve the governing equations. The integrations of vorticity, energy, and concentration equations (3)-(5) are performed using the alternating-direction implicit method (ADI) commonly used for Newtonian fluids. To satisfy the conservation of mass, the Poisson equation (6) is solved by a point successive over-relaxation method (PSOR). For an aspect ratio $A = 24$ (the smallest value of A beyond which mixed convection heat and mass transfer does not change), a uniform grid size of 341×81 is judged enough to model accurately the fluid flow, temperature, and concentration distributions based on trial-and-error calculations conducted to find the best trade-off between computation time and solution accuracy.

Examining the flow structure shows a parallel aspect with regard to the horizontal boundaries of the shallow enclosure; while temperature and concentration fields are linearly stratified in the x -direction independently of the value of Pe . Such observations confirm the existence of an analytical solution based on the parallel flow approximation. For more details on both approaches, numerical and analytical, refer to Tizakast et al. [9].

4. Results and Discussion

The effect of Peclet number Pe on flow characteristics $|\Psi|$, \overline{Nu} , and \overline{Sh} is presented in figure 2 for $Le = 3, N = 1$, and different values of Ra_T . For feeble magnitudes of shear force (generated by the sliding walls) associated with low values of Pe , natural convection regime dominates the overall convection as the fluid flow intensity and heat and mass transfer rates are insensitive to Pe augmentation. Furthermore, the results match the ones obtained in the case of pure natural convection presented with dotted lines, where increasing Ra_T enhances flow characteristics. After a given value of Pe , the three quantities begin to decrease as the shear force and buoyancy forces, acting in opposite directions, compete against each other and hurt the transport phenomenon which indicates that the convective regime shifts from natural to mixed convection. The value of Pe signaling such shift depends on Ra_T , as increasing thermal Rayleigh number strengthens the buoyancy

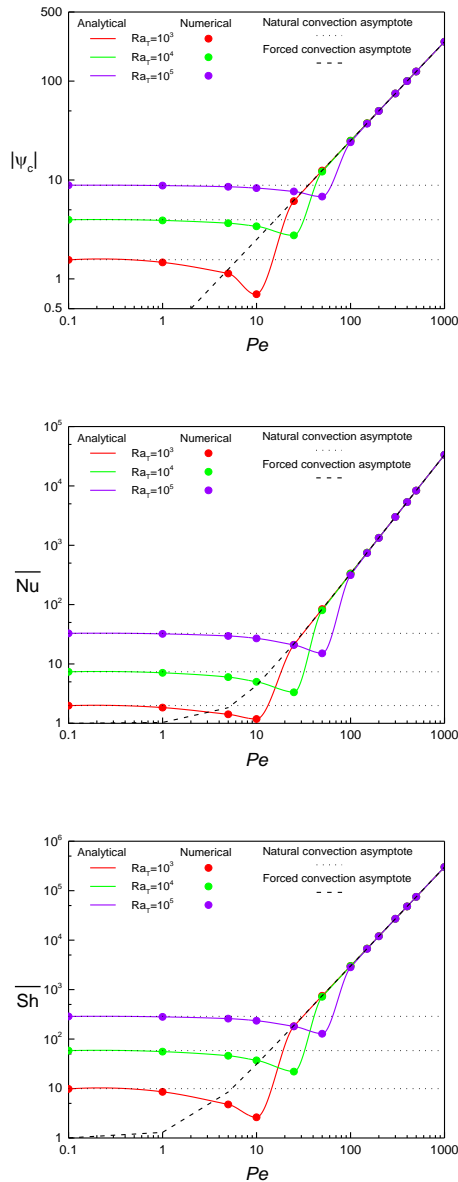


FIGURE 2. Evolutions of stream function at the center of the cavity ($x = A/2$, $y = 1/2$) (top), Nusselt number (middle), and Sherwood number (bottom) with Pecllet number (Pe) for $Le = 3$, $N = 1$, and different values of Ra_T .

forces; thus, delays the transition from natural regime to mixed one as it requires higher Pe value (higher shear force). Next, the illustrated quantities reach a minimum before start rising slowly as the shear force intensifies and overthrow buoyancy forces as the main driving force. Finally, for higher values of Pecllet number, flow characteristics $|\Psi_c|$, \overline{Nu} , and \overline{Sh} increase in a monotonous way as the influence of Ra_T vanishes, indicating that forced regime dominates overall convection as the results agree with the ones obtained in pure forced convection illustrated with dashed lines.

5. Conclusions

The present paper investigates the opposing flow case of double-diffusive mixed convection inside a shallow rectangular cavity filled with Newtonian fluid, both

numerically and analytically. The two approaches show a perfect agreement for a wide range of governing parameters; thus, validating the numerical code, the parallel flow approximation, and the choice of large aspect ratio approximation value $A = 24$. The resulting main conclusions can be summarized as follows:

- Increasing thermal Rayleigh number enhances the input of natural convection (higher buoyancy forces) leading to enhance fluid circulation and heat and mass transfer.
- In mixed convection regime (natural and forced convection are of comparable magnitudes), increasing Pecllet number decreases fluid flow and heat and mass transfer instead of enhancing it given that buoyancy forces and shear one act in opposite directions.
- Higher Pecllet numbers lead to dominant forced convection where flow intensity and heat and mass transfer rates increase linearly with Pe .

References

- [1] K. M. Gangawane et al., *Mixed convection characteristics in rectangular enclosure containing heated elliptical block: Effect of direction of moving wall*, Int J of Therm Sci (2018), pp.100-15.
- [2] H. Lamarti et al., *Numerical simulation of mixed convection heat transfer of fluid in a cavity driven by an oscillating lid using lattice Boltzmann method*, Int J of Heat and Mass Transfer 137 (2019), pp.615–29.
- [3] M. J. H. Munshi et al., *A Numerical Study of Mixed Convection in Square Lid-Driven with Internal Elliptic Body and Constant Flux Heat Source on the Bottom Wall*, J. Sci. Res. 9 (2017), pp. 145-58.
- [4] M. A. Waheed, *Mixed convective heat transfer in rectangular enclosures driven by a continuously moving horizontal plate*. Int J Heat Mass Transfer 52 (2009), pp. 5055–63.
- [5] M. A. R. Sharif, *Laminar mixed convection in shallow inclined driven cavities with hot moving lid on top and cooled from bottom*, Appl Therm Eng 27 (2007), pp. 1036–42.
- [6] S. Sivasankaran et al., *Double diffusive mixed convection in a lid-driven cavity with non-uniform heating on sidewalls*, Sādhanā 42 (2017), pp. 1929–41.
- [7] A. Ababaei et al., *Numerical simulation of double-diffusive mixed convection and entropy generation in a lid-driven trapezoidal enclosure with a heat source*, Numerical Heat Transfer, Part A: Applications (2018), pp. 702-20.
- [8] D. A. Siginer et al., *On the laminar free convection and stability of grade fluids in enclosures*, Int J Heat Mass Transfer 43 (2000), pp. 3391–405.
- [9] Y. Tizakast et al., *Thermosolutal Mixed Convection in Shallow Rectangular Cavity with Imposed Uniform Heat and Mass Fluxes and Filled with Newtonian Power-Law Fluid*, Inter J Heat and Technology, 39 (2021), pp. 669-680.

Thermal and energy performance investigation of a smart double skin facade through CFD simulations

O. IKEN¹, S.D. FERTAH², R. AGOUNOUN¹, I. KADIRI¹, M. RAHMOUNE¹, K. SBAI¹, R. SAADANI¹

1. Université Moulay Ismail, Laboratoire d'Etude des Matériaux Avancés et Applications, FS-EST de Meknès, Morocco.
 2. Thermodynamics and Energy research team, Energy Research Center, Mohammed V University in Rabat, Morocco.
- Email: o.iken@edu.umi.ac.ma

Abstract

In this paper, a smart configuration of a double skin façade with the ability to control the greenhouse effect generation was numerically simulated using a two dimensional Computational Fluid Dynamic (CFD) method. The objective of these simulations is to study the thermal and energy behavior of a DSF integrating Tungsten (W) doped Vanadium dioxide (VO_2) as an optically smart thin material and a high absorbing aluminum nitride (AlN) coating. A parametric study was carried out to analyze the effect of the air cavity thickness on the smart DSF behavior. Results have shown that the smart configuration decreases significantly both heating and cooling loads. Indoor surface temperature has been reduced by around 2 °C during summer and amplified by around 3 °C during a sunny winter day.

Keywords : *Smart DSF, Thermochromic materials, CFD simulations, Heating and cooling loads, Energy efficiency.*

1 Introduction

The urban heat island phenomenon is considered as a climatic effect, which is challenging cities over the world [1]. In fact, it was carried out that it is responsible of amplifying the urban environment temperature causing the deterioration of the indoor comfort of buildings [2]. Thus, preserving the indoor comfort stability needs high levels of energy consumption. This situation leads to the urban heat island. therefore, a vicious circle [3] is created between the necessity to preserve indoor comfort and the amplification of heat island impact.

In this communication, a smart DSF was described as an powerful configuration in terms of energy consumption reduction mostly during winter. This capacity is the result of the greenhouse effect that the DSF generates in the air gap between the glazed external wall and the massive wall. What makes this new DSF smart is the use of VO_2 as a solar transmittance controller on the glazed outdoor side of the DSF [4]. Moreover, an AlN solar heat absorber coating is integrated on the DSF internal facade. This coating is applied to increase the greenhouse effect

produced in the air cavity of this new smart DSF. Numerical simulations based on CFD method was performed to quantify the impact of the smart DSF with VO_2 as a solar transmittance controller and AlN as an absorber coating on building indoor comfort and energy consumption.

2 Methods

2.1 Description of the configuration

The smart DSF proposed in this study is respectively composed from outdoor to the indoor of a tungsten (W) doped VO_2 thin film deposited on a glass substrate, an air gap, an AlN coating, a first cement plaster layer, brick and a second cement plaster layer as shown in Fig.1. A parametric investigation of the air gap thickness effect on the greenhouse phenomenon generation was performed. Four air gap thicknesses were studied. the aspect ratio of the DSF was calculated for each configuration. It has been defined as the air gap and the length of the DSF (1 m) ratio. The thickness values are listed as follows: 10, 25, 50 and 75.

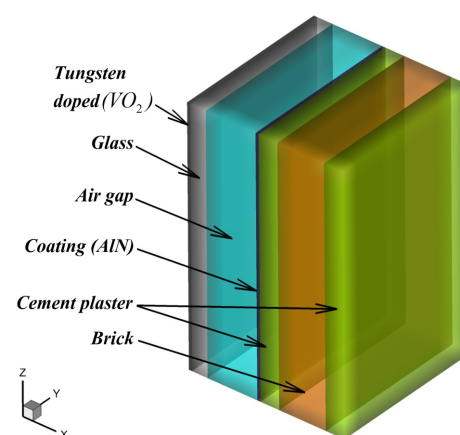


Figure 1: Schematic description of the studied multilayer configuration.

2.2 Physical modeling

The physical model implemented to calculate the heat transfer modes through the wall configuration presented in Fig.1 was described in the following section. The thermal conduction take place in the DSF solid sections such

as the thin vanadium material, the glass, the AlN coating in addition to the brick and the cement plaster. The natural convection occurs in the air gap with different width ($e = 10$ mm, $e = 25$ mm, $e = 50$ mm and $e = 75$ mm). finally, the thermal radiation exchange take place between the internal surface of the glass material and the AlN coating. The commercial CFD code allows the resolution of Navier-Stokes equations in unsteady flow regime (Unsteady Reynolds Averaged Navier Stokes (URANS)[5]). In addition, a coupling between the energy equation and the radiation transport equation (RTE) was performed. The resolution of Navier-Stokes equations aims to calculate the instantaneous evolution of the velocity and pressure fields in the air cavity [6] having variable thickness according to the considered case study. The mass conservation equation is presented in Eq.1 [7]. Where ρ is the density of the air filling the cavity.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \mathbf{u} = 0 \quad (1)$$

The momentum equations of the air filling the enclosure of the DSF is expressed in Eq. 2 [8].

$$\rho \frac{\partial \mathbf{u}}{\partial t} + (\rho \mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \nabla \cdot \bar{\tau} - \rho \beta (T - T_{ref}) g \quad (2)$$

The diffusion/convection equation is solved to show the dynamic evolution of the air filling the enclosure temperature field and the heat passing throughout the different solide sections of the DSF as shown in Eq. 3 [9]. Where C_p is the specific heat and λ is the thermal conductivity.

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T = \nabla \cdot (\lambda \nabla T) \quad (3)$$

The general form of the radiative transfer equation (RTE) for an absorbing, emitting, and scattering medium is expressed in Eq. 4.

$$\frac{dI(\vec{r}, \vec{s})}{ds} + (a + \sigma_s) I(\vec{r}, \vec{s}) = an^2 \frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\vec{r}, \vec{s}') \phi(\vec{s}, \vec{s}') d\Omega' \quad (4)$$

3 Results

3.1 Air velocity

The studied DSF air cavity in this paper is enclosed. Thus, the cavity air momentum is controlled only by the transmitted amount of solar radiative heat, which is regulated by the smart coating W doped VO_2 . During summer, the sol-air temperature is almost above 25°C . Thus, the smart coating is in its metallic reflective state and reflects around 70% of near infrared (NIR) solar radiations [10]. In this case, the cavity air does not receive enough solar radiative heat to accelerate its movement compared with the cold season. Consequently, the greenhouse effect is

minimized in summer [11]. The air velocity maximum value during summer is 0.103 m/s for $e=75$ mm (Fig.2-(a)). The summer's and winter's air velocities comparison has showed the greenhouse effect controlling impact of the smart coating [12]. During winter, the air velocities are considerably higher compared with summer season. This is related to the W doped VO_2 high transmittance (70%) of NIR solar radiations, which increases the greenhouse effect. Winter's air velocity maximum value is 0,142 m/s for $e=75$ mm (Fig.2-(b)). It is noticed that the air mass momentum is highly concentrated near the inner surfaces of the facade [13]. This localization is due to the high radiative heat transfer near the glass substrate and the AlN coating.

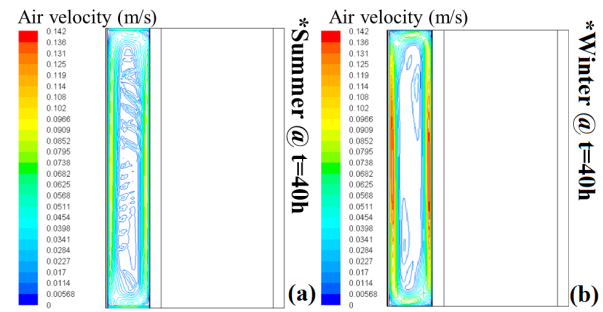


Figure 2: Air velocity during summer and Winter at 8 pm in the smart DSF configuration with $e=75$ mm.

3.2 Contours of temperature

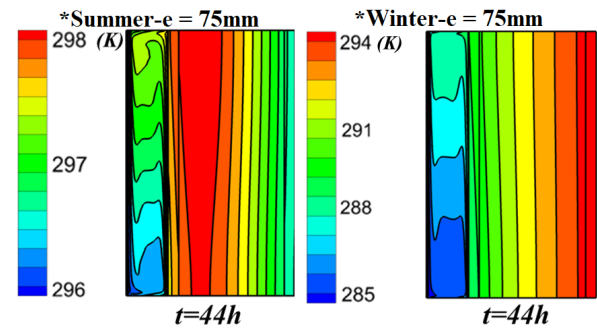


Figure 3: Temperature contours of the smart DSF configuration with a 75 mm air gap at 8 pm.

The 75 mm and 50 mm air gap smart DSF configurations (Fig.3) show practically the same results. The natural convection-radiation interaction effect can be more noticed on the air isotherms shape with more pronounced buoyancy effects close to the heat transfer surfaces. Thus, the greenhouse effect is more pronounced in this configuration. During summer, an average indoor surface temperature of around 25°C is noticed at 8 pm. During winter, high indoor surface temperature is maintained at around 20°C even with the low solar radiation impact at

the selected moments.

3.3 Heating and cooling loads

Transmission loads presented in Fig.4 show the total consumed and gained energy in a summer and a winter day. During summer (Fig.4-(a)), the uninsulated and the 10 mm smart DSF were the highest energy consuming configurations with respectively 1.36 and 0.05 MJ/m². The other smart DSF configurations (25, 50 and 75 mm) have ensured energy gains values around 0.45, 0.69 and 0.81 MJ/m² respectively. During winter (Fig.4-(b)), the uninsulated and the 10 mm smart DSF configuration have presented heating energy demand of around 1.20 and 0.04 MJ/m² respectively. Otherwise, the smart DSF configurations with 25, 50 and 75 mm have presented energy gains values around 0.78, 1.05 and 1.34 MJ/m² respectively.

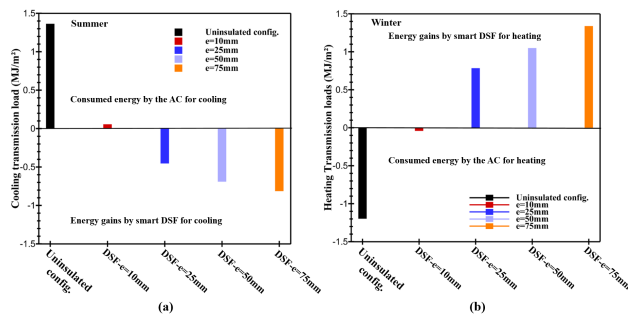


Figure 4: Transmission loads during (a)-summer day (b)-winter day

4 Conclusion

as a conclusion, a smart DSF had been numerically designed by using a W doped VO_2 thin film as a solar radiation controller and an AlN coating as a solar absorber. This smart DSF configuration can be considered as an efficient solution in terms of building thermal insulation.

References

- [1] P. Coseo, L. Larsen, How factors of land use/land cover, building configuration, and adjacent heat sources and sinks explain urban heat islands in chicago, *Landscape and Urban Planning* 125 (2014) 117 – 129.
- [2] C. Shen, X. Li, S. Yan, Numerical study on energy efficiency and economy of a pipe-embedded glass envelope directly utilizing ground-source water for heating in diverse climates, *Energy Conversion and Management* 150 (2017) 878–889.
- [3] X. long Meng et al., A novel absorptive/reflective solar concentrator for heat and electricity generation: An optical and thermal analysis, *Energy Conversion and Management* 114 (2016) 142 – 153.
- [4] O. Iken, R. Agounoun, I. Kadiri, M. Rahmoune, K. Sbai, R. Saadani, Optically smart thin materials for building cooling, in: *Eco-efficient Materials for Reducing Cooling Needs in Buildings and Construction*, Elsevier, 2021, pp. 355–399.
- [5] M. Ilie, Numerical studies of turbulent swirling reacting flows using les and urans, *International Journal of Thermal Sciences* 134 (2018) 89 – 100.
- [6] I. V. Miroshnichenko, M. A. Sheremet, Turbulent natural convection combined with thermal surface radiation inside an inclined cavity having local heater, *International Journal of Thermal Sciences* 124 (2018) 122 – 130.
- [7] T. B. et al., Numerical modeling and optimization of thermal stratification in solar hot water storage tanks for domestic applications: Cfd study, *Solar Energy* 157 (2017) 441 – 455.
- [8] S. ed-Dîn Fertahi et al., Experimental study and cfd thermal assessment of horizontal hot water storage tank integrating evacuated tube collectors with heat pipes, *Solar Energy* 170 (2018) 234 – 251.
- [9] Z. F. et al, A new modelling method for superalloy heating in resistance furnace using fluent, *International Journal of Heat and Mass Transfer* 128 (2019) 679 – 687.
- [10] O. Iken, M. Dlimi, R. Agounoun, I. Kadiri, S. ed Din Fertahi, A. Zoubir, K. Sbai, Numerical investigation of energy performance and cost analysis of moroccan's building smart walls integrating vanadium dioxide, *Solar Energy* 179 (2019) 249 – 263.
- [11] H. Poirazis, Double skin facades for office buildings, literature review (2004).
- [12] Y. Kwak, J.-H. Huh, Development of a method of real-time building energy simulation for efficient predictive control, *Energy Conversion and Management* 113 (2016) 220–229.
- [13] E. O. et al., Double-skin facades : integrated planning, München : Prestel, 2001.

NUMERICAL STUDY OF THE THERMAL BEHAVIOR WITHIN TWO ANISOTROPIC MEDIA IN CYLINDRICAL GEOMETRY.

Zakaria EL HADY¹, Hamza HAMID, Jawad LAHJOMRI, Abdelaziz OUBARRA

Laboratory of Mechanics, Faculty of Science Ain Chock, University Hassan II, Casablanca 20100, Morocco.

zakaria.elhadi-etu@etu.univh2c.ma

Abstract

The impact of the anisotropy within two anisotropic cylindrical media in two-dimensional configuration space on the thermal level and the appearance of the Isotherms was investigated. The existence of the mixed-derivative terms in the thermal heat equations, make it difficult to obtain an analytical solution. For the purpose of overcoming this complicity, a numerical approach based on a linear coordinate transformation was conducted to transform the anisotropic equations to an equivalent isotropic equations form. The alternating directions method (ADI) is carried out for the integration of the thermal heat conduction equations and their boundary and interface conditions. The numerical results presented, are achieved by a digital code which is developed and successfully confirmed.

Keywords : Anisotropic mediums, Cylindrical geometry, Heat conduction , ADI.

1. Introduction

Anisotropic materials has recently drawn great interest in the areas of solar energy automobile, heaters and aerospace industries. This type of materials is oriented in order to meet new needs in term of thermal resistance which is a case that interests us. With the intention to analyze the thermal behaviour of these advanced materials, a comprehensive study of two anisotropic media is at the core of this research.

The investigation of thermal conduction of this type of materials gives rise to partial differential equations that have second-order derivative terms, these terms complicate the anisotropic equations resolution. The study of orthotropic media are described by the absence of this term in the governing equations, countless studies have been considered in this case [1-2], among others, Haji-Sheikh et al [1], who have used the separation of variables method to find a solution of the transfer heat equation. On the contrary, where the medium is described anisotropic by the presence of this term, the researches are rare and as far as we know they deal only with the case of two-dimensional steady state [3-6], and three-dimensional case [7]. Based on the accomplished studies and scientific researches, it's known that there is no analytical solution to the thermal heat equations in

anisotropic media where the configuration is finite. On account of this, a numerical method has to be employed.

2. Mathematical Model

The configuration studied shown in Fig.1 is two anisotropic media of cylindrical geometry of length L and two rays b_1 and b_2 . The ends of the media are maintained at constant temperatures, whereas a radial flux is applied to the lateral surface of medium B.

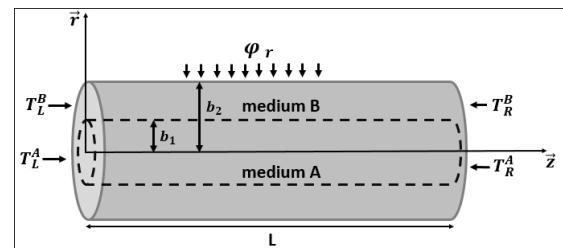


Fig. 1 Three-dimensional representation of the anisotropic cylinders

By taking into account the symmetry of the cylinders, the configuration studied transforms to be two-dimensional.

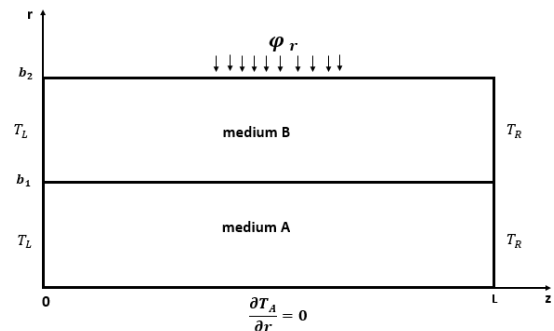


Fig. 2 Two-dimensional representation of the anisotropic cylinders.

The heat diffusion equations for the two anisotropic mediums in the plane (r, z) are expressed by:

$$K_r^i \frac{\partial^2 T^i}{\partial r^2} + 2K_{rz}^i \frac{\partial^2 T^i}{\partial r \partial z} + K_z^i \frac{\partial^2 T^i}{\partial z^2} + K_r^i r \frac{\partial T^i}{\partial r} + \frac{K_{rz}^i}{r} \frac{\partial T^i}{\partial z} = \rho_i C_p^i \frac{\partial T^i}{\partial t} \quad (1)$$

$i = A$ for medium A, $i = B$ for medium B

The radial and axial heat fluxes for the two mediums by the application of Fourier's law are :

$$\begin{cases} \varphi_r^i = K_r^i \frac{\partial T^i}{\partial r} + K_{rz}^i \frac{\partial T^i}{\partial z} \\ \varphi_z^i = -K_{rz}^i \frac{\partial T^i}{\partial r} - K_z^i \frac{\partial T^i}{\partial z} \end{cases} \quad (2)$$

With (K_r^i, K_z^i) are the main thermal conductivities and K_{rz}^i the cross thermal conductivities for the two media.

The above equations are allied to the following boundary and interface conditions :

$$\left. \frac{\partial T^A}{\partial r} \right|_{r=0} = 0 \quad \varphi_r = K_r^B \frac{\partial T^B}{\partial r} \Big|_{r=b_2} + K_{rz}^B \frac{\partial T^B}{\partial z} \Big|_{r=b_2}$$

$$\begin{cases} T^A(b_1, z) = T^B(b_1, z) \\ \varphi_r^A(b_1, z) = \varphi_r^B(b_1, z) \end{cases} \quad \begin{cases} T^A(r, 0) = T^B(r, 0) = T_L \\ T^A(r, L) = T^B(r, L) = T_R \end{cases}$$

Resolving the above equations with the existence of the cross thermal conductivities terms is difficult. Therefore, a linear coordinate transformation is utilized so as to transform the anisotropic equations to an equivalent isotropic equations form.

$$\begin{pmatrix} R \\ Z \end{pmatrix} = \begin{pmatrix} a & 0 \\ c & 1 \end{pmatrix} \begin{pmatrix} r \\ z \end{pmatrix} \quad (4)$$

Where :

$$a = \sqrt{\left(\frac{K_r^B}{K_r^A}\right)^2 - \left(\frac{K_{rz}^B}{K_r^A}\right)^2} = \sqrt{\left(\frac{K_r^A}{K_r^B}\right)^2 - \left(\frac{K_{rz}^A}{K_r^B}\right)^2} \quad \text{and} \quad c = -\frac{K_{rz}^B}{K_r^B} = -\frac{K_{rz}^A}{K_r^A}$$

By incorporating the non-dimensional parameters:

$$Z^* = \frac{Z}{b_2} \quad R^* = \frac{R}{b_2} \quad \theta = \frac{T - T_R}{T_R - T_L} \quad t^* = t \frac{\alpha_{rr}^B}{b_2^2} \quad (6)$$

The non-dimensional forms of above equations became :

$$\frac{1}{R^*} \frac{\partial \theta^i}{\partial R^*} + \frac{\partial^2 \theta^i}{\partial R^{*2}} + \frac{\partial^2 \theta^i}{\partial Z^{*2}} = (m) \frac{1}{a^2} \frac{\partial \theta^i}{\partial t^*} \quad (7)$$

$$m = 1 \quad \text{for medium A, } m = \frac{\alpha_r^A}{\alpha_r^B} \quad \text{for medium B}$$

$$\begin{cases} \varphi_r^i = a \left(\frac{L}{b_2}\right) \left(\frac{K_r^A}{K_z^B}\right) \frac{\partial \theta^i}{\partial R^*} \\ \varphi_z^i = -a \left(\frac{K_{rz}^A}{K_z^B}\right) \left(\frac{L}{b_2}\right) \frac{\partial \theta^i}{\partial R^*} - a^2 \left(\frac{K_r^A}{K_z^B}\right) \left(\frac{L}{b_2}\right) \frac{\partial \theta^i}{\partial Z^*} \end{cases} \quad (8)$$

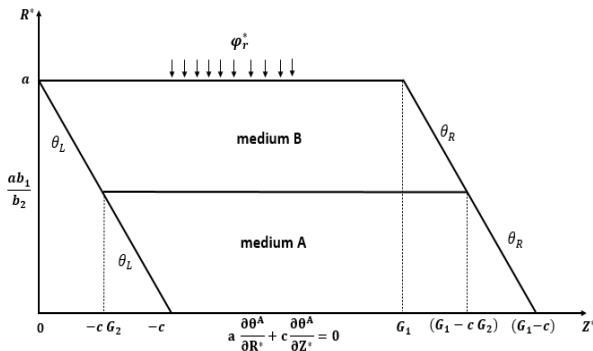


Fig. 3 The studied geometry in the virtual space. The Fig.3 represents the two media in the dimensionless virtual space (R^*, Z^*) .

The boundry and interface conditions are given by :

$$a \frac{\partial \theta^A}{\partial R^*} + c \frac{\partial \theta^A}{\partial Z^*} = 0 \quad -c \leq Z^* \leq G_1 - c \quad R^* = 0$$

$$\varphi_r^* = a \left(\frac{L}{b_2}\right) \left(\frac{K_r^B}{K_z^B}\right) \frac{\partial \theta^B}{\partial R^*} \quad 0 \leq Z^* \leq G_1 \quad R^* = a$$

The continuity of the temperatures and the fluxes at the interface :

$$\theta^A(R^*, Z^*) = \theta^B(R^*, Z^*) \quad , \quad \frac{\partial \theta^A}{\partial R^*} = \left(\frac{K_r^B}{K_r^A}\right) \frac{\partial \theta^B}{\partial R^*}$$

For both oblique straight lines left and right of the two mediums respectively

$$\theta^i(R^*, Z^*) = \theta_L \quad , \quad \theta^i(R^*, Z^*) = \theta_R$$

Solving the above equations system provides access to the temperature profile in the real space inside the two anisotropic mediums using the reverse transformation.

3. Numerical solution and Validation

With the aim of solving the thermal heat equations of this problem, the ADI finite difference method has been utilized. This method is unconditionally stable and leads into two tri-diagonal matrixes that are solved by the THOMAS algorithm.

In order to make certain of the results presented in this paper, a comparison between the numerical results and the analytical solution for the case of a steady-state orthotropic mediums [8] needs to be done.

$$\theta(Z^*, R^*) = \left(1 - \frac{Z^*}{G_1}\right) m + \frac{4\varphi_r^* \delta G_1}{\pi^2} \sum_{n=1,2,3}^{\infty} \frac{I_0\left(\frac{n\pi R^*}{\delta G_1}\right)}{n^2 I_1\left(\frac{n\pi}{\delta G_1}\right)} \sin\left(\frac{n\pi Z^*}{G_1}\right) \quad (9)$$

where

$$\lambda_n = \frac{n\pi}{\delta G_1} \quad , \quad \delta = \sqrt{\frac{K_r}{K_z}} \quad \text{and} \quad \begin{cases} m = 0 \quad \text{for } \theta_L^i = \theta_R^i \\ m = 1 \quad \text{for } \theta_L^i = 1, \theta_R^i = 0 \end{cases}$$

The confirmation of the digital code, was made by comparing the numerical results and the analytical solution for a low value of $K_{rz}/K_r = 0.05$ and for $K_z/K_r = 1, K_r^B/K_r^A = 1$ and for imposed radial flux equals 1. The imposed tempratures are $(\theta_R = 1, \theta_L = 0)$.

The figure 4 demonstrates the good agreement between the numerical and the analytical solutions.

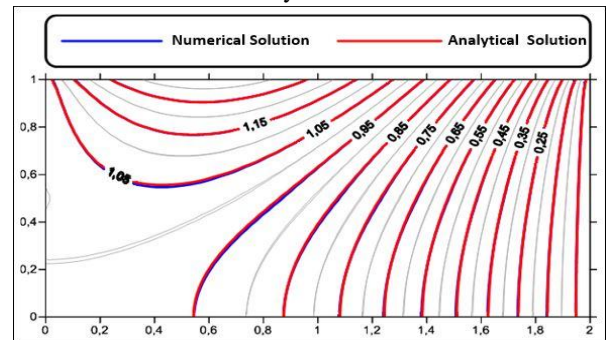


Fig. 4 Validation of the orthotropic mediums.

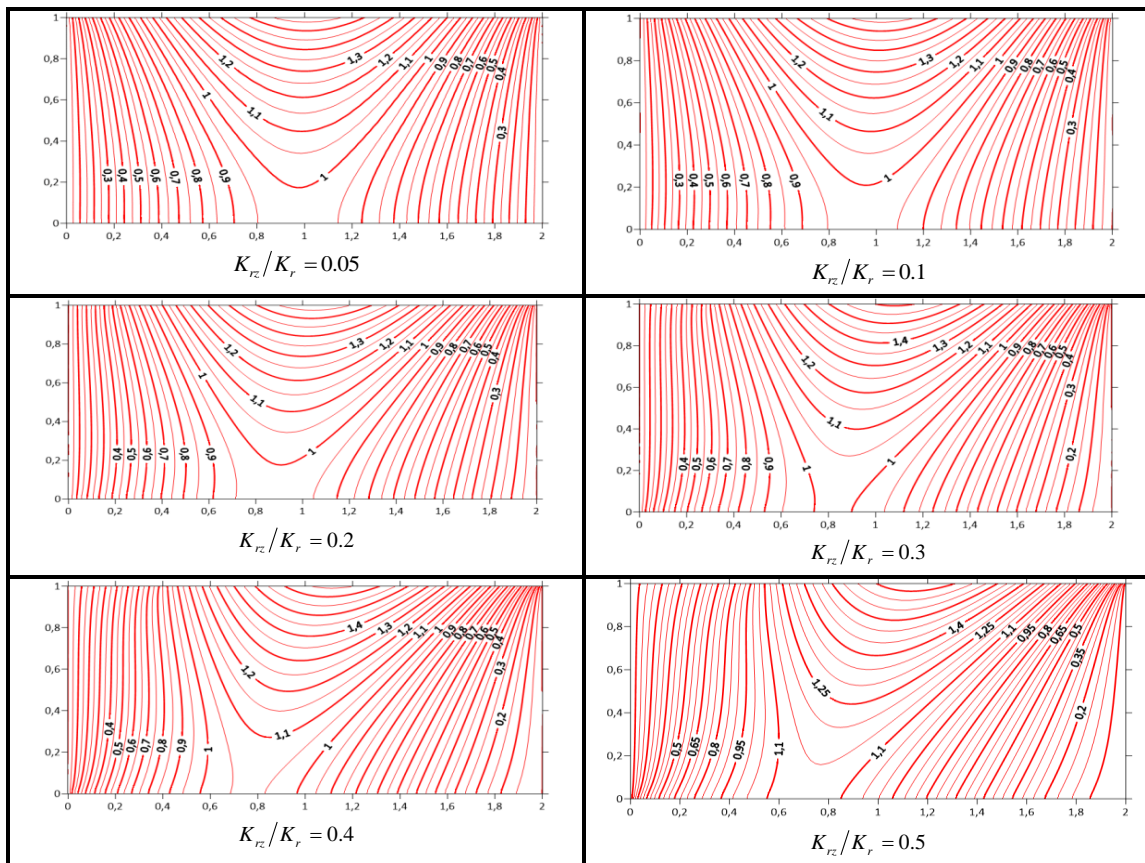


Fig.5 Steady state temperature profiles for $K_z/K_r = K_r^B/K_r^A = \varphi_r^* = 1$

4. Results and Discussion

The numerical code developed provides access to the steady state temperature profiles inside the two anisotropic media in the real space. In order to examine the effect of the anisotropy, that is to say K_{rz}/K_r , different temperature profiles have been presented for different values of K_{rz}/K_r and for the same values of K_z/K_r , K_r^B/K_r^A and for imposed radial flux equals 1.

Through the distribution of the temperatures presented in figure 5, it's quite clear that the Isotherms are affected by the increasing of the anisotropy K_{rz}/K_r . An increase of this ratio breach the symmetry presented for low values of K_{rz}/K_r and increases the thermal level of the isotherms.

5. Conclusion

The thermal behaviour of two anisotropic media in two-dimensional cylindrical geometry was investigated. The Analysis of this behaviour was examined by the impact of the anisotropy on the Isotherms. the numerical solution presented as temprature profiles demonstrate that the anisotropy has an great impact on the thermal level and the shape of the isotherms .

Références

- [1] Haji-Sheikh, A and Beck, JV and Agonafer, D. Steadystate heat conduction in multi-layer bodies. International Journal of Heat and Mass Transfer 46 n.13, p.2363–2379 2003
- [2] Delouei, A. A., Kayhani, M. H., and Norouzi, M. (2012). Exact analytical solution of unsteady axisymmetric conductive heat transfer in cylindrical orthotropic composite laminates. International journal of heat and mass transfer, 55(15-16), 4427-4436.
- [3] Miloevi, N. D., and Raynaud, M. (2004). Analytical solutionof transient heat conduction in a two-layer anisotropic cylindrical slab excited superficially by a short laser pulse. International journal of heat and mass transfer,47(8-9), 1627-1641.
- [4] Norouzi, M., and Rahmani, H. (2015). On exact solutions for anisotropic heat conduction in composite conical shells. International Journal of Thermal Sciences,94, 110-125.
- [5] TR, T., and Akoz, A. Y. (1975). Stationary Temperature and stress Fields in an Anisotropic Elastic Slab, ASME. Journal of Applied Mechanics, 42, 647-650.
- [6] Ma, C. C., and Chang, S. W. (2004). Analytical exact solutions of heat conduction problems for anisotropic multi-layered media. International journal of heat and mass transfer, 47(8-9), 1643-1655.
- [7] Mulholland, G. P., and Gupta, B. P. (1977). Heat transfer in a three-dimensional anisotropic solid of arbitrary shape. ASME Transactions Journal of Heat Transfer, 99, 135-137.
- [8] Ozisik, M. N., 1993, Heat Conduction, 2nd ed., Wiley, New York.

Effect of the Lewis Number on Natural Double Diffusive Convection in Shallow Rectangular Cavities Filled with Nanofluid

B. El hadoui¹, M. Kaddiri², M. Lamsaadi³, H. El harfi⁴

1. Faculty of Sciences and Technologies, Beni-Mellal. bilal.elhadoui@usms.ac.ma
2. Faculty of Sciences and Technologies, Beni-Mellal. mouradkaddiri@usms.ma
3. Poly-disciplinary Faculty, Beni-Mellal. lamsaadima@yahoo.fr
4. Poly-disciplinary Faculty, Beni-Mellal. elharfi.hassan@yahoo.fr

Abstract

In this study, the effect of the Lewis number on natural double diffusive convection in a rectangular cavity horizontally disposed, filled with nanofluid, heated and salted by uniform heat and mass fluxes on its vertical walls while the horizontal ones are assumed thermally adiabatic and solutally impermeable, is studied analytically (parallel flow approximation) and numerically (finite difference method). The results obtained by examining the interest of using nanofluids in the considered configuration were against all expectations, that they led to a degradation of the rates of heat and mass transfers.

Keywords: Heat and mass transfers, Nanofluids, Natural double diffusive convection, Rectangular cavity.

1. Introduction

Among the modes of heat and mass transfers that has grown gradually, is natural double diffusive convection, whose movement of the fluid is induced by variations of its density due to differences of temperature and concentration. This mode occurs in many fields and industrial applications.

An innovative way to improve the thermal conductivity of the conventional fluids, is to suspend nanoscale solid particles in them, resulting in a new type of heat transfer fluids called nanofluid. A considerable number of studies reported enhancement of heat transfer due to the addition of nanoparticles in conventional fluids in various applications. However, the interest of using nanofluids in some configurations may be against the odds, that they lead to a degradation of the rate of heat transfer. This phenomenon depends mainly on the geometry, the governing parameters and the thermophysical properties.

The investigations of this study focus on the effects of the Lewis number and the volume fraction of nanoparticles on the dynamic, thermal and solutal fields.

2. Geometry and Governing Equations

Fig. (1), depicts a rectangular horizontal cavity of length L and height H , filled with Cu-water nanofluid. The vertical

walls are subjected to uniform heat and mass fluxes and the horizontal ones are kept adiabatic and impermeable.

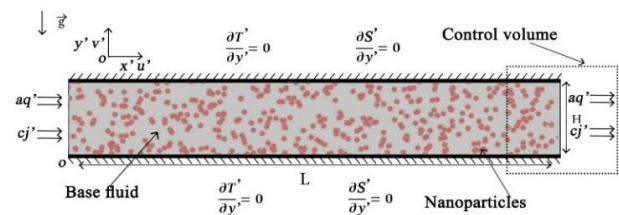


FIGURE 1. Sketch of the geometry and coordinates system.

The nanofluid is Newtonian and incompressible, the heat production due to viscous friction as well as the Soret and Dufour effects are not taken into account, and it is assumed that both the base fluid and nanoparticles are in thermal equilibrium and have the same velocity.

Under the above assumptions and adopting the Boussinesq approximation. The dimensionless governing equations written in terms of vorticity, Ω , temperature, T , concentration, S and stream function, Ψ , are as follows:

$$\frac{\partial \Omega}{\partial t} + \frac{\partial(u\Omega)}{\partial x} + \frac{\partial(v\Omega)}{\partial y} = \frac{\bar{\nu}}{\bar{\alpha}} \text{Pr} \nabla^2 \Omega + \frac{\bar{\beta}}{\bar{k}\bar{\alpha}^2} \text{Ra}_T \text{Pr} \left(\frac{\partial T}{\partial x} + N \frac{\partial S}{\partial x} \right) \quad (1)$$

$$\frac{\partial T}{\partial t} + \frac{\partial(uT)}{\partial x} + \frac{\partial(vT)}{\partial y} = \nabla^2 T \quad (2)$$

$$\frac{\partial S}{\partial t} + \frac{\partial(uS)}{\partial x} + \frac{\partial(vS)}{\partial y} = \frac{1}{\bar{\alpha} \text{Le}} \nabla^2 S \quad (3)$$

$$\nabla^2 \Psi = -\Omega \quad (4)$$

The dimensionless boundary conditions are:

$$u = v = \Psi = \frac{\partial T}{\partial x} + 1 = \frac{\partial S}{\partial x} + 1 = 0 ; x = 0, A \quad (5)$$

$$u = v = \Psi = \frac{\partial T}{\partial y} = \frac{\partial S}{\partial y} = 0 ; y = 0, 1 \quad (6)$$

Where $A = L/H$ and $\bar{f} = \frac{f_{\text{nf}}}{f_f}$ for α, β, k and ν .

In these equations, four dimensionless parameters appeared, namely: the Prandtl, Pr , the thermal Rayleigh, Ra_T , the Lewis, Le , numbers and the buoyancy ratio, N , whose expressions are:

$$\text{Ra}_T = \frac{g\beta_T H^4 q'}{v_f \alpha_f k_f} ; \text{Pr} = \frac{v_f}{\alpha_f} ; \text{Le} = \frac{\alpha_f}{D_{nf}} ; \quad (7)$$

$$N = \frac{\beta_{\text{Snf}} j' k_{\text{nf}}}{\beta_{\text{Tnf}} q' D_{\text{nf}}}$$

The thermophysical properties used for copper nanoparticles and water are presented in Table 1 and they are taken at atmospheric temperature and pressure.

TABLE 1. Thermophysical properties of H₂O and Cu.

	$c_p \left(\frac{\text{J}}{\text{kgK}} \right)$	$\rho \left(\frac{\text{kg}}{\text{m}^3} \right)$	$k \left(\frac{\text{W}}{\text{mK}} \right)$	$\beta \left(\frac{1}{\text{K}} \right) \cdot 10^{-5}$
H ₂ O	4179	997.1	0.613	21
Cu	385	8933	401	1.167

The effective thermophysical properties of the nanofluid are calculated as in [1].

3. Numerical & Analytical Approaches

The discretization scheme used to solve the governing equations is the second order central finite difference scheme with the aid of ADI method.

The dimensionless quantities measuring respectively the average heat and mass transfers in the central part of the cavity, are Nusselt and Sherwood numbers, defined as in [2]:

$$\overline{Nu} = - \int_0^1 \frac{\bar{k}}{\left(\frac{\delta T}{\delta x} \right)_{x=\frac{A}{2}}} dy ; \overline{Sh} = - \int_0^1 \frac{1}{\left(\frac{\delta S}{\delta x} \right)_{x=\frac{A}{2}}} dy \quad (8)$$

The present numerical approach was validated by comparing our results for different parameters with the previous studies. It is evident that the results of the present code are in excellent agreement with the works published as it can be clearly reflected in Table 2.

TABLE 2. Validation of our numerical code with previous studies.

	$ \Psi_{\text{max}} $	\overline{Nu}	
Davis [3]	5.071	2.243	$\text{Pr} = 0.71, \text{Ra}_T = 10^4$
Our code	5.070	2.242	
Ghasemi [1]	11.801	4.968	$\text{Ra}_T = 10^5, \varphi = 6\%, \text{Pr} = 6.2$
Our code	11.776	4.951	

In order to find a compromise between the computational time and the solution accuracy, preliminary numerical tests were carried out for $A = 12$ (the smallest value of the aspect ratio A that satisfies the infinitely long cavity approximation). A uniform grid size of 181×61 has been chosen and estimated to model accurately the fluid flow, temperature and concentration distributions inside the cavity. The numerical results concerning streamlines, isotherms and isoconcentrations are presented in Fig. (2) for $Le = 0.01$ and $Le = 10$ and for various values of φ of Cu. As can be seen, throughout the cavity except the ends, the flow remains parallel to the horizontal walls and the temperature and concentration fields remain linearly stratified in the horizontal direction. Therefore, these observations had been the origin of the simplifications made in the next lines, to transform the differential equations into ordinary equations.

Based on these observations, the following simplifications are used in the center of the cavity:

$$u(x,y) = u(y) ; v(x,y) = 0 ; \Psi(x,y) = \Psi(y) ;$$

$$T(x,y) = \theta_T(y) + C_T \left(x - \frac{A}{2} \right) ; \quad (9)$$

$$S(x,y) = \theta_S(y) + C_S \left(x - \frac{A}{2} \right)$$

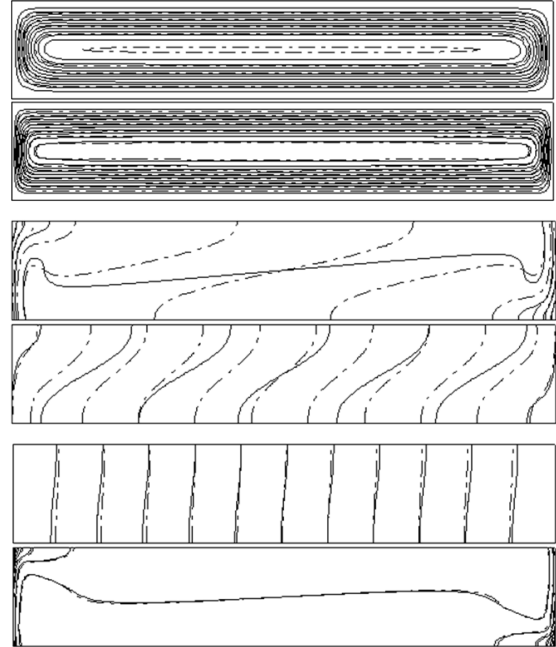


FIGURE 2. Streamlines (top), isotherms (middle) and isoconcentrations (bottom) for $A = 12, \text{Pr} = 7, \text{Ra}_T = 10^4, N = 1$ and various values of Le (top: $Le = 0.01$, bottom: $Le = 10$) and φ (solid lines: $\varphi = 0$, dashed lines: $\varphi = 0.1$) when water-based Cu nanofluid is used.

Where $C_T = \partial T / \partial x$ and $C_S = \partial S / \partial x$ are unknown.

Using the above simplifications Eqs. (9), Eqs. (1) – (3) are simplified and their integration associated with the boundary conditions (5) and (6) leads to:

$$u(y) = 32\Psi_c(2y^3 - 3y^2 + y) \quad (10)$$

$$\Psi(y) = 32\Psi_c \left(\frac{y^4}{2} - y^3 + \frac{y^2}{2} \right) \quad (11)$$

$$\theta_T(y) = \frac{4\Psi_c C_T}{15} (12y^5 - 30y^4 + 20y^3 - 1) \quad (12)$$

$$\theta_S(y) = \frac{\bar{\alpha} Le C_S}{C_T} \theta_T(y) \quad (13)$$

Where Ψ_c is the current function in the center of the cavity and has the following expression:

$$\Psi_c = \Psi \left(\frac{1}{2} \right) = \frac{\bar{\beta}}{\bar{k} \bar{\alpha} \bar{v}} (C_T + N C_S) \frac{\text{Ra}_T}{384} \quad (14)$$

The Nusselt and Sherwood numbers take the following simple forms:

$$\overline{Nu} = - \frac{\bar{k}}{C_T} ; \overline{Sh} = - \frac{1}{C_S} \quad (15)$$

To determine the constants C_T and C_S , a procedure, originally due to Bejan (1983), is used and it leads to:

$$C_T = - \frac{1}{\frac{128}{315} \Psi_c^2 + 1} ; C_S = - \frac{1}{\frac{128}{315} (\bar{\alpha} Le \Psi_c)^2 + 1} \quad (16)$$

By substituting the new expressions of C_T and C_S in Eq. (14), we obtain the following transcendental equation:

$$\left(\frac{128}{315}\bar{\alpha}Le\right)^2 \Psi_c^5 + \frac{128}{315}\Psi_c^3(\bar{\alpha}^2Le^2 + 1) + \Psi_c = -\frac{\bar{\beta}}{k\bar{\alpha}v} \frac{Ra_T}{384} \left(\frac{128}{315}(N + (\bar{\alpha}Le)^2)\Psi_c^2 + 1 + N\right) \quad (17)$$

4. Results & Discussion

The case of imposing uniform heat and mass fluxes, as boundary conditions, and with bigger aspect ratio, A , leads to flow characteristics independent on this latter. Therefore, numerical simulations have been performed to determine the smallest value of A in order to obtain numerical results reasonably close to the analytical ones and we found it to be $A = 12$.

The information drawn from Fig. (2) informs about the effects of the volume fraction of nanoparticles and the Lewis number on the flow, temperature and concentration fields inside the cavity. Throughout the cavity except the ends, the flow remains parallel to the horizontal walls, the temperature and concentration fields are stratified in the horizontal direction, and the unicellular character of the flow with a clockwise direction is always maintained. It is clear that the isotherms seem to be more sensitive to variations in φ than isoconcentrations, especially in the center. The thermal boundary layer gets thicker with increasing the nanoparticles volume fraction, so that the inclination of isotherms with respect to the vertical direction, becomes less inclined. Another effect of φ is that as it increases the intensity of the flow reduces, which means the convection mode weakens. Switching from $Le = 0.01$ to $Le = 10$ leads to the change of the solutal convection (thermal conduction) to the thermal one (solutal diffusion).

The influence of the Lewis number on the quantities $|\Psi_c|$, \bar{Nu} and \bar{Sh} is clarified in Fig. (3) for different values of φ .

On the one hand, the evolution of $|\Psi_c|$ and \bar{Nu} as functions of Le is characterized by two plateaus, upper and lower, whose intensities depend on φ . The upper plateau is associated with a dominant thermal convection compared to the solutal one, given the low values of Le , and the lower plateau implies a dominance of the solutal convection compared to the thermal one, given the high values of Le . For $0.01 < Le < 10$ both $|\Psi_c|$ and \bar{Nu} decrease steeply with the Lewis number. In addition, for large values of Le , the effect of φ becomes less significant compared to the case of small Le .

On the other hand, the variations of \bar{Sh} are characterized by one plateau $\bar{Sh} \approx 1$ for low values of Le , and by an increase which first occurs with a slight slope, thereafter it increases linearly with the increase of Le . From this figure it is clear that the nanoparticle volume fraction has no effect on \bar{Sh} except for the interval $0.01 < Le < 10$ where water has the best mass transfer rate compared to nanofluid.

Conclusion

Natural double diffusive convection in a horizontal shallow enclosure confining nanofluids whose vertical walls are subjected to constant heat and mass fluxes, was studied numerically and analytically to qualitatively predict the characteristics of the flow and heat and mass transfers, as function of the Lewis number and the nanoparticles volume fraction. Under this study we obtained the following results: First of all, it should be noted that the analytical results are in good agreement with the numerical ones.

The use of nanofluid in the configuration in this study has a negative effect on natural double diffusive convection.

The Nusselt number and the intensity of the flow are insensitive to the variation of the Lewis number outside the interval $10^{-2} < Le < 10$, where in this interval they decrease with the increase in this parameter. On the other hand, the Sherwood number increases with Le for $Le > 10^{-2}$.

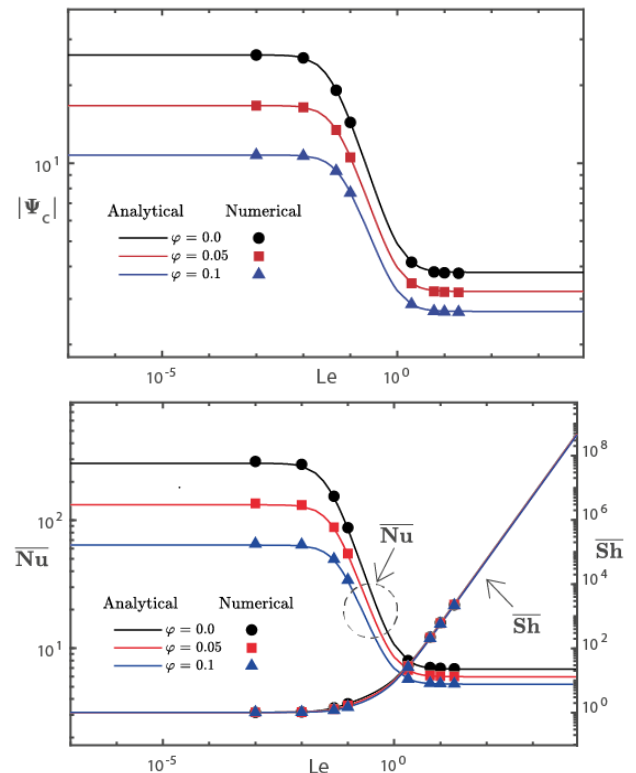


FIGURE 3. Evolutions of the quantities $|\Psi_c|$, \bar{Nu} and \bar{Sh} with various values of Le and φ of water-based Cu nanofluid.

References

1. Ghasemi, B., Aminossadati, S.M., Raisi, A., "Magnetic field effect on natural convection in a nanofluid-filled square enclosure," International Journal of Thermal Sciences. 50, 1748-1756 (2011).
2. Lamsaadi, M., Naimi, M., and Hasnaoui, M., "Natural convection heat transfer in shallow horizontal rectangular enclosures uniformly heated from the side and filled with non-Newtonian power law fluids," Energy Conversion and Management. 47, 2535-2551 (2006).
3. Davis, G. D. V., and Jones, I. P., "Natural Convection in a Square Cavity: A Comparison Exercise," Int. J. Numer. Methods Fluids. 3, 227-248 (1983).

Effet de la variation des matériaux de construction sur le comportement thermique du système de chauffage par le sol.

Effect of the variation of building materials on the thermal behavior of the floor heating system.

A. Charraou¹, S. Oubenmoh¹, R. Saadani¹, A. Mourid¹, M. Rahmoune¹, M. El Alami²

¹ Laboratoire des Etudes des Matériaux Avancés et Applications (LEM2A), FS - EST Meknès, Université Moulay Ismail, BP 11201, Avenue Zitoune, Meknès, Maroc

² Laboratoire de Physique des Matériaux, Microélectronique, Automatique et Thermique, LPMMAT, Faculté des Sciences Aïn Chock de Casablanca, Maroc

Corresponding author: afaf.charraou@gmail.com

Abstract

Radiant floor heating systems have become popular because of their advantages over conventional heating systems especially in residential spaces, commercial and industrial. They are characterized by low energy consumption, better thermal comfort and can generate lower temperature gradients compared to other convection heating systems. In this work, we are interested in predicting the thermal behavior of different types of floor heating systems such as homogeneous floors, single-layer and multilayer under the variation of floor covering and the location of the acoustic insulation using a numerical simulator for three-dimensional calculation in transient state.

Résumé

Les systèmes de chauffage par le sol sont devenus populaires en raison de leurs avantages par rapport aux systèmes de chauffage conventionnels notamment dans les espaces résidentiels, commerciaux et industriels. Ils sont caractérisés par une faible consommation d'énergie, un meilleur confort thermique et permettent de générer des gradients de température plus faibles en comparaison avec les autres systèmes de chauffage par convection. Dans ce travail, on s'intéresse à prédire le comportement thermique de différents types de plancher chauffant tel que le plancher homogène, monocouche et multicouche sous la variation de revêtement de sol et l'emplacement de l'isolation acoustique, en utilisant un simulateur numérique pour le calcul tridimensionnel en régime transitoire.

Mots clés : plancher chauffant, matériaux de revêtement du sol, la température moyenne, simulation numérique.

Keywords: Floor heating, Floor covering materials, average temperature, numerical simulation.

1. Introduction

Au cours des deux dernières décennies les systèmes de chauffage par plancher radiant sont devenus les plus fiables sur le plan énergétique et les plus confortables [1]. L'utilisation de ce type de système est considérée comme un moyen efficace afin d'assurer le confort thermique des occupants dans les bâtiments à faible consommation d'énergie et de réduire les fluctuations de la température, contrairement aux systèmes de chauffage conventionnel.

Dans ce contexte, les systèmes de chauffage par le sol ont fait l'objet de diverses études visant à déterminer l'effet de la variation de plusieurs paramètres sur la température superficielle du plancher. Oubenmoh et al. [2] ont adopté un modèle de simulation numérique basé sur la méthode des volumes finis pour les calculs 2D afin d'étudier l'influence de certains paramètres sur le comportement thermique du système de chauffage par le sol. En outre, Sattari et al. [3] ont montré que les paramètres de conception ont des effets plus importants sur les performances thermiques du système de chauffage par le sol. L'objectif principal de cette étude est d'examiner l'effet de la variation du matériau de revêtement du sol, et l'emplacement de l'isolation acoustique sur la température de la surface du sol qui est considérée parmi l'un des paramètres qui influe le confort thermique des occupants.

2. Modèle de plancher radiant

L'étude a été faite sur un appartement de grand volume situé en premier étage et soumis aux conditions climatiques de la ville de Casablanca. Il est constitué principalement de trois chambres de construction différente, de même taille 2m (largeur) × 2,43m (longueur), construite l'une à côté de l'autre dans lesquelles est noyé le circuit du chauffage, afin d'assurer la circulation d'eau chaude provenant d'une chaudière à une vitesse de 0,75m/s. Dans le cadre de cette étude, nous proposons trois types différents de construction du plancher qui sont respectivement le sol homogène, le sol monocouche et multicouche, comme illustré dans la figure 1.

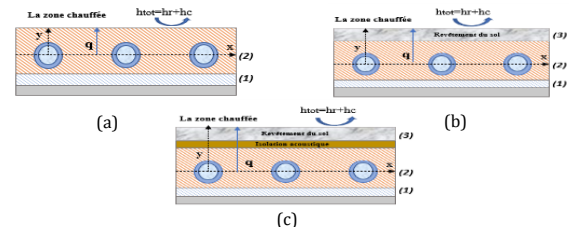


Figure 1 : Vue en coupe du sol homogène(a), monocouche (b) et multicouche(c)

Le sol homogène est composé seulement d'une couche de chape à base du béton en raison de sa bonne conductivité thermique et sa capacité d'être combiné avec un système de chauffage par le sol, tandis que le sol monocouche est constitué d'une seule couche au-dessus de la couche de chape qui peut être du contreplaqué, marbre artificiel ou bien

du PVC. En outre, le sol multicouche se compose une couche d'isolation acoustique de 5 mm d'épaisseur placée entre la couche de chape et le revêtement du sol pour réduire la transmission des bruits d'un local à un autre. L'emplacement de cette couche d'isolation est étudié également afin de sélectionner sa position optimale au niveau du plancher.

Le système de chauffage modélisé se compose principalement de trois couches principales : la couche d'isolation (1), la couche de stockage thermique (2) et la couche de revêtement du sol (3). Les tubes du circuit de chauffage sont enrobés dans la chape de béton formant une chaîne tubulaire de 16 mm de diamètre extérieure. Ce plancher est isolé thermiquement par une couche de polystyrène de 20 mm d'épaisseur ce qui permet d'annuler les échanges de chaleur vers le bas. Le tableau suivant récapitule la composition du plancher bas des cas étudiés et ses propriétés thermo-physiques [4].

Composants	Types	Epaisseur (mm)	Conductivité thermique (W/m.k)	Capacité thermique (J/Kg. K)
Revêtement du sol	Contre-plaqué	10	0.186	1298
	Marbre artificiel	10	1.57	858
	PVC	10	0.79	837
Isolation	Isolation acoustique	5	0.05	5
Tuyau	PPR	4	0.41	1210
La chape	Béton	45	1.65	1000

Tableau 1 : propriétés thermo physiques des matériaux utilisés

3. Formulation mathématique du problème

Tout d'abord, l'échange de chaleur entre l'eau et la surface interne des tuyaux se fait par convection, puis la conduction thermique se produit entre la paroi de tuyaux et les couches de la dalle, et enfin la surface du sol échangera de la chaleur avec l'environnement intérieur par convection et rayonnement. Le flux thermique total q_{tot} qui traverse la surface du plancher est la somme du flux de chaleur convective q_c et du flux de chaleur radiante q_r [5] :

$$q_{tot} = q_c + q_r \quad (1)$$

Le flux de chaleur convective (q_c) indique l'échange de chaleur convective entre la surface du plancher et l'air intérieure :

$$q_c = h_c (t_a - t_s) \quad (2)$$

Dans le cas du chauffage, le coefficient de transfert de chaleur par convection naturelle h_c pour le plancher chauffant peut être calculé par la corrélation suivante [6] :

$$h_c = \frac{2.175}{H^{0.076}} (t_s - t_a)^{0.308} \quad (3)$$

Avec H est le diamètre hydraulique de la surface chauffée, t_s et t_a sont respectivement la température du sol et l'air ambiant intérieure. L'échange de la chaleur rayonnante avec la surface intérieure est calculé par l'équation suivante :

$$q_r = h_r (AUST - t_s) \quad (4)$$

Avec h_r le coefficient de transfert de chaleur rayonnant, $AUST$ est la température moyenne de la surface non chauffée. D'autre part, les équations régissant le comportement thermique et les interactions réelles entre le système intégré au sol et la dalle chauffante sont étudiées par le biais d'un modèle numérique tridimensionnel développé via le logiciel

COMSOL Multiphysiques sous des conditions d'état instationnaire. La résolution de ce système d'équations nécessite d'imposer des conditions initiales et aux limites, qui sont les suivants :

- La température initiale du plancher est supposée être $T_0 = 10^\circ\text{C}$

- Les parois latérales du plancher sont supposées adiabatiques, cette condition aux limites signifie qu'il n'y a pas de flux de chaleur à travers la frontière :

$$-n \cdot (-k\nabla T) = 0 \quad (5)$$

- La condition de type flux sortant est imposée à la sortie de la tuyauterie.

4. Résultats et discussions

Une série des simulations ont été effectués afin de déterminer l'effet de la variation de la construction du plancher sur le comportement thermique du système étudié en régime transitoire. La température de l'eau circulant dans le serpentin incorporé sous la surface du plancher a été variée de 30 à 50 °C avec un intervalle de 5°C.

4.1. Effet de revêtement du sol

Les résultats montrent l'effet de la variation de la composition structurale de la couverture du sol sur la distribution de la température superficielle au bout de six heures de fonctionnement du système de chauffage. L'objectif est de déterminer parmi les revêtements du sol sélectionnés, le plus apte à transférer une grande quantité de chaleur vers la pièce à chauffer, et atteindre une température dans la marge de confort thermique.

Il est clair d'après ces résultats que la température de la surface du sol dans le cas d'un plancher homogène à base du béton oscille entre 20,9 et 28,9 °C pour des températures d'alimentation entre 35 et 45 °C, ceci permet d'apporter un confort appréciable au niveau de la pièce et d'augmenter le temps de réponse du système. Donc, les températures d'alimentation les plus élevées entraînent une augmentation plus rapide de la température au niveau du sol et une diminution du temps de chauffage.

Lorsque la chape de béton est recouverte par les revêtements du sol, la température moyenne de la surface du plancher varie de 16 à 22°C pour le contreplaqué, de 19 à 28,1°C pour le marbre artificiel et de 18,5 à 27,1°C pour le PVC pour une température d'alimentation allant de 30 à 50 °C comme illustre le tableau 2. On peut constater que le revêtement en contreplaqué a montré la plus faible fluctuation de la température superficielle en raison de sa faible conductivité thermique par rapport aux autres types de revêtement du sol. En revanche, le revêtement en marbre artificiel a montré une fluctuation dans l'intervalle de la température de confort grâce à sa conductivité thermique plus élevée que celle du contreplaqué et du PVC, de plus, ce type de revêtement est plus adapté avec l'architecture du bâtiment en raison de ces qualités esthétiques.

Revêtement du sol	30°C	35°C	40°C	45°C	50°C
Sans revêtement	20,9	23,5	26,2	28,9	31,5
Contreplaqué	16,0	17,5	19,0	20,5	22,0
Marbre artificiel	19,0	21,4	23,6	25,8	28,1
PVC	18,5	20,7	22,8	24,9	27,1

Tableau 2 : la température moyenne de la surface du sol (°C) après 6h de fonctionnement pour le cas du plancher homogène et monocouche.

La figure 2 montre la répartition de la température du sol du système avec contreplaqué, marbre et PVC lorsque la température de l'eau d'alimentation est de 40 °C. De toute évidence, la distribution de température sur le système avec une couche homogène est plus uniforme et la température moyenne est plus élevée que celle avec plancher multicouche. En résumé, un plancher avec un matériau épaisse peut réduire la température de l'eau d'alimentation et uniformiser la distribution de la température de surface du sol, ce qui peut non seulement économiser l'énergie de chauffage, intégrer la source de chaleur à basse température, mais également éliminer le défaut de distribution inégale de la température de surface. Cependant, selon les recherches de Cho [7], les matériaux de finition à haute résistance thermique conservent la chaleur plus longtemps, et diminuent les fluctuations de température.

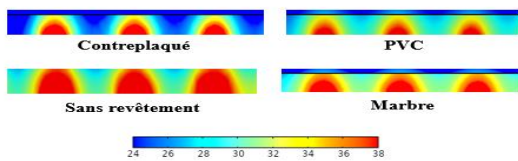


Figure 2 : Distributions de la température d'une coupe horizontale du sol.

4.2. Impact de la position de l'isolation acoustique

L'ajout d'une isolation phonique dans la structure du plancher chauffant a un véritable avantage pour l'habitation, dans la mesure où elle a un grand rôle à jouer dans l'isolation phonique et thermique. Dans cette partie, on s'intéresse à étudier l'impact de la position d'isolation acoustique sur la distribution température superficielle pour les différents cas étudiés. La figure 3 montre les différentes positions d'isolation au niveau du sol multicouche à base de marbre artificiel pour une température d'alimentation de 50°C.

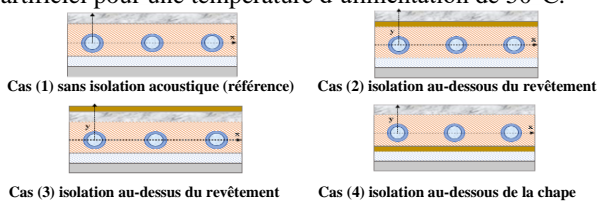


Figure 3 : position d'isolation acoustique au niveau du plancher multicouche

Il est clair d'après les résultats obtenus par simulation numérique dans la figure 4, que l'emplacement de l'isolation acoustique au-dessous du circuit de chauffage pour le cas (2) et (3) entraîne une baisse notable de la température superficielle par rapport au cas de référence sans isolation phonique, car elle permet d'empêcher le transfert de chaleur vers la surface du plancher grâce à sa faible conductivité

thermique. En outre, concernant le cas (4) où l'isolation acoustique est placée au-dessous du circuit de chauffage, la température moyenne au niveau du sol atteint une valeur de 27,7 °C en régime permanent, qui est située dans la plage de la température de confort, cela nous permet de conclure que l'emplacement et le type d'isolation permet d'atténuer les bruits passant d'un étage à l'autre et d'éliminer les déperditions de chaleur vers le bas du plancher.

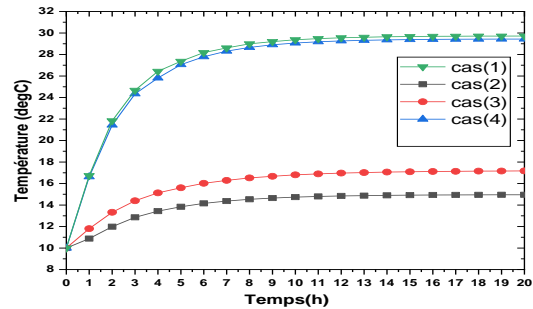


Figure 4 : L'effet de la variation de l'emplacement d'isolation acoustique sur la température du sol.

Conclusion

Le but de ce travail est d'étudier le comportement thermique de différents types de construction du plancher en faisant varier le matériau de revêtement du sol et l'emplacement de l'isolation acoustique. L'analyse des résultats obtenus montre que l'insertion du marbre artificiel permet d'obtenir des températures oscillant autour de la température de confort thermique par rapport aux autres types de revêtement du sol. De plus, en comparant l'effet de la variation de l'emplacement d'isolation acoustique sur le comportement thermique du plancher multicouche, nous avons constaté que l'insertion d'une couche d'isolation au-dessous du circuit de chauffage, offre à la fois une isolation thermique et acoustique.

Références

- [1] V. Golkarfard, et al., *Energy Consumption of Floor Heating System in Building*, Proceedings of the Second International Conference on Heating, Ventilation and Air-Conditioning June 1-3, 2010, Olympic Hotel, Tehran, Iran ICHVAC2-1098.
- [2] S. Oubenmoh, et al., *Some particular design considerations for optimum utilization of under floor heating systems*. Case Studies in Thermal Engineering 12 (2018) 423–432.
- [3] Sattari, B. Farhanieh, *A parametric study on radiant floor heating system performance*, Renewable Energy 31 (2006) 1617–1626
- [4] Gook-Sup Song, *Buttock responses to contact with finishing materials over the ONDOL floor heating system in Korea*, Energy and Buildings 37 (2005) 65–75
- [5] Lun Zhang, Xiao-Hua Liu, Yi Jiang, *Simplified calculation for cooling/heating capacity, surface temperature distribution of radiant floor*. Energy and Buildings 55 (2012) 397–404
- [6] Hazim B. Awbi, *Calculation of convective heat transfer coefficients of room surfaces for natural convection*, energy and Buildings 28 (1998) 219-227.
- [7] K.-H. Cho, *The thermal transfer characteristics of the floor panel for the ONDOL heating system*, Journal of Korean Society of Living Environment Systems 13 (3) (1996) 5–15 (in Korean).

NUMERICAL STUDY OF A WALL PROTECTED BY WATER FILM FROM AN INCIDENT RADIATIVE HEAT FLUX

M. AIT FARIA, H. HAMZA, K. ZNIBER, J. LAHJOMRI, A. OUBARRA

FSAC, simohammed.aitfaria@gmail.com

ABSTRACT

A Mathematical model has been performed to explore the effects of thermal protection by a liquid film of a structure exposed to radiant heat flux. A numerical modal of combined radiation and conduction heat transfer is developed. The film evaporation has been taken into account using the analytical solution of the mass diffusion equation associated with Nusselt falling film theory. To integrate the equations of the model including interfaces and boundary conditions, the Alternating Direction Implicit Method (ADI) was used. The effects of various factors on the absorption of radiation and the growing temperature of the wall were explored.

Keywords: Thermal protection, heat transfer, water film, radiant heat flux.

1. INTRODUCTION:

The flow of a vertical or inclined liquid film under gravity is used in a variety of industrial applications, including process engineering, safety of nuclear reactors (passive cooling), and fire protection engineering. Cooling of a hot surface, wetted-wall columns, desalination, evaporators, film reactors and thermal shielding of a surface from thermal radiation from fires are just a few examples. (Sherwood et al., 1975, Wu et al., 2007, Yih, 1986, Levand Strachan, 1989). In the present work we will focus on the two previous applications, in order to study the mechanisms thermal protection of a structure exposed to an incident radiant heat flux by a liquid water film.

Radiant energy transfer, and attenuation of radiation in Water, have been the topic of numerous investigations (Viskanta and Toor, 1972, Cengel and Özişik, 1984, Viskanta and Toor, 1978, Lazaridis et al., 1986, Dombrovsky et al., 2011, Snider and Viskanta, 1975, Snider and Viskanta, 1974). In the visible and near-infrared regions of the spectrum, water is transparent. Yet, it is practically opaque in the infrared section of the spectrum $\lambda c > 3$ (Viskanta and Toor, 1972, Snider and Viskanta, 1975). According to (Cengel and Özişik, 1984), the air-water interface is a poor reflector, reflecting just 6.6 percent of typical incident energy back to the atmosphere; nevertheless, the water-air interface reflects 47.7% of energy returning to the water from deeper portions of the pond.

In the literature, a considerable number of analytical and numerical investigations of heat and mass transfer in water films with radiant heating and/or subjected to solar irradiation, have been described (Song et al., 1999, Lazaridis et al., 1986, Qahtan et al., 2014, Yan and Soong, 1995, Yang and Yuen, 2018). The latent heat flow was 5 to 15 times superior than the sensible heat flux, according to B. Song's numerical calculations (Song et al., 1999), and evaporation dominated the heat and mass transfer process from the water film to the gas layer.

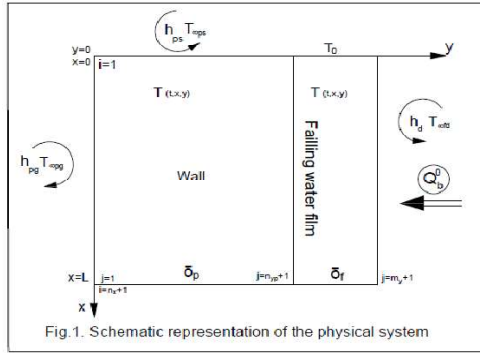
Some experimental studies have looked into the effectiveness of water films in radiation attenuation (Kuznetsov et al., 2018, Brissinger et al., 2014). Following (Brissinger et al., 2014), the energy eliminated by water absorption was anticipated for incoming radiation from sources up to 1473 K and was found to be in the range of 60–95% for films ranging in thickness from 100 μ m to 1 mm.

The majority of previous research has focused on heat transfer in liquid films in a variety of applications, such as heating surfaces, cooling surfaces, and solar collectors. However, apart from (Aubert et al., 2013, Fu et al, 1990), the thermal protection of a structure by a water film has not been the subject of many studies. (Fu et al, 1990) has studied the performance of a water film as a thermal protection in one-dimensional system, and it has shown that using water film to protect a structure from an incident heat flux has been confirmed.

In this paper heat transfer in a wall, insulated by a laminar water layer with a radiative heat flux exerted is investigated. The current study compares heat transfer in a wall exposed to radiation flux without any thermal protection to heat transfer in a wall shielded by a falling water film to evaluate the efficacy of radiation energy attenuation by water film. Furthermore, the impacts of different factors such as the water film flow rate, Biot number, incident radiation flux and water film cooling effect are examined.

2. PROBLEM FORMULATION AND ANALYSIS

Figure 1 illustrates the physical model of this study. Gravity trickles the water film along a solid wall. The heat flux incident upon the water film free surface is partially transmitted across the water film and it is principally absorbed by the film and the wall surface.



3. HYPOTHESIS

The following assumptions are made when developing the mathematical formulation of the problem.

- The research problem is in two-dimensional x-y plane.
- According to the Nusselt falling film theory, the liquid film is incompressible, has a constant thickness, and has a plan free surface. It is designed to be transparent in the visible and near infrared regions of the spectrum ($0 \leq \lambda < \lambda_c$) and opaque in the infrared part of the spectrum ($\lambda_c \leq \lambda \leq \infty$). Moreover, the solubility of air in water film is minimal.
- Heat radiation is supposed in the Y-direction without absorption by air.
- The radiant energy incident on the film's free surface is presumed to be uniform, and the film's spectral radiation flux is assumed to be known along its length.
- Thermodynamic and transport properties of the wall and water are assumed to be constant.
- The water-wall interface is thought to be a diffuse radiation reflector.

To develop the mathematical formulation of the problem, the following assumptions are made.

- The study problem is in two-dimensional x-y plane ;
- The liquid film is incompressible having a uniform thickness and plan free surface obedient submitted to Nusselt falling film theory. It is supposed to be transparent in the visible and in the near infrared part of the spectrum ($0 \leq \lambda < \lambda_c$) and opaque in the infrared part of the spectrum ($\lambda_c \leq \lambda \leq \infty$).

Furthermore the solubility of air in water film is negligible.

- Heat radiation is supposed in the Y-direction without absorption by air.
- The radiant energy incident on the free surface of the film is assumed uniform and the spectral radiation flux is considered to be known along the film.
- Thermodynamic and transport properties of the wall and water are supposed constant.

4. GOVERNING EQUATION:

By introducing the following non-dimensional parameters

$$t^* = \frac{\alpha_f t}{\delta_f^2}; \quad \eta = \frac{y}{\delta_p + \delta_f} = \frac{y}{\delta_T}; \quad \chi = \frac{x}{L}$$

$$\theta = \frac{T - T_0}{Q_b^0 \frac{\delta_f}{k_f}}; \quad V = \frac{\delta_f^2 u}{\alpha_f L};$$

The unsteady dimensionless equations governing the study problem associated to initial and boundary conditions are written as follows:

- Water Film

$$\frac{\partial \theta_f}{\partial t^*} = \frac{\delta_f^2}{L^2} \frac{\partial^2 \theta_f}{\partial X^2} + \frac{\delta_f^2}{\delta_T^2} \frac{\partial^2 \theta_f}{\partial \eta^2} + \frac{\delta_f}{\delta_T} \frac{\partial Q_{rad}^*}{\partial \eta} - V \frac{\partial \theta_f}{\partial X} \quad (1)$$

Where V is the dimensionless water film velocity depending on the film thickness and calculated by the integral of motion equation (Yang and Yuen, 2018):

$$V(\eta) = F \left(\frac{\delta_f}{\delta_T} \eta - \frac{1}{2} \eta^2 \right) \quad (2)$$

Q_{rad}^* represents the dimensionless local radiation flux in the water film, expressed as (Lazaridis et al., 1986) :

- Wall

$$\frac{\partial \theta_w}{\partial t^*} = \frac{\delta_f^2 \alpha_p}{L^2 \alpha_f} \frac{\partial^2 \theta_w}{\partial \chi^2} + \frac{\delta_f^2 \alpha_p}{\delta_T^2 \alpha_f} \frac{\partial^2 \theta_w}{\partial \eta^2} \quad (3)$$

- $\eta = 0, 0 \leq X \leq 1:$

$$\frac{\partial \theta_w}{\partial \eta} \Big|_{\eta=0} = B_{iog} (\theta_{pg} - \theta_{\infty g}) \quad (4)$$

$B_{iog} = \frac{h_{pg} \delta_T}{k_p}$ is the Biot number on the Wall-Air interface,

- $\eta = \frac{\delta_p}{\delta_f + \delta_p}, 0 \leq X \leq 1:$

$$\frac{\partial \theta_f}{\partial \eta} \Big|_{\eta=\frac{\delta_p}{\delta_f + \delta_p}} + \frac{\delta_T}{\delta_f} Q_{\delta_p}^* = \frac{k_p}{k_f} \frac{\partial \theta_w}{\partial \eta} \Big|_{\eta=\frac{\delta_p}{\delta_f + \delta_p}} \quad (5)$$

$Q_{\delta_p}^*$ is the dimensionless radiant flux absorbed by the Wall-Water film interface [1].

- $X = 0, 0 \leq \eta \leq \frac{\delta_p}{\delta_f + \delta_p}$

$$\frac{\partial \theta_w}{\partial X} \Big|_{X=0} = B_{ios} (\theta_{ps} - \theta_{\infty s}) \quad (6)$$

$B_{ios} = \frac{Lh_{ps}}{k_p}$ is the Biot number on the wall-Air interface,

- $X = 0, \frac{\delta_p}{\delta_f + \delta_p} \leq \eta \leq 1: \theta_{f0} = 0$
- $X = 1, 0 \leq \eta \leq 1 \frac{\partial \theta_w}{\partial X} \Big|_{X=1} = \frac{\partial \theta_f}{\partial X} \Big|_{X=1} = 0$ (7)

- $0 \leq X \leq 1, \eta = 1 :$

$$\frac{\partial \theta_f}{\partial \eta} \Big|_{\eta=1} = \frac{\delta_T}{\delta_f} Q_{\delta T}^* - Bi_{od}(\theta_{fd} - \theta_{\infty d}) - \frac{\delta_T}{\delta_f} Q_{evp}^* \quad (8)$$

$B_{iod} = \frac{h_d \delta_T}{k_f}$ is the Biot number on the Film-Air interface.

$Q_{\delta T}^*$ is the amount of the incident radiant flux absorbed by the water film surface in the infrared part of the spectrum ($\lambda_c \leq \lambda \leq \infty$), [3], and Q_{evp}^* represents the dimensionless required energy for water film evaporation.:

$$Q_{evp}^* = \frac{\dot{m}_v L_{fg}}{Q_b^0}$$

The mass evaporation rate \dot{m}_v is calculated by solving analytically the mass diffusion equation in the y-direction, [1] and [2]:

5. RESULTS AND DISCUSSION:

4.1 Wall thermal effect

To assess the thermal radiation attenuation generated by a water film flowing along a wall, we will compare the temperature evolution in a wall without protection as against a wall protected by a water film under identical conditions. Figures 2,3 and 4 illustrate the evolution of dimensionless temperature with time, as well as transverse and axial temperature distributions at various locations.

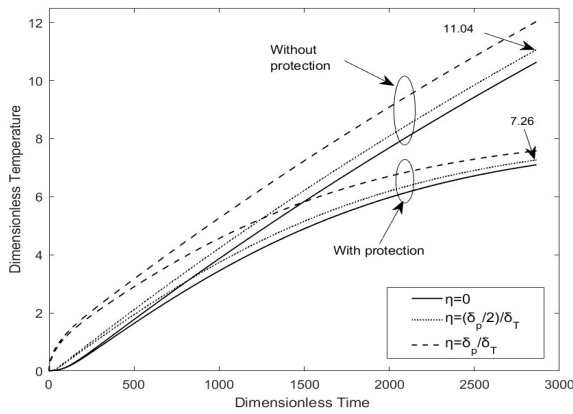


Figure 2: Comparison of dimensionless temperature variation as a function of dimensionless time, at $X=0.5$, for $Q_b^0 = 0.85, \delta_f^* = 3.48 \times 10^{-3}, \theta_0 = \theta_i = 0, \theta_{\infty g} = \theta_{\infty d} = \theta_{\infty s} = 1.87$

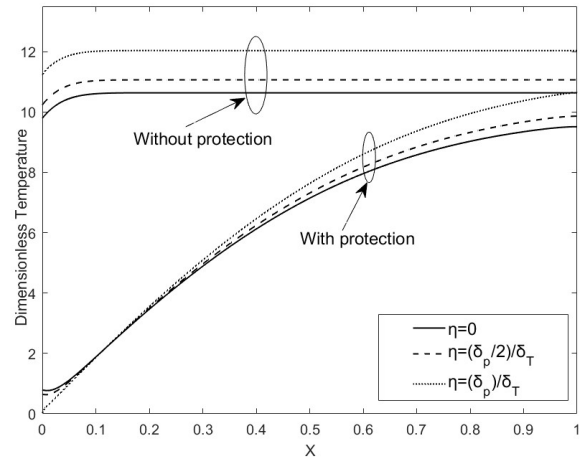
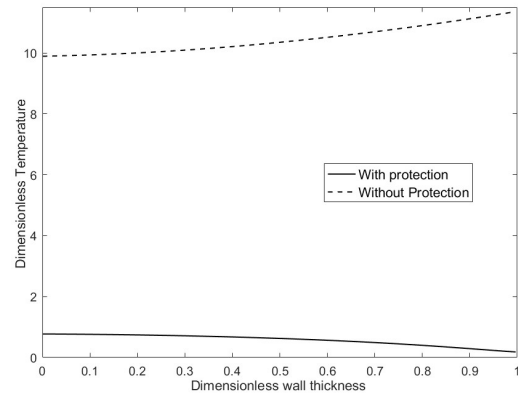


Figure 3: Comparison of axial dimensionless temperature distribution at $t^* = 2865$, in a protected and without protection wall for $Q_b^0 = 0.85, \delta_f^* = 3.48 \times 10^{-3}, \theta_0 = \theta_i = 0, \theta_{\infty g} = \theta_{\infty d} = \theta_{\infty s} = 1.87$

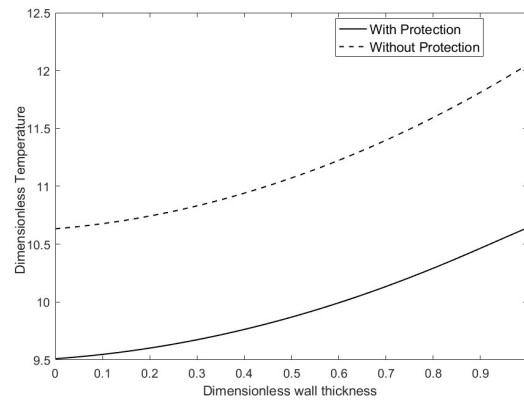


Figure 4: Comparison of transverse dimensionless temperature distribution at $t^* = 2865$, in a protected and without protection wall for $Q_b^0 = 0.85, \delta_f^* = 3.48 \times 10^{-3}, \theta_0 = \theta_i = 0, \theta_{\infty g} = \theta_{\infty d} = \theta_{\infty s} = 1.87$

According to figure 2, the dimensionless temperature profiles are similar in both cases. The results indicate that after some time ($t^*=200$), the temperature deviation between the protected wall and without protection increases gradually at all selected locations. The maximum reached

temperatures in the middle of wall thickness ($\eta = \frac{\delta_p}{2\delta_T}$) are 7.26 and 11.04 in the protected and unprotected wall respectively. This result reveals clearly the impact of the water film on the dimensionless temperature rising. It prevents at least an increase of the order of 52% in the middle of the wall.

Figure 3 displays the axial temperature distribution following $t^*= 2865$ of incident radiant heat flux exposure, the dimensionless temperature distributions are similar in each case, however continuous radiant heating raises all the temperature in different locations. The deviation of the temperatures between the protected and unprotected wall at $X=0$ (top) are much higher than those at $X=L$ (bottom), this can be explained by the impact of the boundary conditions and also the effect of the water cooling on the short region near to the inlet.

The transverse dimensionless temperatures distribution are presented on the Figure 4, it shows that the profiles are also similar except in the top of the wall in which the temperature decreases through the wall thickness due to the impact of the film on the region near to the inlet. It can also be noticed that the highest temperature was attained at the face exposed to the radiative heat flux. By comparing both results, the water film prevents at least an increase of the order of 13% in the wall bottom.

The Analysis of Figure 2-4 shows and as expected the temperature of the wall without being protected at various locations is higher than with a thermal protection. The water film seems to be very efficient at protecting the wall, as it provides a significant drop in temperature for a very little amount of water ($\delta_f^* = 3,48 \cdot 10^{-3}$) by primarily using convection and water semi-transparent properties; thus, to protect a wall or structure from an incident radiative heat flux using water film is available and effective.

4.2 Water film cooling effect

To examine the effect of wall cooling caused by the water film, we assume that the wall dimensionless temperature is 30.65 at $t^*=0$ (when the wall temperature hits 30.65 degrees due to radiant heating, water film is triggered.). Figure 5 illustrates the temperature variations in the middle of the wall and the water as a function of dimensionless time.

As water film absorbs heat not only from the direct radiant heat flux but also from the wall in the early time, the temperature of the water film rises faster than the temperature of the wall.

However, as time passes, the tendency of temperature distribution variation will be comparable. It should also be noted that once the film is activated, cooling occurs very quickly owing to the difference between the wall temperatures at $t^*=0$ and the inlet film temperature.

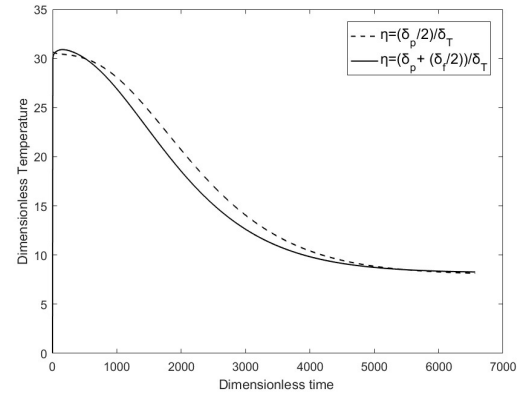


Figure 5: Dimensionless temperature as a function of time for $Q_b^{0*} = 1$, $\delta_f^* = 3.48 \times 10^{-3}$, $\theta_i = 30.65$, $\theta_{\infty g} = \theta_{\infty d} = \theta_{\infty s} = 1.87$

CONCLUSION

The present paper studied heat transfer in a falling water film exposed to a radiant flux. The focus was on exploring the thermal radiation attenuation by the water film in a thermal protection application. The resulting dimensionless governing equations were solved numerically. Based on the findings, it appears that the thermal protection efficacy of water films has been proven, even at low thicknesses (low water flow rates).

REFERENCES

- [1] Song B, Inaba H, Horibe A, Ozaki K, Heat, mass and momentum transfer of a water film flowing down a tilted plate exposed to solar irradiation, 1999.
- [2] Song B, Inaba H, Horibe A, Ozaki K. Heat, mass and momentum transfer of a water film flowing down a tilted plate exposed to solar irradiation. Int J ThermSci. 1999;38:384–397.
- [3] L. Anastas, Robert J. Copeland J.A, Temperature distribution in a solar irradiated liquid film flowing over a solid wall, 1985.
- [4] Jiann C. Y. Walter W. Y. Heat transfer in a falling laminar liquid film with in-depth radiation absorption 2018.

NUMERICAL SOLUTION OF HEAT TRANSFER INSIDE AN ANISOTROPIC PLATE MEDIUM SUBMITTED TO A LATERAL FLUX

R.IDMOUSSA¹, H. HAMZA, J. LAHJOMRI¹, A.OUBARRA¹

¹Laboratory of Mechanics, Faculty of Science Aïn Chock, University Hassan II, Casablanca 20100, Morocco
rabiaa.idmoussa@gmail.com

Abstract

In this paper, a numerical solution for bidirectional anisotropic plate heat transfer is presented. The medium is subject at ends to an imposed temperature while an external flux is maintained through the side surface. The mathematical difficult for the anisotropic heat conduction problem is caused by the complex form of the governing equation. Therefore a linear coordinate transformation is used in order to alleviate this difficulty by converted to an equivalent isotropic problem. The finite-difference method (ADI) is used to integrate the isotropic equation. Good agreement between the numerical results with analytical solution has been found. The aim of this paper is to establish the axial and radial temperature profiles for steady state anisotropic medium as a function of main thermal conductivities, anisotropy and the external flux.

Keywords: Anisotropic plate, Main thermal conductivities, Cross thermal conductivity, linear coordinate transformation.

1. Introduction

Heat conduction anisotropic materials are prevalent choices in many fields of modern engineering science. These anisotropic materials are abundant either naturally in the form of crystals, and sedimentary rocks or industrial, as fibrous materials. Studying of this kind of problem in thermal is quite difficult because of nonlinearity term (cross-derivative term) presented in the governing energy equation. Analytical or semi-analytical solutions presented in the literature for conductive anisotropic medium are limited compared to the isotropic and orthotropic case. In this field the publications may be divided into studies under transient [1-3] or steady state [4-7]. In the case of finite geometries the analytical solution for anisotropic medium, has not still found. Hence is essential to resort a numerical analysis. Three basic methods have been proposed to deal numerically this problem: element method (FEM), boundary finite element method (BEM) [8-9] and finite difference method [10-11]. The latter method is adopted by our study in order to obtain temperature distribution for the steady state anisotropic medium, as a function of all parameter studies: main thermal conductivities, anisotropy and the external heat flux.

2. Model formulation

The study concerns anisotropic rectangular plate of length L and width $2b$ schematised on Fig.1. The two ends of

plate are subject to a same temperature while the lateral surface is maintained at an external flux.

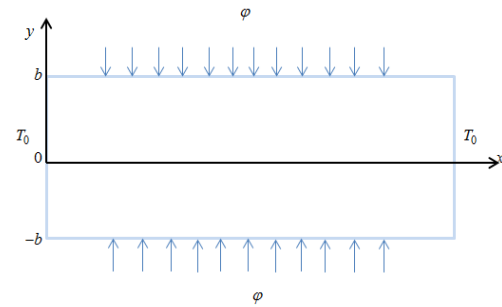


Fig. 1: Anisotropic plate geometry.

The conduction heat transfer equation is given by the following differential equation:

$$K_{xx} \frac{\partial^2 T}{\partial x^2} + 2K_{xy} \frac{\partial^2 T}{\partial x \partial y} + K_{yy} \frac{\partial^2 T}{\partial y^2} = \rho c_p \frac{\partial T}{\partial t} \quad (1)$$

With K_{xx} , K_{yy} and K_{xy} are respectively, the main and cross thermal conductivities according to directions x and y .

The Fourier relation for an anisotropic medium in a Cartesian system is given by:

$$\varphi_x = -K_{xx} \frac{\partial T}{\partial x} - K_{xy} \frac{\partial T}{\partial y} \quad (2)$$

$$\varphi_y = -K_{xy} \frac{\partial T}{\partial x} - K_{yy} \frac{\partial T}{\partial y} \quad (3)$$

The above equation (1) is associated to the following spatio-temporal boundary conditions :

$$T(x, y, t = 0) = T(x = 0, y, t) = T(x = L, y, t) = T_0 \quad (4)$$

$$y = \pm b \text{ and } 0 \leq x \leq L \quad K_{xx} \left(\frac{\partial T}{\partial x} \right)_{y=\pm b} + K_{xy} \left(\frac{\partial T}{\partial y} \right)_{y=\pm b} = \varphi \quad (5)$$

A special linear coordinate transformation [5] is introduced in view to eliminate the cross-derivatives and reduce the governing equation to a canonical form.

$$\begin{pmatrix} X \\ Y \end{pmatrix} = \begin{pmatrix} 1 & a \\ 0 & \gamma \end{pmatrix} \begin{pmatrix} x \\ y \end{pmatrix} \quad (6)$$

$$\text{With } a = -\frac{K_{xy}}{K_{yy}} \quad ; \quad \gamma = \frac{\sqrt{(K_{xx}K_{yy} - K_{xy}^2)}}{K_{yy}}$$

The governing equations (1)-(3) above becomes

$$K \frac{\partial^2 T}{\partial X^2} + K \frac{\partial^2 T}{\partial Y^2} = \rho c_p \frac{\partial T}{\partial t} \quad (7)$$

With: $K = K_{yy}\gamma^2$

$$\varphi_x = -K \frac{\partial T}{\partial X} - K_{xy}\gamma \frac{\partial T}{\partial Y} \quad (8)$$

$$\varphi_y = -K_{yy}\gamma \frac{\partial T}{\partial Y} \quad (9)$$

To generalize the problem, we introduce the following non-dimensional parameters:

$$X^* = \frac{X}{b}, \quad Y^* = \frac{Y}{b}, \quad \theta = \frac{T - T_0}{T_0}, \quad t^* = \frac{t\alpha}{b^2}$$

$$\alpha = \frac{K}{\rho c_p}$$

The non-dimensional form of equations can be written as follows:

$$\frac{\partial^2 \theta}{\partial X^{*2}} + \frac{\partial^2 \theta}{\partial Y^{*2}} = \frac{1}{\gamma} \frac{\partial \theta}{\partial t^*} \quad (10)$$

$$\varphi_x^* = -K \frac{\partial \theta}{\partial X^*} - \frac{K_{xy}\gamma}{K} \frac{\partial \theta}{\partial Y^*} \quad (11)$$

$$\varphi_y^* = -\gamma \frac{\partial \theta}{\partial Y^*} \quad (12)$$

$$\theta(X^*, Y^*, t^* = 0) = \theta(X^*, Y^* = 0, t^*) = \theta(X^*, Y^* = G, t^*) = \theta_0 = 0 \quad (13)$$

$$y = \pm \gamma \text{ and } 0 \leq x \leq G - 2a \quad \varphi^* = \frac{\varphi}{K \cdot T_0} \quad (14)$$

These non-dimensional equations are associated with the domain of Fig.3 in the plane (X^*, Y^*)

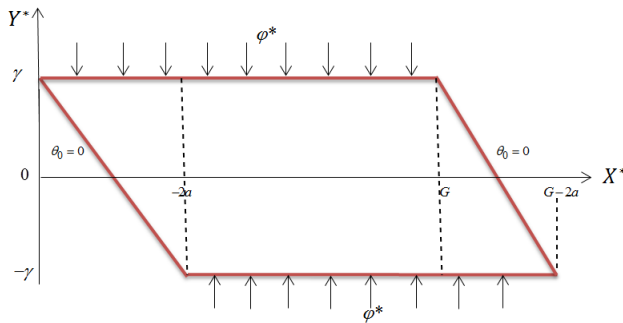


Fig. 2: Geometric representation of anisotropic plate in new space (X^*, Y^*) .

3. Numerical solution and validation

Globally, the analytical solutions of conductive heat transfer problems of anisotropic plate present mathematical difficulties, particularly in finite geometry. Therefore a numerical modeling has been developed and solving by the finite-difference method (ADI) [12]. This method is unconditionally stable and gives rise to a two tri-diagonal matrix resolved by the THOMAS algorithm. The verification of this current solution is examined by comparing it with the analytical solution (equation 15) established for steady-state orthotropic medium given by:

$$\theta(x^*, y^*) = \frac{4\delta G \varphi^*}{\pi^2} \sum_{n=1,3} \frac{1}{n^2} \sin\left(\frac{n\pi x^*}{G}\right) \left\{ \begin{array}{l} \frac{1}{\text{th}(2\lambda_n)} \text{ch}\left(\frac{n\pi y}{G\delta}\right) - \text{sh}\left(\frac{n\pi y}{G\delta}\right) \\ + \frac{1}{\text{sh}(2\lambda_n)} \text{ch}\left(\frac{n\pi y}{G\delta}\right) \end{array} \right\} \quad (15)$$

$$\text{With: } \delta = \frac{K_{xx}}{K_{yy}}; \quad \lambda_n = \frac{n\pi}{\delta G}; \quad G = \frac{L}{b}$$

The distribution of the steady state dimensionless temperature given by equation (1) is confronted to the corresponding of numerical model profile obtained, for the case $(K_{xx}/K_{yy} = 0.5, K_{xy}/K_{yy} = 0.01, \varphi^* = 1)$.

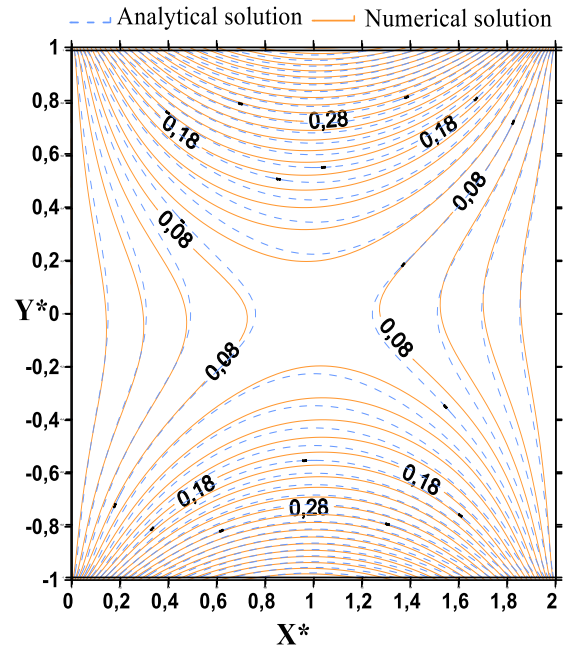


Fig. 3: Analytical and numerical solution in steady state for orthotropic medium $(G=2)$.

4. Results and discussion

By using the numerical solution developed, we can evaluate the effect of the study parameters (external heat flux, main thermal conductivities ratio and anisotropy) on thermal behavior by tracing the radial and axial temperature profiles in the middle and at mid-height of the anisotropic plate for form factor (G) equal to 2. The below figures relate to a main thermal conductivities ratio (K_{xx}/K_{yy}) equals respectively 0.5, 1 and 2 parameterized by lateral external density flux (φ^*) equals to 0.5 and 1, the anisotropic character worth 0.1 and 0.6. The values retained for this study are in the range of anisotropic materials cited by Ma and Chang [5]. We notice from the numerical results schematized in Fig.4 that the anisotropy has a marked influence on the radial and axial distribution of temperature.

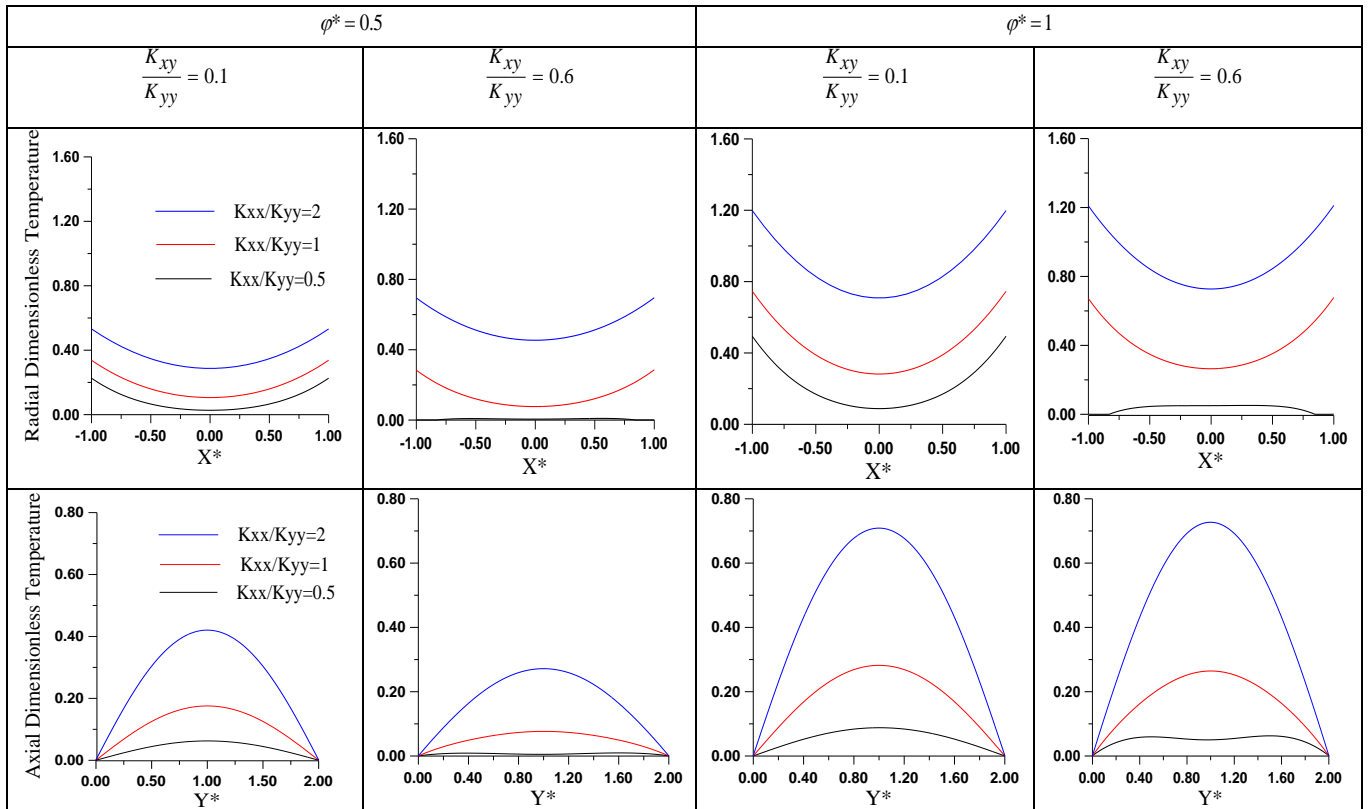


Fig. 4: Radial and axial dimensionless temperature profile in the middle and at mid-height of anisotropic plate.

5. Conclusion

In this paper, a numerical solution for heat conduction in anisotropic plate is got by using a coordinate transformation technique and resolved by alternating directions method (ADI). This code has been successfully validated and used to examine the impact of all study parameters, mainly thermal conductivities, anisotropy and the external heat flux on the thermal behavior of the anisotropic medium.

References

- [1] D. Reitzle, S. Simeon Geiger, A. Kienle, *Semi-analytical solution of the time-dependent heat equation for three-dimensional anisotropic multilayered media*, J.of heat Mass transfer 134 (2019), 984–992.
- [2] M.N. Ozisik, S.M. Shouman, *Transient Heat Conduction in an Anisotropic Medium in Cylindrical Coordinates*, J. of the Franklin Institute 309.6 (1980), 457-472
- [3] J.Padovan, *Transient temperature distribution of an anisotropic half space*, J.AIAA. 11, 4(1973), 565-566.
- [4] Y.P.Chang, *Analytical solution for heat conduction in anisotropic media in infinite semi-infinite, and two-plate-bounded regions*, J.of Heat Mass Transfer 20 (1977), 1019–1028.
- [5] C.C.Ma, S.W.Chang, *Analytical exact solutions of heat conduction problems for anisotropic multilayered media*, J.of Heat Mass Transfer 47(2004), 1643–1655.
- [6] G.P. Mulholland, B.P.Gupta, *Heat transfer in a three-dimensional anisotropic solid of arbitrary shape*, J.of Heat Transfer 99 (1977), 135-137.
- [7] M. Norouzi, H. Rahmani, A.Birjandi, A.Joneidi, *A general exact analytical solution for anisotropic non-axisymmetric heat conduction in composite cylindrical shells*, J.of Heat Mass Transfer 93(2016), 41-56.
- [8] Y.C., Shiah, C.L.Tan, *BEM treatment of two dimensional anisotropic field problems by direct domain mapping*, J.of Engineering Analysis with Boundary element 20 (1997), 347-351.
- [9] Shiah,Y.C, and Tan, C.L, *BEM treatment of three-dimensional anisotropic field problems by direct domain mapping*, J.of Engineering Analysis Boundary element 28 (2004), 43-52.
- [10] I. Aslib, H. Hamza, J. Lahjomri, K. Zniber, A.Oubarra, *Numerical solution of unsteady heat transfer in anisotropic cylindrical geometry*, J.of Thermal Science Engineering Application 8(2016), 0310131- 0310139.
- [11] I. Aslib, H. Hamza, N. Hanchi, J. Lahjomri, A.Oubarra, *Numerical study of transient and steady-state anisotropic cylindrical pin fin thermal behavior*, Thermal Science Engineering Application 11(2019), 0310071-03100712.
- [12] S.V.Patankar, *Numerical Heat Transfer and Fluid Flow*, New York, McGraw-Hill, ISBN, (1980) 0-07-048740.

Investigating wave energy potential in Casablanca-Mohammedia coast using SWAN model

M. El hou¹, L. Mouakkir², S. Mordane³

1. Faculté des sciences Ben M'sik, elhou.mohammad@gmail.com

2. Faculté des sciences Ben M'sik, mouakkir_laila@yahoo.fr

3 Faculté des sciences Ben M'sik, mordanesoumia@gmail.com

Abstract

This work focuses on the investigation of wave potential in the Casablanca-Mohammedia coast. The wave climate was firstly calculated based on 20-years wave data. Subsequently, the wave model SWAN was implemented in the study area. The wave model was applied to compute the patterns of wave field for a typical wave conditions. The results of this study confirms the abundance of wave energy in the area with an annual mean value of about 22kW/m. Additionally, the wave simulations revealed an important wave energy hot-spot site, situated in front of Sidi Rahal.

Keywords: *Wave energy, Numerical wave modeling, SWAN model, Casablanca-Mohammedia coast.*

1. Introduction

Morocco is experiencing a significant growth in electricity consumption, with an average rate of 6.9 % per year [1]. This growth, in combination with the global warming problems and the need to conserve fossil energy resources, requires the exploitation of renewable and sustainable energy resources. To reach a global and sustainable solution to these problems, the strategic and general conditions of the country are oriented towards the development of renewable energies. In this respect, the Moroccan government has put in place an ambitious energy plan, called the National Energy Strategy [2] to increase the input of renewable energy resources into the national grid.

Over the last few years, a big interest has been given to marine renewable energy (MRE). Among the potential sources of MRE wave energy is regarded as one of the most promising. It is a condensed form of ocean wind energy [3], constituting an abundant resource of highest marine energy density in the coastal areas [4].

Therefore, the assessment of wave energy propagation is a very important task not only for its exploitation as a power resource but also for its destructive effects in the coastal zones [5]. The necessity of wave energy suitable for countries or regions with an extensive coastline and high waves approaching the shore [6].

The Moroccan coast is longer than 3500 Km, it can be seen as a considerable renewable energy source. Morocco has a huge potential source of marine renewable energy, it has a view on the Mediterranean Sea and Atlantic Ocean, which makes it a country with several sites for the exploitation of wave energy.

According to [7,11], The average wave power obtained along the Moroccan Atlantic coast is considerable (up to 30 kW/m with an average annual wave energy at 262 MWh/m). The spatial distribution of wave energy shows

great variability, with a high energy stretch in the central part of this area. The present work focuses on the wave energy resource in the Casablanca-Mohammedia Coast, which is characterised thoroughly in terms of wave directions, heights and periods using ECMWF (European Center for Medium range Weather Forecasts) ERA5 data-set.

Due to advances in computer science, wave energy assessment uses numerical models capable of accounting for the physics of wave propagation in the shallow waters of coastal regions, and therefore more realistic assessments of wave energy potential.

The present study is based on wave modelling using a state-of-the-art wave model SWAN. The latter computes the transformation of the offshore wave field as it propagates into the coastal area.

This study is structured as follows: In section 2, we first briefly describe the study area, then we present the numerical model features and set-up, the sources of wave data. The analysis results are illustrated in Section 3. Finally the main conclusions from the work are discussed in section 4.

2. Data and Methods

2.1 Study area

The area concerned by the simulation includes the coast of Casablanca-Mohammedia located at the northwest of Morocco on the Atlantic coastline. This coastal stretch has a NW-NE orientation, and is consequently exposed to waves coming from NW sector. The model zone extends longitudinally from 7°2'W to 8°W, and at latitude from 33°45'N to 34°N. This area is characterized by a regular bathymetry showing a slope going down to the sea. The isobath is appreciably parallel to the coast, they vary from 5m at the coast and 293 m offshore (fig.1). This quasi-regular isobaths configuration is perturbed in the extreme south by the presence of a submarine ridge extending about 11Km from coastline. Moreover, we notice the presence of three very marked capes (Cape Dar Bouazza, Cape Elhank, Cape Fedala), these latter are elongated seawards by submarine ridges (D1, D2, D3 and D4). There exist also six shoals (H1, H2, H3, H4, H5, H6) [8] where water depth is nearly 28m. The bathymetry information, provided through the "General Bathymetric Chart of the Ocean" (GEBCO) [12]. After interpolation of bathymetric data, we obtained in the horizontal plane a regular grid of 1km x 1km, resulting in a horizontal configuration with 74 points in the x-direction, and 61 points in the y-direction (Fig.1).

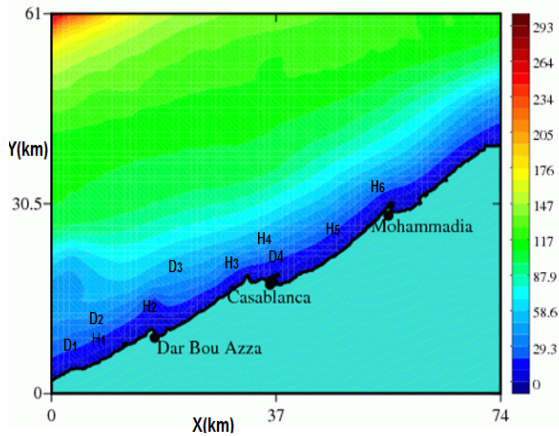


Fig.1 The bathymetry of the study area, depth contours marked in meters, red diamond symbol indicates the buoy location, red star indicates ERA5 data-set node used for wave climate calculation

2.2 SWAN wave model and used data

The wave simulations in this study were performed based on the third generation model SWAN version 41.31 [9,10]. It is a numerical wave model that provides realistic estimates of wave parameters in open seas, coastal areas, lakes, and estuaries from given wind, bottom, and current conditions [9]. In the SWAN model, for the control equation for wave description, the dynamic spectrum balance equation is adopted based on the theory of linear and random surface gravity waves. In the flow field, the random waves are presented in two-dimensional dynamic spectral density rather than two dimensional energy spectral density.

The study of data on the wave climate in the area of interest was drawn up using ERA5 database. The wave time series spans twenty years and covers the period 2001-2020. The wave climate analysis was based on a standard wave parameter (significant wave height H_s , peak wave period T_e , and mean direction θ_m). A Datawell buoy observations, provided by the General Direction of Meteorology, were exploited to test the performance of SWAN model to simulate waves nearshore. The location of the buoy is indicated in the figure 1.

The model was executed in both stationary and nonstationary modes. The spectral directions cover the full circle. The resolution in θ -space is 10° for the variable θ and 32 frequency. The lowest frequency is fixed at 0.0521 Hz and the highest at 1 Hz. The wind condition, in the stationary mode, was considered nil. Physical processes during the wave simulation are wave generation, quadruplet nonlinear interaction, dissipation by friction on the bottom, the parameterization of the friction used for our simulations is that of Madsen. The simulations also take into account the effects of diffraction, as well as wave-wave interactions.

3. Results and discussions

3.1 Wave climate

Figure 2 depicts the wave rose off Casablanca-Mohammadia coast. The deep-water waves reaching the area of interest come mainly from NW and NNW directions. The

dominant heights are in the interval [1.0m-2.0m] with about 62% of probability. As far as wave periods concerning, the most frequent are in the interval [12s-15s] with about 35% of frequency, followed by those in the intervals [10s-12s] and [8s-10s] respectively (frequencies 27% and 20%). The calculated wave statistics confirm that both observed mean significant wave height and wave energy period are significantly higher during winter ($H_s=2.3m, T_e=11.8s$) than in summer ($H_s=1.3m, T_e=8s$).

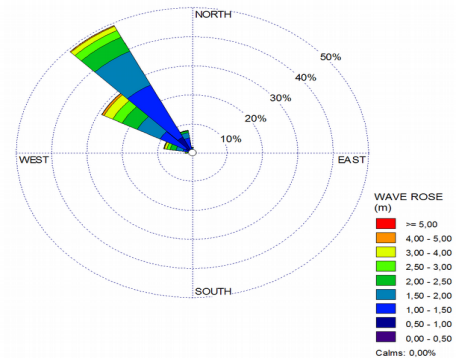


Fig.2 Wave rose offshore the studied zone

3.2 Wave simulations

The model was, at first, tested in a real conditions. Figure 3 presents the model wave heights compared to buoy data during eight day starting from 01/01/2008. it shows a good agreement between model and observed H_s . Afterwards, it was applied to determine the location of the nearshore hotspots in the Casablanca-Mohammadia coast. For this purpose, the nearshore wave patterns were computed in two case studies. They represent a characteristic winter and summer wave situations.

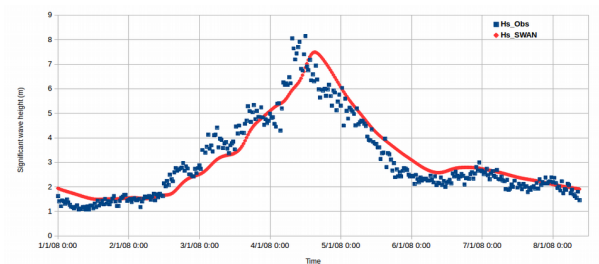


Fig.3: SWAN computed wave height compared to buoy data observation from 01/01/08 at 0000GMT to 08/01/08 at 0900GMT.

For the simulation results, the first case study I, represents a typical winter wave condition ($H_s=2.3m, T_p=12.4s, \theta=304^\circ$). The wave heights are generally in the interval 2-2.2 m. In the extreme south, nearshore Sidi Rahal, H_s exceeds 2.3 m. Regarding the wave energy flux, in the-nearshore the values are in the range 20-25 kW/m. We note a remarkable area of energy concentration located in the southern part of the coast in front of Sidi Rahal area, where wave energy rises significantly to 30-35 kW/m. This increase is equivalent to about 17% of the offshore wave energy.

The second case study II represents a typical summer wave condition ($H_s=1.3m, T_p=8.5s, \theta=321^\circ$). The wave height as well as wave flux in the nearshore region are

significantly low with H_s between 1-1.2 m (4-6 kW/m), except in the southern extremity where H_s arises slightly above 1.2 m (about 7 kW/m). In conclusion, it is apparent that as the waves propagate from the sea to the coast, the wave height gradually decreases, however, this general trend is not verified in some coastal areas where the bathymetry leads to a concentration of wave energy and therefore an increase in wave height. This remark is observed in the vicinity of the Cape Dar Bouazza, precisely around the ridges D1 and D2 as well as the shoals H1 and H2. This observation is clearly visible on the longitudinal sections that pass through the three graphical capes, i.e., a decrease in height as it moves from the open sea to the coast (figure 4). **Fig.4:** Wave energy flux along five vertical sections passing through Cape Dar Bouazza (blue), Tamaris (red), Cape Al Hank (yellow), Cape Fedala (green) and El mansouria (brown) for the first case study (winter).

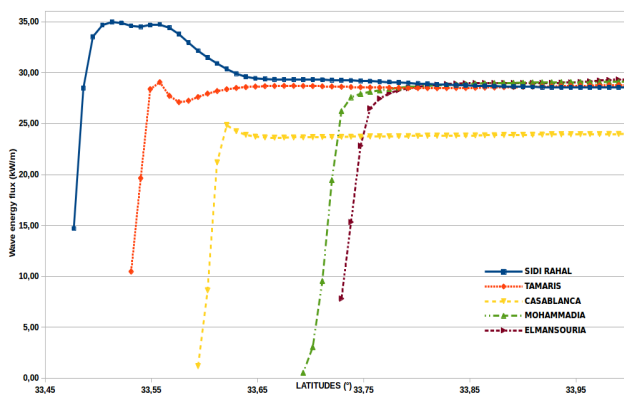


Fig.4: Wave energy flux along five vertical sections passing through Cape Dar Bouazza (blue), Tamaris (red), Cape Al Hank (yellow), Cape Fedala (green) and El mansouria (brown) for the first case study (winter).

4. Conclusions

The present study presents an analysis of the potential wave energy resource in Casablanca-Mohammedia coast.

The wave climate has been drawn based on 20-years wave data provided by ECMWF. The SWAN model was executed in a stationary mode to simulate wave patterns.

The spatial distribution of wave energy in the study area is examined under different wave conditions (winter and summer). The results of this study shown in term of wave energy power analysis confirms the abundance of wave energy in the area with an annual mean potential of about 22kW/m. Additionally, an important hot-spot site was identified to be taken in consideration when investigating wave farm implementation. The hotspot site is located in the southern limit of studied coast, in front of Sidi Rahal area.

In sum, the wave energy simulations presented in this study will enable the estimation of the capacity and profitability of wave energy farms in the Casablanca-Mohammedia coastal area. Therefore, the results of the

long-term assessment using finer resolution bathymetry is a necessary tool to identify suitable sites for the implementation of wave energy farms by the private sector or public institutions.

References

- [1] ONE, Programme d'Electrification Rurale a meteorological perspective, Energy Strategy Reviews 6 (2015) pp. 39–47.
- [2] A. Akpinar, B. Bingölbali, G.P. Van Vledder, Long-term analysis of wave power potential in the Black Sea, based on 31-year SWAN simulations, Ocean Eng. 130 (2017) pp. 482–497.
- [3] A. Clement et al., Wave energy in Europe: current status and perspectives, Renew. Sustain. Energy Rev. 6 (2002) pp. 405–431
- [4] K. Amarouche, N.-E.-I. Bachari, F. Houma, Study of the coastal wave energy propagation using {GIS} and hydrodynamic model, PROCEEDING B. Int. Symp. Global, 2010.
- [5] A. Alhamwi, D. Kleinhans, S. Weitemeyer, T. Vogt, Moroccan National energy strategy reviewed from GIS Appl. Geogr. Geosci., Çanakkale Onsekiz Mart University, Canakkale, Turkey, 2017, pp. 55-64.
- [6] D. Zhang, W. Li, Y. Lin, Wave energy in China: current status and perspectives, Renew. Energy 34 (2009) pp. 2089-2092.
- [7] J.P. Sierra; C. Martín; C. Mösso; M. Mestres; R. Jebbad, Wave energy potential along the Atlantic coast of Morocco. Renew. Energy 96 (2016) pp. 20–32.
- [8] L. Mouakkir, H. Smaoui, S. Mordane and M. Chagdali, Three-dimensional modelling of hydrodynamics with wave-current interaction : Application to the Casablanca-Mohammedia coast, Journal Physical and Chemical News 55 (2010) pp. 74-80
- [9] SWAN team, SWAN User Manual, SWAN Cycle III version 41.20AB, Delft University of Technology, Delft, 2018.
- [10] N. Booij, R.C. Ris, L.H. Holthuijsen, A third-generation wave model for coastal regions: 1, Model description and validation, J. Geophys. Res. 104(C4) (1999) 7649–7666.
- [11] H. Bouhrim & A. El Marjani, Wave Energy Assessment Along the Moroccan Atlantic Coast, Journal of Marine Science and Application 18 (2019) pp. 142–152
- [12] P. Weatherall, K. Marks, M. Jakobsson, T. Schmitt, S. Tani, J.E. Arndt, M. Rovere, D. Chayes, V. Ferrini, R. Wigley, GEBCO_2014 Grid, version 20150318, General Bathymetric Chart of the Oceans (2015)

EVAPORATION OF A SALTWATER FILM INSIDE A VERTICAL TUBE

A. CHAREF^{1,2}, M. FEDDAOUI², M. NAJIM^{2,3}, M. RAHMOUNE¹, R. SAADANI¹,
M. HISSOUF²

1. Laboratory of Advanced Materials and Applications, EST, Moulay Ismail University, Meknes, Morocco.

2. GEMS laboratory, ENSA, Ibn Zohr University, Agadir, Morocco.

3. FST, University of Hassan II Mohammedia, Morocco.

Corresponding author: charef.adil@gmail.com

Abstract

A numerical analysis is conducted to examine numerically the combined heat and mass transfer through studying liquid film evaporation of saltwater in a vertical tube. The saltwater film is heated with uniform heat flux. A complete two-phase model, based on the conservation of mass, momentum, and energy in each phase is realized. Results are presented for slightly saline water and highly saline water. The results indicate that the evaporation process increases with salinity increasing. In addition, heat and mass transfer are enhanced by increasing the heat flux of the wall.

Keywords: Evaporation, Saltwater, Salinity, Heat and mass transfer, Desalination.

1. Introduction

The shortage of freshwater is today a challenge that threatens a large part of the population in many parts of the world. To solve this problem, desalination has been proved to be a feasible and effective method. Solar desalination systems which are based on the humidification-dehumidification concept are considered the most promising among solar desalination technologies [1]. Numerous solar systems have been developed based on the evaporation of liquid films. Several studies have been conducted to investigate the evaporation process of liquid films. Najim et al [2] investigated the salinity effect on heat and mass transfer during the evaporation of a liquid film through a vertical tube. They analyzed the variation of salinity across the tube and compared the influence of the operating conditions on the effectiveness of the desalination process. Abraham et al [3] numerically studied the heat transfer characteristics of evaporation from a liquid film falling on the outside of a tube. The findings of their study indicated that the convective heat transfer performance of evaporation increases with the feed rate, while it decreases with the diameter of the tube. Feddaoui et al [4] carried out a detailed computational work that reported better conditions to get good heat and mass transfer like higher heat flux and lower inlet water flow. Orfi et al [5] investigated air humidification by evaporation from a thin liquid film moving in a vertical channel. The results indicate that the heat transfer at the interface can be greater than the heat flux imposed on the wall. Qi et al. [6] presented numerically and experimentally the heat transfer of seawater desalination with a liquid film on the external of an elliptical tube.

Yang et al. [7] conducted an experimental study of falling film evaporation outside horizontal tubes.

Many articles exist in the literature related to film evaporation for desalination. Liu et al. [8] used the VOF model to simulate the falling film of saline wastewater and pure water into vertical pipes at different structural parameters and mass flow rate. They analyzed the flow and heat transfer process's difference of both pure water and saline wastewater along with falling film evaporator. Shen et al [9] studied falling film evaporation in a vertical tube. Their results show that thicker film produces larger thermal resistance. Kouhikamali et al. [10] conducted a numerical study in the case of falling film evaporation of a multi-effect desalination plant.

The current paper presents a numerical investigation of the heat and mass transfer during evaporation of a saltwater film in a vertical tube. The aim of this study is to reveal the salinity effects on the evaporation process.

2. Mathematical model

The geometry of the present study is a vertical tube with a uniform heat flux in which a falling liquid film of saltwater is evaporated (Fig. 1). The liquid film enters with inlet temperature T_{L0} , inlet mass flux Γ_0 , and inlet salinity S_0 . The gas flow enters with inlet velocity u_0 , inlet temperature T_0 , and inlet relative humidity Hr .

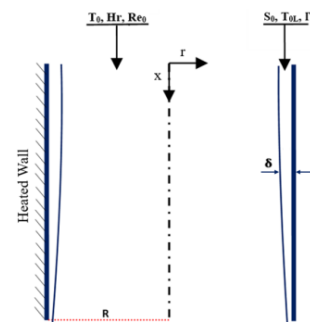


Figure 1: Physical model

2.1 Governing equations

Taking into account some assumptions, heat and mass transfer in the liquid film and gas flows are described by the governing equations and interfacial conditions given by Groff et al. [11].

For the liquid phase

$$\frac{\partial}{\partial x}(\rho_L u_L) + \frac{1}{r} \frac{\partial}{\partial r}(r \rho_L v_L) = 0 \quad (1)$$

$$\frac{\partial}{\partial x}(\rho_L u_L u_L) + \frac{1}{r} \frac{\partial}{\partial r}(r \rho_L v_L u_L) = -\frac{dp}{dx} + \frac{1}{r} \frac{\partial}{\partial r}(r \mu_L \frac{\partial u_L}{\partial r}) + \rho_L g \quad (2)$$

$$\frac{\partial}{\partial x}(\rho_L u_L C_{p,L} T_L) + \frac{1}{r} \frac{\partial}{\partial r}(r \rho_L v_L C_{p,L} T_L) = \frac{1}{r} \frac{\partial}{\partial r}(r \lambda_L \frac{\partial T_L}{\partial r}) \quad (3)$$

The salinity value is updated at each section by considering the evaporated quantity of liquid film as follows:

$$S_x = \frac{S_0 \Gamma_0}{\Gamma_0 - 2\pi \int_0^x \dot{m}_l (R - \delta_x) dx} \quad (4)$$

For the gas phase

$$\frac{\partial}{\partial x}(\rho_G u_G) + \frac{1}{r} \frac{\partial}{\partial r}(r \rho_G v_G) = 0 \quad (5)$$

$$\frac{\partial}{\partial x}(\rho_G u_G u_G) + \frac{1}{r} \frac{\partial}{\partial r}(r \rho_G v_G u_G) =$$

$$-\frac{dp_d}{dx} + \frac{1}{r} \frac{\partial}{\partial r}(r \mu_G \frac{\partial u_G}{\partial r}) + (\rho_G - \rho_0)g$$

$$\frac{\partial}{\partial x}(\rho_G u_G C_{p,G} T_G) + \frac{1}{r} \frac{\partial}{\partial r}(r \rho_G v_G C_{p,G} T_G) =$$

$$\frac{1}{r} \frac{\partial}{\partial r}(r \lambda_G \frac{\partial T_G}{\partial r}) + \frac{1}{r} \frac{\partial}{\partial r}[r \rho_G D (C_{p,v} - C_{p,a}) \Gamma_G] \frac{\partial w}{\partial r}$$

$$\frac{\partial}{\partial x}(\rho_G u_G w) + \frac{1}{r} \frac{\partial}{\partial r}(r \rho_G v_G w) =$$

$$\frac{1}{r} \frac{\partial}{\partial r}(r \rho_G D \frac{\partial w}{\partial r}) \quad (8)$$

At every axial location, the overall mass balance in the gas flow and liquid film should satisfied the two criterions:

$$\frac{\Gamma_0}{2\pi} = \int_{R-\delta_x}^R (r \rho_L u_L) dr - \int_0^x \rho_g v_l (R - \delta_x) dx \quad (9)$$

$$\frac{(R - \delta_0)^2}{2} \rho_0 u_0 = \int_0^{R-\delta_x} (r \rho_L u_L) dr \quad (10)$$

$$+ \int_0^x \rho_g v_l (R - \delta_x) dx$$

Where $\dot{m}_l = -\rho_g v_l$ is the rate of vapour generation.

2.2 Boundary conditions

The boundary conditions used in this study are:

❖ at the inlet of the tube ($x = 0$)

$$T_G = T_{G0}, w_G = w_{G0}, u_G = u_{G0}; \quad (11)$$

$$T_L = T_{L0}, \Gamma = \Gamma_0, S = S_0$$

❖ At the tube center ($r = 0$)

$$v_G = 0; \frac{\partial u_G}{\partial r} = \frac{\partial T_G}{\partial r} = \frac{\partial w}{\partial r} = 0 \quad (12)$$

❖ at the interface ($r = R - \delta_x$)

$$u_l(x) = u_{G,l} = u_{L,l} \quad (13)$$

$$T_l(x) = T_{G,l} = T_{L,l}$$

$$\tau_l = \left[\mu \frac{\partial u}{\partial r} \right]_{L,l} = \left[\mu \frac{\partial u}{\partial r} \right]_{G,l} \quad (14)$$

$$\left[\lambda \frac{\partial T_L}{\partial r} \right]_L = \left[\lambda \frac{\partial T_G}{\partial r} \right]_{-G} + \dot{m}_l h_{fg} \quad (15)$$

❖ at the tube wall ($r = R$)

$$u_L = 0; -\lambda_L \frac{\partial T_L}{\partial r} = Q_w \quad (16)$$

3. Numerical resolution

The governing equations of flows with heat and mass exchanges coupled with the accurate boundary conditions are solved by a numerical finite difference method. The backward difference is used for axial convective terms while the central difference is applied in the transverse direction for both convective and diffusive terms. The discretization of each system of equations makes a tridiagonal matrix, which is resolved by the algorithm of Thomas [12]. Furthermore, to ensure the accuracy of the numerical simulation, a non-uniform grid in both radial and axial directions is used. The numerical model is validated by comparing our results with the previously published ones.

4. Results and Discussion

This study is carried out to investigate numerically heat and mass transfer during liquid film evaporation of saltwater inside a vertical tube. The range of each parameter for the analysis is listed in table 1. The chosen common parameters are: the tube length of 1m, Reynolds number of 2000, and inlet gas temperature of 20°C.

Tableau 1: Ranges of the physical parameters.

Inlet salinity S_0 (g/kg)	3 – 8 – 35
Wall heat flux Q_w (KW.m ⁻²)	8 – 10 – 12

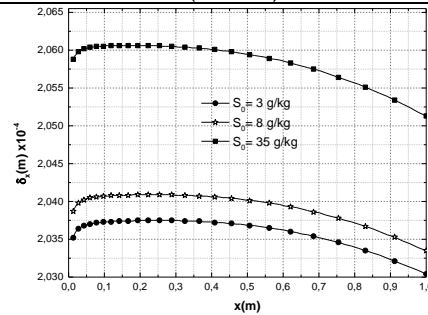


Figure 2: Influence of S_0 on liquid film thickness along the tube.

The effect of the inlet salinity S_0 on the liquid film thickness δ_x along the tube is shown in Fig. 2. From this figure, it is observed that as the liquid goes downstream, the film thickness decreases notably near the tube exit. The rise in water salinity gives the high evaporating mass flux. In addition, with high salinity, the density of the liquid increases compared to slightly saline water. This means that with the same inlet Γ_0 , highly saltwater has a bigger film thickness. The curves of Fig. 3, show the evolution of the sensible heat flux along the tube. It is noted that slightly saline water presents a little more sensible heat fluxes compared to highly saltwater. In addition, the differences rise as the fluid moves toward the exit. This is apparently due to the thermal conductivity which becomes lower with the increase in salinity and the amount of evaporated water increases. The variation of axial evolution of latent heat flux is shown in Fig.4. It is found that with salinity increasing, the latent heat flux becomes more important in comparison to slightly saline water. The salinity improves latent heat transfer, Moreover, it is interesting to note that the latent heat flux dominates this evaporation process since it is much larger than the sensible heat flux.

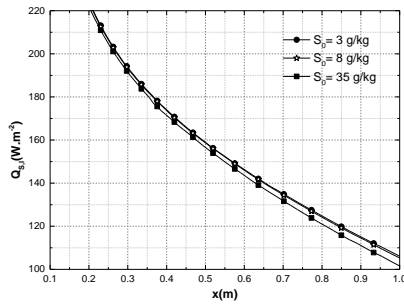


Figure 3: Axial evolution of sensible heat flux.

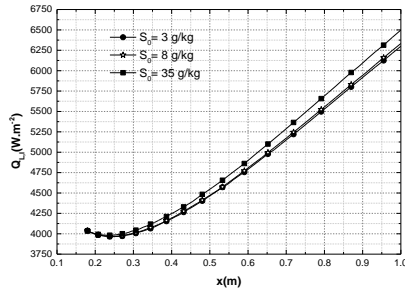


Figure 4: axial evolution of latent heat flux.

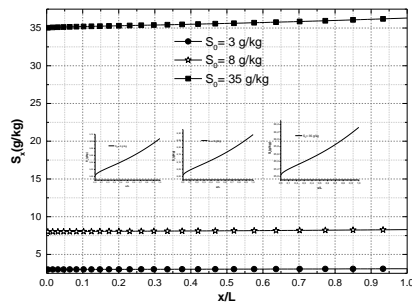


Figure 5: Salinity variation along the tube.

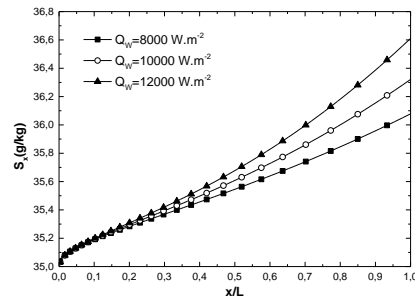


Figure 6: Influence of Q_w on the salinity along the tube.

The variations of the salinity along the tube for different S_0 are presented in Fig. 5. It is shown that the salinity S_x increases from the inlet to the exit of the tube especially with high salinity. In fact, when increasing the salinity, the evaporation mass flux increases too, which in turn enhances the evaporation process, therefore a thicker δ_x is obtained as seen before in Fig 2.

Figure 6 illustrates the effect of the imposed heat flux Q_w on the salinity variation along the tube. The overall analysis of this figure reveals that S_x is increased along the tube by increasing Q_w . This result can be justified by the fact that increasing Q_w leads to extra energy stored in the saltwater liquid film which enhances heat transfer along the tube. Consequently, the amount of evaporated

saltwater increases. This explains why evaporation is enhanced for a bigger Q_w .

5. Conclusion

The evaporation of saltwater liquid film inside a vertical tube is examined numerically. The external wall of the tube is subjected to a uniform heat flux. The effects of salinity on the combined heat and mass transfer during the mechanism of evaporation are presented and examined. Based on the numerical results obtained, it is observed that as salinity increases, evaporation is improved. Finally, the evaporation mechanism is enhanced for a greater imposed heat flux.

Références

- [1] Kang, H., Yang, Y., Chang, Z., Zheng, H., & Duan, Z. (2014). Performance of a two-stage multi-effect desalination system based on humidification–dehumidification process. *Desalination*, 344, 339-349.
- [2] Najim, M., Charef, A., & Meftah, H. (2017). Computational study of saline water film evaporation in a vertical tube. *Desalination*, 408, 81-91.
- [3] Abraham, R., & Mani, A. (2015). Heat transfer characteristics in horizontal tube bundles for falling film evaporation in multi-effect desalination system. *Desalina*, 375, 129–137.
- [4] Feddaoui, M., Mir, A., & Belahmidi, E. (2003). Cocurrent turbulent mixed convection heat and mass transfer in falling film of water inside a vertical heated tube. *International Journal of Heat and Mass Transfer*, 46(18), 3497-3509.
- [5] Orfi, J., Debbissi, C., Mohamed, A. B., & Nasrallah, S. B. (2004). Air humidification by free convection in a vertical channel. *Desalination*, 168, 161-168.
- [6] Qi, C. H., Feng, H. J., Lv, H. Q., & Miao, C. (2016). Numerical and experimental research on the heat transfer of seawater desalination with liquid film outside elliptical tube. *International Journal of Heat and Mass Transf*, 93, 207-216.
- [7] Yang, L., & Shen, S. (2008). Experimental study of falling film evaporation heat transfer outside horizontal tubes. *Desalin*, 220, 654-660.
- [8] Liu, B., Wang, X., Liu, Y., Gao, Y., Ma, Z., & Xue, J. (2020). Simulation analysis of flow velocity and liquid film of saline wastewater in falling film evaporation. *Environmental Technology & Innovation*, 19, 100790.
- [9] Shen, Y., Shi, C., Zhang, L., Bu, S., Xu, W., Zhang, Z., & Liu, X. (2021). Falling film evaporation in a vertical tube with a new type of liquid distributor designed using the brachistochrone principle. *Vacuum*, 187, 110023.
- [10] Kouhikamali, R., Abadi, S. N. R., & Hassani, M. (2014). Numerical investigation of falling film evaporation of multi-effect desalination plant. *Applied thermal engineering*, 70, 477-485.
- [11] Groff, M. K., Ormiston, S. J., & Soliman, H. M. (2007). Numerical solution of film condensation from turbulent flow of vapor–gas mixtures in vertical tubes. *International Journal of Heat and Mass Transfer*, 50(19-20), 3899-3912.
- [12] Patankar, S.V., *Numerical heat transfer and fluid flow*, Hemisphere/McGraw-Hill, New York, 1980.

Evaluation expérimentale de l'effet du taux d'eau de gâchage sur le comportement thermique du plâtre

M. TOUIL¹, O. RAHMOUN¹, A. LACHHEB¹, M. DLIMI¹, A. CHAREF¹, R. SAADANI¹

1. Laboratoire d'étude des matériaux avancés et applications, FSM-ESTM, Université Moulay Ismail, Meknès, Maroc.
Email: mohamed.touil.as@gmail.com

Résumé:

L'industrie du bâtiment est l'un des secteurs les plus énergivores au monde. Par conséquent, le perfectionnement des méthodes de production des matériaux conventionnels apparaît comme l'une des solutions efficaces pour remonter à ce défi. Le gypse est considéré comme l'un des matériaux les plus utilisés dans la construction en raison de ses propriétés avantageuses. C'est dans ce contexte le présent article comprend dans son intégralité une étude expérimentale visant à évaluer l'effet du taux d'eau de gâchage sur ses propriétés et son comportement thermique. Les résultats obtenus ont démontré que l'augmentation de la quantité d'eau permet d'améliorer les caractéristiques thermo-physiques et de maximiser la capacité d'isolation de la matrice plâtreuse.

Mots clés: Plâtre; Eau de gâchage; Propriétés thermo-physiques; Comportement thermique.

1 Introduction

Le bâtiment est un domaine qui interfère directement avec l'environnement extérieure. Par conséquent, il consomme une part importante de l'énergie pour maintenir le confort thermique des occupants [1]. Dans ce contexte, la recherche et le développement de nouvelles méthodes de production des matériaux à faible consommation est devenu l'un des principaux objectifs de la stratégie de construction durable.

Le gypse est considéré comme l'un des matériaux conventionnels largement utilisé dans l'industrie de la construction depuis l'antiquité dans de nombreuses applications de la décoration, grâce à ses propriétés avantageuses par rapport à d'autres matériaux, telles que sa faible consommation d'énergie, ses propriétés thermo-physiques, sa résistance au feu, et également son aspect esthétique [2] [3]. Cependant, d'un point de vue thermique, malgré ses propriétés efficaces en matière d'isolation et de régulation de l'humidité. Il reste un matériau affectant l'efficacité énergétique. Ceci implique la nécessité d'améliorer les méthodes et les conditions de sa fabrication.

Par ailleurs, l'étude des qualités thermiques de la matrice de plâtre est indispensable avant sa mise en œuvre dans le bâtiment, afin de mesurer ses performances par rapport aux modèles déjà existants. L'objectif principale de la présente étude est d'examiner expérimentalement l'effet de deux dif-

férents rapports d'eau de gâchage (50 et 70%) sur ses propriétés thermo-physiques et son comportement thermique.

2 Matériaux et méthodes

2.1 Matrice en plâtre

Le matériau utilisé dans cette expérience est un plâtre extrait des roches situées dans les environs de la ville de Safi, Maroc. C'est un composé poreux de sulfate de calcium dihydraté, fabriqué industriellement à partir du gypse, dont la formule chimique est $CaSO_4 \cdot 2H_2O$. Lorsque ce dernier est chauffé à $100^\circ C$, il perd sa teneur en eau et se transforme en sulfate de calcium hémihydraté $2CaSO_4 \cdot H_2O$. La solubilité du plâtre dans l'eau permet de produire des pièces solides et dures.

2.2 Préparation des échantillons

Les échantillons développés sont formés à partir de gypse préparé avec deux différents taux d'eau de gâchage (E/P). Le premier est préparé avec une consistance plus sèche de 50% (P:0.5) et un autre formulé avec une proportion standard de 70% (P:0.7) (Figure 1).



Figure 1: Echantillons préparés.

Le processus de fabrication est basé sur plusieurs étapes. Tout d'abord, la quantité d'eau nécessaire au mélange est ajoutée dans un récipient, suivie d'une lente répartition de la poudre de plâtre à la surface de l'eau. Ensuite, un malaxage manuel de la pâte est effectué jusqu'à l'obtention d'un mélange homogène. Ensuite, ce dernier est versé dans des moules de dimensions $25 \times 25 \times 4 \text{ cm}^3$, puis il subit un séchage forcé dans une étuve à une température de $50 \pm 1^\circ C$ jusqu'à enlever toute la

teneur en eau présente dans sa structure. Ainsi, les pièces fabriquées sont conservées dans des conditions ambiantes pendant 28 jours avant la caractérisation thermo-physique.

2.3 Instrumentation expérimentale

La mesure de la conductivité et l'évaluation du comportement thermique des échantillons sont identifiés à l'aide d'une maison modèle très isolée de dimensions $40*40*40 \text{ mm}^3$ (Figure 2).

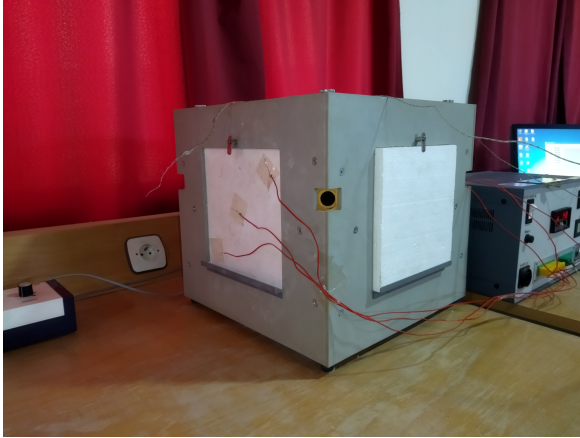


Figure 2: Equipement expérimental.

Cette boîte est équipée de murs latéraux remplaçables avec une ouverture carrée sur chaque côté du mur de $21*21 \text{ mm}^2$ pour permettre l'étude de plusieurs type de matériaux. En effet, les plâtres étudiés ont été fixés dans la face intérieure d'une manière à sceller l'ouverture. La température à l'intérieur a été augmentée au moyen d'une ampoule électrique de 100 W reliée à un thermostat à régulation thermique, produisant en régime permanent une température interne d'environ $50 \text{ }^\circ\text{C}$. Après plusieurs heures, le flux thermique était stationnaire, et la température de la chambre (T_b), des faces interne et externe de l'échantillon (T_{in}) et (T_{ex}), et celle du laboratoire (T_l) ont été mesurées à l'aide de thermocouples. La conductivité thermique a été calculé à l'aide la loi de Fourier:

$$\lambda = \frac{P}{S} \frac{e}{(T_{in} - T_{ex})} \quad (1)$$

Où S et e sont respectivement la surface et l'épaisseur de l'échantillon en m.

la valeur P/S est déterminée à travers la moyenne des flux thermiques surfaciques d'air à l'intérieur et à l'extérieur de la boîte à travers les formules 2 et 3:

$$\left(\frac{P}{S}\right)_{in} = h(T_b - T_{in}) \quad (2)$$

$$\left(\frac{P}{S}\right)_{ex} = h(T_{ex} - T_l) \quad (3)$$

Où h est le coefficient de transfert de chaleur dans le cas d'un mouvement naturel de l'air en contact avec une parois en $\text{W}/\text{m}^2\text{K}$ [4].

2.4 Fiabilité du processus expérimental

Avant de commencer le processus de prise de mesures, il est nécessaire de vérifier l'ampleur de la validité de la méthodologie expérimentale. Pour ce faire, un test a été effectué sur un échantillon commercial standard de bois, qui présente des propriétés thermo-physiques bien connues. Le tableau 1 présente une comparaison entre sa conductivité thermique standard et celle mesurée expérimentalement. En effet, les résultats sont approximativement les mêmes, ce qui montre la fiabilité de la cellule de mesure.

Matériau	Conductivité standard λ [$\text{W}/\text{m}\cdot\text{K}$]	Conductivité expérimentale λ [$\text{W}/\text{m}\cdot\text{K}$]
Bois	0.12	0.14

Tab. 1: Comparaison entre la conductivité thermique standard de l'échantillon commercial (bois) et sa mesure expérimentale.

3 Résultats et discussion

3.1 Propriétés thermo-physiques

D'après l'analyse des résultats de la figure 3, on constate que l'augmentation de la quantité d'eau de gâchage de 50 à 70% entraîne une diminution de la conductivité thermique de 0.401 à $0.38 \text{ W}/\text{m}\cdot\text{K}$, et une réduction de sa densité de 1407 à $1162 \text{ kg}/\text{m}^3$. Ainsi, une matrice en plâtre préparée avec un ratio de mélange élevé possède une bonne capacité d'isolation et de retardement de la transmission de chaleur dans sa structure. Elle offre également un gain en légèreté qui permettra de réduire le poids des bâtiments après son implantation [5].

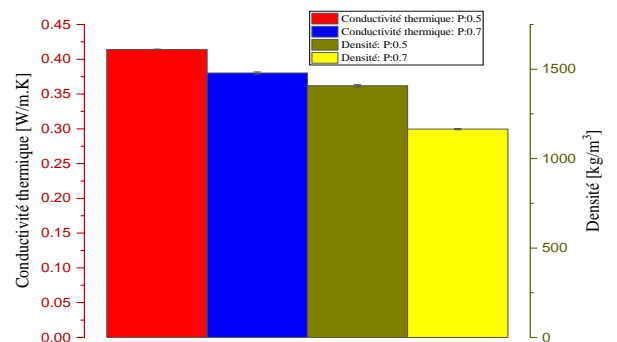


Figure 3: Propriétés thermo-physiques en fonction du taux de l'eau de gâchage.

3.2 Comportement thermique

L'étude de la performance thermique des matériaux de construction est nécessaire pour évaluer leur comportement dans un environnement réel.

La figure 4 représente l'évolution de la température de la face extérieure des échantillons développés sous une source de chaleur uniforme de 100 W à l'intérieur de la chambre témoin.

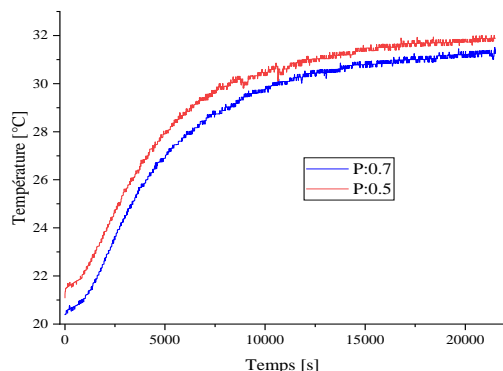


Figure 4: Evolution de la température extérieure de l'échantillon en fonction du type de plâtre.

En effet, le modèle P:0.7 nécessite relativement plus de temps pour atteindre le régime permanent (5h et 12 min). Alors que la configuration P:0.5 favorise un transfert de chaleur plus important et permet d'atteindre l'équilibre thermique en 4 heures et 22 minutes. Cela confirme l'effet apporté par les quantités élevées de l'eau de gâchage sur la capacité du plâtre à bloquer le passage de la chaleur.

La figure 5 représente la réponse thermique de la face extérieure par rapport à la température de la surface intérieure.

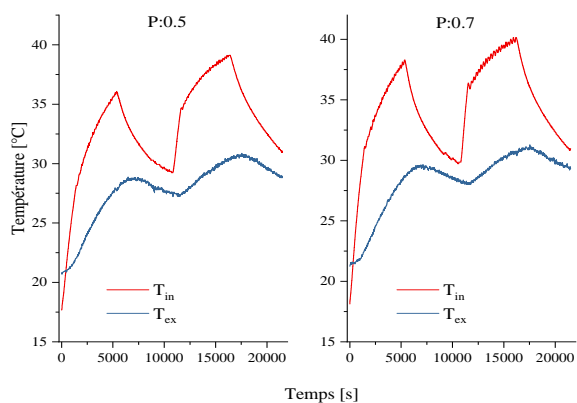


Figure 5: Le déphasage thermique en fonction du type de plâtre.

D'après son examen, il apparaît que le déphasage du flux de chaleur est plus élevé pour le plâtre produit avec une consistance standard. Il atteint ainsi une valeur moyenne de 1640 s par rapport au modèle fabriqué avec la consistance sèche, qui présente un décalage temporel moyen de 1152 s. Ces résultats peuvent être expliqués par l'effet de la diminution de la conductivité thermique.

4 Conclusion

Cet article présente une solution simple et alternative pour le développement des propriétés thermiques du plâtre, via l'augmentation du taux d'eau nécessaire pour le gâchage de la poudre de gypse. Toutefois, les résultats obtenus ont révélé que ce procédé permet de réduire la conductivité thermique et la légèreté du plâtre, ainsi que de lui offrir la possibilité de retarder la diffusion de la chaleur dans sa propre structure.

References

- [1] National Agency for the Development of Renewable Energy and Energy Efficiency (ADEREE), *Thermal regulation of the construction in Morocco*, Morocco.
- [2] T. Freire, A. Santos Silva, M. R. Veiga and J. de Brito., *The 19th century revivalisms in the Portuguese architecture: Characterization of the interior plaster coatings of Monserrate Palace in Sintra*, Proc. 4th Hist. Mortars Conf. HMC2016. (2016) 187-194.
- [3] M. Arıkan and K. Sobolev, *The optimization of a gypsum-based composite material*, Cement and Concrete Research 32 (2002) 1725-1728.
- [4] N. F. Medina and M. M. Barbero-Barrera , *Mechanical and physical enhancement of gypsum composites through a synergic work of polypropylene fiber and recycled isostatic graphite filler*, Construction and Building Materials 131 (2017) 165-177.
- [5] S. Gutiérrez-González, J. Gadea, A. Rodríguez, C. Junco, and V. Calderón, *Lightweight plaster materials with enhanced thermal properties made with polyurethane foam wastes*, Construction and Building Materials 28 (2011) 653-658.

Amélioration du transfert de masse à une interface liquide/solide à l'aide d'écoulements générés par des ultrasons

V. BOTTON^{1,2}, S. MIRALLES¹, N. EL GHANI¹, S. MARCELIN², B. TER-OVANEISSIAN²,
H. BEN HADID¹, D. HENRY¹

1. Laboratoire de Mécanique des Fluides et d'Acoustique, CNRS/Université de Lyon, Ecole Centrale de Lyon / Université Lyon 1 / INSA Lyon, ECL, 36 Avenue Guy de Collongue, 69134 Ecully Cedex, France, valery.botton@insa-lyon.fr
2. Laboratoire MATEIS UMR CNRS 5510, Université Lyon 1 / INSA Lyon, Bat L. de Vinci, 21 Avenue Jean Capelle, 69621 Villeurbanne cedex, France.

Abstract

The approach developed in this work is based on water experiments in which we introduce electrochemical techniques in order to quantify and understand the mass transfer phenomena at the wall and their improvement by acoustic streaming. Measurements are performed in a glass tank containing a transparent electrolyte. Acoustic waves are used to generate a steady flow: the acoustic beam is directed towards a wall on which several electrodes are set. The beam impinges the wall with an angle and it is reflected on this wall. This results in an original quasi-steady jet flow presented in a former paper [2]. The electrodes enable us to characterize the mass transfer by diffusion through the ions concentration boundary layer. We experimentally and numerically observe that the mass flux is greatly improved by the convective flow induced by the ultrasound beam. We propose an order of magnitude model to explain the observed scaling law. All this is presented with more details in our recent article [4].

Mots clefs : *Transfert de masse en paroi, Acoustic Streaming, Electrochimie, analyse d'échelle.*

1. Introduction

Dans le cadre de l'exploitation de l'énergie solaire par l'intermédiaire des cellules photovoltaïques, la croissance de la production du Silicium Photovoltaïque (Si-PV) rend l'innovation essentielle pour, d'une part, mettre en place des techniques efficaces pour la purification du Si-PV, et d'autre part, améliorer les moyens de modélisation de ces procédés trop lourds et coûteux. Un des problèmes majeurs rencontrés lors de la production à grande échelle de Si-PV est le contrôle de la concentration en dopants dans le silicium lors de la phase de solidification, afin de garantir une meilleure homogénéité du matériau fini. La mise au point de techniques de brassage est une solution envisagée pour évacuer les impuretés accumulées au front de solidification et les répartir de manière homogène, ce

qui permettra la production de lingots de Si-PV de grandes dimensions et à des coûts réduits.

Dans ce travail, on s'intéresse particulièrement au brassage par ultrasons, car cette technique a l'avantage d'être non intrusive et localisée. Le brassage par ultrasons consiste à générer un écoulement *via* la propagation d'une onde ultrasonore au sein du fluide, ce phénomène étant plus connu sous le terme anglais *Acoustic Streaming*. Des travaux récents en eau [1, 2, 3], ont permis de mieux comprendre le comportement de ce type d'écoulement. Outre les caractéristiques de l'écoulement, il est important d'estimer et de modéliser l'influence de l'*Acoustic streaming* sur le transfert de masse à l'interface solide/liquide, qui est la grandeur la plus pertinente en matière de ségrégation. Dans ce travail, nous nous intéressons donc à mettre en place un démonstrateur en eau, dans lequel des mesures électrochimiques seront réalisées pour mettre en évidence le transfert de masse à la paroi en présence d'ultrasons.

2. Dispositif expérimental

Nous nous intéressons à caractériser le phénomène de transfert de masse par des méthodes électrochimiques. En effet, le transfert de masse par diffusion à travers la couche limite des impuretés contenues dans le Si-PV est analogue au transfert de masse par diffusion des ions dans une cellule électrochimique (figure 1).

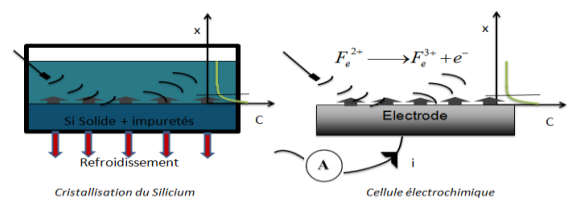


Figure 1: Analogie entre le phénomène de ségrégation des impuretés lors de la solidification du Si-PV (à gauche) et le transfert d'ions dans une cellule électrochimique (à droite)

Le dispositif ASTRID (*Acoustic STReaming Investigation Device*) [1] consiste en une cuve parallélépipédique composée de deux chambres séparées par une paroi

absorbante (figure 2). Une étude expérimentale et numérique précédente nous a apporté une bonne connaissance de l'écoulement dans cette configuration [2]. Dans la première chambre, remplie d'eau déminéralisée, est introduite une source acoustique de 29 mm de diamètre nominal, fonctionnant à la fréquence de 2MHz et dont on peut faire varier la puissance de 0 à 8W. La deuxième chambre de l'aquarium est remplie de solution électrolytique (ferri-ferrocyanure de potassium).

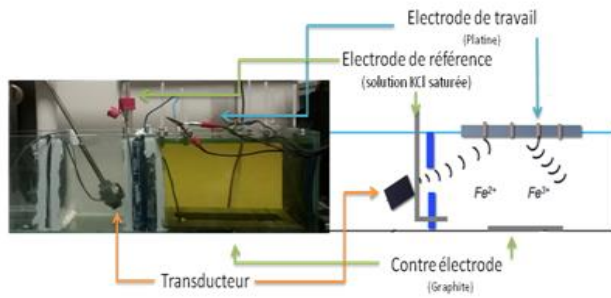


Figure 2: Vue de côté du dispositif expérimental

Des mesures électrochimiques locales sont réalisées par 7 électrodes de travail en platine situées sur la paroi supérieure vers laquelle le faisceau acoustique est dirigé. Cette paroi modélise donc le front de solidification. La contre-électrode en graphite est placée au fond de la cuve et l'électrode de référence en calomel saturé est placée sur une paroi latérale.

Les protocoles électrochimiques consistent à imposer une différence de potentiel entre une électrode de travail et la contre-électrode. Différents phénomènes physiques (diffusion, migration) vont induire des échanges d'électrons et le déplacement des espèces chimiques Fe^{2+}/Fe^{3+} . La mesure du courant dans différents protocoles expérimentaux électrochimiques (potentiocinétique, SIE, ...) permet de caractériser les phénomènes physiques qui se développent dans la cellule électrochimique.

3. Résultats expérimentaux typiques

La potentiocinétique consiste à appliquer à l'électrode de travail une augmentation linéaire du potentiel avec le temps. Pour chaque potentiel (E), le courant (i) à l'électrode est mesuré. Cette méthode électrochimique permet d'obtenir des informations sur les propriétés de diffusion des espèces électrochimiques et sur la cinétique des réactions transitoires.

Le tracé des courbes de polarisation $i(E)$ permet de mettre en évidence des plateaux liés à la diffusion des espèces électroactives (Fe^{2+} ou Fe^{3+}). La présence du plateau correspond à une limitation diffusionnelle, c'est-à-dire que tous les ions qui arrivent à la surface de l'électrode de travail sont instantanément consommés.

Dans ce régime, on mesure le courant limite (Cf. valeurs plateaux sur la figure 3), qui est donc directement lié au flux des ions à travers la couche limite de concentration.

L'amélioration du transfert de masse par rapport au cas diffusif pur sera alors mesuré par le nombre de Sherwood suivant :

$$Sh = \frac{I_{lim}}{I_{lim,diff}} \quad (1)$$

où I_{lim} est le courant limite observé en présence de convection forcée par les ultrasons et $I_{lim,diff}$ le courant limite que l'on aurait en l'absence de convection.

On trace donc les courbes intensité-potentiel au niveau de chaque électrode pour différentes puissances acoustiques. La figure 3 représente un résultat typique pour une électrode.

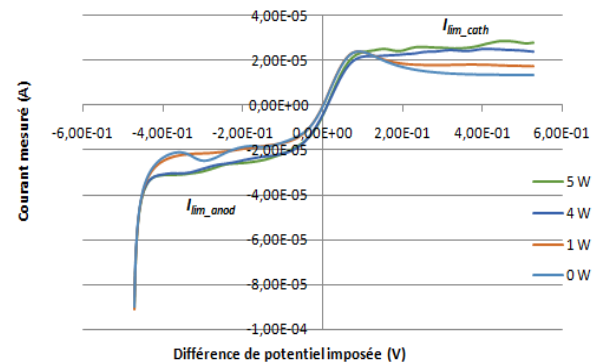


Figure 3 : Intensité mesurée en fonction de la différence de potentiel appliquée à une électrode, lors d'un essai potentiocinétique pour différentes puissances acoustiques (vitesse de balayage = 2 mV/s).

L'augmentation observée du courant limite lorsqu'on augmente la puissance traduit l'amélioration du transfert de masse par l'*Acoustic streaming*. Un exemple typique pour une électrode est représenté sur la figure 4.

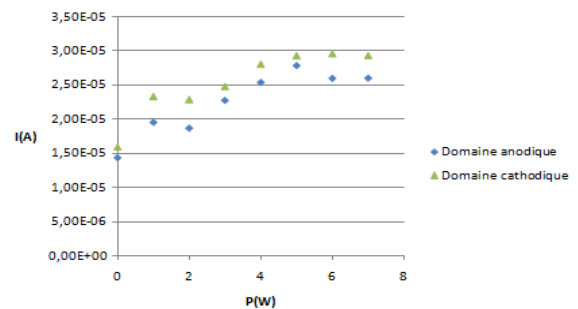


Figure 4: Exemple typique de l'influence de la puissance acoustique sur le courant limite pour une électrode.

4. Modélisation numérique et analyse d'échelles

Un modèle numérique de cette expérience a été réalisé à l'aide du logiciel StarCCM+™. Dans ce modèle, pour rendre compte de la présence du faisceau acoustique, on ajoute un terme de forçage dans les équations de Navier-Stokes pour un fluide incompressible. Ce terme de forçage est calculé sous Matlab™ par un modèle simple de propagation acoustique linéaire prenant en compte la diffraction et l'atténuation du son dans l'eau. La concentration en ions est considérée comme un scalaire

passif qui suit une équation de convection-diffusion. Un des intérêts de cette approche numérique est qu'elle donne accès dans le même temps au transfert de masse à la paroi, et aux différents champs caractérisant l'hydrodynamique de ce problème, dont en particulier la contrainte de cisaillement sur la paroi. Un résultat typique est donné sur la figure 5 et illustre la corrélation entre le champ de concentration (figure 5b) et le champ de contrainte (figure 5a) au niveau de la paroi où sont positionnées les différentes électrodes numérotées de E0 à E7.

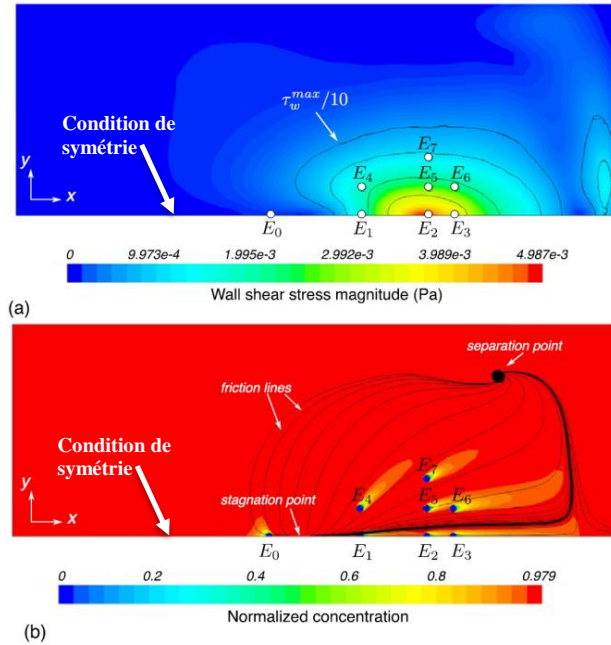


Figure 5: Résultat typique du modèle numérique, correspondant à une puissance acoustique de 1,099 W. (a) Isovaleurs de la contrainte visqueuse sur la paroi. (b) Isovaleurs de la concentration normalisée sur la même paroi. Les 7 électrodes sont représentées et numérotées de E1 à E7. (Figure adaptée de [4])

Cette corrélation entre transfert de masse et transfert de quantité de mouvement est bien connue des expérimentateurs utilisant la polarimétrie, méthode électrochimique permettant une mesure indirecte des contraintes pariétales. Ces derniers utilisent notamment la relation de Leveque pour relier le courant mesuré à la contrainte pariétale [4]. En nous basant d'une part sur une estimation en ordre de grandeur de l'évolution de la contrainte en fonction de la puissance acoustique et d'autre part sur la loi de Leveque, nous avons développé un modèle prédisant le taux de transfert de masse à la paroi en fonction de la puissance acoustique. Il prédit que le nombre de Sherwood, Sh , doit être dans un intervalle $[0,1^{1/3}Sh^0; Sh^0]$ (figure 6), avec Sh^0 donné par la relation suivante :

$$Sh^0 = a (Sc Gr_{ac})^{1/3}, \quad (2)$$

où a est une constante connue dépendant de la géométrie (incidence, diamètre et longueur du faisceau, surface de l'électrode), Sc , le nombre de Schmidt des ions réactifs

dans la solution vaut 1200, et Gr_{ac} est le nombre de Grashof acoustique. Comme pour la convection naturelle, il compare les effets de la force volumique à ceux de la viscosité :

$$Gr_{ac} = \frac{32}{\pi} \frac{\alpha}{\rho c \nu^2} d_s P_{ac},$$

où α et c sont respectivement le coefficient d'atténuation et la célérité du son dans l'eau, ρ et ν sont la masse volumique et la viscosité cinématique de l'eau, et enfin d_s et P_{ac} sont le diamètre et la puissance de la source acoustique.

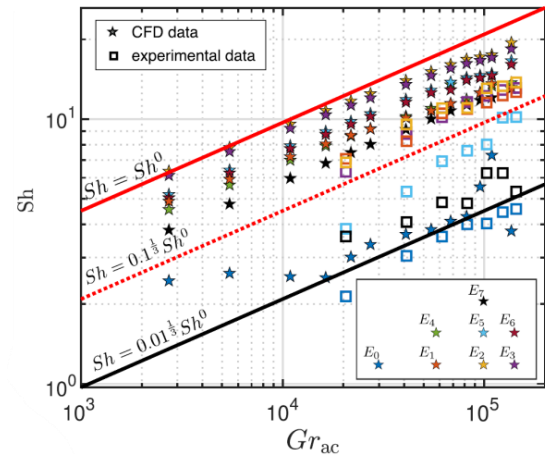


Figure 6: Comparaison des données obtenues numériquement (étoiles) et expérimentalement (carrés) à l'expression obtenue par une analyse en ordre de grandeur (lignes). Notez que $0,1^{1/3} \sim 1/2$ et $0,01^{1/3} \sim 1/5$ (Figure issue de [4]).

5. Conclusion

Ces mesures mettent bien en évidence l'amélioration significative qu'apporte l'*Acoustic streaming* aux transferts de masse à la paroi. La complémentarité des approches expérimentale, numérique et par analyse d'échelle permet d'avoir un tableau cohérent, même si des différences persistent.

Références

- [1] B. Moujded, *Caractérisation expérimentale et théorique des écoulements entraînés par ultrasons*, Thèse, INSA Lyon, 2013.
- [2] B. Moudjed *et al.*, *Y-shaped jets driven by an ultrasonic beam reflecting on a wall*, *Ultrasonics*, 68 (2016), pp. 33-42.
- [3] B. Moujded *et al.*, *Scaling and dimensional analysis of acoustic streaming jets*, *Physics of Fluids*, 26 (2014), 093602.
- [4] N. El Ghani, S. Miralles, V. Botton, D. Henry, H. Ben Hadid, B. Ter-Ovanesian, S. Marcellin, *Acoustic streaming enhanced mass transfer at a wall*, *International Journal of Heat and Mass Transfer*, 172 (2021), 121090.

EFFET DES PARAMETRES EXTERNES SUR LES PERFORMANCES ELECTRIQUES D'UN GENERATEUR PHOTOVOLTAÏQUE

M. ER-RAKI¹, M. HASNAOUI², S. HASNAOUI², F.Z. AMATOUL¹

⁽¹⁾ Ecole Supérieure de Technologie, Essaouira, Université Cadi Ayyad, Maroc.

m.erraki@uca.ac.ma ; fz.amatoul@uca.ma

⁽²⁾ LMFE, FSSM, Unité affiliée au CNRST (URL-CNRST No 16), Université Cadi Ayyad, Marrakech, Maroc.

hasnaoui@uca.ac.ma; safae.hasnaoui@edu.uca.ac.ma

Résumé

Lorsqu'on parle d'énergie photovoltaïque, on doit garder à l'esprit les différents facteurs, tant internes qu'externes, qui peuvent affecter la génération optimale des modules. C'est dans ce contexte que s'inscrit cette étude qui propose un test expérimental sur l'effet de l'éclairement, de la température et de l'ombrage sur les performances électriques d'un générateur photovoltaïque. Les résultats de cette étude ont montré que, contrairement à l'éclairement, qui affecte positivement les performances des panneaux photovoltaïques, la température et l'ombrage ont un effet néfaste sur ceux-ci.

Mots clés: Conversion photovoltaïque, Eclairement, Température, Ombrage, Rendement.

1. Introduction

Le photovoltaïque est une technologie permettant de convertir l'énergie solaire en énergie électrique par l'intermédiaire de cellules photovoltaïques, disposées en modules appelés panneaux photovoltaïques. Après un départ très modeste, cette technologie a connu récemment une évolution très forte, ce qui a engendré une forte augmentation de la demande pour ce type d'énergie pour de nombreuses applications [1-3]. Cette évolution a été rendue possible grâce aux recherches variées et continues menées dans le domaine des matériaux photovoltaïques [4], sur les algorithmes MPPT [5], mais aussi sur le contrôle des facteurs pouvant affecter la performance du système photovoltaïque [6-8]. En s'inspirant de ces études et bien d'autres, il a été prouvé que le photovoltaïque réagit de manière sensible à de nombreux paramètres aussi bien internes qu'externes.

Dans le présent travail, on s'intéresse plus particulièrement à l'étude expérimentale de l'effet de quelques paramètres externes sur la productivité électrique d'un générateur photovoltaïque. Particulièrement, la présente étude est focalisée sur l'étude de l'effet de l'ensoleillement, de la température de fonctionnement et de l'ombrage sur les performances électriques d'un panneau photovoltaïque. Les résultats obtenus lors des tests expérimentaux sont présentés et discutés.

2. Partie théorique

Un photo-générateur, ou photopile, souvent appelé cellule photovoltaïque, est un composant électronique qui produit de l'électricité une fois exposé à la lumière grâce à l'effet photoélectrique. Son schéma électrique équivalent, illustré par la figure 1, est constitué principalement par une diode, modélisant la jonction PN et montée en parallèle avec une source de courant modélisant la conversion du flux lumineux en énergie électrique. La résistance shunt, R_{sh} , montée parallèlement à la diode correspond aux pertes directes à travers la jonction. La résistance R_s , correspond entre autres aux pertes engendrées par la résistivité des couches [9].

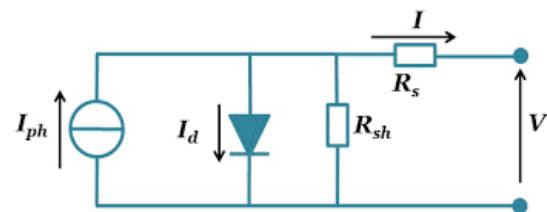


Figure 1: Schéma équivalent d'un photo-générateur

Aux deux bornes du photo-générateur, l'énergie électrique produite est récupérée sous forme d'une tension V et d'un courant I , couplés de manière non linéaire comme indiqué par l'équation suivante [10] :

$$I = I_{ph} - I_s \left(e^{\frac{q(V+R_s I)}{nKT}} - 1 \right) - \frac{(V+R_s I)}{R_{sh}} \quad (1)$$

Les paramètres I_{ph} , I_s , q , n , K et T désignent respectivement le photo-courant, le courant de saturation de la diode, la charge élémentaire, le facteur d'idéalité de la diode, la constante de Boltzmann et la température effective des cellules.

Le photo-courant I_{ph} varie en fonction de l'ensoleillement, G , et de la température de fonctionnement de la cellule, T , selon l'équation suivante :

$$I_{ph} = \frac{G}{G_{ref}} \left[I_{ph_{ref}} + C_T (T - T_{ref}) \right] \quad (2)$$

Où G_{ref} , $I_{ph_{ref}}$ et T_{ref} sont respectivement l'irradiation solaire, la température de référence et le photo-courant dans

les conditions STC ($T=25\text{ }^{\circ}\text{C}$, $G = 1000\text{ W/m}^2$ et AM 1.5). C_T étant le coefficient de variation du courant en fonction de la température (donné dans la fiche technique du panneau).

De point de vue efficacité, les performances d'un panneau photovoltaïque sont évaluées via son facteur de forme, noté $FF(\%)$, défini par :

$$FF(\%) = \frac{P_{max}}{V_{co} \times I_{cc}} \quad (3)$$

Dans l'équation précédente, P_{max} représente la puissance maximale générée par le panneau, V_{co} la tension en circuit ouvert et I_{cc} le courant de court-circuit du panneau.

En plus du facteur de forme, la capacité de production électrique d'une cellule PV peut également être estimée à travers son rendement défini comme étant le rapport de la puissance électrique maximale fournie et la puissance lumineuse incidente :

$$\eta(\%) = \frac{P_{max}}{E \times A} \quad (4)$$

Où E est l'éclairement solaire incident en W/m^2 et A est la surface du module PV en m^2 .

Le rendement d'un générateur photovoltaïque varie en fonction de sa température de fonctionnement comme suit :

$$\eta = \eta_{ref} [1 - C_T (T - T_{ref})] \quad (5)$$

Où η_{ref} est le rendement de référence du module sous les conditions standards STC.

3. Diapositif expérimental

La maquette expérimentale est composée d'un panneau photovoltaïque au silicium poly-cristallin dont les caractéristiques électriques sont détaillées dans le tableau 1.

Tableau 1: Caractéristiques du panneau étudié.

Type de cellules	Polycristalline
Nom	TESLA SOLAR
P_{mpp}	250Wc
U_{mpp}	30,3V
I_{mpp}	8,26 A
V_{CO}	37,4 V
I_{CC}	8,65 A
η (%)	15,5%

En face du panneau photovoltaïque sont exposés des lampes halogènes qui génèrent un éclairage uniforme sur le panneau. La partie mesure nécessite en outre un rhéostat, un multimètre et un thermocouple. Quant à la partie ombrage, un cache permettant de régler l'ombrage à des taux différents a été utilisé.

4. Résultats et discussions

Dans cette section, les principaux résultats obtenus lors des tests expérimentaux sont discutés. Ils portent sur

l'influence des paramètres externes (taux d'irradiation, température et ombrage) sur les performances électriques d'un panneau photovoltaïque.

Les variations des caractéristiques (I-V) et (P-V) du module sont illustrées sur la figure 2 pour une température fixée à $T=25\text{ }^{\circ}\text{C}$ et différentes valeurs de l'éclairement. On note que le courant de court-circuit croît de manière proportionnelle à l'éclairement (avec un coefficient de proportionnalité de l'ordre de 0.0086) alors que la tension est peu sensible aux variations de ce dernier. L'effet positif du taux d'insolation sur la puissance électrique du panneau est manifeste ($P_{max} \propto E$, et le coefficient de proportionnalité est de l'ordre de 0.2).

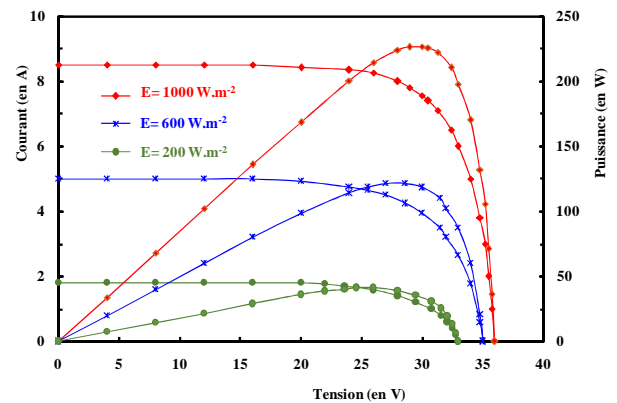


Figure 2: Caractéristiques électriques du module pour différentes valeurs de l'éclairement.

La figure 3 montre l'influence de la température sur les caractéristiques électrique du module. L'examen des résultats présentés montre que la température de la cellule a un effet néfaste sur les caractéristiques photovoltaïques et contrôle la qualité et les performances de la cellule solaire. En effet, le courant augmente légèrement ($+0.05\text{ } \%/^{\circ}\text{C}$) à mesure que la température augmente. En contrepartie, la température affecte négativement la tension du circuit ouvert. En augmentant la température, la tension du circuit ouvert subit une diminution de $0.3\text{ } \%/^{\circ}\text{C}$. Par conséquent, la puissance maximale du générateur chute de manière non négligeable ($-0.4\text{ } \%/^{\circ}\text{C}$).

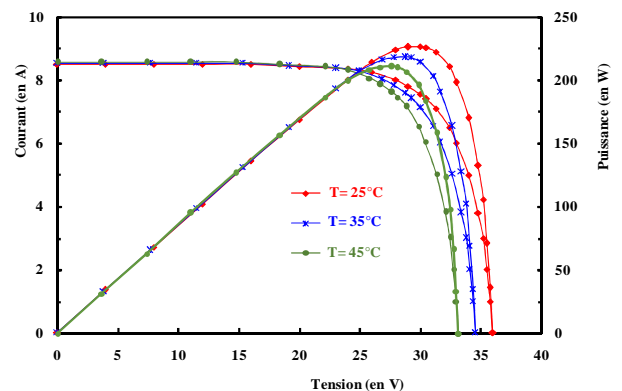


Figure 3: Caractéristiques électriques du module pour différentes valeurs de la température.

L'effet de l'ombrage sur les caractéristiques (I-V) et (P-V) est illustré sur la figure 4. Le module est placé dans des conditions d'ombrage uniforme, à différents taux et sous un éclairage et une température fixés. On peut noter une diminution du courant de court-circuit qui accompagne une augmentation du pourcentage d'ombrage, où il a subi une diminution de 28.6 % de sa valeur initiale, pour un ombrage de 25 % du module. Par conséquent, la puissance de sortie diminue également avec l'augmentation de l'ombrage. Dans des conditions d'ombrage uniforme, toutes les cellules solaires du panneau reçoivent la même quantité d'insolation pour un taux d'ombrage donné. Par conséquent, la réduction résultante de la lumière solaire se produit de manière uniforme pour toutes les cellules. De ce fait, le courant de court-circuit, qui est directement proportionnel à l'insolation reçue, est affecté en conséquence avec le taux d'ombrage.

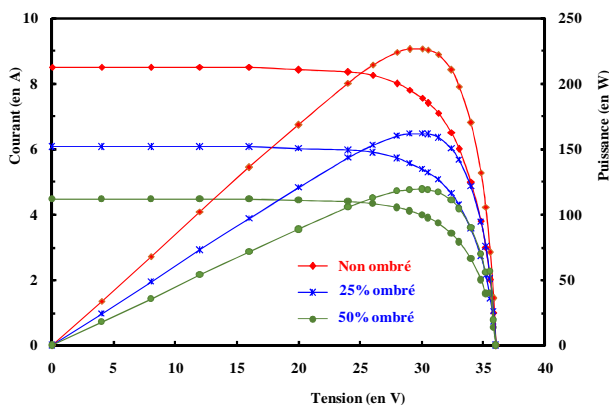


Figure 4: Caractéristiques électriques du module pour différents taux d'ombrage.

5. Conclusion

Les résultats de cette étude ont montré que l'amélioration des performances électriques d'un photogénérateur passe nécessairement par le contrôle des paramètres (internes et externes) dont dépend le rendement global de la chaîne de conversion photovoltaïque. Outre l'influence de l'ensoleillement et de la nature cristalline du matériau utilisé sur le rendement d'un panneau solaire photovoltaïque, la température de fonctionnement et même l'ombrage ont leurs propres effets à prendre en compte lors du fonctionnement des panneaux photovoltaïques. En termes de valeurs, l'étude a montré que le courant de court-circuit croît proportionnellement à l'éclairage, avec un coefficient de proportionnalité de l'ordre de 0.0086. De même, on a trouvé que toute augmentation de la température des cellules d'un °C conduit à une diminution de la tension V_{co} de 0.3% et de la puissance maximale de 0.4%. Quant à l'effet de l'ombrage, les résultats trouvés ont montré que le courant de court-circuit a subi une diminution de 28.6 % de sa valeur initiale, pour un ombrage de 25 % du module.

Références

- [1] L. Joshi, N.C. Narayanan, J. Venkateswaran, C.S. Solanki, P. Kumar, Adoption of solar photovoltaic lighting in rural India: Role of localization strategy, *Energy and Buildings*, vol. 202, 109370, 2019.
- [2] G. Liu, Sustainable feasibility of solar photovoltaic powered street lighting systems, *International Journal of Electrical Power & Energy Systems*, vol. 56, pp. 168-174, 2014.
- [3] J. Reça, C. Torrente, R. López-Luque, J. Martinez, Feasibility of standalone direct pumping PV system for irrigation in mediterranean greenhouses, *Renewable Energy*, vol. 85, pp. 1143-1154, 2016.
- [4] S. Chander, A. Purohit, A. Sharma, S.P. Nehra, M.S. Dhaka, A study on photovoltaic parameters of monocrystalline silicon solar cell with cell temperature, *Energy Reports*, vol. 1, pp. 104-109, 2015.
- [5] M.F.N. Tajuddin, M.S. Arif, S.M. Ayob, Z. Salam, Perturbative methods for maximum power point tracking (MPPT) of photovoltaic (PV) systems: a review, *International Journal of Energy Research*, vol. 39(9), pp. 1153-1178, 2015.
- [6] F. Zaouia, A. Titaouinea, M. Becherifb, M. Emzianec, A. Abouboua, A combined experimental and simulation study on the effects of irradiance and temperature on photovoltaic modules, *Energy Procedia*, vol. 75, pp. 373-380, 2015.
- [7] F. Belhachat et C. Larbes, Modeling, analysis and comparison of solar photovoltaic array configurations under partial shading conditions, *Solar Energy*, vol. 120, pp. 399-418, 2015.
- [8] G. Shobana, P. Sornadeepika, Dr. R. Ramaprabha, Global Maximum Power Point Tracking of Photovoltaic Array under Partial Shaded Conditions *International Journal of Engineering*, vol. 2(3), pp. 219-223, 2013.
- [9] K. Keshavani, J. Joshi, V. Trivedi, M. Bhavsar, Modelling and Simulation of Photovoltaic Array Using Matlab/Simulink, *International Journal of Engineering Development and Research*, vol. 2(4), pp. 3742-3751, 2014.
- [10] A. Yahfdhou, A. Mahmoud, I. Youm, Modeling and optimization of photovoltaic generator with Matlab/Simulink, *International Journal of I Tech and E Engineering*, vol. 3(4), pp. 108-111, 2013.

MODÉLISATION ET SIMULATION DES TRANSFERTS THERMIQUES À TRAVERS UN MUR TROMBE EN RÉGIME STATIONNAIRE

Z. CHARQUI¹, M. BOUKENDIL¹, L. EL MOUTAOUAKIL¹, R. HIDKI¹, Z. ZRIKEM¹, A. ABDELBAKI¹

1. Université Cadi Ayyad, Faculté des Sciences Semlalia, Département de Physique, LMFE, B.P. 2390, Marrakech, Maroc

m.boukendil@uca.ac.ma

Résumé

Ce travail est consacré à l'étude du transfert de chaleur dans un mur Trombe non ventilé en régime permanent. Les trois modes de transfert de chaleur sont impliqués. À savoir : la conduction, la convection et le rayonnement surfacique. La surface externe du mur massif est soumise à un flux solaire variant de 0 à 1000W/m². Les surfaces verticales extrêmes échangent avec les environnements interne et externe avec des coefficients d'échange globaux h_e et h_i . Les surfaces horizontales sont isolées. Les effets des conductivités du mur massif et le vitrage sur le transfert de chaleur dans une telle structure sont étudiés. Les résultats de simulation montrent que les flux utiles et perdus peuvent être optimisés en optant pour une grande conductivité du mur massif. Quant à la conductivité du vitrage, ces flux n'y sont pas sensibles.

Mots-clés : Convection naturelle laminaire ; Conduction ; Rayonnement surfacique ; Irradiation solaire ; Régime stationnaire ; Mur Trombe ;

1. Introduction

Le mur Trombe est un système solaire passif inventé par Edward Morse dans les années quatre-vingt du dix-neuvième siècle. Il a été conçu principalement pour réduire la charge de chauffage permettant de maintenir le confort thermique dans les habitats pendant les saisons froides. Depuis son apparition, les chercheurs en ont dérivé plusieurs variantes afin d'améliorer ses performances thermiques et de l'adapter aux différents climats du globe. Dans la littérature, on trouve le mur Trombe composite, qui a été développée en vue de minimiser les pertes substantielles de chaleur courantes dans le mur Trombe classique [1]. On y trouve également, avec un grand nombre de publications, le mur Trombe PCM dont la structure est très similaire à celle du mur Trombe classique, sauf qu'il est muni d'un

matériau à changement de phase afin d'améliorer sa capacité de stockage thermique [2]. Pour la même raison, des chercheurs [3] suggèrent de remplacer le béton usuel dont est construit le mur massif par de l'eau en raison de sa grande capacité thermique. On obtient ainsi le mur Trombe à eau. Le mur Trombe PV est aussi une variante à laquelle les chercheurs se sont beaucoup intéressés ces dernières années [4]. Cet intérêt s'explique par le fait qu'elle peut fournir de l'énergie sous forme de chaleur et d'électricité simultanément.

En dépit du développement de toutes ces variantes, il a été montré que le mur Trombe non ventilé fonctionne bien meilleur dans plusieurs cas. De ce fait, on se propose de l'étudier numériquement en analysant son comportement thermique vis-à-vis de l'irradiation solaire qu'il absorbe et les températures auxquelles il est exposé.

2. Modèle physique

Le mur Trombe considéré est schématisé dans la figure 1. Il est constitué d'un mur massif et une vitre, d'épaisseurs respectives $ex_1 = 4\text{mm}$ et $ex_2 = 10\text{cm}$, séparés par une lame d'air de largeur hx et de hauteur H . Les parois verticales échangent par convection et par rayonnement avec les milieux intérieur et extérieur de températures respectives $T_i = 20^\circ\text{C}$ et $T_e = 30^\circ\text{C}$. Les coefficients d'échanges globaux choisis correspondent au climat de la ville de Marrakech. Soient $h_i = 8,3\text{W/m}^2\text{K}$ et $h_e = 17\text{W/m}^2\text{K}$. Quant aux parois horizontales, une condition d'adiabaticité est y imposée. Le rapport de forme de la lame d'air est noté $A = H/hx$. La surface extérieure du mur massif est soumise à un flux solaire net S variant de 0 à 1000W/m².

L'écoulement dans la lame d'air est supposé laminaire et bidimensionnel. Le fluide est Newtonien, incompressible et transparent à l'égard du rayonnement de surface. Les surfaces de la lame d'air sont supposées grises et diffuses avec des émissivités différentes. La dissipation visqueuse est négligeable. Les propriétés thermophysiques de l'air sont indépendantes de la température, sauf la densité

dans le terme de poussée, pour lequel on adopte l'approximation de Boussinesq.

Les équations régissant le mouvement du fluide et la conservation de l'énergie dans la lame d'air, la vitre et le mur massif s'expriment par [3] :

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0 \quad (1)$$

$$U \frac{\partial U}{\partial X} + V \frac{\partial U}{\partial Y} = -\frac{\partial P}{\partial X} + \text{Pr} \left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2} \right) \quad (2)$$

$$U \frac{\partial V}{\partial X} + V \frac{\partial V}{\partial Y} = -\frac{\partial P}{\partial Y} + \text{Pr} \left(\frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2} \right) + \text{RaPr}\theta_f \quad (3)$$

$$U \frac{\partial \theta_f}{\partial X} + V \frac{\partial \theta_f}{\partial Y} = \frac{\partial^2 \theta_f}{\partial X^2} + \frac{\partial^2 \theta_f}{\partial Y^2} \quad (4)$$

$$\frac{\partial^2 \theta_s}{\partial X^2} + \frac{\partial^2 \theta_s}{\partial Y^2} = 0 \quad (5)$$

Où Eq. (1) est l'équation de continuité, Eq. (2) et Eq. (3) sont respectivement les équations de la quantité de mouvement suivant les directions X et Y, Eq. (4) et Eq. (5) sont respectivement les équations de la conservation de l'énergie dans les milieux fluides et solides.

Le flux radiatif net $Q_{r,k}(r_k)$ échangé par l'élément de surface dS_k , d'émissivité ϵ_k et situé à la position r_k sur la surface k est déterminé par la méthode des radiosités qui consiste à résoudre les deux systèmes d'équations suivants [3] :

$$J_k(r_k) = \epsilon_k \left(1 + \frac{\theta_k(r_k)}{\theta_{\text{ref}}} \right)^4 + (1 - \epsilon_k) \sum_{j=1}^4 \int_{S_j} J_j(r_j) dF_{dS_k-dS_j} \quad (6)$$

$$Q_k(r_k) = J_k(r_k) - \sum_{j=1}^4 \int_{S_j} J_j(r_j) dF_{dS_k-dS_j} \quad (7)$$

Où $J_k(r_k)$ est la radiosité de l'élément de surface dS_k , $\theta_{\text{ref}} = k_f T_i / (SL)$ est une température de référence et dF est le facteur de forme entre les éléments de surface dS_k et dS_j situés respectivement aux positions r_k et r_j .

Dans la présente étude, on s'intéresse aux flux de chaleur moyens échangés par le système à travers ses surfaces latérales. Ces quantités sont définies comme suit :

$$Q_i(t) = -\frac{S}{N_k A} \int_0^A \frac{\partial \theta_s}{\partial X} \Big|_{X=1} dY \quad (8)$$

$$Q_e(t) = \frac{S}{N_k A} \int_0^A \frac{\partial \theta_s}{\partial X} \Big|_{X=1} dY \quad (9)$$

Avec $N_k = \frac{k_f}{k_s}$

3. Méthode de résolution et validation

La méthode des volumes finis est utilisée pour discrétiser les équations du modèle mathématique. Le couplage

pression-vitesse est résolu par l'algorithme SIMPLE. Le schéma en loi de puissance est adopté pour le couplage convection-diffusion. Un programme numérique écrit en langage Fortran est élaboré et un algorithme à matrice tri-diagonale (TDMA) est utilisé pour résoudre le système d'équations algébriques ligne par ligne avec une procédure itérative jusqu'à la satisfaction du critère de convergence.

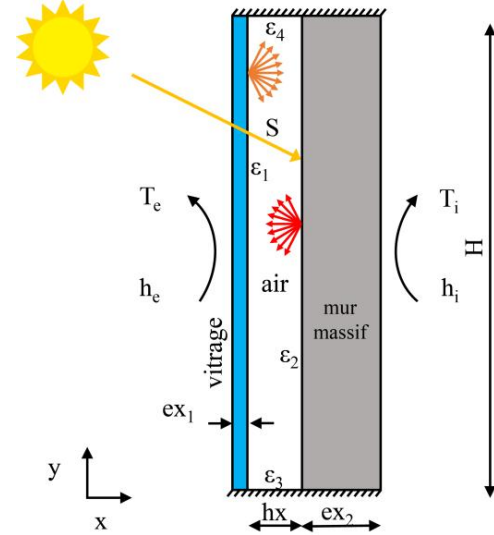


Figure 1. Configuration étudiée

La validité du code de calcul a été confirmée en comparant les résultats qu'il fournit avec ceux publiés par Heinrich Manz [5]. Ces auteurs ont analysé le transfert de chaleur par convection naturelle dans des cavités à grand rapport de forme. Comme le montre le tableau 1, une bonne concordance est observée entre le nombre de Nusselt du présent travail et celui trouvé par Heinrich Manz [5]. Les déviations enregistrées sont dues au fait que le présent solveur utilise le schéma en loi de puissance, tandis que Heinrich Manz [5] ont utilisé le schéma Upwind pour la discrétisation des termes convectifs.

Table 1. Comparaison entre le nombre de Nusselt du présent travail et celui trouvé par Heinrich Manz [5]

	Ra	Nu (code)	Nu [5]	Deviation
A = 20	1000	1,022	1,028	0,58%
	3685	1,131	1,139	0,70%
	22940	1,769	1,759	0,57%
	85674	2,590	2,593	0,11%
A = 80	1000	1,005	1,019	1,37%
	3685	1,032	1,046	1,34%
	23094	1,239	1,236	0,24%

4. Résultats et discussion

La figure 2a révèle que le flux utile Q_i augmente linéairement avec l'irradiation solaire S. En outre, il reste

toujours positif même si le flux solaire est inférieur à sa valeur critique $S_c = 56,1\text{W/m}^2$ (la valeur pour laquelle l'écoulement dans la lame d'air change de sens de rotation). Ceci peut être expliqué par le fait que la différence de température $T_e - T_i$ est toujours positive quelle que soit la valeur de S . Quant à l'influence de la conductivité du mur massif, on remarque que Q_i subit une augmentation appréciable en augmentant K_s , notamment pour les grandes valeurs du flux solaire. Ce comportement s'explique par l'homogénéisation des températures des surfaces de la lame d'air.

Pour ce qui est du flux perdu Q_e , on constate qu'il est négatif pour $S < S_c$ et pour toutes les valeurs considérées de la conductivité K_s . Ce qui donne un transfert de chaleur de l'extérieur vers l'intérieur du local. Dès que $S > S_c$, la situation s'inverse puisque la température de la face externe du mur massif devient supérieure à celle de la vitre. Il est évident que pour $S = S_c = 56,1\text{W/m}^2$, l'échange de chaleur entre les surfaces de la lame d'air devient nul. Similairement au flux utile Q_i , la figure 2a montre également que le flux perdu Q_e subit une augmentation linéaire avec l'irradiation solaire. Cette augmentation est de plus en plus favorisée que la conductivité K_s est faible. Compte tenu de tous ces résultats, on peut conclure que les flux utile et perdu peuvent être optimisés en optant pour une grande conductivité du mur massif.

La figure 2b montre que le flux utile Q_i et celui perdu Q_e sont pratiquement indépendants de la conductivité du vitrage, et ce quel que soit le flux solaire S . À vrai dire, ils subissent une très faible variation, en particulier pour les grandes valeurs du flux solaire. À titre d'exemple pour $S = 1000\text{W/m}^2$, Q_i passe de $438,4\text{W/m}^2$ à $436,5\text{W/m}^2$ quand on varie K_v de $0,8\text{W/mK}$ à $1,2\text{W/mK}$, soit une variation maximale de juste $0,44\%$. Pour la même valeur du flux solaire, le flux Q_e subit une augmentation du même ordre de grandeur. Ce résultat est attendu étant donné la faible épaisseur du vitrage qui ne représente que 3% de la largeur totale du mur Trombe étudié ($ex_1 = 4\text{mm}$).

5. Conclusion

Dans ce travail, on a présenté une étude numérique portant sur le transfert de chaleur combiné dans un mur Trombe non ventilé. Un modèle mathématique traduisant le principe de conservation de la masse, la quantité de mouvement et l'énergie a été adopté. Un solveur numérique basé sur la méthode des volumes finis a été validé et utilisé pour résoudre les équations gouvernantes. Les résultats de simulation montrent que les flux utile et perdu peuvent être optimisés en augmentant la conductivité du mur massif. Quant à la conductivité du vitrage, ces flux n'y sont pas sensibles.

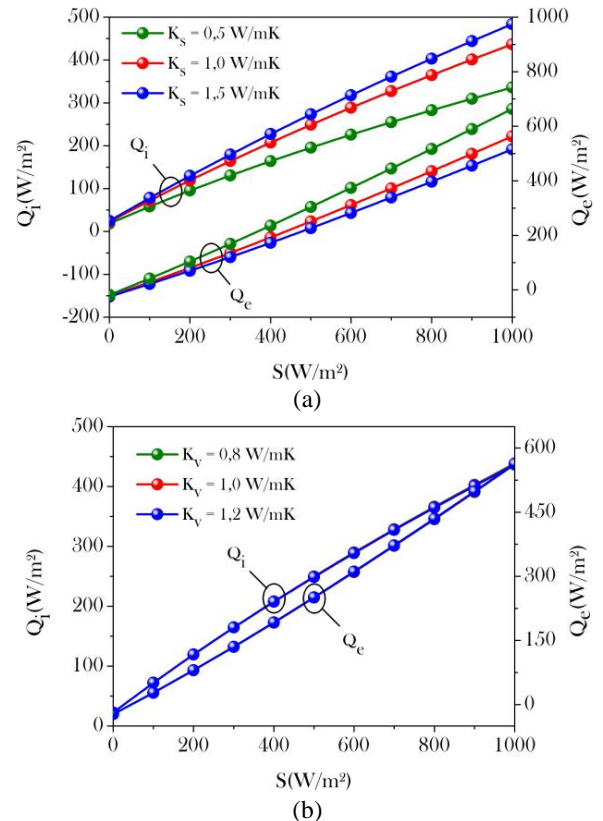


Figure 2. Effet de la conductivité du mur massif (a) et le vitrage (b) sur les flux entrant Q_i et sortant Q_e pour différentes valeurs de l'irradiation solaire

Remerciement

Le premier auteur tient à remercier le Centre National pour la Recherche Scientifique et Technique CNRST pour son soutien financier, bourse numéro 6UCA2020.

Références

- [1] Q. Ma, H. Fukuda, T. Kobatake, M. Lee, *Study of a double-layer Trombe wall assisted by a temperature-controlled DC fan for heating seasons*, Sustainability, vol. 9 (2017), 2179.
- [2] N. Zhu, S. Li, P. Hu, F. Lei, R. Deng, *Numerical investigations on performance of phase change material Trombe wall in building*, Energy, vol. 187 (2019), 116057.
- [3] Z. Charqui, L. El Moutaouakil, M. Boukendil, R. Hidki, *Numerical study of heat transfer in a tall, partitioned cavity confining two different fluids: application to the water Trombe wall*, International Journal of Thermal Sciences, vol. 171 (2022), 107266.
- [4] O. K. Ahmed, K. I. Hamada, A. M. Salih, *Enhancement of the performance of Photovoltaic/Trombe wall system using the porous medium: Experimental and theoretical study*, Energy, vol. 171 (2019), 14-26.
- [5] H. Manz, *Numerical simulation of heat transfer by natural convection in cavities of facade elements*, Energy and Buildings, vol. 35 (2003), 305-311;

SIMULATION NUMÉRIQUE DU COMPORTEMENT THERMIQUE D'UN MUR TROMBE EN RÉGIME VARIABLE

Z. CHARQUI¹, M. BOUKENDIL¹, L. EL MOUTAOUAKIL¹, R. HIDKI¹, Z. ZRIKEM¹, A. ABDELBAKI¹

1. Université Cadi Ayyad, Faculté des Sciences Semlalia, Département de Physique, LMFE, B.P. 2390, Marrakech, Maroc

m.boukendil@uca.ac.ma

Résumé

Dans ce papier, on se propose d'analyser numériquement le transfert de chaleur dans un mur Trombe non ventilé. Le mur échange par convection et rayonnement avec deux ambiances interne et externe et absorbe un flux solaire variable à travers sa surface extérieure. Les équations de conservation régissant le problème ont été discrétisées par la méthode des volumes de contrôle et résolues par l'algorithme SIMPLE. Les résultats obtenus indiquent que l'épaisseur du mur massif atténue les variations du flux entrant tandis qu'elle n'a pratiquement aucun effet sur le flux sortant, et ce sans ou avec film solaire.

Mots-clefs : *Convection naturelle turbulente ; Conduction ; Rayonnement surfacique ; Irradiation solaire ; Régime variable ; Mur Trombe ;*

1. Introduction

À l'échelle internationale, le secteur du bâtiment est responsable sur la consommation d'environ 40 % de l'énergie totale, dont une bonne partie est utilisée principalement pour subvenir aux demandes de chauffage, de refroidissement et de ventilation des habitats. Ainsi, la réduction de cette grande consommation d'énergie tout en assurant le confort thermique aux habitants constitue l'un des principaux enjeux qui préoccupent les intervenants dans le secteur de l'énergie. Dans ce sens, il est souvent recommandé d'intégrer un système solaire appelé « mur Trombe » dans les habitats en raison de son caractère passif et durable.

Durant les deux dernières décennies, le mur Trombe a été étudié massivement dans la littérature sous différentes configurations en raison de ces multiples avantages. On y trouve par exemple : le mur Trombe composite [1], le mur Trombe à changement de phase

[2], le mur Trombe photovoltaïque [3], le mur Trombe à eau [4]...

Néanmoins, les publications relatives au mur Trombe classique s'avèrent relativement en pénurie. Pourtant, il a été montré dans plusieurs cas qu'un tel système solaire fonctionne bien meilleur que les autres variantes. De ce fait, on se propose de l'étudier numériquement en analysant son comportement thermique vis-à-vis de l'irradiation solaire qu'il absorbe et les températures auxquelles il est exposé.

2. Modèle physique

Une représentation schématique du système étudié est illustrée dans la figure 1. Il s'agit d'un mur Trombe non ventilé de largeur L et de hauteur H . Il est composé principalement d'un mur massif séparé de l'ambiance extérieure par un simple vitrage. Le système entier échange par convection et rayonnement avec les milieux intérieur et extérieur de températures respectives T_i et T_e . Les parois horizontales sont supposées isolées thermiquement. Pour la saison hivernale, le flux solaire tape directement sur la face externe du mur massif tandis que pour la saison estivale, le flux solaire tape uniquement sur le vitrage extérieur grâce à un film amovible qui masque l'intégralité de l'irradiation.

L'écoulement d'air est considéré turbulent vu que le rapport de forme de la lame d'air est très important ($A = 24,5$). Le mouvement du fluide est supposé bidimensionnel, Newtonien, incompressible et sans dissipation visqueuse. L'air est supposé transparent à l'égard du rayonnement surfacique. Toutes les propriétés thermophysiques sont indépendantes de la température, sauf la densité dans le terme de poussée, qui est approximée par le modèle de Boussinesq.

Les équations régissant le mouvement du fluide et la conservation de l'énergie sont données par :

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial u_i}{\partial x_j} - \rho \overline{u_i' u_j'} \right) - \rho \beta (T - T_0) g_i \quad (2)$$

$$\frac{\partial(\rho c_p T)}{\partial t} + \frac{\partial(\rho c_p u_j T)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} - \rho c_p \overline{u_j' T'} \right) \quad (3)$$

$$\frac{\partial(\rho c_p T)}{\partial t} = \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right) \quad (4)$$

Les équations (2-3) font apparaître deux inconnues supplémentaires autres que les champs classiques (u, v, p , et T) qui sont les tensions de Reynolds $\rho \overline{u_i' u_j'}$ et le flux de chaleur turbulent $\rho c_p \overline{u_j' T'}$. Pour traiter ces termes, les auteurs ont adopté le modèle SA (Spalart et Allmaras), dont les équations ont été exposées et détaillées dans des papiers publiés précédemment par la même équipe [5]. Pour des raisons de limitation, la validation de ce modèle ne sera pas abordée dans le présent papier puisqu'il a été déjà validé extensivement dans le passé. Les conditions aux limites associées aux équations ci-dessus s'expriment par :

Pour les surfaces internes de la lame d'air :

$$u = v = 0 \quad (5)$$

Pour les surfaces horizontales extrêmes :

$$\left. \frac{\partial T_p}{\partial y} \right|_{y=0} = \left. \frac{\partial T_p}{\partial y} \right|_{y=H} = 0 \quad (6)$$

Pour les surfaces verticales extrêmes :

$$-k_v \left. \frac{\partial T_v}{\partial x} \right|_{x=0} = -h_e (T_v - T_e(t)) \quad (7)$$

$$-k_p \left. \frac{\partial T_p}{\partial x} \right|_{x=L} = h_i (T_p - T_i) \quad (8)$$

Pour les interfaces solide-fluide :

$$T_s = T_f \quad (9)$$

$$-k_s \frac{\partial T_s}{\partial \eta} + S(t) = -k_f \frac{\partial T_f}{\partial \eta} \pm q_r \quad (10)$$

La dernière équation contient le terme $S(t)$ qui représente le flux absorbé par le mur quand le film solaire est enlevé. Par contre quand ce dernier est inséré, ce terme est annulé de l'équation (10) et ajouté à l'équation (7). Le flux radiatif q_r est déterminé par le biais de la méthode des radiosités [4].

Dans la présente étude, on s'intéresse aux flux de chaleur moyens échangés par le système à travers ses surfaces latérales. Ces quantités sont définies comme suit :

$$Q_i(t) = \frac{1}{H} \int_0^H k_p \left. \frac{\partial T}{\partial x} \right|_{x=L} dy \quad (11)$$

$$Q_e(t) = \frac{1}{H} \left(\int_0^{\delta_p} k_p \left. \frac{\partial T}{\partial x} \right|_{x=0} dy + \int_{\delta_p}^{\delta_p+h} k_v \left. \frac{\partial T}{\partial x} \right|_{x=0} dy + \int_{\delta_p+h}^{2\delta_p+h} k_p \left. \frac{\partial T}{\partial x} \right|_{x=0} dy \right) \quad (12)$$

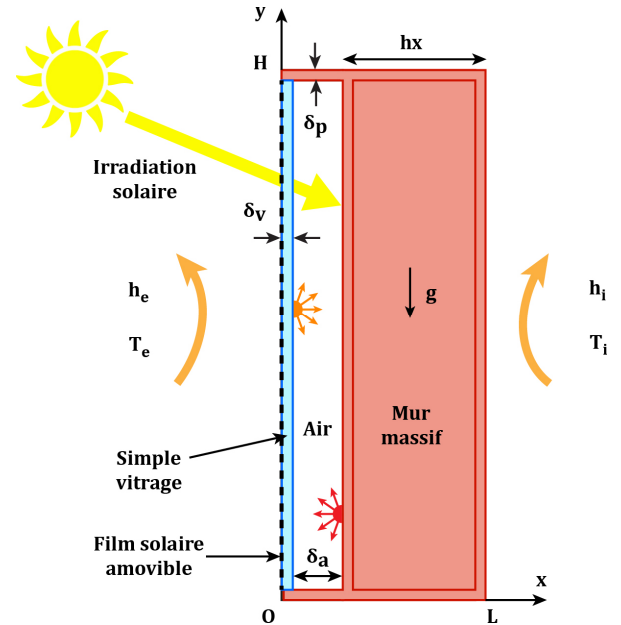


Figure 1. Configuration étudiée

3. Méthode de résolution

Afin de résoudre les équations gouvernantes (1-4), la méthode des volumes finis et l'algorithme SIMPLE ont été utilisés en les implémentant dans un code de calcul écrit en langage Fortran. Le schéma en loi de puissance et le schéma aux différences centrées ont été choisis pour traiter les termes convectifs et diffusifs apparaissant dans lesdites équations lors de leur discrétisation sur un maillage non uniforme. Les termes transitoires ont été traités par un schéma explicite du premier ordre. Le système d'équations algébrique obtenu a été ensuite résolu itérativement par le biais de l'algorithme de Thomas.

Les résultats qui suivent sont presque indépendants du maillage, le pas de temps et les conditions initiales. C'est la raison pour laquelle aucune information n'est donnée sur ces trois paramètres.

4. Résultats et discussion

La figure 2 présente les variations horaires des flux entrant et sortant pour les deux saisons considérées. Les courbes obtenues indiquent que ces grandeurs sont caractérisées par des variations temporelles périodiques avec une période identique à celle des excitations thermiques (24 heures). Ce résultat logique est prévisible étant donné que le mur Trombe étudié est un système

physique passif. Pour la saison d'hiver (film solaire enlevé), l'amplitude du flux entrant Q_i subit une atténuation appréciable en augmentant la largeur du mur massif h_x . Une telle atténuation est beaucoup plus accentuée pendant la période diurne ($S \neq 0$). Quant au flux sortant Q_e , son amplitude est pratiquement indifférente à h_x durant tout le cycle. Cela s'explique par le fait qu'une grande partie de ce flux est transportée par rayonnement surfacique vers l'extérieur. D'un autre côté, le système solaire considéré introduit un déphasage temporel (retard) de 6h entre le flux entrant et l'irradiation solaire. Un tel déphasage devient 9h en augmentant la largeur du mur massif à 20cm. En d'autres termes, en doublant la largeur du mur massif, on gagne trois heures supplémentaires de retard. La figure 2 montre également qu'en couvrant le vitrage extérieur par un film noir (saison d'été), le flux entrant Q_i se limite de varier entre $29,3W/m^2$ et $51,3W/m^2$ pour $h_x = 20cm$. Autrement dit, ce système n'empêche pas uniquement la surchauffe du bâtiment pendant les périodes chaudes, mais il réduit aussi la charge de climatisation de manière substantielle. Cela s'explique par le rôle d'isolation que joue la lame d'air dans ce cas de figure. D'un autre côté, la couverture du vitrage extérieur par un film noir permet de renvoyer une grande partie de l'irradiation solaire à l'ambiance extérieure par convection et rayonnement. Cela se voit nettement en examinant les variations raides du flux Q_e . En augmentant la largeur h_x , il s'avère que les flux entrant et sortant subissent presque les mêmes variations enregistrées pour la configuration d'hiver. Au contraire, le déphasage temporel devient plus important d'environ une heure. Ce résultat est prévisible vu que la lame d'air, dans ce cas, ralenti la propagation de la chaleur vers l'ambiance intérieure.

5. Conclusion

Le comportement thermique d'un mur Trombe non ventilé en régime variable a été étudié dans le présent papier. Le modèle mathématique adopté est en vertu du principe de conservation de la masse, la quantité de mouvement et l'énergie. Un solveur numérique basé sur la méthode des volumes finis a été validé et utilisé pour résoudre les équations gouvernantes. Les résultats des simulations numériques indiquent que la largeur du mur massif atténue les variations du flux entrant au local. Au contraire le flux sortant est pratiquement insensible à ce paramètre. En outre, la largeur du mur massif favorise le retard temporel introduit par le système entre le flux entrant et l'irradiation solaire, ce qui est recherché surtout pour la saison d'hiver.

Remerciement

Le premier auteur tient à remercier le Centre National pour la Recherche Scientifique et Technique CNRST

pour son soutien financier, bourse numéro 6UCA2020.

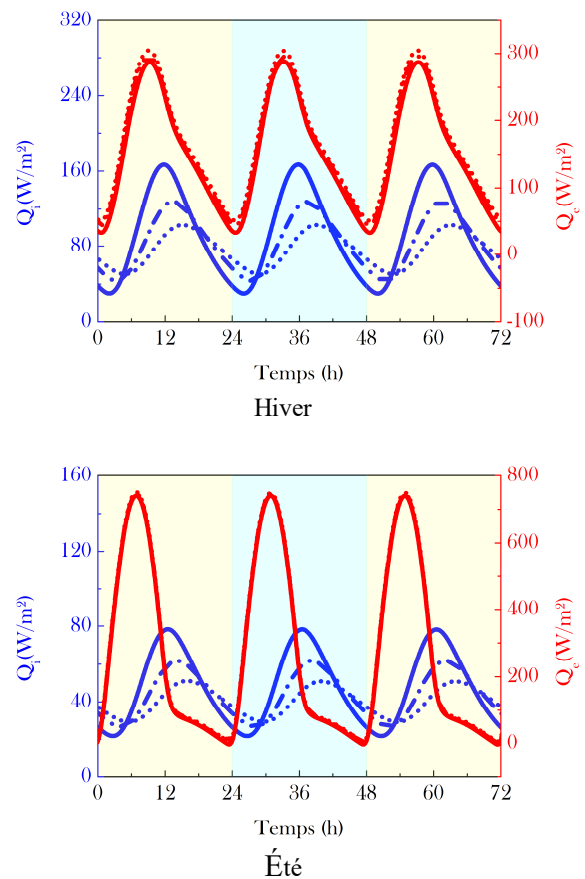


Figure 2. Effet de la largeur du mur massif sur les variations horaires des flux entrant Q_i et sortant Q_e : - $h_x = 10cm$; --- $h_x = 15cm$; ... $h_x = 20cm$

Références

- [1] Q. Ma, H. Fukuda, T. Kobatake, M. Lee, *Study of a double-layer Trombe wall assisted by a temperature-controlled DC fan for heating seasons*, Sustainability, vol. 9 (2017), 2179.
- [2] N. Zhu, S. Li, P. Hu, F. Lei, R. Deng, *Numerical investigations on performance of phase change material Trombe wall in building*, Energy, vol. 187 (2019), 116057.
- [3] O. K. Ahmed, K. I. Hamada, A. M. Salih, *Enhancement of the performance of Photovoltaic/Trombe wall system using the porous medium: Experimental and theoretical study*, Energy, vol. 171 (2019), 14-26.
- [4] Z. Charqui, L. El Moutaouakil, M. Boukendil, R. Hidki, *Numerical study of heat transfer in a tall, partitioned cavity confining two different fluids: application to the water Trombe wall*, International Journal of Thermal Sciences, vol. 171 (2022), 107266.
- [5] L. El Moutaouakil, Z. Zrikem, A. Abdelbaki, *Performance of various RANS eddy-viscosity models for turbulent natural convection in tall vertical cavities*, Heat and Mass Transfer, vol. 50 (2014), 1103-1113.

CONVECTION NATURELLE DANS UNE CAVITÉ CARRÉE CONTENANT DEUX BLOCS GÉNÉRATEURS DE CHALEUR

R. HIDKI, L. EL MOUTAOUAKIL, M. BOUKENDIL, Z. CHARQUI, Z. ZRIKEM, A. ABDELBAKI

LMFE, Department of Physics, Cadi Ayyad University, Faculty of Sciences Semlalia, B.P. 2390 Marrakesh, Morocco

m.boukendil@uca.ac.ma

Résumé

Ce travail présente une étude numérique de la convection naturelle laminaire dans une cavité carrée contenant deux blocs générateurs de chaleur. La paroi droite est refroidie uniformément avec une température constante T_c , tandis que les autres parois sont maintenues adiabatiques. Les deux blocs génèrent deux puissances volumiques uniformes Q_1 et Q_2 pouvant être différentes. Les effets des paramètres influençant l'écoulement et le transfert de chaleur dans la cavité sont discutés. À savoir : les deux nombres de Rayleigh et le rapport des conductivités. Les résultats obtenus montrent que l'augmentation de la quantité de chaleur d'un bloc permet de refroidir l'autre bloc. De plus, l'augmentation du rapport des conductivités provoque un bon refroidissement des blocs.

Mots-clés: Cavité fermée; Blocs chauffants; Convection naturelle; Simulation numérique

1. Introduction

Le modèle de la cavité contenant de blocs carrés actifs ou inactifs peut trouver ses applications dans plusieurs domaines d'ingénierie. Tels que le refroidissement des appareils électroniques, les échangeurs de chaleurs, dans le bâtiment, etc. Ce qui justifie l'intérêt accordé à ce type de modèle [1-5]. Dans ces travaux, les auteurs ont étudié le cas d'un bloc carré inactif [1], le cas d'un ou plusieurs blocs isothermes [2,3] et le cas d'un bloc carré générant une puissance volumique uniforme [5].

L'objectif de travail est d'étudier numériquement la convection naturelle induite par deux blocs carrés générant des puissances volumiques uniformes, dans une enceinte refroidie par l'un de ses côtés. Les effets des deux nombres de Rayleigh et le rapport des conductivités sur les caractéristiques dynamique et thermique de l'écoulement sont examinés.

2. Modèle mathématique et Méthode numérique

Le problème considéré, Figure 1, est une cavité carrée, remplie d'air ($Pr = 0,71$) et refroidie par le côté droit. Les parois restantes sont considérées adiabatiques. A

l'intérieur de la cavité, ils se trouvent deux blocs carrés de côté $W = 0,25$ qui génèrent, de l'intérieur, des puissances volumiques uniformes. Ces blocs sont placés selon la médiane verticale de la cavité.

Les propriétés physiques de l'air sont constantes par rapport à la température, à l'exception de la densité dans le terme de flottabilité, en utilisant l'approximation de Boussinesq. En adaptant ces approximations aux équations de continuité, de quantité de mouvement et d'énergie, on obtient le système sans dimension suivant :

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0 \quad (1)$$

$$U \frac{\partial U}{\partial X} + V \frac{\partial U}{\partial Y} = -\frac{\partial P}{\partial X} + Pr \nabla^2 U \quad (2)$$

$$U \frac{\partial V}{\partial X} + V \frac{\partial V}{\partial Y} = -\frac{\partial P}{\partial Y} + Pr \nabla^2 V + Ra_m Pr \theta \quad (3)$$

$$U \frac{\partial \theta}{\partial X} + V \frac{\partial \theta}{\partial Y} = ((1 - \delta) + K\delta) \nabla^2 \theta + \frac{Ra_i}{Ra_m} \delta \quad (4)$$

δ est un symbole qui vaut 0 dans le milieu fluide et 1 dans le milieu solide.

Les variables et paramètres sans dimension apparaissant dans les équations (1)-(4) sont définis comme suit :

$$(X, Y) = \frac{(x, y)}{L}, (U, V) = \frac{(u, v)L}{\alpha_f}, \theta = \frac{(T - T_c)}{\Delta T}, \quad (5)$$

$$\Delta T = \frac{Q_m L^2}{k_f}, Pr = \frac{\nu_f}{\alpha_f}, Ra = \frac{g \beta L^3 \Delta T}{\nu_f \alpha_f}, K = \frac{k_s}{k_f}$$

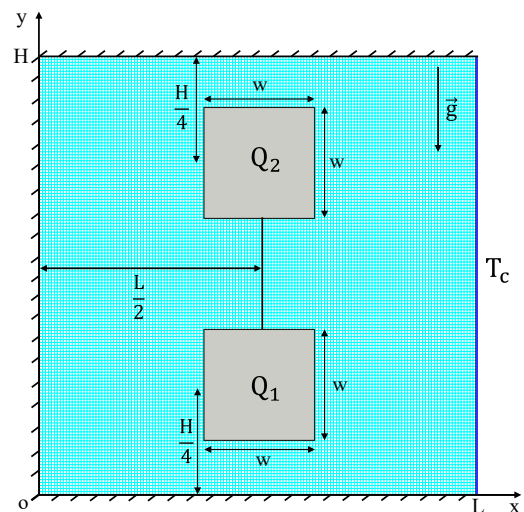


Figure 1. Configuration étudiée.

Les équations (1)-(4) ont été résolues par la méthode des volumes finis basée sur l'algorithme SIMPLE. Le code numérique utilisé a été déjà validé avec succès avec d'autres résultats par Hidki et al. [4].

3. Résultats and discussion

La Figure 2 donne les lignes de courant (à gauche) et les isothermes (à droite) pour différentes valeurs de Ra_1 et Ra_2 . Le rapport des conductivités est fixé à 0,1. Les lignes de courant illustrent que l'écoulement s'intensifie par augmentation de Ra_1 ou Ra_2 , autrement dit lorsque le nombre de Rayleigh moyen, Ra_m , augmente. Pour des nombres de Ra réduits la structure de l'écoulement est presque symétrique vis-à-vis de la médiane horizontale de la cavité, et cette symétrie s'estompe dès que les nombres de Rayleigh dépassent suffisamment 10^4 . Cette structure est constituée de deux cellules horaires; la première (la plus grande) encercle les deux objets chauffants et la deuxième, de taille plus réduite, se développe sur le côté droit des deux blocs générateurs de chaleur. Ceci montre que la façon dont les blocs sont arrangés à l'intérieur de la cavité est déterminante. Afin de contribuer au refroidissement des deux blocs simultanément, la petite cellule glisse sur les deux objets sur le côté le plus proche de la paroi froide. De plus, on signale que cette cellule tend à s'approcher du bloc qui correspond au nombre de Rayleigh le plus élevé. Dans le cas où $Ra_1 = Ra_2 = Ra_m$, avec Ra_m suffisamment élevé, la cellule est généralement plus proche du bloc 2. Il est à noter que ce bloc est l'objet le plus difficile à refroidir, vu qu'il est placé dans un coin relativement chaud (en haut de la cavité).

La Figure 2 montre également que les isothermes sont plus concentrées dans le bloc correspondant au nombre de Rayleigh le plus élevé. A la limite, lorsque $Ra_1 \sim Ra_m \gg Ra_2$, le bloc 2 devient presque isotherme et inversement si $Ra_2 \sim Ra_m \gg Ra_1$. Pour le cas particulier où $Ra_1 = Ra_2 = Ra_m$, les isothermes se resserrent au niveau de chaque bloc sous forme d'anneaux concentriques indiquant que la température atteint son maximum au milieu des blocs. Notons aussi que pour des valeurs de Ra_1 et Ra_2 relativement faibles les isothermes sont pratiquement verticales près de la paroi froide indiquant que le transfert de chaleur local est uniforme sur cette paroi (régime conductif). Lorsque l'un des nombres de Ra_i (i.e. Ra_m) est élevé, les isothermes s'approchent de la partie supérieure de la paroi froide et inversement en bas de cette surface. Cela signifie que le transfert de chaleur local croît du bas en haut de la paroi active par augmentation de Ra_m .

On présente dans la Figure 3 les profils de vitesse obtenus sur la médiane verticale de la cavité, pour différentes combinaisons de (Ra_1, Ra_2) , le rapport des conductivités est fixé à 0,1. Cette figure montre que le

fluide est presque au repos pour des nombres de Rayleigh faibles (régime de conduction dominante). Ensuite, lorsqu'on augmente Ra_i ($i = 1$ ou 2) jusqu'à ce qu'il dépasse suffisamment la valeur 10^4 , correspondant à la limite du régime conductif, l'écoulement devient très intense pour permettre à l'air d'évacuer la quantité de chaleur (proportionnelle à Ra_m) générée. Lorsque Ra_1 est trop élevé ($Ra_1 = 10^6$), la vitesse de l'air dans la cavité devient indépendante de Ra_2 ; un fait qu'on pourra attribuer à la saturation thermique de l'air par l'énorme quantité de chaleur générée par le bloc 1.

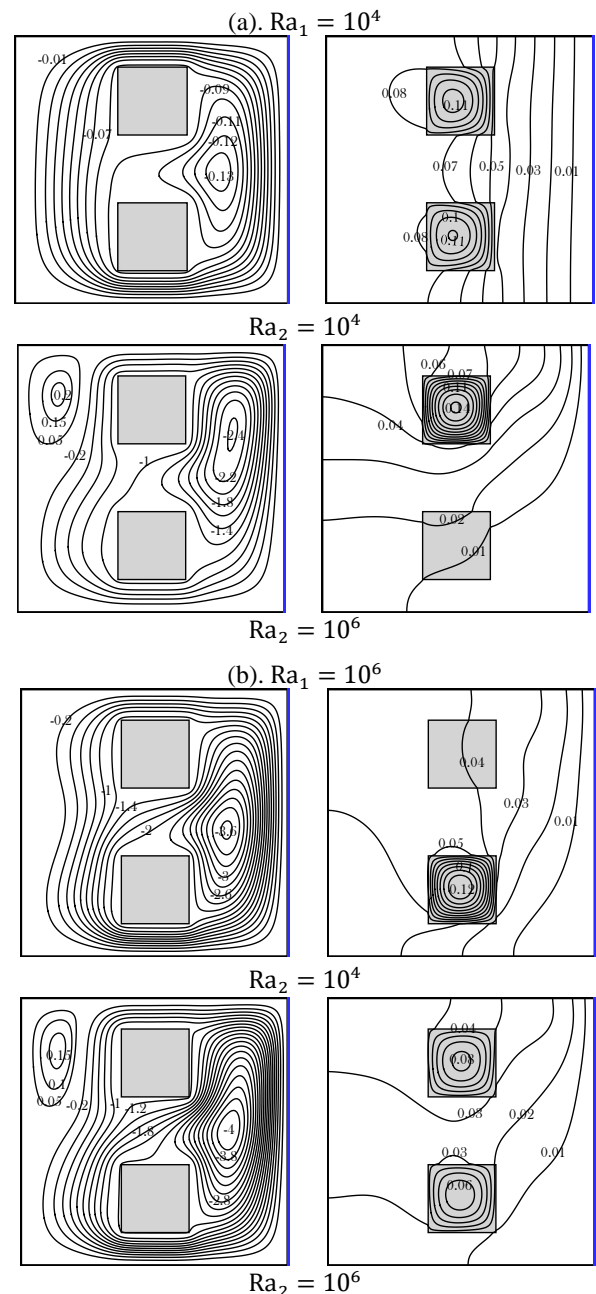


Figure 2. Lignes de courant (droite) et isothermes (gauche) obtenues pour différentes combinaisons de (Ra_1, Ra_2) .

Afin de monter l'effet du rapport des conductivités ($0,01 \leq K_i \leq 10$), la Figure 4 présente les lignes de

courant et les isothermes pour $Ra_1 = Ra_m = 10^6$. Les résultats montrent que la structure de l'écoulement reste pratiquement inchangée en augmentant le rapport des conductivités K_i . L'effet de ce dernier intervient plutôt dans la distribution du champ de température au sein des deux blocs générateurs de chaleur. En effet, pour $K_i \sim 0,01$, il y a un encombrement des isothermes sous forme d'anneaux concentriques de même intensité au niveau de chaque bloc, en indiquant que la température atteint son maximum au milieu des deux objets. L'air, dans ce cas, est pratiquement isotherme à une température réduite, vu que la conductivité des blocs est très petite devant celle du fluide. En revanche, pour $K_i \sim 10$, on constate que les gradients de température sont plus intenses dans le milieu fluide, tandis qu'ils sont pratiquement nuls dans les blocs (blocs isothermes). Notons également que le point le plus chaud dans la cavité se trouve dans le bloc 2 pour toutes les valeurs de $K_i \geq 0,1$.

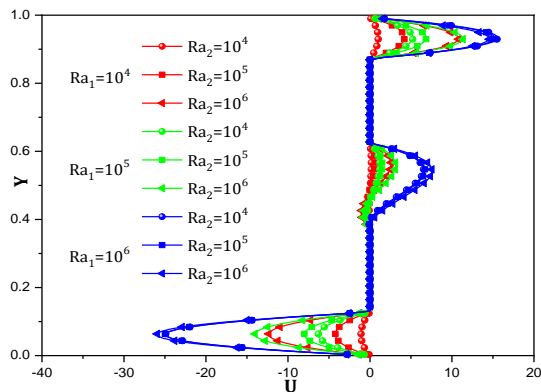


Figure 3. Profils de vitesse sur la médiane verticale de la cavité obtenus pour différentes combinaisons de (Ra_1, Ra_2) .

4. Conclusion

Dans ce travail, une étude numérique par la méthode des volumes finis est réalisée afin d'analyser l'échange de chaleur par convection naturelle autour de deux blocs générateurs de chaleur dans une cavité carrée refroidie par l'un de ses côtés.

Les résultats obtenus montrent que les deux nombres de Rayleigh affectent significativement l'écoulement et le champ température dans le domaine fluide. Par contre, le rapport des conductivités est un paramètre qui a plus d'effet dans le domaine solide que dans le fluide. Pour des valeurs élevées de ce rapport, les résultats montrent qu'on s'approche des conditions de validité du modèle des blocs isothermes. Ces résultats montrent également que la température des blocs chute considérablement au fur et à mesure que K_i augmente. En outre, la température la plus chaude est notée au niveau du bloc 2 qui est l'objet difficile à refroidir.

Références

- [1] J. M. House, C. Beckermann, T. F. Smith, Effect of a centered conducting body on natural convection heat transfer in an enclosure, *Numerical Heat Transfer, Part A*, 18 (1990), 213-225.
- [2] S. M. Dash, T. S. Lee, Natural convection in a square enclosure with a square heat source at different horizontal and diagonal eccentricities, *Numerical Heat Transfer, Part A*, 68 (2015), 686-710.
- [3] M. Sheikholeslami, K. Vajravelu, Lattice Boltzmann method for nanofluid flow in a porous cavity with heat sources and magnetic field, *Chinese J. Phys.*, 56 (2018), 1578-1587.
- [4] R. Hidki, L. El Moutaouakil, Z. Charqui, M. Boukendil, Z. Zrikem, Natural convection in a square cavity containing two heat-generating cylinders with different geometries. *Materials Today: Proceedings*, 45 (2021), 7415-7423.
- [5] J. Y. Oh, M. Y. Ha, K. C. Kim., Numerical study of heat transfer and flow of natural convection in an enclosure with a heat-generating conducting body, *Numerical Heat transfer, Part A*, 31 (1997), 289-303.

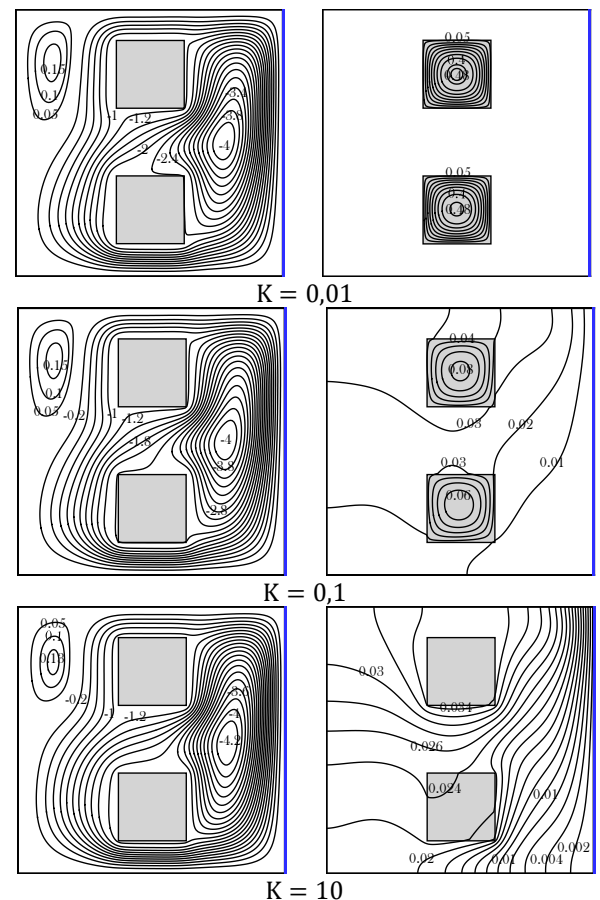


Figure 4. Lignes de courant (droite) et isothermes (gauche) obtenues pour différentes valeurs de K_i .

MIXED CONVECTION IN A VENTILATED SQUARE CAVITY WITH A CIRCULAR HEAT-GENERATING CYLINDER

R. HIDKI, L. EL MOUTAOUAKIL, M. BOUKENDIL, Z. CHARQUI, Z. ZRIKEM, A. ABDELBAKI

LMFE, Department of Physics, Cadi Ayyad University, Faculty of Sciences Semlalia, B.P. 2390 Marrakesh, Morocco

m.boukendil@uca.ac.ma

Abstract

Mixed convection in a ventilated cavity containing a heat-generating cylinder is carried out. This configuration can be used for the cooling of electronic components. The differential equations are discretized by the finite volume method (FVM) and solved by the SIMPLE algorithm. The effects of Richardson ($0.1 \leq Ri \leq 10$), Reynolds ($50 \leq Re \leq 500$) numbers, and cylinder position on the dynamic and thermal characteristics are studied for $Pr = 0.71$, $K = 1$, and $D = 0.4$. The results show that Re has a significant influence on the flow and their increase causes a good cooling of the circular heat-generating cylinder.

Keywords: Mixed convection; Ventilated cavity; Finite volume method; Heat-generating cylinder; Numerical simulation

1. Introduction

In recent decades, mixed convection (MC) in ventilated cavities (VC) equipped with heating cylinders has attracted the great attention of many scientific researchers due to its multiple practical applications. Among them: air conditioning, cooling of electronic devices, heat exchangers, solar collector, and so on. This justifies the interest given to this type of flow [1, 2].

Chamkha et al. [3] performed a numerical study of MC in a VC containing an isothermal square cylinder. The authors studied the effect of the outlet position for different values of the Richardson (Ri) and Reynolds (Re) numbers. The authors found that the top position of the outlet port causes a minimum temperature in the cavity. Chatterjee and Mishra [4] considered the case of two isothermal circular cylinders in a VC. They concluded that Ri has a slight influence on the average Nusselt number. Gupta et al. [5] analyzed the MC in a differentially heated cavity containing a circular conductive cylinder. The authors showed that, for low values of Ri , the increase in the conductivity of the cylinder (K) causes a decrease in the Nusselt number while the average temperature in the cavity increases.

The same configuration has been reconsidered by [6] in the case of a square conductive cylinder. Mamun et al. [7] considered a hollow circular cylinder placed in the center of a square VC. The authors concluded that the thermal conductivity does not influence the flow for $Ri = 0$. The case of a square heat-generating cylinder has been considered by Ahammad et al. [8] in a VC heated from below. They concluded that the Nusselt number reaches its maximum for the small size of the cylinder.

This literature review showed that the MC in a VC with heating cylinders had been the subject of several studies because of its importance in several practical fields, such as the cooling of electronic equipment. It can also be seen that most of these studies deal with the case of an isothermal or a simple conductive cylinder. On the other hand, fewer studies have been found in the case of a heat-generating cylinder. Therefore, this work aims to study the air-cooling by MC of a heat-generating circular cylinder in a VC and cooled from the right side. For this purpose, the effect of (Ri, Re) on the dynamic and thermal characteristics is discussed.

2. Mathematical model

The geometry of the considered physical problem with appropriate boundary conditions is represented in Figure 1. It highlights a circular cylinder placed in a square air-ventilated cavity ($Pr = 0.71$). The cylinder generates a uniform volume power $Q(W/m^3)$ and has a diameter of $D = d/L = 0.4$. The inlet and outlet ports are placed on the vertical walls of the cavity. The size of two openings (inlet and outlet) is set to equal $h_i = h_o = 0.1L$. The air enters the cavity with a velocity u_i . The cavity is cooled by its left vertical wall, while other walls thermally insulate.

The physical properties of air are constant with respect to temperature, except for the density in the buoyancy term, using the Boussinesq approximation. By adapting these approximations to the equations of continuity, momentum, and energy the following dimensionless system is found:

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0 \quad (1)$$

$$U \frac{\partial U}{\partial X} + V \frac{\partial U}{\partial Y} = -\frac{\partial P}{\partial X} + \frac{1}{Re} \nabla^2 U \quad (2)$$

$$U \frac{\partial V}{\partial X} + V \frac{\partial V}{\partial Y} = -\frac{\partial P}{\partial Y} + \frac{1}{Re} \nabla^2 V + Ri\theta \quad (3)$$

$$U \frac{\partial \theta}{\partial X} + V \frac{\partial \theta}{\partial Y} = \frac{(1-\delta) + K\delta}{RePr} \nabla^2 \theta + \frac{\delta K}{RePr} \quad (4)$$

δ is a symbol that equals 0 in the fluid medium and 1 in the solid medium.

The dimensionless variables and parameters appear in Eqs. (1)-(4) are defined as follows:

$$(X, Y) = \frac{(x, y)}{L}, (U, V) = \frac{(u, v)}{u_i}, \theta = \frac{(T - T_c)}{\Delta T}, \Delta T = \frac{QL^2}{k_s}$$

$$Re = \frac{u_i L}{\nu}, Pr = \frac{\nu_f}{\alpha_f}, Ra = \frac{g\beta L^3 \Delta T}{\nu_f \alpha_f}, Ri = \frac{Ra}{Re^2 Pr}$$

$$K = \frac{k_s}{k_f}$$

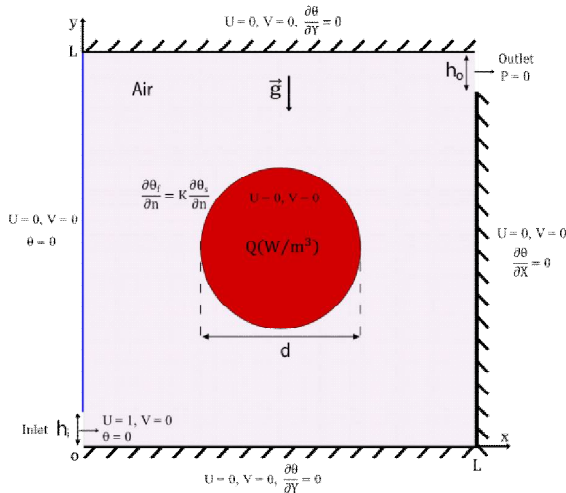


Figure 1. Studied Configuration.

3. Numerical method and validation

The governing equations with appropriate boundary conditions were discretized by FVM. A numerical code based on the SIMPLE algorithm written in FORTRAN software was used to solve these discretized equations. The convergence criterion imposed on all variables is 10^{-4} . The mesh used is uniform in both directions of size 150^2 .

The present numerical code has been successfully validated by comparing our results with Gupta et al. [5]. The comparative results in terms of streamlines and isotherms for $Ri = 5$, $Re = 100$, $Pr = 0.71$, $K = 5$, and $D = 0.4$ are presented in Figure 2. The results show good agreement between our data and those of Gupta et al. [5].

4. Results and discussion

Figure 3 gives the streamlines for different values of (Ri, Re) . For $Re = 50$ and whatever Ri , the streamlines show that the external air covers the entire cavity, showing a symmetry with respect to the line joining the inlet and outlet. This shows that the cold external air entering the cavity does not interact with the inner warmer air. A drastic change is noticed on the

streamlines when Re increases to 500. In fact, the main flow is located on the right half of the cavity, and a large vortex occupies the remaining part and confines the heat-generating cylinder. This shows that the inlet air begins to interact with the internal air. It can also be seen that the effect of Ri on the streamlines is negligible for all considered values of Re .

Under the same conditions, Figure 4 shows that, for $Re = 50$, the isotherms are distributed throughout the domain and that the thermal gradients are more intense below the heat-generating cylinder. The increase of Ri ($Re = 50$) causes a slight increase in the temperature of the cylinder, and a thermal plume develops above the cylinder; this remark has been already provided by Ahammad et al. [8]. When $Re = 500$, the air is almost isothermal, except for the air around the cylinder; this is due to the forced convection, which dominates the natural convection in the cavity. We can also add that this time the effect of Ri is not felt due to the forced convection

Figure 5 shows the maximum temperature profile in the cavity for different values of (Ri, Re) . For low values of Re , the maximum temperature θ_{max} increases with increasing Ri until it reaches its maximum value at $Ri = 5$, then decreases in the region $5 \leq Ri \leq 10$; the same observation has been made by [8]. The increase in Re weakens this variation. When Ri is fixed, the temperature θ_{max} decreases rapidly with increasing Re . Indeed, when Re varies from 50 to 200 θ_{max} decreases by 33%. We can also add that the lowest temperature is found for high values of (Ri, Re) .

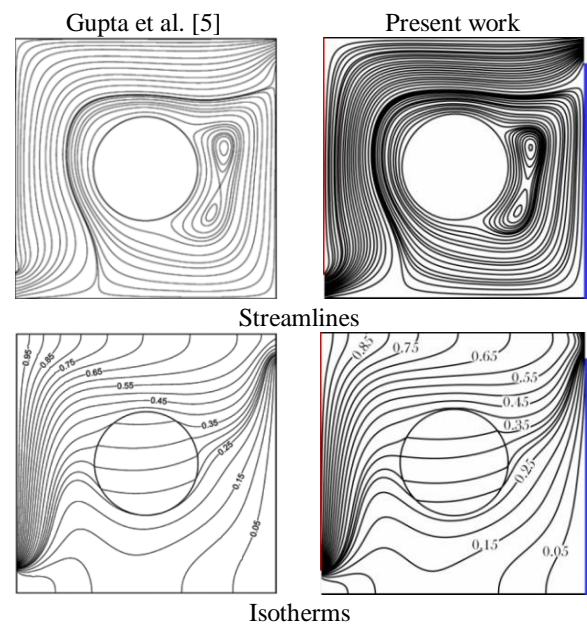


Figure 2. Streamlines and isotherms.

5. Conclusion

In this work, a numerical study by FVM on the cooling of a circular heat-generating cylinder by MC has been simulated. The cylinder is placed inside a square VC. The effects of Richardson and Reynolds numbers on the streamlines, isotherms and maximum temperature are studied. The main highlights of this study are:

- The variation of the Reynolds number strongly influences the streamlines, and they are practically invariant to the Richardson number.
- The highest value of (Ri, Re) causes a good cooling of the heat-generating cylinder.

References

- [1] L. Boutina, R. Bessaïh, Numerical simulation of mixed convection air-cooling of electronic components mounted in an inclined channel, *Appl. Therm. Eng.*, 31 (2011), 2052–2062.
- [2] M. A. Hssain, R. Mir, Y. El Hammami, Numerical Simulation of the Cooling of Heated Electronic Blocks in Horizontal Channel by Mixed Convection of Nanofluids, *J. Nanomater.*, 2020 (2020), 1-11.
- [3] A. J. Chamkha, S. H. Hussain, Q. R. Abd-Amer, Mixed convection heat transfer of air inside a square vented cavity with a heated horizontal square cylinder, *Numer. Heat Transf. Part A Appl.*, 59 (2011), 58–79.
- [4] D. Chatterjee, R. Mishra, Numerical Investigation of Transient Magnetohydrodynamic Mixed Convection in a Ventilated Cavity Containing Two Heated Circular Cylinders, *Heat Transf. Eng.*, 39 (2018), 1052–1066.
- [5] S. K. Gupta, D. Chatterjee, B. Monda, Investigation of mixed convection in a ventilated cavity in the presence of a heat conducting circular cylinder, *Numer. Heat Transf. Part A Appl.*, 67 (2015), 52–74.
- [6] M. M. Rahman, M. Alim, S. Saha, M. Chowdhury, A numerical study of mixed convection in a square cavity with a heat conducting square cylinder at different locations, *J. Mech. Eng.*, 39 (2008), 78–85.
- [7] M. A. H. Mamun, M. M. Rahman, M. M. Billah, R. Saidur, A numerical study on the effect of a heated hollow cylinder on mixed convection in a ventilated cavity, *Int. Commun. Heat Mass Transf.*, 37 (2010), 1326–1334.
- [8] M. U. Ahammad, M. M. Rahman, M. L. Rahman, Mixed Convection Flow and Heat Transfer Behavior inside a Vented Enclosure in the Presence of Heat Generating Obstacle, *Int. J. Innov. Appl. Stud.*, 3 (2013), 967–978.

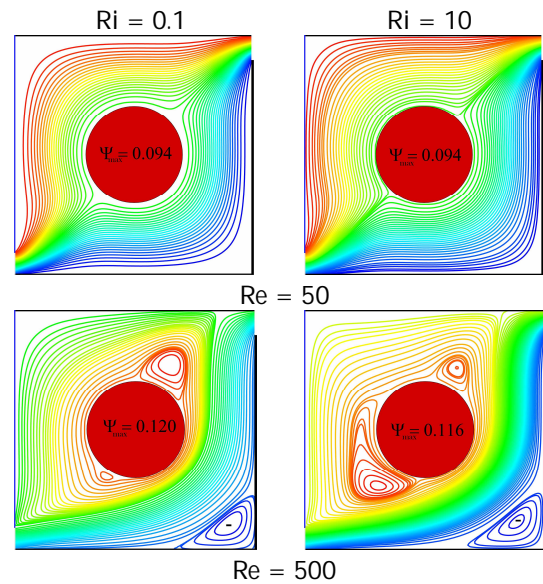


Figure 3. Streamlines for different combinations of (Ri, Re).

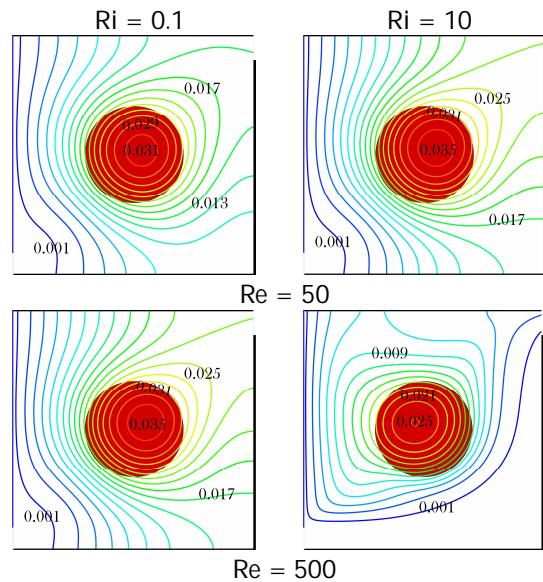


Figure 4. Isotherms for different combinations of (Ri, Re).

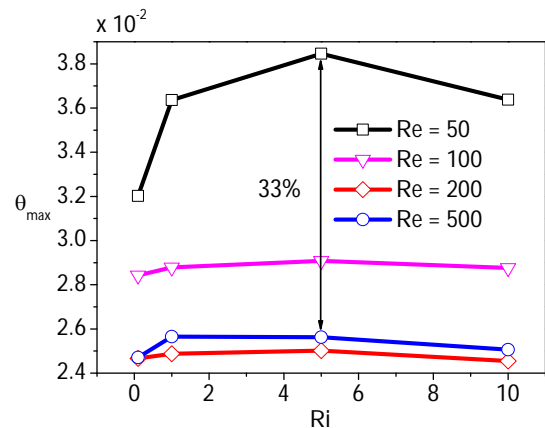


Figure 5. Maximum temperature for different combinations of (Ri, Re).

Effet de la finesse et du taux de remplacement des cendres sur les propriétés physiques de mortier de ciment

Effect of fineness and ash replacement rate on physical properties of mortar

A. LACHHEB¹, M. TOUIL¹, Y. BANDADI¹, R. SAADANI¹, A. SBAIBI², M. RAHMOUNE¹

1. Laboratoire d'Etude des Matériaux Avancées et Applications, FSM-ESTM, Université Moulay Ismail, Meknès, Maroc.

2. Faculté des Sciences et Techniques Mohammedia, Université Hassan II Casablanca, Maroc.

Abstract

The construction industry is a large consumer of cement which is used in concrete and mortar manufacture. Cement substitution would reduce its economic and environmental cost. This study presents an experimental investigation of ash fineness and replacement rate effect on the properties of cement mortar. The ash was ground into two different fineness values of 4500 cm²/g and 6500 cm²/g and used as replacement rate ranging from 10 up to 50% by weight of cement. Test results show that the use of ash in the mortar resulted in a significant improvement in thermomechanical properties and a substantial decrease in water absorption.

Résumé

L'industrie de la construction est une grande consommatrice de ciment qui entre dans la fabrication des béton et mortier. La substitution du ciment permettrait de réduire son coût économique et environnemental. Cette étude présente une enquête expérimentale sur l'effet de la finesse et du taux de remplacement des cendres sur les propriétés du mortier de ciment. Les cendres ont été broyées en deux valeurs de finesse différentes de 4500 cm²/g et 6500 cm²/g et utilisées comme taux de remplacement allant de 10 jusqu'à 50 % en poids de ciment. Les résultats des tests montrent que l'utilisation de cendres dans le mortier a entraîné une amélioration significative des propriétés thermomécaniques et une diminution substantielle de l'absorption d'eau.

Mots clefs : *Cendres, ciment, mortier, méthode des boîtes, banc H10KL, propriétés thermomécaniques.*

Keywords : *Ash, cement, mortar, box method, H10KL bench, thermomechanical properties.*

1. Introduction

De nos jours, le recyclage des déchets et leur utilisation dans le secteur de la construction deviennent une tendance majeure pour améliorer l'efficacité énergétique des bâtiments. Cette tendance vise à réduire la forte consommation d'énergie du secteur du bâtiment [1].

D'un autre côté, la production du ciment consomme beaucoup d'énergie, détruit l'environnement écologique et nous apporte un sérieux effet de serre. Afin de remédier à ces problèmes, de nombreuses recherches se sont penchées sur le sujet afin d'étudier quels seraient les effets sur le mortier ou le béton si une partie du ciment qui les compose était substitué par des cendres [2, 3]. Ils ont montré que l'incorporation des cendres dans les matériaux à base de ciment vise non seulement à réduire la consommation des matières premières et de l'énergie mais aussi à améliorer les propriétés physico-chimiques de ces matériaux ainsi que leur durabilité. Dans notre cas, l'objectif est d'étudier l'effet de dosage et de la finesse des cendres de la centrale thermique JORF Lasfar sur les propriétés physiques du mortier de ciment.

Pour cela les cendres ont été broyées en deux valeurs de finesse différentes de 4500 cm²/g et 6500 cm²/g pour substituer le ciment avec des pourcentages qui varient entre 0% et 50% en poids de ciment. Initialement, le pouvoir d'absorber et de retenir l'eau par ces échantillons est déterminé à travers des essais d'absorption d'eau. Par la suite, des caractérisations thermomécaniques ont été réalisées pour estimer les paramètres thermophysiques et mécaniques des mortiers de cendres. Enfin, une étude comparative de l'ensemble de ces résultats permettra de choisir le dosage et la finesse les plus efficaces et adéquats à incorporer dans le mortier afin de l'intégrer dans l'enveloppe du bâtiment.

2. Méthodes et matériels

2.1 Préparation des matériaux

Les échantillons sont préparés à température ambiante, dans un premier temps le sable est déposé, le ciment et les cendres sont coulés, puis les trois composants sont mélangés à l'état sec afin d'homogénéiser le mélange. Après on creuse le tas dans le cratère et on verse régulièrement la quantité d'eau nécessaire jusqu'à ce que le mélange atteigne la bonne consistance, puis on malaxe pendant 2 minutes pour obtenir un mortier de ciment prêt à l'emploi. Ce mortier de ciment est coulé dans des moules de dimensions [27 x 27 x 4] cm³ pour la caractérisation thermique et de [16 x 4 x 4] cm³ pour la caractérisation mécanique et par la suite séché dans une étuve maintenue à 70°C.

Afin d'étudier l'influence de dosage et de la finesse des cendres sur les propriétés physiques du mortier, une série de mélanges avec deux valeurs de finesse et un niveau différent de remplacement du ciment par des cendres ont été conçus. Le rapport massique eau-ciment était donc de 0,5 pour tous les mélanges de mortiers.

2.2 Absorption d'eau

La capacité d'absorption d'eau des matériaux élaborés est déterminée en mesurant l'augmentation de masse d'un échantillon séché à l'étuve à 70 °C (M_s) après avoir été immergé dans l'eau pendant 24 heures et séché superficiellement (M_a).

Le rapport entre l'augmentation de la masse et la masse de l'échantillon sec, exprimé en pourcentage, est défini comme étant l'absorption (Abs), soit :

$$Abs = \left(\frac{M_s - M_a}{M_s} \right) \times 100\% \quad (1)$$

2.3 Mesure des propriétés thermiques

Les caractéristiques thermiques des matériaux de construction sont des facteurs décisifs dans la performance énergétique des bâtiments.

La caractérisation thermique de nos échantillons est réalisée par la méthode des boîtes.



Fig. 1 : Méthode de boîtes EI700.

La conductivité thermique (λ) des mortiers de cendres avec différents pourcentages a été mesuré en utilisant la première boîte de la méthode après l'obtention du régime permanent. Elle est calculée par l'équation suivante :

$$\lambda = \frac{e}{S(T_C - T_F)} \left[\frac{V^2}{R} - c(T_B - T_A) \right] \quad (2)$$

Tandis que, la diffusivité (α) est mesurée par la méthode du flash dans la deuxième boîte du dispositif expérimental. Elle est estimée par la moyenne des trois valeurs calculées par les expressions suivantes [4] :

$$\begin{aligned} a_1 &= \frac{e^2}{t_{5/6}^2} [1,15 t_{5/6} - 1,25 t_{2/3}] \\ a_2 &= \frac{e^2}{t_{5/6}^2} [0,76 t_{5/6} - 0,926 t_{1/2}] \\ a_3 &= \frac{e^2}{t_{5/6}^2} [0,618 t_{5/6} - 0,862 t_{1/3}] \end{aligned} \quad (3)$$

2.4 Mesure des propriétés mécaniques

La caractérisation mécanique des échantillons est réalisée par le banc H10KL caractérisé par une capacité de charge maximale de 10 KN.



Fig. 2 : Banc H10KL de mesure des résistances à la compression et flexion.

Les essais de flexion 3 points, ont été réalisés en appliquant une charge croissante au milieu de l'éprouvette, maintenue sur deux supports avec une vitesse de chargement de 50 N/s, jusqu'à sa rupture en deux morceaux distincts. La détermination de la résistance maximale à la flexion (R_f) est déduite de l'équation suivante :

$$R_f = \frac{3 FL}{2 e^3} \quad (4)$$

Les demi-prismes de l'éprouvette obtenus après rupture en flexion seront rompus en compression. La résistance à la compression a été calculée en divisant la force (F) par la surface de l'échantillon (S).

$$R_c = \frac{F}{S} \quad (5)$$

3. Résultats et discussions

3.1 Absorption de l'eau

L'absorption d'eau est utilisée pour déterminer la quantité d'eau absorbée dans des conditions spécifiées. La figure 3 montre l'absorption d'eau des mortiers de ciment simples et des mortiers de cendres avec différentes valeurs de finesse et des taux de remplacement des cendres.

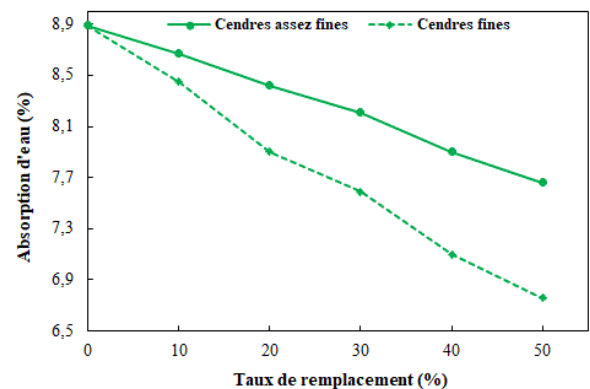


Fig. 3 : Absorption d'eau des mortiers en fonction de dosage et de la finesse des cendres

Comme le montre la figure 3, l'absorption d'eau des mortiers de cendres est inférieure à celle du mortier de ciment simple. Cela indique que l'incorporation des cendres entraîne une réduction de l'absorption d'eau.

De plus, l'absorption d'eau des mortiers de cendres a diminué avec une augmentation du taux de remplacement des cendres. Cela se traduit par la diminution des particules poreuses.

Les résultats montrent également que la réduction est améliorée avec des cendres plus fines en raison de la surface plus lisse des particules de cendres plus fines.

3.2 Conductivité et diffusivité thermiques

L'effet de dosage et de la finesse des cendres sur les propriétés thermiques du mortier de ciment a été étudié particulièrement sur la conductivité et la diffusivité thermiques.

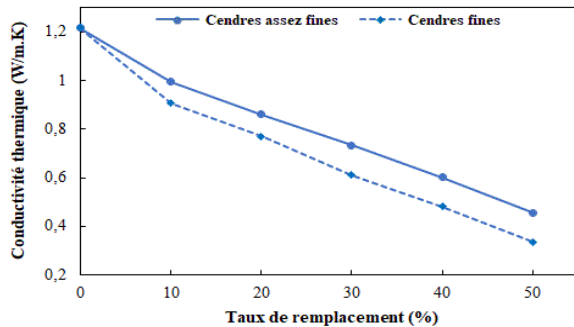


Fig. 4 : Effet de dosage et de la finesse des cendres sur la conductivité thermique du mortier.

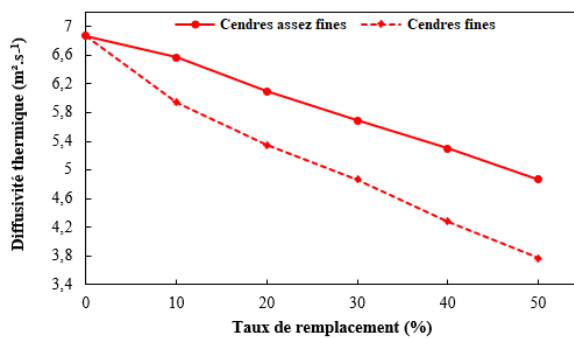


Fig. 5 : Effet de dosage et de la finesse des cendres sur la diffusivité thermique du mortier.

D'après ces résultats, on peut voir que plus le pourcentage de cendres volantes est élevé, plus la conductivité et la diffusivité du mortier de cendres sont faibles. Ceci montre que plus le composite contient des cendres, moins il peut transmettre de chaleur, ce qui lui confère une importance en termes d'isolation thermique. Ces résultats montrent également que les propriétés thermiques des mortiers de cendres décroissent en fonction de la finesse des cendres. Ceci s'explique par le fait que l'augmentation de la finesse s'accompagne d'une augmentation de la porosité du milieu qui engendre un taux d'air important au sein du milieu poreux qui possède des propriétés thermiques inférieures à celles de la matrice solide.

3.3 Résistances à la compression et la flexion

L'effet des cendres sur les propriétés mécaniques du mortier de ciment, à savoir la résistance à la compression et la flexion, ont été analysés.

Les résultats obtenus montrent que le remplacement d'une partie du ciment par les cendres peut améliorer de manière sensible les résistances mécaniques à la

compression et à la flexion surtout pour des valeurs de substitution de l'ordre de 30%. Au-delà de cette valeur, on observe des légères améliorations des résistances.

Il est montré également que plus l'échantillon est fin, plus les performances mécaniques sont élevées. En effet, les éléments les plus fins rassemblent le plus de matière active et possèdent une plus grande vitesse de prise.

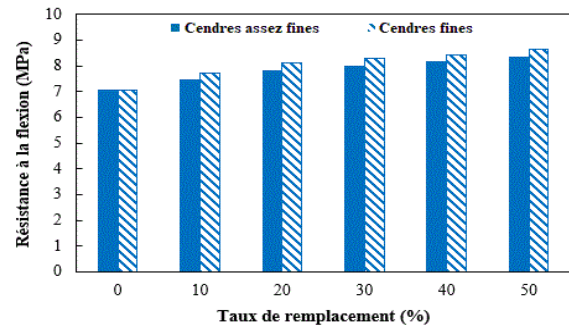


Fig. 6 : Variation de la résistance à la flexion des mortiers de cendres en fonction du taux de dosage.

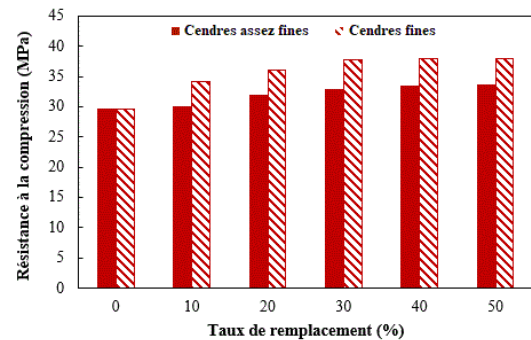


Fig. 7 : Variation de la résistance à la compression des mortiers de cendres en fonction du taux de dosage.

Conclusion

A l'issue de ce travail, il est montré que le mortier de cendres est un composite prometteur par rapport au mortier conventionnel. Sur la base des résultats, la finesse et le taux de remplacement des cendres ont une influence significative sur les propriétés du mortier de ciment. En effet, l'augmentation du pourcentage de cendres fines dans la formulation du mortier améliore leurs propriétés thermomécaniques.

Références

- [1] A., Lachheb et al., *Thermal insulation improvement in construction materials by adding spent coffee grounds: An experimental and simulation study*, Journal of Cleaner Production, 209 (2019), pp. 1411-1419.
- [2] Y. Gao et al., *Investigation on microstructure and microstructural elastic properties of mortar incorporating fly ash*, Cement and Concrete Composites, 86 (2018), pp. 315-321.
- [3] B. Turhan et al., *Properties of mortars with fly ash as fine aggregate*, Construction and Building Materials, 93 (2015), pp. 782-789.
- [4] A. DEGIOVANNI, *Diffusivité et méthode flash*, Revue Générale de Thermique, n°185, (1977), pp. 420-442.

Modélisation théorique et simulation numérique de phénomène de la convection thermosolutale, tenant compte de l'effet Soret et Dufour, dans une cavité poreuse cylindrique remplie d'un nanofluide

M. El HAMMA¹, M.TAIBI^{1,2}, A. RTIBI¹, K. GUERAOU¹ & H. RACHID²

1. Equipe de Modélisation et de Simulation en Mécanique des Fluides, FSR, Université Mohamed V, Rabat.

2. Groupe de Mécanique et Energétique, FSAC, Université Hassan II, Casablanca.

Abstract

In this article, we present a theoretical and numerical study on the phenomena of combined heat and mass transfer (thermosolutal), in a porous, isotropic and saturated media filled with a nanofluid (aluminum nanoparticles) in thermal equilibrium with a basic fluid (water) in a cylindrical enclosure. The side walls of the enclosure are rigid, impermeable and adiabatic while the horizontal walls are maintained at uniform temperature and concentration.

The nanofluid flow occurring in porous layers is described by BRINKMAN-FORCHHEIMER's extended DARCY's law using the BOUSSINESQ approximation. The finite volume method was used to discretize the equations modeling the phenomenon. We thus studied the effect of the various parameters on the rate of heat transfer and concentration.

Résumé

Dans cet article, nous présentons une étude théorique et numérique sur les phénomènes du transfert combiné de chaleur et de masse (thermosolutale), dans un milieu poreux, isotrope et saturé rempli d'un nanofluide (nanoparticules d'aluminium) en équilibre thermique avec un fluide de base (eau) dans une enceinte cylindrique. Les parois latérales de l'enceinte sont rigides, imperméables et adiabatiques tandis que les parois horizontales sont maintenues à température et à concentration uniformes. L'écoulement du nanofluide se produisant dans les couches poreuses est décrit par la loi de Darcy étendue de Brinkman-Forchheimer en utilisant l'approximation de Boussinesq. La méthode des volumes finis a été utilisée pour discrétiser les équations modélisant le phénomène. Nous avons ainsi étudié l'effet des différents paramètres sur le taux de transfert de chaleur et de concentration.

Mots clés : *Nanofluides, Milieux poreux, Thermosolutale, Extension de la loi de Darcy, Effet Soret et Dufour.*

Keywords: *Nanofluid, Porous Media, Thermosolutal Convection, Extension of the Law of Darcy, Soret and Dufour Effects*

1. Introduction

Les phénomènes des transferts de chaleur et de masse dans les cavités remplies de milieux poreux ont été développés ces dernières années à cause de leur diverse application : l'isolation thermique des bâtiments, la migration de mélange en isolant fibreux, la climatisation des locaux, etc., [1, 2].

Beaucoup de chercheurs se sont intéressés à la modélisation des phénomènes de la convection double diffusion, naturelle et artificiel dans les milieux poreux. Le flux convectif dans une cavité remplie de milieu poreux saturés à une large gamme d'application pratique [2, 3, 4, 5, 6].

L'originalité du présent travail réside dans le fait qu'il tient en compte d'une part de l'influence de la géométrie cylindrique de la cavité d'étude et d'autre part, de l'effet Soret et Dufour sur le transfert de chaleur et de masse dans un milieu poreux saturé par un nanofluide métallique de nanoparticules d'aluminiums et fluide de base binaire. Les paramètres de contrôle considérés sont : les nombres de Rayleigh, de Prandtl, le rapport des poussées, le rapport d'aspect géométrique le nombre de Soret, Dufour et la fraction volumique des nanoparticules.

2. Formulation mathématique du problème

La géométrie étudiée est une cavité cylindrique, ayant un rapport d'aspect géométrique : $A = \frac{L}{R}$ remplie par un milieu poreux isotrope et saturé par un nanofluide métallique d'aluminium. Le fluide de base est un fluide binaire. Les propriétés thermo-physiques du nanofluide sont supposées constantes et l'approximation de Boussinesq a été utilisée. L'écoulement du nanofluide

dans les couches poreuses est décrit par le modèle de Darcy-Brinkman-Forchheimer. Les équations, sous forme adimensionnelles, régissant notre problème sont données par :

$$\begin{aligned} \frac{\partial(UT)}{\partial r} + \frac{\partial(WT)}{\partial z} + \frac{UT}{r} &= \frac{\partial^2 T}{\partial r^2} + \frac{\partial^2 T}{\partial z^2} + \frac{1}{r} \frac{\partial T}{\partial r} \\ + \varepsilon Du \left(\frac{\partial^2 C}{\partial r^2} + \frac{\partial^2 C}{\partial z^2} + \frac{1}{r} \frac{\partial C}{\partial r} \right) \\ \frac{\partial(UC)}{\partial r} + \frac{\partial(WC)}{\partial z} + \frac{UC}{r} &= \frac{\varepsilon}{Le} \left(\frac{\partial^2 C}{\partial r^2} + \frac{\partial^2 C}{\partial z^2} + \frac{1}{r} \frac{\partial C}{\partial r} \right) \\ + \frac{\varepsilon}{Le} Sr \left(\frac{\partial^2 T}{\partial r^2} + \frac{\partial^2 T}{\partial z^2} + \frac{1}{r} \frac{\partial T}{\partial r} \right) \\ \frac{1}{\varepsilon^2} \left(\frac{1}{r} \frac{\partial(rU\Omega)}{\partial r} + \frac{\partial(W\Omega)}{\partial z} - \frac{U\Omega}{r} \right) &= -\bar{\lambda} \frac{Pr}{Da} \Omega \\ - \bar{\lambda}^2 R_T Pr \left(\frac{\partial T}{\partial r} + N \frac{\partial C}{\partial r} \right) \\ + \frac{\bar{\lambda} Pr}{\varepsilon} \left(\frac{\partial^2 \Omega}{\partial r^2} + \frac{1}{r} \frac{\partial \Omega}{\partial r} + \frac{\partial^2 \Omega}{\partial z^2} - \frac{\Omega}{r^2} \right) \\ - \frac{C_F}{\sqrt{Da}} |\vec{V}| \Omega + \frac{C_F}{\sqrt{Da}} \left(W \frac{\partial |\vec{V}|}{\partial r} - U \frac{\partial |\vec{V}|}{\partial z} \right) \\ \left\{ \begin{array}{l} U = \frac{1}{r} \frac{\partial \psi}{\partial z} \\ W = -\frac{1}{r} \frac{\partial \psi}{\partial r} \\ \Omega = \frac{1}{r} \frac{\partial^2 \psi}{\partial r^2} - \frac{1}{r^2} \frac{\partial \psi}{\partial r} + \frac{1}{r} \frac{\partial^2 \psi}{\partial z^2} \end{array} \right. \end{aligned}$$

Où il apparaît les nombres sans dimensions, caractéristiques de notre problème, suivants :

$$\begin{aligned} R_T &= \frac{\rho_0(1-\varphi)g\beta_T\Delta T R^3}{\mu_{nf}\alpha_{nf}} \\ Pr &= \frac{\nu_{nf}}{\alpha_{nf}} \quad Da = \frac{K}{R^2} \quad N = \frac{\beta_s\Delta C}{\beta_T\Delta T} \\ Le &= \frac{\alpha}{D} \quad Du = \frac{k_{TC}\Delta C}{\alpha\Delta T} \quad Sr = \frac{k_{CT}\Delta T}{D\Delta C} \quad \bar{\lambda} = \frac{\lambda_{nf}}{\lambda_e} \end{aligned}$$

Conditions aux limites associées

À l'équation précédente établie, on adjoint les conditions aux limites de types thermique, solutale et dynamique.

3. Méthode numérique de résolution

Le problème étudié est régi par un système d'équations aux dérivées partielles fortement non linéaires et couplées et donc sa résolution analytique est pratiquement impossible, sauf dans certains cas particuliers, donc un recours aux méthodes numériques s'avère obligatoire. Pour cela, nous avons opté pour la méthode des volumes finis [9] pour la discrétisation de toutes les équations qui gouvernent notre phénomène. À l'issue de cette discrétisation, on obtient des équations algébriques auxquelles on ajoute les conditions aux limites et on procède à leur résolution par la méthode de double balayage (ADI) [9]. L'équation donnant la

fonction de courant est résolue par la méthode de Sur-Relaxation simultanée *SOR* [7, 8].

4. Résultats et discussion

4.2 Influence du nombre de Rayleigh thermique, R_T

La figure 1, montre l'influence du nombre de Rayleigh thermique, R_T , sur le transfert de chaleur et de masse, pour trois valeurs du paramètre de forme A et pour : $N = 1$, $Le = 10$, $Pr = 0.71$, $Da = 0.001$, $Sr = 1$, $\varphi = 0.05$ et $Du = 1$. Le nombre de Rayleigh thermique R_T , varie de 10 à 10^4 . On remarque sur ces figures, qu'au fur et à mesure que le rapport d'aspect géométrique A augmente les valeurs de Nu et de Sh moyen augmentent aussi. Cet accroissement résulte de l'élévation des forces de volume, thermique et solutale. Les résultats obtenus sont compatibles avec ceux obtenus par d'autres auteurs [6, 7, 8, 9].

4.3 Influence du nombre de Soret et Dufour, Sr et Du

Les figures 2 et 3 montrent respectivement l'influence du nombre de Soret sur le transfert de masse et du nombre de Dufour sur le transfert de l'énergie pour : $R_T = 10^4$, $Pr = 0.71$, $Le = 10$, $A = 3$, $\varphi = 0.05$, $Da = 0.001$ et $Du = 1$ pour différentes valeurs de rapport poussés N . On remarque sur la première figure que le nombre de Sherwood moyen décroît quand le nombre de Soret augmente. Cette augmentation du nombre de Soret conduit à une diminution de diffusivité massique et donc par conséquent la diminution de taux de transfert massique. Ces résultats sont compatibles avec ceux d'autres auteurs [2, 3, 5, 8].

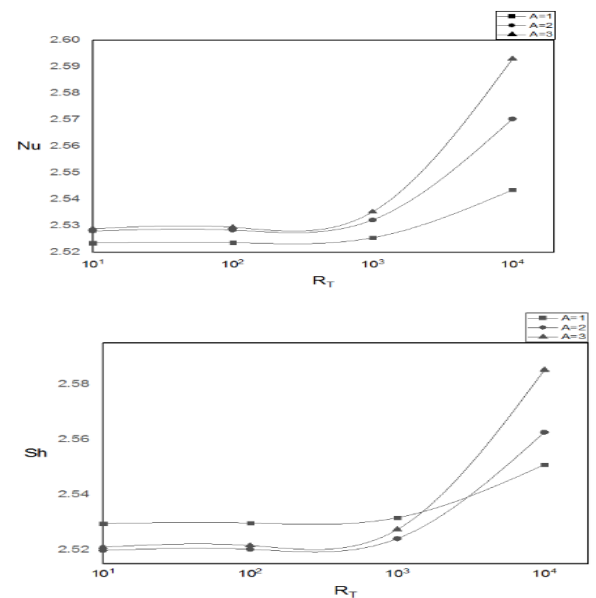


Figure 1 : Influence de R_T sur Nu et Sh pour $N = 1$, $Pr = 0.71$, $Le = 10$, $\varphi = 0.05$, $Da = 0.001$, $Sr = 1$, $Du = 1$

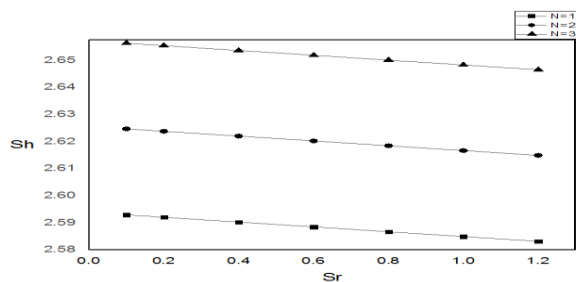


Figure 2 : Influence de Sr sur Sh pour : $A = 3$, $R_T = 10^4$, $Le = 10$, $\varphi = 0.05$, $Pr = 0.71$, $Da = 0.001$, $Du = 1$ et pour trois valeurs de N

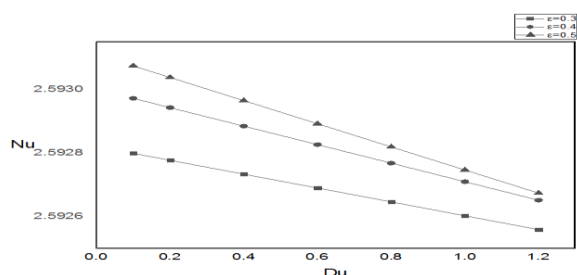


Figure 3: Influence de Du sur Nu pour : $A = 3$, $R_T = 10^4$, $Le = 10$, $\varphi = 0.05$, $Pr = 0.71$, $Da = 0.001$, $Sr = 1$ et pour trois valeurs de porosité

5. Conclusion

À la suite de ce travail on note que les nombres de NUSSELT et de SHERWOOD moyens augmentent avec l'augmentation du nombre de RAYLEIGH. Ceci est expliqué par l'importance de l'intensité de l'écoulement avec l'augmentation du nombre de RAYLEIGH, et par conséquent l'augmentation des nombres des NUSSELT et de SHERWOOD moyens. Cette croissance est plus importante pour des valeurs élevées du nombre de rapport d'aspect géométrique.

Le nombre de NUSSELT moyen diminue avec l'augmentation du nombre de DUFOUR, à cause de la diminution de la diffusivité thermique. Quant au nombre de SHERWOOD moyen, il diminue pour des valeurs plus importantes du nombre de SORET à cause de la diminution de diffusivité massique.

Références

[1] M. A. Sheremet, I. Pop & A. Ishak. Double-Diffusive Mixed Convection in a Porous Open Cavity Filled with a Nanofluid Using Buongiorno's Model. Springer Science Business Media Dordrecht, 2015..

[2] I. Filahi, M. Bourich, M. Hasnaoui & A. Amahmid. Analytical and numerical study of Soret and Dufour effects on thermosolutal convection in a horizontal

Brinkman porous layer with a stress-free upper boundary. Mathematical Problems in Engineering, 2020.

- [3] M. El Hamma, A. Rtibi, M. Taibi, K. Gueraoui, M. Bernatchou. Theoretical and Numerical Study of Thermosolutal Convection in a Cylindrical Porous Cavity Filled with a Nanofluid and Taking into Account Soret and Dufour Effects. International Journal on Engineering Applications (I.R.E.A.). 10. N. 1. pp 56-65. 2022
- [4] I. Pop & D.B. Ingham. Convective Heat Transfer: Mathematical and Computational Modelling of Viscous Fluids and Porous Media". Pergamon, Oxford, 2001.
- [5] N. Hadidi & Y. Ould -Amer. Convection thermosolutale bidimensionnelle dans une cavité poreuse renfermant deux couches poreuses disposées verticalement". Mécanique Appliquée et Théorique Vol. 2, N°4. pp. 385-396, 2011.
- [6] M. Sammouda. Modélisation théorique et numérique du phénomène de la convection naturelle et thermosolutale dans les milieux poreux à porosité variable. Thèse de Doctorat, Université Mohamed V, Morocco, 2012.
- [7] Sheng Chen et al. Double diffusion natural convection in a square cavity filled with Nanofluid. International Journal of Heat and Mass Transfer, 95, pp. 1070–1083, 2016.
- [8] M. El Hamma, M. Taibi, A. Rtibi, K. Gueraoui, M. Bernatchou. effect of magnetic field on thermosolutal convection in a cylindrical cavity filled with nanofluid, taking into accountsoret and dufour effects. JP Journal of Heat and Mass Transfer. 26 , 1-26 2022.
- [9] S.V. Patankar. "Numerical heat transfert and fluid flow". Hemisphere, New York, 1980.

Étude expérimentale des propriétés thermiques des Schistes Bitumineux de TIMAHDIT-Maroc

H. SGHIOURI El Idrissi¹, A. SAMAOUALI¹, M. TAIBI^{1,2} & S. OUHIMMOU²

¹ Université Mohammed V, Faculté des Sciences, Département de Physique, Centre d'Energie, Rabat, Maroc

² Université Hassan II, Faculté des Sciences Ain Chock, Laboratoire de Mécanique, Casablanca, Maroc

Résumé

Ce travail présente une étude expérimentale réalisée sur les échantillons de roches qui sont nommées « Schistes bitumineux » prélevés au gisement de TIMAHDIT au MAROC. Nous nous sommes intéressés dans ce travail à analyser le comportement thermique de ces roches. Aussi dans cette étude, notre intérêt s'est focalisé sur la détermination des propriétés des couches *X, Y et M*, à savoir leur conductivité thermique, leur diffusivité thermique, leur Chaleur spécifique et leur l'effusivité Thermique. La compréhension de processus de la transformation thermique et capacité de stockage en fonction de la variation des pourcentages des matières organique a été étudiée à la base de la détermination de la masse volumique.

Mots-clés : Conductivité thermique – Diffusivité thermique – Chaleur Spécifique – Effusivité Thermique – Masse volumique – Température – Matière organique

Abstract

This work presents an experimental study carried out on the samples of rocks which are named "Bituminous shales" taken from the deposit of Timahdit in Morocco. We are interested in this work to analyze the thermal behavior of these rocks. Also, in this study, our interest focused on the determination of the properties of the X, Y and M layers, namely their thermal conductivity, their thermal diffusivity, their specific heat and their thermal effusivity. The understanding of the thermal transformation process and storage capacity according to the variation of the percentages of organic matter has been studied on the basis of the determination of the density.

Keywords: Thermal conductivity – Thermal diffusivity – Specific heat – Thermal Effusivity – Density – Temperature – Organic matter.

1. Introduction

Face à la diminution des découvertes des champs classiques du pétrole, les groupes pétroliers se tournent vers des ressources non-conventionnelles : les schistes bitumineux. Bien que ces roches soient relativement peu connues, elles sont de plus en plus convoitées. Les schistes bitumineux sont des roches sédimentaires aux grains fins susceptibles de

produire de l'huile en quantité commerciale lorsqu'on les soumet à un traitement de pyrolyse [1, 2].

À travers le Maroc les schistes bitumineux sont présents dans une dizaine de localités, situées dans les régions du Rif, du Moyen-Atlas, du SOUSS, de Tadla, d'Essaouira et d'Errachidia. Les ressources les plus importantes et les plus étudiées sont situées à TIMAHDIT, TARFAYA et TANGER. Dans le cadre de la valorisation supplémentaire de ces ressources énergétiques nationales, nous nous sommes intéressés à caractériser le comportement thermique des schistes bitumineux de gisement de TIMAHDIT. Le développement des proximités physiques basé sur l'analyse de comportement thermique des schistes est d'une importance pratique pour déterminer leur caractère énergétique à savoir la fourniture d'électricité et de chauffage. L'originalité de notre travail provient d'une part, de l'étude de la variabilité de la teneur en matière organique pour les trois couches *M, X et Y* des schistes bitumineux et d'autre part, la détermination de la composition lithologique de ces couches de schistes bitumineux et de leurs propriétés physiques suite à des mesures effectuées sur des échantillons prélevés de différentes couches du gisement. Pour chaque couche, à l'état sec, cinq échantillons sont utilisés pour la détermination de la conductivité thermique λ , de la diffusivité thermique D , l'effusivité thermique E et la Chaleur spécifique C_p . Les méthodes CT-METER et D'ARCHIMEDE sont utilisées respectivement pour déterminer les propriétés thermiques et la masse volumique ρ des schistes.

2. Échantillonnage

2.1. Description Géologique

Le site à Timahdit est situé dans une région montagneuse du moyen atlas marocain à 33, 24°N et 5,06°O avec 1800 m au-dessus du niveau de la mer. Ce gisement est riche en roches volcaniques de type basalte et des schistes bitumineux qui sont constitués de carbonates, de quartz et d'argile associées au kérogène de types II (matière organique) [2, 3, 4]. Selon leur position par rapport au plan de stratification, les schistes bitumineux sont généralement divisés en quatre types de couches lithologiques *M, X, Y et T* qui ont été corrélées sur tout le gisement. En passant d'une couche à l'autre, la variation de la teneur en matière organique qu'ils

contiennent gènère une variation de la quantité d'huiles pouvant être extraites.

2.2 Préparation des échantillons

À l'aide d'une scie électrique, des échantillons de dimensions $4,0\text{ cm} \times 3,5\text{ cm} \times 4,0\text{ cm}$ ont été obtenus en coupant les blocs de roche des trois couches M, X et Y . Pour l'étude du schiste à l'état sec, cinq échantillons par couche ont été séchés sous ventilation pendant 48 heures, dans une étuve électrique à une température constante de 60°C . Ainsi, Quinze échantillons ont été préparés pour réaliser nos tests thermiques.

3. Résultats et discussions

Des analyses par chromatographie ont permis de confirmer que les taux de matière organique MO montre que la couche Y contient plus de matière organique que les couches M et X [2,4]. Des mesures effectuées dans [4, 5, 6] ont permis de montrer le classement des couches par ordre croissant vis-à-vis de teneur en MO et décroissant vis-à-vis de la masse volumique pour chaque couche de la roche étudiée comme le montre le tableau suivant.

Tableau 1 : Classement des couches en fonction de la teneur en Matière Organique et de la masse volumique

Couches	M	X	Y
Matière Organique (%)	15,5	18,5	21,6
Masse Volumique ρ (kg /m ³)	2115	1911	1756

Les résultats obtenus à l'aide de la méthode d'Archimède qu'on a utilisée sont regroupés dans le Tableau 1 ci-dessus pour la masse volumique relative aux trois couches des schistes bitumineux à la température ambiante. On remarque sur ce tableau que le contenu en matière organique des couches M, X et Y augmente respectivement lorsque la masse volumique, ρ décroît. Ces résultats montrent la spécificité de chaque couche des schistes bitumineux. Dans les paragraphes qui suivent nous allons présenter les propriétés thermiques λ, D, E et C_p relatives aux trois couches M, X et Y à l'état sec quand la température varie de la température ambiante jusqu'à 50°C , selon le protocole expérimental exposé plus haut. On a procédé à une valeur moyenne sur les cinq échantillons prélevés aléatoirement pour chaque couche. Les résultats obtenus suite à cette étude, sont visualisés sur les figures 1, 2, 3 et 4.

3.1. Profil de la conductivité thermique

La figure 1, illustre le profil de la conductivité thermique λ en fonction de la température pour les trois couches. La conductivité thermique diminue au fur et à mesure que la température augmente et ceci pour les trois couches. On note que la couche M

favorise le transfert de la chaleur beaucoup plus que les autres couches X et Y . Ainsi la conductivité pour les trois couches est classée comme suit : $\lambda_Y < \lambda_X < \lambda_M$. La masse volumique et la conductivité thermique varient dans le même sens.

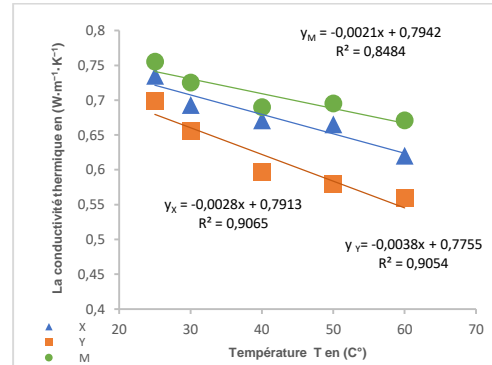


Figure 1 : Évolution de la conductivité thermique en fonction de la température pour les trois couches des Schistes bitumineux : X, Y et M

3.2. Profil de la chaleur Spécifique

On remarque sur cette figure 2 que la chaleur spécifique C_p augmente lorsque la température augmente et ceci pour les trois couches. On note que la chaleur spécifique relative à la couche M est plus grande que celle des deux autres couches. À la base de ce graphique on peut tirer le classement suivant : $C_{pM} < C_{pX} < C_{pY}$. On remarque que la capacité de la couche M à accumuler de l'énergie sous forme thermique est beaucoup plus importante que les autres couches X et Y . Donc une grande capacité thermique signifie un stockage important d'énergie, moyennant une légère augmentation de la température. La variation de la chaleur spécifique est en sens inverse de celle de la masse volumique.

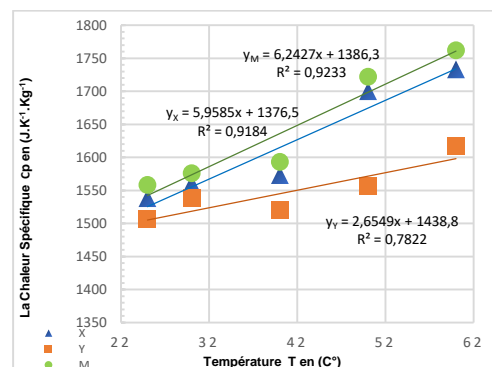


Figure 2 : Variation de la chaleur Spécifique en fonction de la température pour les trois couches des schistes bitumineux : X, Y et M .

3.3. Profil de la diffusivité thermique

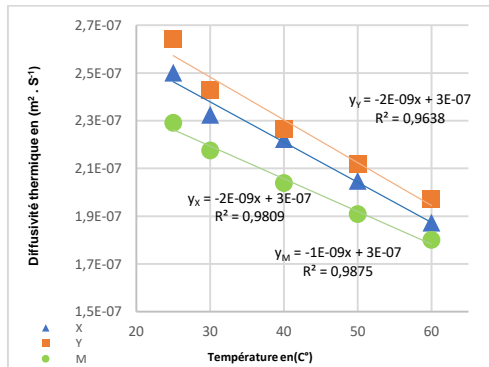


Figure 3 : Évolution de la diffusivité thermique avec la température pour les trois couches des schistes bitumineux : X, Y et M.

Sur la figure 3 on a illustré l'évolution de la diffusivité thermique $D = \frac{\lambda}{\rho \times C_p}$ en fonction de la température, T pour les trois couches M, X et Y. Sur cette figure, on remarque que lorsque la température croît, les valeurs de la diffusivité diminuent d'une manière linéaire et ceux, pour les trois couches. On note aussi que les valeurs de la diffusivité thermique relative à la couche Y sont plus importantes que les deux autres couches. Ceci est justifié car la vitesse de propagation de la chaleur par conduction dans la couche Y est plus rapide que la couche X et Y. Ainsi, on peut dresser le classement de la diffusivité des couches étudiées selon l'inégalité suivante : $D_M < D_X < D_Y$.

Finalement on précise que la masse volumique et la diffusivité thermique varient en sens inverse ; c'est-à-dire, lorsque la masse volumique augmente, la diffusivité diminue et ceux pour toutes les couches.

3.4. Variation de l'effusivité thermique

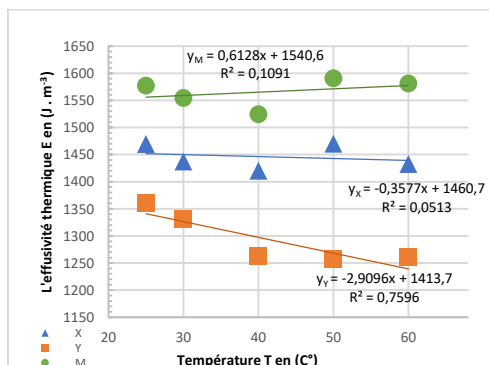


Figure 4 : Variation de l'effusivité thermique avec la température pour les trois couches des schistes bitumineux : X, Y et M

La figure 4 montre la variation de l'effusivité thermique E en fonction de la température pour les trois couches est donnée par l'équation

suivante : $E = \sqrt{\rho \times \lambda \times C_p}$. On note sur cette figure que lorsque la température augmente l'effusivité diminue d'une manière remarquable pour la couche Y, puis une légère diminution pour la couche X et M. La couche M résulte une effusivité plus importante que les deux autres couches, donc la couche M a la capacité d'échanger de chaleur avec son environnement. Lorsqu'on passe de la couche M à la couche Y, l'effusivité thermique diminue de la façon suivante : $E_Y < E_X < E_M$. L'effusivité thermique varie dans le même sens que la masse volumique et la conductivité thermique.

4. Conclusion

Notre étude a porté sur l'analyse du comportement thermique des roches dites « schistes bitumineux » prélevés au gisement de TIMAHDIT-Maroc. Dans cette étude on a mis l'accent sur l'influence de la température sur certaines propriétés comme la conductivité thermique, la diffusivité thermique, la chaleur spécifique et effusivité thermique d'une part et la variation de la matière organique avec la masse volumique d'autre part.

Références

- [1] J.R. Dyni. Oil shale , *BP Statistical Review of World Energy*. Chimie / Chemistry, Vol. 4, pp. 361-366, June 2010.
- [2] A. Saoiabi, A. Doukkali, M. Hamad, A. Zrined, M. Ferhat & Y. Debysier. *Comportement thermique des schistes bitumineux de timahdit(Maroc)*. Comptes Rendus de l'Académie des Sciences - Série IIC - Chimie, Mai 2001, 4 (5): 361-366, DOI: 10.1016 / S1387-1609 (01) 01239-7.
- [3] O. Bekri. *Caractérisation et propriétés physico-chimique des schistes bitumineux de timahdit et tarfaya*. Thèse de l'Université Mohamed V, 1996.
- [4] H. sghiouri El Idrissi, Y. el Rhaffari, A. Samaouali, S. El Alami, Y. Geraud. *Caractérisation des propriétés pétrophysiques des couches de schiste bitumineux de Timahdit au Maroc*. Geosciences, N°10 (9), pp. 337, 2020.
- [5] L. Ely Cheikh Ould Mohamed. *Nouvelle approche de l'étude cinétique non-isotherme de pyrolyse et l'oxydation des Schistes bitumineux d'origines Marocaines*. Thèse de Doctorat, Faculté des Sciences de Rabat, Maroc, 2018.
- [6] M. Hraita, Y. EL Rhaffari, A. Samaouali, Y. Geraud, M. Boukalouch. *Caracterizarea Petrofizica: Petrografica si Mineralogica*. Romanian Journal of Materials ,2014, Vol. 4, pp. 365 – 374.

Optimisation des performances énergétiques d'un bâtiment marocain isolé avec un matériau biosourcé

Energy performances optimization of a Moroccan building insulated with a biobased material

M. DLIMI*, M. TOUIL, R. AGOUNOUN, R. SAADANI, M. RAHMOUNE

Laboratoire d'Etude des Matériaux Avancés et Applications, FSM-ESTM, Université Moulay Ismail, Meknès-Maroc.

*dlimi.maryam@gmail.com

Abstract

This study deals with the integration of thermal insulation in the roof and exterior walls of a Moroccan building. The insulation material studied is hemp wool, an ecological material whose use allows the storage of Carbon and contributes to reduce greenhouse gas emissions. In order to evaluate the dynamic thermal performance of the studied building, a life cycle cost analysis is performed based on the annual heating and cooling requirements specific to the climate of Meknes determined using a numerical code developed under Matlab. Optimal insulation thicknesses, energy savings and payback periods are determined for a 20-year building life.

Résumé

Cette étude porte sur l'intégration de l'isolation thermique dans la toiture et murs extérieurs d'un bâtiment marocain. Le matériau d'isolation étudié est la laine de chanvre, un matériau écologique dont l'utilisation permet le stockage du Carbone et contribue à réduire les émissions de gaz à effet de serre. Afin d'évaluer la performance thermique dynamique du bâtiment étudié, une analyse du coût de cycle de vie est réalisée sur la base des besoins annuelles de chauffage et climatisation spécifiques au climat de Meknès déterminés en se basant sur un code numérique développé sous Matlab. Les épaisseurs d'isolation optimales, les économies d'énergie et les périodes de retour sur investissement sont déterminées pour une durée de vie du bâtiment de 20 ans.

Mots clefs : économies d'énergie ; laine de chanvre ; épaisseur optimale ; période de retour sur investissement.

Keywords: energy savings; hemp wool; optimum insulation thickness; payback period.

1. Introduction

Dans le cadre du développement durable, les nouvelles réglementations thermiques dans le secteur du bâtiment orientent les consommateurs vers l'utilisation des

matériaux d'isolation biosourcés. Contribuant à la réduction des émissions de gaz à effet de serre et au stockage du carbone, ces matériaux sont issus des biomasses végétale et animale. Grâce à sa structure poreuse et à ses propriétés thermiques, le chanvre est de plus en plus utilisé dans l'écoconstruction. Il peut être utilisé comme béton de chanvre à l'intérieur d'une structure à ossature bois, comme laine de chanvre pour l'isolation des murs ou encore comme mortier chaux-chanvre pour l'enduit. Dans la présente étude, nous nous sommes intéressées à la laine de chanvre comme matériau d'isolation biosourcé. En effet, en considérant les besoins annuels de chauffage et de climatisation spécifiques à la ville de Meknès, une analyse du coût de cycle de vie a été menée afin de calculer les épaisseurs économiques optimales pour l'isolation de la toiture et des murs extérieurs d'un bâtiment marocain typique.

2. Analyse et modélisation

2.1 Présentation des configurations étudiées

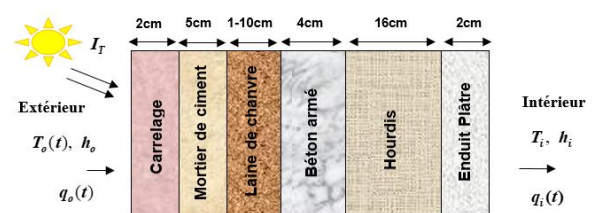


Figure 1 : Structure d'une toiture marocaine typique.

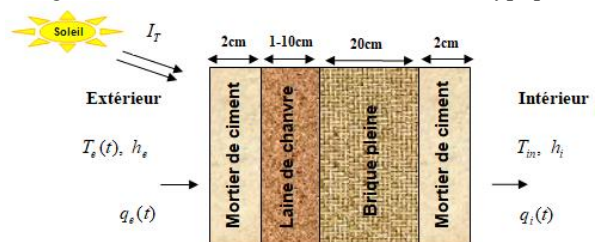


Figure 2 : Structure d'un mur extérieur marocain typique.

Deux structures de murs composites sont considérées : la première concerne une toiture et la seconde décrit une configuration de murs extérieurs typiquement marocains.

Dans les deux structures, une couche d'isolation variant de 1 à 10 cm est introduite et située sur le côté extérieur des murs multicouches.

2.1 Modèle mathématique

Considérant un transfert de chaleur monodimensionnel, aucune génération de chaleur, une résistance d'interface négligeable et des propriétés thermophysiques constantes, le transfert de chaleur en régime transitoire à travers chaque couche de la paroi externe est régi par l'équation de chaleur monodimensionnelle (1D)

$$\text{exprimée par [1] : } k_j \frac{\partial^2 T_j}{\partial x^2} = \rho_j c_j \frac{\partial T_j}{\partial t} \quad (1)$$

Avec k_j , c_j et ρ_j désignent respectivement, la conductivité thermique, la chaleur spécifique et la densité de la couche j . T_j représente la température, x et t sont les coordonnées spatiales et temporelles.

La solution du problème en régime stationnaire est prise comme condition initiale, tandis que les conditions aux limites extérieure et intérieure sont respectivement exprimées par [1] :

$$-k_1 \left(\frac{\partial T}{\partial x} \right)_{x=0} = h_e (T_{sa}(t) - T_{x=0}) \quad (2)$$

$$-k_M \left(\frac{\partial T}{\partial x} \right)_{x=L} = h_i (T_{x=L} - T_{in}) \quad (3)$$

où h_e et h_i représentent respectivement les coefficients d'échanges thermiques externe et interne combinant les effets de convection et de rayonnement. $T_{sa}(t)$ et T_{in} représentent les températures extérieure et intérieure auxquelles notre paroi est soumise. Alors que $T_{x=L}$ et $T_{x=0}$ représentent les températures surfaciques extérieure et intérieure de la paroi multicouche.

L'équation de chaleur monodimensionnelle en régime transitoire est résolue en utilisant la méthode des différences finies et un code numérique développé sous Matlab [2].

La résolution de l'équation de chaleur nous a permis de calculer la variation journalière du flux de chaleur de la surface intérieure de notre paroi et qui est exprimé par [3] : $q_i = h_i (T_{x=L} - T_{in})$ (4)

Ensuite en intégrant le flux de chaleur journalier sur une période de 24h, les besoins de chauffage et de climatisation mensuels sont calculés et puis additionnés pour obtenir les besoins de chauffage et de climatisation annuels.

2.2 Modèle économique : méthode d'analyse du coût de cycle de vie

L'épaisseur économique optimale représente la valeur offrant le coût total le plus bas.

En utilisant les besoins annuels de chauffage et de refroidissement comme données d'entrée, et en se basant sur la méthode de la valeur actuelle prenant en considération la variation des taux d'intérêt d et d'inflation i , un modèle économique est défini pour la détermination de l'épaisseur optimale de l'isolation. Ainsi, pour une durée de vie de bâtiment donnée, le coût de la consommation d'énergie est exprimé par [4] :

$$C_{enr} = \left(PWF \left(\frac{Q_{cl}}{COP} \frac{C_{el}}{(3.6 \times 10^6)} + \frac{Q_{ch}}{H_u \eta_s} C_{el} \right) \right) \quad (5)$$

PWF est le facteur de la valeur actuelle défini comme :

$$PWF = \frac{N}{1+i} \quad i = d \quad (6)$$

Q_{cl} , Q_{ch} , C_{el} et COP représentent respectivement les besoins annuels de climatisation et de chauffage par unité de surface, le coût de l'électricité et les performances du système de climatisation. H_u et η_s représentent respectivement la valeur calorifique et l'efficacité du système.

Le coût total est donc calculé à travers la somme du coût de la consommation d'énergie et du coût du matériau isolant. Le coût total par unité de surface de la paroi est exprimé par [4] :

$$C_t = \left(PWF \left(\frac{Q_c}{COP} \frac{C_{el}}{(3.6 \times 10^6)} + \frac{Q_h}{H_u \eta_s} C_{el} \right) \right) + C_{ins} L_{ins} \quad (7)$$

C_{ins} et L_{ins} décrivent respectivement le coût du matériau isolant par unité de volume et son épaisseur appropriée. Les économies d'énergie résultantes sont calculées à partir de la différence entre le coût total de la paroi sans isolation et le coût total de la paroi isolée avec une épaisseur d'isolation optimale [4]. Les économies d'énergie annuelles sont ensuite obtenues en divisant le résultat de la différence entre les coûts énergétiques de la paroi non isolée et de la paroi avec une épaisseur d'isolation optimale par le facteur de la valeur actuelle PWF . Par ailleurs, le rapport entre le coût d'isolation et les économies d'énergie annuelles définit la période de

$$\text{retour sur investissement } P_b = \frac{C_i \cdot L_{ins}(opt)}{A_s} \quad (8)$$

3. Résultats et discussions

Compte tenu des cinq orientations étudiées, les figures 3,4,5,6 et 7 illustrent la variation du coût de la consommation énergétique, du coût de l'isolation et du coût total en fonction de l'épaisseur de l'isolation.

D'après les allures de chaque courbe tracée, nous remarquons que l'augmentation de l'épaisseur d'isolation entraîne une augmentation linéaire du coût de l'isolation et une diminution du coût de l'énergie. L'épaisseur optimale d'isolation correspond donc à la valeur minimale indiquée par la courbe du coût total représentant la somme des coûts d'énergie et d'isolation. Ainsi, les valeurs des épaisseurs d'isolation optimales sont obtenues égales à 7 cm pour la toiture, 4 cm pour les orientations sud, est et ouest et 2 cm pour le nord.

De plus, des économies d'énergie de l'ordre de (18.11, 6.62, 9.79 1.72 et 10.23) \$/m² ont été réalisées lors de l'intégration des épaisseurs d'isolation optimales dans la toiture et les murs extérieurs orientés respectivement vers le sud, l'ouest, le nord et l'est. Concernant les périodes de retour sur investissement, celles-ci ont été déterminées égales à (3.03, 7.15, 5.51, 20.42 et 5.34) années pour la toiture et les différentes orientations respectives des murs extérieurs.

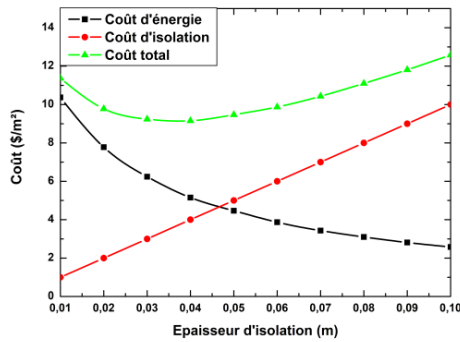


Figure 3 : Variation des coûts pour l'orientation Sud.

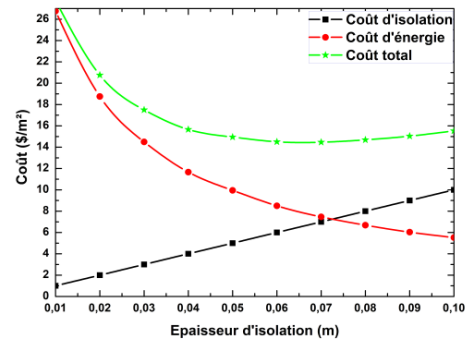


Figure 7 : Variation des coûts pour la toiture.

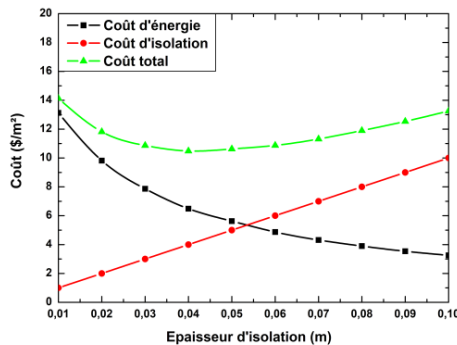


Figure 4 : Variation des coûts pour l'orientation Ouest.

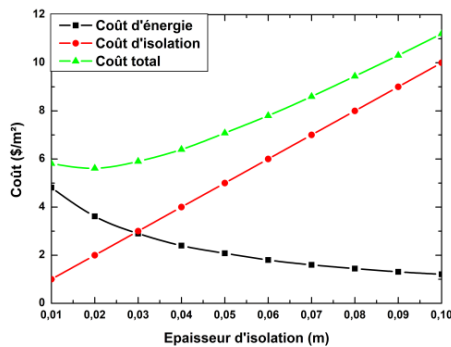


Figure 5 : Variation des coûts pour l'orientation Nord.

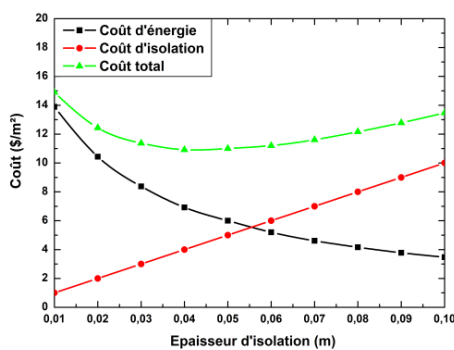


Figure 6 : Variation des coûts pour l'orientation Est.

Conclusion

Dans cette étude, nous avons évalué la performance thermique dynamique d'un bâtiment marocain dont la toiture et les murs extérieurs ont été isolés avec de la laine de chanvre. En effet, en se basant sur la méthode d'analyse du coût de cycle de vie, les épaisseurs économiques optimales, les économies d'énergie et les périodes de retour sur investissement ont été déterminées en utilisant les besoins annuels de chauffage et de climatisation spécifiques à la ville de Meknès comme données d'entrée. L'épaisseur d'isolation optimale la plus faible a été obtenue par le mur orienté vers le nord. Néanmoins, cette orientation offre des économies d'énergie négligeables et une période de retour sur investissement très élevée. Finalement, la configuration de toiture optimisée présente un coefficient de transmission $U=0.46 \text{ W/m}^2\cdot\text{K}$ satisfaisant ainsi les exigences de la réglementation thermique de construction marocaine qui impose un coefficient de transmission pour la toiture $U=0.65 \text{ W/m}^2\cdot\text{K}$.

Références

- [1] Threlkeld J.L 1998 *Thermal Environmental Engineering*, Englewood Cliffs, NJ: Prentice-Hall.
- [2] Ozisik M.N 1994 *Finite Difference Methods in Heat Transfer*, Boca Raton, Florida: CRC Press.
- [3] M. Dlimi et al. *Effect of the integration of hemp wool as an insulation material for the construction of the roof and external walls of a typical Moroccan building*, IOP Conf. Ser.: Mater. Sci. Eng., 783, 2020.
- [4] M. Dlimi, *Modélisation et caractérisation numérique du comportement thermique d'enveloppes de bâtiments isolés par un matériau d'isolation biosourcé*, Thèse de doctorat, Université Moulay Ismail, 2020.

ÉTUDE NUMÉRIQUE DE LA CONDENSATION D'ÉTHANOL EN FILM LIQUIDE DANS UNE MEMBRANE HYDROPHILIQUE LE LONG D'UN CANAL VERTICAL

Amir BOUSSOUF¹, Nabila LABSI¹, M'barek FEDDAOUI², Youb Khaled BENKAHLA¹,
Karima SELLAMI¹

(boussouf_amir@yahoo.fr)

1. Laboratoire des Phénomènes de Transfert, FGMGP, USTHB, Bab Ezzouar, Alger, Algérie.

2. LGEMS, Ecole Nationale des Sciences Appliquées, d'Agadir, Université Ibn Zohr, Maroc.

Résumé

L'objectif de cette étude numérique est l'amélioration du transfert massique et thermique lors de la condensation d'éthanol en film liquide dans une membrane poreuse le long d'un canal vertical. La paroi poreuse est supposée isotherme alors que l'autre est sèche et adiabatique. Nous avons examiné les effets de la porosité, l'épaisseur de la membrane et des conditions ambiantes sur le débit du condensat obtenu. Les résultats numériques montrent que le débit d'éthanol liquide obtenu à la sortie du canal pendant la condensation est augmenté par la présence de la membrane hydrophilique accolée à la paroi.

Mots clés : Canal vertical, Condensation, Membrane fine, Éthanol, Paroi poreuse, film liquide, Condensat.

1. Introduction

La condensation se produit lorsque la vapeur est refroidie en dessous de sa température de saturation et que les molécules de vapeur subissent un changement de phase. Ce phénomène se produit dans de nombreuses applications du génie thermique, par exemple dans la réfrigération, les moteurs chimiques, le dessalement et les batteries combustibles.

Des études expérimentales sur la condensation de vapeur pure par convection forcée sur des plaques verticales et inclinées recouvertes d'une couche poreuse ont été effectuées par Renken et al. [1, 2]. Leurs résultats montrent que l'existence d'une couche poreuse contribue à augmenter les coefficients de transfert thermique par rapport à la surface classique non recouverte.

Une étude numérique de la condensation en film sur une plaque verticale poreuse a été effectuée par Ma et Wang [3]. Ils ont montré que l'épaisseur du film de condensat et le nombre de Nusselt local augmentent avec l'épaisseur du revêtement poreux.

Chaynane et al. [4] ont réalisé une étude numérique et analytique de la condensation en film liquide d'une vapeur pure saturée isotherme sur une

paroi poreuse d'une plaque inclinée. Ils ont montré l'importance de l'utilisation d'une couche poreuse sur l'amélioration des échanges thermiques.

El Hammami et al. [5] ont modélisé les transferts de chaleur et de masse qui régissent la condensation en film laminaire d'une vapeur d'eau dans un canal incliné à paroi poreuse. Ils ont montré que la configuration verticale favorise la condensation de la vapeur d'eau en film liquide. Ils ont aussi montré que le taux de condensation à la sortie du canal diminue avec l'épaisseur de la couche poreuse.

Al-Nimr and AlKam [6] ont mené une étude de la condensation en film le long d'une plaque verticale dans un milieu poreux. Ils ont développé une relation de l'épaisseur du film ainsi que du coefficient de transfert thermique.

L'objectif de la présente étude est d'améliorer la quantité du condensat obtenu lors de la condensation d'un mélange gazeux (éthanol-air) en film liquide (éthanol) en utilisant une membrane fine hydrophilique en Polytétrafluoroéthylène (PTFE) à l'une des parois du canal vertical. Les effets de la porosité, l'épaisseur de la membrane fine et des conditions ambiantes sur le débit du condensat obtenu sont analysés.

Nomenclature

C_p	Chaleur spécifique	$[J.kg^{-1}.K^{-1}]$
D	Diffusivité massique	$[m^2.s^{-1}]$
d	Épaisseur de membrane	$[m]$
g	Accélération gravitationnelle	$[m.s^{-2}]$
H	Largeur du canal	$[m]$
h_{fg}	Chaleur latente	$[J.kg^{-1}]$
K	Perméabilité de la membrane	$[m^2]$
k	Conductivité thermique	$[W.m^{-2}.K^{-1}]$
L	Longueur du canal	$[m]$
P	Pression	$[Pa]$
T	Température	$[K]$
U_0	Vitesse d'entrée des gaz	$[m.s^{-1}]$
V	Vitesse	$[m.s^{-1}]$
W	Fraction massique de la vapeur	

Lettres Grec

ε	Porosité	
Γ	Débit massique du liquide	[kg.m ⁻¹ .s ⁻¹]
ρ	Masse volumique	[kg.m ⁻³]
μ	Viscosité dynamique	[Pa.s]
Φ	Saturation de l'air en éthanol	

Indices

0	À l'entrée du canal	
e	Effective	
I	Interface gaz-liquide	
L	Liquide	
m	Mélange air-éthanol	
s	Solide	
w	Paroi	
(x,y)	Coordonnée cartésienne	[m]

2. Formulation mathématique

Le problème concerne un canal vertical de deux parois parallèles, séparées par la distance $H = 0.03$ m, la paroi gauche est recouverte avec une membrane fine poreuse hydrophilique en PTFE d'épaisseur d . La deuxième paroi est adiabatique et sèche. Un flux d'air-éthanol descendant laminaire dans un canal de longueur $L = 1$ m avec une vitesse d'entrée uniforme U_0 , une température T_0 , et une la saturation de l'air en éthanol Φ_0 (figure 1).

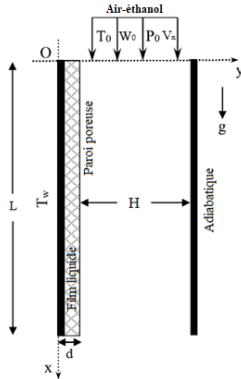


Figure 1 : Schématisation du problème physique

2.1. Equations générales de conservation

• Dans le mélange gazeux

$$\frac{\partial(\rho_m V_x)}{\partial x} + \frac{\partial(\rho_m V_y)}{\partial y} = 0 \quad (1)$$

$$\frac{\partial(\rho_m V_x^2)}{\partial x} + \frac{\partial(\rho_m V_x V_y)}{\partial y} = -\frac{\partial p}{\partial x} + \rho_m g + \frac{\partial}{\partial x} \left(\mu_m \frac{\partial V_x}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu_m \frac{\partial V_x}{\partial y} \right) \quad (2)$$

$$\frac{\partial(\rho_m V_y V_x)}{\partial x} + \frac{\partial(\rho_m V_y^2)}{\partial y} = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left(\mu_m \frac{\partial V_y}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu_m \frac{\partial V_y}{\partial y} \right) \quad (3)$$

$$\frac{\partial(\rho_m V_x C_p T)}{\partial x} + \frac{\partial(\rho_m V_y C_p T)}{\partial y} = \frac{\partial}{\partial x} \left(k_m \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(k_m \frac{\partial T}{\partial y} \right) \quad (4)$$

$$\frac{\partial(\rho_m V_x W)}{\partial x} + \frac{\partial(\rho_m V_y W)}{\partial y} = \frac{\partial}{\partial x} \left(\rho_m D_m \frac{\partial W}{\partial x} \right) + \frac{\partial}{\partial y} \left(\rho_m D_m \frac{\partial W}{\partial y} \right) \quad (5)$$

• Dans le milieu poreux

$$0 = -\rho_L g + \frac{\partial}{\partial y} \left(\frac{\mu_L}{\varepsilon} \frac{\partial V_x}{\partial y} \right) - \frac{\mu_L V_x}{K} - \frac{\rho_L C}{\sqrt{K}} V_x^2 \quad (6)$$

$$\frac{\partial}{\partial x} (\rho_L V_x C_{p,L} T) = \frac{\partial}{\partial y} \left(k_e \frac{\partial T}{\partial y} \right) \quad (7)$$

La conductivité thermique effective k_e est définie comme suit : $k_e = \varepsilon k_L + (1 - \varepsilon) k_S$

La conductivité thermique de la membrane k_S est calculée comme suite [7] :

Les constantes α et β pour des différents types de membrane, sont données dans le tableau 1.

Tableau 1 : Les constantes α et β pour des différents types de membrane [8].

Polymère	α	β
Polypropylène	12.5	-23.5
Polyethersulfone	4.17	1.45
Polyfluorure de vinylidène	5.77	0.914
Polytétrafluoroéthylène	5.77	8.914

2.2. Conditions aux limites

A l'entrée du canal : $x = 0$

Pour le mélange gazeux $d < y < d+H$

$$V_{x,m} = U_{0,m}, \quad V_{y,m} = 0, \quad T = T_{0,m}, \quad \phi = \phi_{0,m} \quad (8)$$

$$\text{Pour le liquide } 0 < y < d, \quad T = T_{0,L}, \quad \Gamma_{0,L} = 0 \quad (9)$$

Sur les parois :

$$y = 0, \quad V_{x,L} = 0, \quad \left. \frac{\partial T}{\partial y} \right|_{y=0} = 0 \quad (10)$$

$$y = d+H, \quad V_{x,m} = V_{y,m} = 0, \quad \left. \frac{\partial T}{\partial y} \right|_{y=d+H} = \left. \frac{\partial W}{\partial y} \right|_{y=d+H} = 0 \quad (11)$$

A l'interface liquide-air : $y = d$

$$V_{x,I} = V_{x,I,m} = V_{x,I,L}, \quad T_I = T_{I,m} = T_{I,L} \quad (12)$$

$$\tau_I = \left[\mu \frac{\partial V_x}{\partial y} \right]_{I,L} = \left[\mu \frac{\partial V_x}{\partial y} \right]_{I,m} \quad (13)$$

$$\left[-k_e \frac{\partial T}{\partial y} \right]_{I,L} = \left[-k \frac{\partial T}{\partial y} \right]_{I,m} - \dot{m}_I h_{fg} \quad (14)$$

A la sortie du canal : $d < y < d+H$ et $x = L$

$$\frac{\partial V_x}{\partial x} = \frac{\partial V_y}{\partial x} = \frac{\partial T}{\partial x} = \frac{\partial W}{\partial x} = 0 \quad (15)$$

Les bilans massiques globaux à satisfaire, défini comme :

$$\int_d^{d+H} \rho_g V_x dy = \rho_{0g} U_0 H - \int_0^L \rho V_c dx$$

3. Résolution numérique

Les équations gouvernantes ainsi que les conditions aux limites associées sont discrétisées avec la méthode des volumes finis proposée par Patankar [9], résolues en utilisant un balayage ligne par ligne. Le couplage pression-vitesse est traité avec l'algorithme SIMPLER [9].

4. Résultats et discussions

La Figure 2 montre l'influence de l'épaisseur d et de la porosité ε de la membrane sur le débit du liquide condensé m_c avec des conditions d'entrée du gaz. Ces profils montrent que le débit condensé augmente considérablement le long du canal sauf pour la porosité $\varepsilon = 0.5$ le débit se stabilise à mi-chemin du canal. On note aussi qu'une augmentation de l'épaisseur de la membrane améliore le débit condensé et peut atteindre

84% lorsqu'on double l'épaisseur. L'effet de la porosité montre également une amélioration du débit condensé en particulier pour les membranes à porosité élevée.

La Figure 3, vise à identifier l'effet de deux paramètres, le nombre de Reynolds Re et la pression du gaz à l'entrée P sur l'évolution du débit du condensat en fonction de la longueur du canal.

Le débit condensé m_c est plus favorisé à basse pression à l'entrée et pour des nombres de Reynolds élevés.

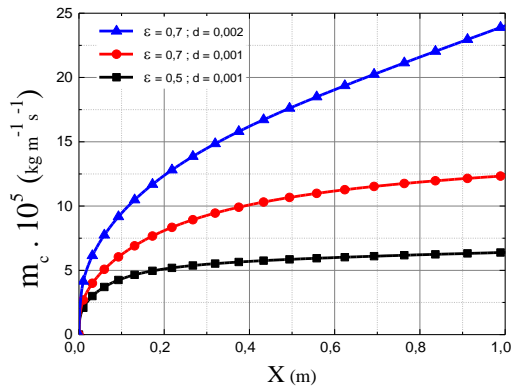


Figure 2 : Evolution du débit de condensat cumulé pour différentes valeurs d'épaisseur et de porosité de la membrane ($T_0 = 40^\circ\text{C}$, $Re = 1000$, $\Phi_0 = 0.6$, $P = 1$ atm)

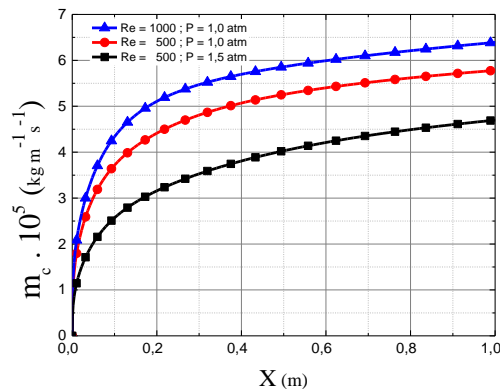


Figure 3 : Evolution du débit de condensat cumulé pour différentes valeurs de Re et de la pression du gaz à l'entrée ($T_0 = 40^\circ\text{C}$, $\Phi_0 = 0.6$, $\varepsilon = 0.5$, $d = 0.001$ m)

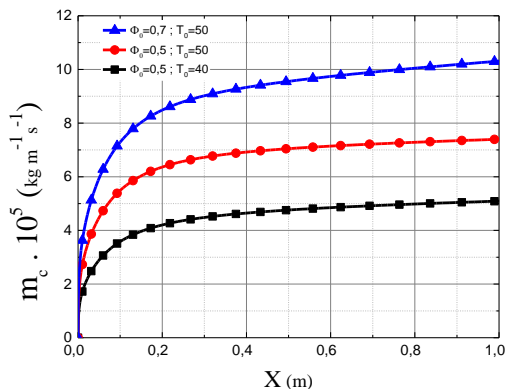


Figure 4 : Evolution du débit de condensat cumulé pour différentes valeurs de température et la saturation de l'air en éthanol à l'entrée ($Re = 1000$, $\varepsilon = 0.5$, $P = 1$ atm, $d = 0.001$ m)

La Figure 4 présente l'impact de la saturation de l'air en éthanol Φ_0 et de sa température T_0 à l'entrée du canal sur le débit du condensat cumulé. Les résultats montrent que la condensation d'éthanol en liquide est favorisée lorsque la saturation et sa température sont augmentées.

5. Conclusion

Le travail présent est une étude numérique de la condensation du vapeur d'eau en film liquide dans une membrane poreuse. Les effets de la porosité, l'épaisseur de la membrane et des conditions d'entrées sur le débit du condensat sont examinés. Les principaux résultats sont brièvement résumés comme suit :

- L'augmentation de l'épaisseur de la membrane d , la porosité ε , le nombre de Reynolds Re , la température T_0 et la saturation Φ_0 de l'air en éthanol à l'entrée du canal favorise la condensation.
- Lorsque le système opère à pression élevée le débit condensé est défavorisé.

Références

- [1] K.J. Renken, C.D. Mueller, *Measurements of enhanced film condensation utilizing a porous metallic coating*, J Thermophys Heat Transf, Vol. 7, (1993), P. 148 – 152.
- [2] K.J. Renken, M. Aboye, *Analysis of film condensation within inclined thin porous layer coated surfaces*, Int J Heat Fluid Flow, Vol. 14, (1993), P. 48 – 53.
- [3] X. Ma, B. Wang, *Film condensation heat transfer on a vertical porous layer coated plate*, J Sci China Technol Sci, Vol. 41, (1998), P. 169 – 175.
- [4] R. Chaynane, M. Asbik, H. Boushaba, B. Zeghmami, A. Khmou, *Study of laminar film condensation on the wall of an inclined porous plate*, Mech Industry, Vol. 5, (2004), P. 381 – 391.
- [5] Y. El hammami, M. Feddaoui, T. Mediouni, S. Senhaji, A. Mir, *Numerical simulation on mixed convection heat and mass transfers in evaporating alcohol liquid film*, Revue International d'Héliotechnique, Vol. 42, (2010), P 38 – 43.
- [6] M. A. Al-Nimr, M. K. Alkam, *Film condensation on a vertical plate imbedded in a porous medium*, Appl Energy, Vol. 56, (1997), P. 47 – 57.
- [7] S. O. Olatunji, L. M. Camacho, *Heat and Mass Transport in Modeling Membrane Distillation Configurations: A Review*. Frontiers in Energy Research, Vol. 6, (2018).
- [8] I. Hitsov, T. Maere, K. De Sitter, C. Dotremont and I. Nopens, *Modelling approaches in membrane distillation: a critical review*, Separation and Purification Technology, Vol. 142, (2015), P. 48 –64.
- [9] S.V. Patankar, *Numerical heat transfer and fluid flow*, Hemisphere, 1980.

Etude du comportement thermique des nanofluides (NTC/Glycérol) à partir de leurs propriétés thermo-physiques

S. LAHLOU¹, R. SEHAQUI¹

1. Department of Physics, Faculty of Sciences Ain Chock, Hassan II University, 20470 Casablanca, Morocco
sara.lahlou20@gmail.com

Résumé

L'utilisation des nanofluides comme fluide caloporteur est un nouveau domaine encore en phase de recherche. Plusieurs paramètres influençant l'échange thermique entrent en jeu tels que la taille, la forme...

Dans notre présente étude, nous nous sommes intéressés à l'étude de l'effet des nanotubes de carbone en suspension avec du glycérol sur les performances thermiques d'un échangeur à serpentin. On a comparé au début deux types de fluides NTC/Glycérol et eau afin d'identifier la différence en terme de transfert de chaleur. Ensuite, l'effet de la dispersion thermique dans l'amélioration du transfert de chaleur. Les résultats prometteurs de cette étude montrent que l'utilisation de nanofluides à base de nanotubes de carbone offre clairement une amélioration des performances thermiques par rapport aux fluides de base conventionnels.

Keywords: *Nanofluid, thermal behavior, serpentine exchanger, heat transfer fluids*

1. Introduction

Les fluides caloporteurs conventionnels usuels tels que l'eau, l'éthylène-glycol et l'huile sont caractérisés par des propriétés thermiques relativement limitées (des conductivités thermiques très faibles) qui limitent parfois leur capacité de transfert de chaleur, ce qui réduit l'efficacité des systèmes thermiques mis en jeu.

Xie et al. [1] et Xie et Chen [2] ont mesuré expérimentalement la conductivité d'un nanofluide à base de MWCNT dispersés dans de l'éthylène-glycol pour des concentrations volumiques de 0.2%, 0.6% et 1%. Leurs résultats montrent une amélioration de la conductivité thermique maximale de l'ordre de 12% à la fraction volumique 1%.

Le domaine des nanofluides est un champ de recherche très vivant et leur application dans les processus industriels devient de plus en plus répandue pour leurs remarquables propriétés optiques, magnétiques, diélectriques ou électromagnétiques. L'influence d'un certain nombre de paramètres, tels que la taille, la forme ainsi que les phénomènes aux interfaces entre liquide et particules sont encore mal comprises et caractérisées.

Dans notre présent travail, nous nous sommes intéressés à l'apport potentiel des nanoparticules dans le domaine des transferts thermiques. Le fil directeur de ce projet est d'étudier l'effet des nanofluides à base de nanotubes de carbone (NTC) en suspension avec le glycérol sur les performances thermiques dans un échangeur de chaleur à

serpentin en modifiant les propriétés thermo physiques des fluides caloporteurs en injectant les nanotubes.

La deuxième partie a été consacrée aux équations et corrélations qui vont nous permettre de décrire les performances thermiques dans l'échangeur de chaleur à Serpentin en calculant les principaux paramètres influençant le transfert thermique après une étude expérimentale qui a été faite dans l'échangeur de chaleur à serpentin pour évaluer l'effet de l'utilisation des nanofluides sur les performances thermiques de l'échangeur.

2. Matières premières et matériels

2.1 Matières premières

Pour le développement des nanofluides testés dans ce travail, nous pensons aux produits suivants :

- Nanotube de carbone multicouche (pureté > 95%)
- Glycérol, liquide choisi comme fluide caloporteur dans laquelle les nanoparticules peuvent rester en suspension stable grâce à sa viscosité élevée.
- Eau pure

2.2 Protocole d'élaboration

Le nanofluide obtenu à partir des étapes suivantes :

- Ajout de la poudre de nanotubes de carbone dans le glycérol tout en continuant de chauffer et de remuer en présence d'une agitation magnétique de 10 tours/sec pendant 1 heure selon les différentes fractions volume souhaité : [0.1%-1%] et les différentes températures [30-80°C].

La suspension est soumise à une agitation magnétique à l'aide d'un bain thermostaté.

2.3 Matériels et méthodes :

Les tests thermiques sont menés à l'aide d'un échangeur à serpentin pour assurer la caractérisation thermique.



Surface d'échange	Coefficient d'échange k_c	Diamètre moyenne D_H	Diamètre intérieure D	Diamètre extérieure d_0
0.6 (m ²)	238 (w/m ² .°C)	267 (mm)	30 (mm)	274 (mm)

3 Résultats et discussions :

Une étude expérimentale dans un échangeur de chaleur à Serpentin a été menée afin d'évaluer l'effet de l'utilisation de nanofluides à base de nanotubes de carbone en suspension avec le glycérol sur les performances thermiques de l'échangeur.

3.1 Etude préliminaire :

Lors de cette première étude, l'effet de l'antigel (glycérol dans notre cas) ainsi que l'effet des nanotubes de carbone va être étudié afin de déterminer leur influence sur le coefficient d'échange convectif h en fonction de la fraction volumique. Afin de pouvoir identifier cela, on va d'abord comparer les nanofluides NTC/Glycérol et NTC/Eau pure puis on va étudier l'effet des NTC sur l'amélioration du coefficient h pour différentes fractions volumiques et ceci en comparant trois types de nanoparticules à savoir : NTC/Glycérol, AL2O3 /Glycérol et Cu/Glycérol.

Le tableau 1 représente les caractéristiques thermophysiques des nanoparticules ainsi que les fluides de base qu'on va étudier :

	K(W/m.K)	C_p (J/Kg.K)	ρ (Kg/m ³)	μ (Pa.s)
NTC	5000	900	1800	
Glycérol	0.3	2430	1109	1.49
Eau	0.604	4186	1000	0.001
Al ₂ O ₃	40	765	3970	
Cu	400	385	8933	

Tableau1 : les caractéristiques thermophysiques des nanoparticules/FB

3.1.1 Effet du glycérol :

Afin d'étudier l'effet du glycérol sur le coefficient d'échange h , on va tracer la variation du coefficient h en fonction de la fraction volumique ϕ pour les deux cas : NTC/Glycérol et NTC/Eau. Ceci nous permettra de cerner la différence de l'influence des deux types de fluides caloporteurs sur h et déterminer le meilleur choix pour améliorer le transfert de chaleur dans l'échangeur.

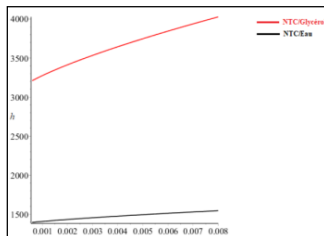


Figure 1 : Variation du coefficient d'échange convectif h en fonction de la fraction volumique ϕ dans une bobine hélicoïdale pour le nanofluide NTC/Glycérol et le nanofluide NTC/Eau

La figure 1 représente la variation du coefficient d'échange convectif h en fonction de la fraction volumique ϕ dans une bobine hélicoïdale pour le nanofluide NTC/Glycérol et le nanofluide NTC/Eau.

Les résultats montrent, après comparaison entre les deux types de nanofluides NTC/Glycérol et NTC/Eau (Pur) que, d'une part, le coefficient d'échange convectif augmente linéairement en fonction de la fraction volumique pour les deux types de nanofluides. D'autre part, que, l'amélioration du coefficient d'échange convectif est plus importante dans le cas de l'utilisation de NTC/Glycérol.

L'augmentation du coefficient d'échange convectif h en fonction de la fraction volumique est plus prononcée dans le cas de NTC/Glycérol par rapport à l'utilisation de NTC/Eau, ce qui confirme que les performances thermiques sont plus prononcées dans le cas où le glycérol est le fluide caloporteur. Ceci est dû à la grande viscosité dynamique du glycérol (par rapport à l'eau pure) et sa conductivité thermique réduite.

3.1.2 Effet de nanotubes de carbone :

Afin de cerner le rôle des NTC et son influence sur le coefficient d'échange h , on va étudier l'évolution du coefficient d'échange convectif h dans une bobine hélicoïdale pour différents types de nanoparticules en gardant le glycérol comme fluide de base. Les nanoparticules utilisées seront : Cu, Al2O3 ainsi que le glycérol.

Le coefficient de transfert thermique total est calculé selon la relation suivante :

$$H = \left(\frac{1}{h_h} + \frac{1}{h_{io,nf}} + \frac{x}{k_c} + R_t + R_\alpha \right)^{-1}$$

Avec : R_t : Le facteur d'encrassement du côté du tube.

R_α : le facteur d'encrassement du côté de la coque.

x : l'épaisseur de la paroi de la bobine.

K_c : le coefficient d'échange convectif du côté de la bobine.

h_h : le coefficient d'échange convectif du fluide de base (le glycérol).

$h_{io,nf}$: le coefficient de transfert de chaleur à l'intérieur de la bobine en fonction du diamètre extérieur de la bobine, tel que :

$$h_{io,nf} = h_{ic,nf} \left(\frac{D}{d_0} \right) \quad x = (d_0 - D)/2$$

$h_{ic,nf}$: le coefficient de transfert de chaleur du tube circulaire.

Avec d_0 et D le diamètre extérieur et intérieur de la bobine respectivement.

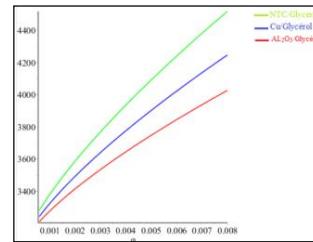


Figure 2 : Variation du coefficient d'échange convectif h en fonction de la concentration des nanoparticules pour différents types de nanofluides : NTC/Glycérol, Al2O3/Glycérol, Cu/Glycérol.

La figure 2 représente la variation du coefficient d'échange convectif h en fonction de la concentration des nanoparticules pour différents types de nanofluides NTC/Glycérol, Al2O3/Glycérol, CU/Glycérol. Les résultats montrent que le coefficient d'échange convectif augmente avec l'augmentation de la concentration des nanoparticules, ceci montre d'avantage l'importance de l'utilisation des nanofluides d'une part. D'autre part, l'amélioration du coefficient d'échange convectif est plus importante dans le cas du nanofluide NTC/Glycérol par rapport aux autres types étudiés. Ceci explique le choix des NTC comme nanoparticules dans ce projet.

3.2 Etude numérique des performances thermiques du nanofluide :

Afin d'étudier l'effet de la dispersion thermique sur l'intensification du transfert de chaleur des nanofluides à base de NTC, une étude numérique sur les caractéristiques du nanofluide NTC/Glycérol (Type : NC700/ diamètre :9.2nm) a été réalisée. Pour ce faire, on étudiera l'évolution du nombre de Nusselt en fonction du nombre de Reynolds pour différentes fractions volumiques dans une bobine hélicoïdale, et ceci en régime turbulent.

3.2.1 Approche conventionnelle :

Dans le but d'étudier les performances thermiques de nanofluides en régime turbulent et de voir l'influence de la dispersion thermique sur le transfert de chaleur, on commence par l'approche conventionnelle. Dans cette approche, l'effet de la dispersion thermique des nanoparticules ainsi que l'effet moléculaire sont négligés, le nanofluide est considéré alors comme fluide pur.

En régime turbulent, le nombre de Nusselt dans un tube circulaire est évalué selon la corrélation de Colburn :

$$Nu_D = 0.023.Pr^{1/3}Re_D^{0.8}$$

Le coefficient d'échange convectif dans une bobine hélicoïdale hic,nf est calculé comme suit :

$$h_{ic,nf} = h_{inf}(1 + 3.5 \frac{D}{D_h})$$

Avec D_h le diamètre moyen de l'hélice de l'échangeur de chaleur à Serpentin et D le diamètre intérieur.

En utilisant ces expressions, on va étudier la variation du nombre de Nusselt en fonction du nombre de Reynolds pour différentes fractions volumiques allant de 0.0006 % jusqu'à 0.006 % pour le nanofluide NTC/Glycérol utilisé dans notre étude, puis on le comparera avec l'eau pure afin de voir l'effet de l'ajout de NTC/Glycérol sur les performances thermiques dans l'échangeur de chaleur à serpentin.

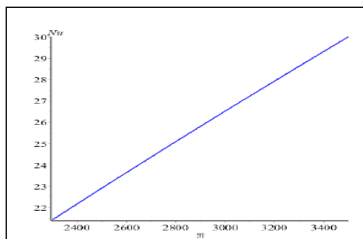


Figure3: Evolution du nombre de Nusselt en fonction de Reynolds pour l'eau pure avec ϕ variant de 0.0006% jusqu'à 0.006% en régime turbulent.

On remarque dans la figure 3 que le nombre de Nusselt augmente significativement en fonction de Reynolds, ce qui montre l'influence du régime d'écoulement sur le transfert thermique.

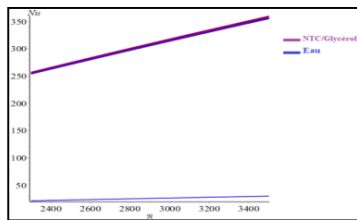


Figure 4: Variation du nombre de Nusselt en fonction de Reynolds pour l'eau et le nanofluide NTC/glycérol avec ϕ variant de 0.0006% jusqu'à 0.006% en régime turbulent.

La courbe sur la figure 4 décrit la variation de Nusselt en fonction de Reynolds pour l'eau pure et le nanofluide NTC/Glycérol, on remarque que le nombre de Nusselt augmente en fonction de Reynolds pour les deux cas mais de façon différente : l'augmentation est plus prononcée dans le cas de NTC/Glycérol, ce qui montre que le choix du fluide caloporteur influence grandement l'augmentation du nombre de Nusselt en fonction du nombre de Reynolds.

L'utilisation du NTC/Glycérol donne de meilleurs résultats en performances thermiques.

3.2.2 Approche conventionnelle modifiée :

On passe au cas avec dispersion thermique : approche conventionnelle modifiée. Cette approche tient compte autant pour la dispersion thermique que pour l'effet moléculaire des nanotubes de carbone. Pour cela, on travaillera avec la formule de Li Xuan qui évalue le nombre

de Nusselt pour les nanofluides en tenant compte de l'effet de la dispersion thermique des nanoparticules dans un échangeur de chaleur à serpentin. Le nombre de Nusselt est calculé avec la formule de Li Xuan :

$$Nu_{nf} = 0.0059[1 + 7.628\phi^{0.6886}Pe_d^{0.001}]Re_{nf}^{0.9288}Pr_{nf}^{0.4}$$

Avec ϕ : la fraction volumique des nanoparticules, Pe : le nombre de Peclet et Re : le nombre de Reynolds.

Dans cette formule, le nombre de Peclet nous permettra de décrire l'effet de la dispersion thermique des nanoparticules.

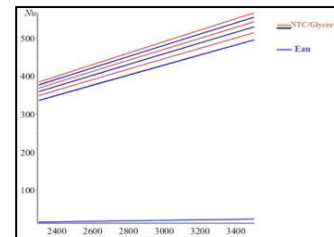


Figure 5 : Variation du nombre de Nusselt en fonction du nombre de Reynolds pour le nanofluide NTC/Glycérol et pour l'eau avec ϕ variant de 0.0006 % jusqu'à 0.006 %

La courbe sur la figure 5 représente la variation du nombre de Nusselt en fonction du nombre de Reynolds pour le nanofluide NTC/Glycérol ainsi que pour l'eau déminéralisée en variant la fraction volumique (de 0.0006% à 0.006%). Les résultats montrent premièrement, une forte augmentation du nombre de Nusselt en fonction du nombre de Reynolds, ceci est dû à l'intensification des mouvements browniens qui est due à la présence des NTC et des forts débits d'écoulement. Cette augmentation devient plus prononcée en augmentant la fraction volumique.

Deuxièmement, le nanofluide NTC/Glycérol est plus performant en comparaison avec l'eau pure : L'évolution du nombre de Nusselt en fonction du nombre de Reynolds est très faible par rapport à celle du NTC/Glycérol. Troisièmement, la dispersion thermique des nanoparticules NTC permet d'intensifier le transfert thermique pour le nanofluide caloporteur.

3. Conclusion:

Dans ce travail, nous avons présenté d'abord une étude préliminaire sur les échanges thermiques dans un échangeur de chaleur à serpentin utilisant des nanofluides à base de NTC comme agents thermiques. Les premiers résultats obtenus montrent l'effet remarquable de l'utilisation des nanotubes de carbone ainsi que l'effet de l'antigel (glycérol) sur l'amélioration du coefficient d'échange convectif dans l'échangeur de chaleur.

Les résultats confirment que le nanofluide offre des performances thermiques supérieures à celles de l'eau pure et peut donc aider à réduire la surface totale de transfert de chaleur, le nombre de tours et la puissance de pompage de l'échangeur de chaleur en fournissant la même capacité d'échange thermique.

Références

- [1] Xie H., Lee H., Youn W., Choi M., Nanofluids containing multiwalled carbon nanotubes and their enhanced thermal conductivities, *J. Appl. Phys.*, 94 (2003), 4967 - 4971.
- [2] Xie H., Chen L., Adjustable thermal conductivity in carbon nanotube nanofluid *Physics Letters A*, 373(2009), 1861-1864.

A novel design of solar dryer based-ETC technology for agricultural applications: Thermal performance assessment

H. Allouhi ^{a*}, A. Allouhi ^a, A. Jamil ^a

^aEcole Supérieure de Technologie de Fès, U.S.M.B.A, Route d'Imouzzer, BP 242 Fez, Morocco

*Corresponding author: hamza.allouhi@usmba.ac.ma

Abstract

Solar drying is considered among the most suitable techniques that can assist agriculture producer to enhance both quantity and quality of the product at accessible costs. In this paper, a new model of a solar dryer based-ETC technology was designed and simulated. Thermal analysis based on a steady state mathematical model was conducted to assess the energy performance of the system. Parametric investigations were performed at the level of the ETC to reveal the impact of various parameters on the global system outputs. The proposed configuration reveals the advantages of the ETC technology which could be suitably integrated in large scale and industrial drying applications.

Keywords: *Solar drying; ETC; Thermal efficiency; Output collector temperature.*

1. Introduction

Recently, solar energy has exhibited several advantages in supplying eco-friendly and clean energy. Alternatives based solar technologies have shown a very high potential in covering the global energy requirements [1]. One of the most developing usages of solar radiation is solar dryer specially in locations which enjoy high potential [2]. Drying is considered as the most important process employed in several domains mainly in the preservation of agricultural and marine products such as vegetables, grains, fruits and fish [3]. In this regard, solar dryer systems are widely recommended as alternative to reduce the rate of the conventional energy resource uses [4]. In such process, heat is supplied to remove moisture. The heat transfer in solar drying process can be executed by two modes namely, the direct and indirect drying techniques [5]. In the direct mode, the product is situated in a drying chamber with transparent cover. Sunrays will be directly converted to thermal heat gained by the drying chamber which will be then employed to extract the humidity from the product. This technique does not present fabrication complexity and could be made at lower costs. However, the rate of drying is practically low since it is mainly dependent on weather conditions [6]. On the other hand, indirect solar drying mode has achieved efficient yields and high drying rates can be

reached since thermal collection is better ensured via the use of solar thermal collectors. These dryer systems, in turn require expensive costs for construction and maintenance. One advantage of this dryer class is that product is not exposed directly to the sun, so its quality is better preserved [7]. In this sense, numerous works based on experimental data support, developed design and improved theoretical models were performed by the research community to evolve strategies that can promote the implementation of this type of systems [8]. Lingayat et al. [9] proposed an indirect solar dryer based Flat Plat Collector (FPC) with an area of 2 m² to reduce moisture of agricultural products. It has been revealed that the thermal efficiency of the collector and drying chamber reached 31.5% and 22.38%, respectively. Lakshmi et al. [10] experimentally assessed a solar drying system composed of two solar air heaters in series. The drying and the average thermal efficiencies of the overall system were found to be 33.5% and 38.2%, respectively. Vigneshkumar et al. [11] carried out the investigation of an indirect solar dryer based on FPC with phase change material (PCM) for drying sliced potatoes. The integration of PCM with the collector remarkably improved the system performance after sunshine period and allowed increasing the amount of removed moisture from potatoes by 5.1%/day. Wang et al. [12] analyzed the performance of an indirect passive solar dryer based principally on an Evacuated Tube Collector (ETC) with an auxiliary heater for drying mango. The experimental findings revealed that the average thermal efficiency at four different locations of the drying room was ranging from 30.9% to 33.8%. In the current work, a novel design of a solar dryer based-ETC technology is proposed. Thermal analysis based on a steady state mathematical model is developed in order to characterize the energy performance of this solar dryer. This special configuration can be applied for large scale and industrial drying applications in developing countries to promote the agriculture sector.

2. System description:

2.1. Dryer Structure:

The proposed solar dryer mainly consists of a drying chamber equipped with ETCs applied in two sides for

efficient solar energy capture. The first side is oriented towards the South direction and the second side is towards the North. Both collectors are tilted at an optimal inclination (30°) recommended in the studied regions (Fez, Morocco). Specifically, the design involves the following substructures:

- Support: made of steel frame on which the entire system will be implemented.
- Absorber plate: made of aluminium for better solar radiation absorption.
- Heat pipe: each tube contains a heat pipe. Each side of the dryer is made up with 7 tubes.
- Heat pipe support: is the box containing the absorber plate and heat pipes. It contains an air ventilation channel for heat exchange and the exterior. The exterior surface is covered by transparent glass cover.
- Perforated shelves: on which the product to be dried will be introduced. Perforations will allow hot air from the collector output to flow within the drying chamber.

The overall structure has been engineered using CATIA V5 environment. A perspective view of the studied system is depicted in **Fig. 1**.

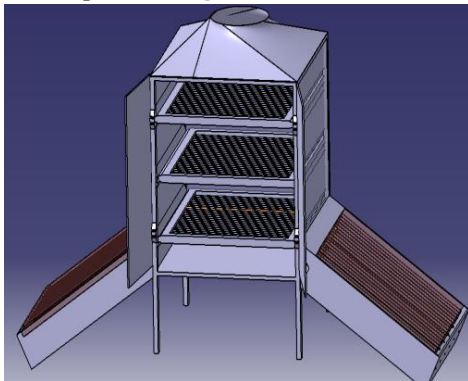


Figure 1: Illustrative view of the proposed model.

2.2. Thermal analysis modelling:

During this study, the phenomenon of heat and mass transfer within the drying chamber was not taken into consideration. The main objective was to analyze the solar thermal collector part by proposing an ETC-based configuration for heat production. Since the collector is designed with two identical surfaces, the analysis focused essentially on only one part, this can be extrapolated to the entire surface by considering the variation of the incident radiation when changing the orientation of the collector from the South to the North direction. It is worth of mention that the double-collector configuration allowed ensuring both high solar collected energy and high heated air flow rate and subsequently a reduced drying time. The investigation of the collector performance was established under MATLAB software through a thermal model. The Matlab code was programmed to solve all the thermal energy balance equations at the level of each solar collector components.

Geometrical and physical properties of the designed system as well as temperatures at the level of the glazing, absorbing plate and heat pipe have been identified using differential equations and specific details furnished in Ref [13]. Then, the thermal efficiency was accordingly calculated considering the received solar radiation (G_t), the collector surface (A), the air circulation flow rate (\dot{m}) and the specific heat capacity (c). The related equation can be expressed as follows:

$$\eta = \frac{\dot{Q}_u}{\dot{Q}_s} = \frac{\dot{m} c (T_{out} - T_{in})}{A G_t}$$

\dot{Q}_u and \dot{Q}_s describe the useful thermal power and global solar radiation, respectively. T_{in} and T_{out} represent the inlet and outlet air temperatures.

3. Results and discussion:

In this section, a parametrical study has been performed to visualize the effect of the variation of several parameters namely the solar insolation (G), the inlet collector temperature (T_{in}) and the mass flow rate ($Flow_rate$) on the output temperature (T_{out}) and thermal efficiency of the collector. In this direction, three parametrical scenarios were conducted. Firstly, the inlet collector temperature and the mass flow rate were fixed 25°C and 0.0458 kg/s , respectively and the effect of the variation of the solar insolation on the thermal efficiency and the output temperature of the collector was presented as depicted in **Fig. 2**. Solar radiation was ranged between 100 W/m^2 and 1000 W/m^2 . It was observed that the output collector temperature increases with each increment of the solar radiation reaching a value of about 307 K while the thermal efficiency undergoes a slight variation for solar radiation values lower than 200 W/m^2 . Afterwards, the variation of the solar radiation seems to be marginal and does not report practically any change on the thermal system efficiency.

Secondly, the solar insolation was fixed 800 W/m^2 with the same value of the mass flow rate. In this scenario, the analysis focused on the impact of the inlet collector temperature on the thermal efficiency and the output temperature (see **Fig. 3**). Increasing the value of inlet temperature leads to maximal output achievements. A remarkable improvement of the thermal efficiency was detected while increasing the inlet temperature.

Finally, the effect of the mass flow rate on the thermal efficiency and the output temperature of the collector was visualized while fixing the input parameters as mentioned before (see **Fig. 4**). This scenario considers varying the mass flow rate from 0.02 kg/s to 0.1 kg/s . It is clearly observed that incrementing the value of the mass flow rate acts positively on thermal efficiency. On the other hand, it can be deduced that the increase of such parameter results in a decrease of the output temperature. Accordingly, an optimum value of the mass flow rate was detected and found to be 0.035 kg/s , coinciding with a value of about 307 K and 0.28 of the

output collector temperature and the thermal efficiency, respectively.

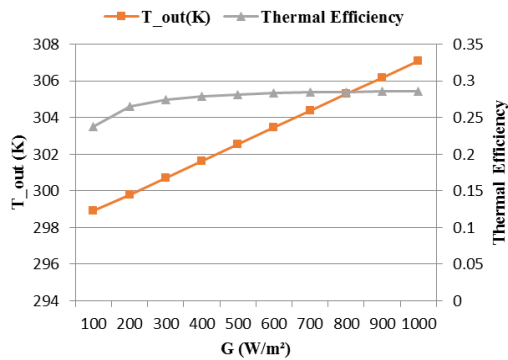


Figure 2: Effect of the solar radiation variation on the output system performances.

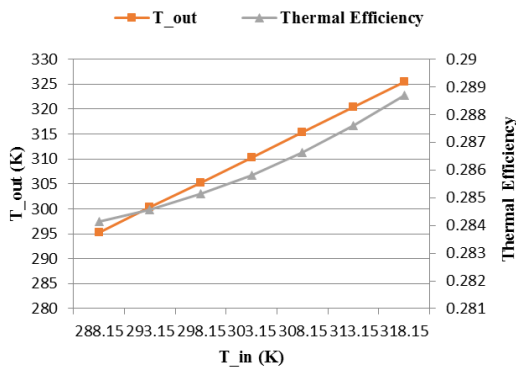


Figure 3: Impact of the inlet collector temperature variation on the output system performances.

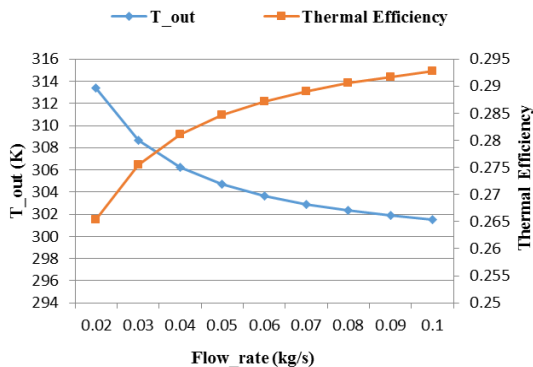


Figure 4: Thermal efficiency and output collector temperature profile versus the mass flow rate variation.

Conclusion:

A novel design of indirect solar dryer based on ETC technology was proposed and thermally assessed. The suggested system can ensure promising yields since the design allowed capturing high amount of solar radiation. From the economic and environmental perspectives, the proposed system can be considered among the suitable alternatives for drying process and can promote the agriculture sector especially in developing countries.

References

- [1] S. Kumar, B. Bhattacharyya, and V. K. Gupta, "Present and future energy scenario in India," *J. Inst. Eng. Ser. B*, vol. 95, no. 3, pp. 247–254, 2014.
- [2] M. R. Nukulwar and V. B. Tungikar, "A review on performance evaluation of solar dryer and its material for drying agricultural products," *Mater. Today Proc.*, vol. 46, pp. 345–349, 2021.
- [3] A. M. Asemu, N. G. Habtu, M. A. Delele, B. Subramanyam, and S. Alavi, "Drying characteristics of maize grain in solar bubble dryer," *J. Food Process Eng.*, vol. 43, no. 2, p. e13312, 2020.
- [4] A. Fudholi *et al.*, "Techno-economic of solar drying systems with water based solar collectors in Malaysia: a review," *Renew. Sustain. Energy Rev.*, vol. 51, pp. 809–820, 2015.
- [5] H. El Hage, A. Herez, M. Ramadan, H. Bazzi, and M. Khaled, "An investigation on solar drying: A review with economic and environmental assessment," *Energy*, vol. 157, pp. 815–829, 2018.
- [6] M. Zarezade and A. Mostafaeipour, "Identifying the effective factors on implementing the solar dryers for Yazd province, Iran," *Renew. Sustain. Energy Rev.*, vol. 57, pp. 765–775, 2016.
- [7] O. Prakash, V. Laguri, A. Pandey, A. Kumar, and A. Kumar, "Review on various modelling techniques for the solar dryers," *Renew. Sustain. Energy Rev.*, vol. 62, pp. 396–417, 2016.
- [8] A. K. Bhardwaj, R. Kumar, and R. Chauhan, "Experimental investigation of the performance of a novel solar dryer for drying medicinal plants in Western Himalayan region," *Sol. Energy*, vol. 177, pp. 395–407, 2019.
- [9] A. Lingayat, V. P. Chandramohan, and V. R. K. Raju, "Design, development and performance of indirect type solar dryer for banana drying," *Energy Procedia*, vol. 109, pp. 409–416, 2017.
- [10] D. V. N. Lakshmi, P. Muthukumar, A. Layek, and P. K. Nayak, "Performance analyses of mixed mode forced convection solar dryer for drying of stevia leaves," *Sol. Energy*, vol. 188, pp. 507–518, 2019.
- [11] N. Vigneshkumar *et al.*, "Investigation on indirect solar dryer for drying sliced potatoes using phase change materials (PCM)," *Mater. Today Proc.*, vol. 47, pp. 5233–5238, 2021.
- [12] W. Wang, M. Li, R. H. E. Hassanien, Y. Wang, and L. Yang, "Thermal performance of indirect forced convection solar dryer and kinetics analysis of mango," *Appl. Therm. Eng.*, vol. 134, pp. 310–321, 2018.
- [13] A. Allouhi and M. Benzakour Amine, "Effect analysis on energetic, exergetic and financial performance of a flat plate collector with heat pipes," *Energy Convers. Manag.*, vol. 195, no. April, pp. 274–289, 2019, doi: 10.1016/j.enconman.2019.04.081.

Forced Convection of Jeffery-Hamel Flow through a Convergent Channel with Step-Change in Wall Temperature and Streamwise Conduction

Convection de l'Écoulement de Jeffery-Hamel à travers un canal convergent avec conduction radiale et un saut de température à la paroi

M. DARIF, E. ESSAGHIR *¹, Y. HADDOUT, A. OUBARRA, J. LAHJOMRI*²

Université Hassan II, Faculté des sciences Ain Chock, Laboratoire de mécanique,
Km 8 route d'El Jadida BP 5366 Maarif, Casablanca, Maroc

*¹ elhoucine.essaghir3@gmail.com *² lahjomri@hotmail.com

Abstract

This work is devoted to the determination of the analytical solution of the problem of the laminar forced convection of the Jeffery-Hamel flow through a convergent channel. The physical characteristics have been calculated under the assumption that the flow is symmetric and purely radial. The analytical solution is largely based on the strong approach of self-adjoint formalism developed by Papoutsakis and Ramkrishna for the Graetz problem extended to Jeffery-Hamel flow. This extension has been done by using a new matrix operator of three dimensions in the Hilbert space. The analysis reveals that the streamwise conduction in the flow and the aperture angle between the two plane walls of the channel have a substantial impact on the physical parameters within the warming section..

Keywords: *Convergent channel; Jeffery-Hamel flow; self-adjoint formalism; streamwise conduction.*

Résumé

Ce travail est consacré à la détermination de la solution analytique du problème de la convection forcée laminaire de l'écoulement de Jeffery-Hamel à travers un canal convergent. Les caractéristiques physiques ont été calculées sous l'hypothèse que l'écoulement soit symétrique et purement radial. La solution analytique est largement basée sur l'approche forte du formalisme auto-adjoint développé par Papoutsakis et Ramkrishna pour le problème de Graetz étendu pour l'écoulement de Jeffery-Hamel. Cette extension a été réalisée en utilisant un nouvel opérateur matriciel à trois dimensions dans l'espace de Hilbert. L'analyse révèle que la conduction longitudinale dans l'écoulement et l'angle d'ouverture entre les deux parois planes du canal ont un impact substantiel sur les paramètres physiques au sein de la section de réchauffement.

Mots clés : *canal convergent ; Ecoulement de Jeffery-Hamel ; Formalisme autoadjoint ; Conduction radiale*

1. Introduction

L'étude du transfert de chaleur dans l'écoulement de Jeffery-Hamel est d'une grande importance à la fois en théorie et en pratique ; puisqu'il a de nombreux domaines d'application, notamment dans la technologie de fabrication de tôles plastiques et métalliques. Depuis le premier travail de Jeffery 1915 puis Hamel en 1917, de nombreux auteurs ont examiné l'écoulement Jeffery-Hamel en déterminant le profil de vitesse pour un canal convergent, un canal divergent ou mixte [1-4]. Les différentes approches analytiques et numériques utilisées pour étudier ce problème ont démontré que la vitesse est très sensible à la variation de l'angle d'ouverture du canal et du nombre de Reynolds. Au point de vue thermique, les études antérieures de ce problème ont été réalisées dans le cadre des approximations de la couche limite [5] en négligeant la conduction radiale dans la direction de l'écoulement. D'autres travaux ont supposé que le champ de température à l'entrée du canal convergent est pleinement développé [2-5]. Notons que malgré le grand nombre de travaux sur le problème de Graetz étendu à l'écoulement de Hagen-Poiseuille en canal à plaques droites parallèles ou en tube, la plupart des études antérieures ont été réalisées dans le cas particulier où la conduction dans la direction radiale de l'écoulement est négligeable aussi que les effets de l'angle d'ouverture du canal sur le transfert de chaleur n'ont pas été pris en compte. Cependant, les effets de la conduction radiale dans l'écoulement peuvent jouer un rôle important pour des faibles valeurs des nombres de Reynolds et de Péclet. En effet, C'est la discontinuité de la paroi vers l'entrée de la zone chauffée qui pose problème lorsqu'on s'approche de la singularité [6]. Ces oscillations produisent des erreurs considérables dans la caractéristique thermique [6, 7]. Ainsi, une solution analytique est nécessaire pour résoudre cette singularité.

L'objectif de la présente étude est d'offrir une solution semi-analytique du problème de Graetz étendu à l'écoulement de Jeffery-Hamel dans un canal convergent avec une température de paroi uniforme. La présente

analyse prend en considération les effets de la conduction dans le sens de l'écoulement radial. L'approche est basée sur la méthode de formalisme autoadjoint résultant de la décomposition de l'équation de l'énergie en un système de deux équations aux dérivées partielles de premier ordre. Ceci a été réalisé en utilisant l'opérateur matriciel et un produit interne adéquat entre les deux vecteurs dans l'espace de Hilbert. En calculant un nombre adéquat de valeurs propres précises, les oscillations de Gibbs sont supprimées et donc la singularité est résolue efficacement. Les effets du changement de l'angle d'ouverture entre les deux parois, des nombres de Reynolds (ou de Péclet) et de Prandtl, sur les paramètres physiques seront étudiés en détail.

2. Formulation Mathématique :

2.1 Description du problème et équations :

On considère l'écoulement unidirectionnel, permanent, glissant, pleinement développé d'un fluide Newtonien entre deux plans semi-infinis inclinés et séparés par un angle d'ouverture 2ψ .

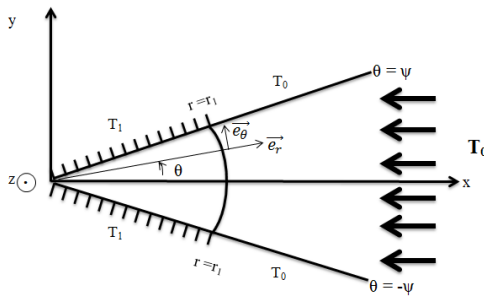


Figure 1 : Configuration géométrique

La figure 1 montre la configuration géométrique et le système des coordonnées cylindriques utilisés. Le fluide entre avec une température uniforme T_0 très loin du point formé par ces deux plans. Les parois du canal sont chauffées avec une température uniforme T_1 pour $0 \leq r < r_1$ et portées à la température T_0 pour $r \geq r_1$. r_1 désigne la longueur de la section de chauffage. Dans le cas d'un écoulement incompressible ayant des propriétés physiques qui sont supposées constantes, et en négligeant la dissipation visqueuse tout en tenant compte de la symétrie de l'écoulement par rapport à l'axe x , les équations de la quantité de mouvement et d'énergie écrites sous forme adimensionnelle tenant compte de la conduction radiale dans la direction de l'écoulement sont données par:

$$U''''(\eta) - 2\psi Re U(\eta) U'(\eta) + 4\psi^2 U'(\eta) = 0 \quad (1)$$

$$U(\eta) \frac{\partial \theta}{\partial \xi} = \frac{\psi}{Pe} \frac{\partial}{\partial \xi} \left[(1 - \xi) \frac{\partial \theta}{\partial \xi} \right] + \frac{1}{\psi Pe (1 - \xi)} \frac{\partial^2 \theta}{\partial \eta^2} \quad (2)$$

Les conditions aux limites appropriées écrites sous forme adimensionnelles sont données par :

$$U(0) = 1, U'(0) = 0, U(1) = 0 \quad (3)$$

$$\frac{\partial \theta}{\partial \eta} = 0 \text{ pour } \eta = 0 \quad (4)$$

$$\theta \rightarrow 0, \text{ pour } \xi \rightarrow -\infty \quad (5)$$

$$\theta \rightarrow 1, \text{ pour } \xi \rightarrow 1 \quad (6)$$

$$\theta(\xi, 1) = \begin{cases} 1 & \text{pour } 0 < \xi < 1 \\ 0 & \text{pour } \xi \leq 0 \end{cases} \quad (7)$$

Dans lesquelles les paramètres et les variables adimensionnels sont définis par :

$$\eta = \frac{\theta}{\psi}, \xi = 1 - \frac{r}{r_1}, U(\eta) = \frac{u_r(r, \theta)}{u_r(r, 0)} = \frac{F(\theta)}{|F(0)|}, \Theta = \frac{T - T_0}{T_1 - T_0} \quad (8)$$

où $u_r(r, \theta) = \frac{F(\theta)}{r}$ est le champ de vitesse de l'écoulement de Jeffery-Hamel et $Pe = Re Pr$ est le nombre de Péclet, $Re = u_r(r, 0) r \psi / \nu = |F(0)| \psi / \nu$, le nombre de Reynolds et $Pr = \nu / \alpha$, le nombre de Prandtl. $F(0) = Q/2 \psi \int_0^1 U(\eta) d\eta$ est la vitesse maximale au centre qui est reliée au débit par unité de longueur du canal défini par $Q = 2 \int_0^\psi r u_r d\theta$.

2.2 Solution analytique

En introduisant le flux axial de l'énergie [8-9] :

$$\varphi(\xi, \eta) = \int_0^\eta \left[U(\eta') \theta(\xi, \eta') - \frac{\psi}{Pe} (1 - \xi) \frac{\partial \theta}{\partial \xi} \right] d\eta' \quad (9)$$

Permet de décomposer l'équation de l'énergie (2) en un système de deux équations aux dérivées partielles de premier ordre :

$$Pe \psi (1 - \xi) \frac{\partial \theta}{\partial \xi} = Pe^2 U(\eta) \theta - Pe^2 \frac{\partial \varphi}{\partial \eta} \quad (10)$$

$$Pe \psi (1 - \xi) \frac{\partial \varphi}{\partial \xi} = \frac{\partial \theta}{\partial \eta} \quad (11)$$

En appliquant la méthode de formalisme autoadjoint [8-9] nous pouvons obtenir la solution du problème sous forme d'un développement en série de fonctions propres $\phi_{n1}^+(\eta)$ et $\phi_{n1}^-(\eta)$ associées respectivement aux valeurs propres λ_n et β_n [7]:

Pour $\xi \leq 0$

$$\theta(\xi, \eta) = - \sum_{n=1}^{\infty} A_n \exp\left(-\frac{\lambda_n^2}{Pe \psi} \ln(1 - \xi)\right) \phi_{n1}^+(\eta) \quad (15)$$

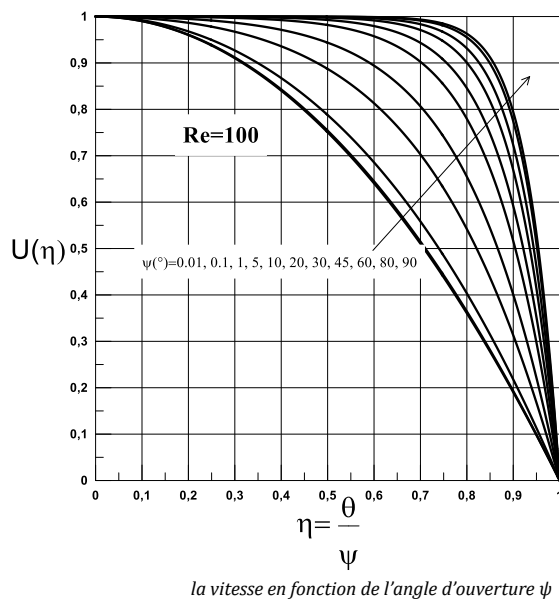
Pour $0 < \xi < 1$

$$\theta(\xi, \eta) = 1 - \sum_{n=1}^{\infty} B_n \exp\left(\frac{\beta_n^2}{Pe \psi} \ln(1 - \xi)\right) \phi_{n1}^-(\eta) \quad (16)$$

$$\text{Avec } A_n = \frac{2}{\lambda_n \left[\frac{d\phi_{n1}^+(1)}{d\lambda_n} \right]} \text{ et } B_n = - \frac{2}{\beta_n \left[\frac{d\phi_{n1}^-(1)}{d\beta_n} \right]}$$

3. Résultats et discussion

L'effet de l'angle d'ouverture ψ sur la vitesse de l'écoulement est représenté sur la figure 2. On remarque que pour des faibles valeurs de ψ le profil de vitesse est parabolique et ressemble à celui de l'écoulement de Poiseuille. Cependant, l'augmentation du nombre d'angle ψ fait croître la vitesse et par conséquent un profil plat s'établit au centre de la conduite, à l'exception de la région proche des parois du canal dans laquelle nous avons des gradients de vitesse importants. En effet, l'accroissement de l'angle ψ fait augmenter le gradient de pression favorable. Dans cette situation, la vitesse de l'écoulement croît et par conséquent l'épaisseur de la couche limite se verra diminuée.



La figure 3 montre la variation des profils de température pour l'écoulement thermiquement développer dans un canal à plaques convergentes pour $Pe=5$ avec différentes valeurs de distances longitudinales ξ et pour un angle d'ouverture $\psi = 30^\circ$. La comparaison de ces profils démontre clairement l'influence de la conduction dans le sens de l'écoulement en particulier pour des grandes valeurs de l'angle d'ouverture. En effet, la température la plus faible se produit au niveau de l'axe du canal, par contre la température la plus élevée se manifeste près des parois. Cependant, l'accroissement de l'angle ψ conduit vers une augmentation de la conduction thermique dans le sens du courant. Les profils de température sont établis dans la région aval de la section de chauffée à $\xi = 0.99$ pour un angle d'ouverture de 30° , et s'homogénéisent en amont de la partie chauffée en $\xi = -0.05$ pour $\psi = 30^\circ$.

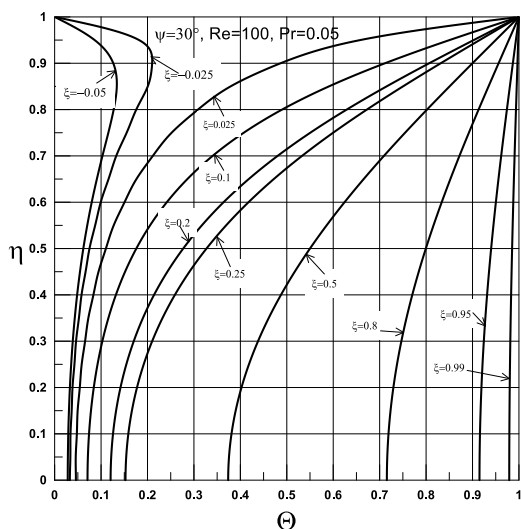


Figure 3 Effet des distances longitudinales sur les profils de température

4. Conclusion

Dans le présent travail, nous avons étudié de manière semi-analytique le développement thermique du problème de l'écoulement laminaire de Jeffery-Hamel dans un canal convergent d'un fluide newtonien, y compris la conduction dans le sens du courant avec une température de paroi uniforme. Les résultats obtenus sont considérés comme faisant partie de la solution des travaux précédents en considérant l'impact de la conduction dans le sens du courant. De plus, les résultats thermiques de cette étude sont d'une grande importance pratique et théorique, car ils peuvent donner un aperçu de plusieurs processus de production dans l'industrie et l'ingénierie, tels que la fabrication de plastiques et de tôles.

Références

- [1] H. Schlichting, J. Kestin, Boundary layer theory, Springer, 1961..
- [2] I. C. Das, Q. N. T. Nasa. Analysis of magnetohydrodynamic jeffery-hamel flow in a convergent-divergent channel using Cu-water nanofluid. Journal of Engineering Science 12.2 (2021) 79-92.
- [3] A. M. Jasim., New Analytical Study for Nanofluid between Two Non-Parallel Plane Walls (Jeffery-Hamel Flow). Journal of Applied and Computational Mechanics. 7, 1 (2021) 213-224.
- [4] S. Kumbinaraiaiah, K. R. Raghunatha. Numerical solution of the Jeffery-Hamel flow through the wavelet technique, Heat Transfer 51.2 (2022): 1568-1584
- [5] B. Weigand (Ed.), Analytical methods for heat transfer and fluid flow problems, Springer, Berlin, Heidelberg, 2015: pp. 121-171.
- [6] J. Lahjomri, A. Oubarra, Analytical solution of the Graetz problem with axial conduction, Journal of Heat Transfer. 121 (1999) 1078-1083.
- [7] Y. Haddout, E. Essaghir, A. Rafiki, A. Oubarra, J. Lahjomri, The Graetz Problem Extended to Jeffery-Hamel Flow through a Convergent Plate Channel with Step-Change in Wall Temperature and Streamwise Conduction, International Communications in Heat and Mass. 134 (2022): 105986.
- [8] Papoutsakis E., Ramkrishna D. & Lim H.C., The extended Graetz problem with Dirichlet wall boundary conditions, Appl. Sci. Res. 36 (1980) 13-34
- [9] Ku H. C. & Hatzivramidis D., Chebyshev expansion methods for the solution of the extended Graetz problem, J. of Computational Physics 56 (1984) 495-512

Thème 10 :
Interaction fluide-structure

Modelling of DEXA wave energy converter under irregular wave conditions

A. JABRALI¹, R. KHATYR², J. KHALID NACIRI³

Laboratory of Mechanics, Faculty of Sciences Ain Chock, B.P 5366, Maarif, Hassan II University, Casablanca, Morocco

Corresponding authors: E-Mail: ⁽¹⁾ jabraliahmed91@gmail.com

⁽²⁾ khatyrrabha@gmail.com

⁽³⁾ nacirih2c@gmail.com

Abstract

This paper studies a DEXA wave energy converter under irregular wave. The considered system consists of two articulated rigid bodies where the energy recovery process relies on the relative movement of parts of the WEC set in motion by wave action. The study focuses on the effect of the irregular wave characteristics on the recovered energy. The results show that the wave frequency has an impact on power capture by the WEC, and its efficiency.

Keywords: *Wave energy converter; wave characteristics; multi-body articulated system; irregular wave; DEXA Wave; Marine renewable energy.*

1. Introduction

In recent decades, marine renewable energies are receiving increasing attention worldwide. Among the various types of ocean renewable energy, wave energy is the most distinguished one [1-2] as they do not require complex and expansive platforms for harvesting energy. So far, several types of energy recovery devices have been developed and are in operation. However, due to the complexity of the interactions between randomly behaving waves and the recovery system, the field of wave energy conversion is still considered in research and improvement phase to achieve breakeven points. Zhang and Yang [3] studied the power capture performance of an oscillating body WEC with nonlinear snap through power take off (PTO) systems in irregular waves. The Results show that the nonlinear snap through PTO system can increase the power captured by the WEC in irregular waves compared to linear PTO system. Ji et al. [4] analyzed a two-body oscillating wave energy converter by varying seven different system model parameters. They found that the PTO stiffness coefficient affects the performance of WEC in the regular and irregular waves. The present study focuses on the performance of a DEXA wave device in irregular waves. The considered floating WEC is a freely floating wave energy converter system developed in 2009 in Denmark [5]. This system consists of two articulated rigid bodies where the energy is recovered by taking advantage of the relative movement between the two parts set in motion differently by the wave. The forces due to the fluid action on the solid parts of the system are modelled by an Archimedean thrust exerted

on the submerged parts of the system and a Morison Force [6], which makes it possible to consider the effects of added mass as well as a drag between the solid and the fluid. The system of five nonlinear differential equations governing the plane movement of the WEC is solved numerically by a fourth-order Runge-Kutta method [7]. The current study investigated the wave frequency effect on the recovered energy by applying a signal filter. This filter applied through modeling the irregular wave by Fourier function. The results show that there are some frequencies that have more impact than others on energy recovery. This thus makes it possible, for the case of irregular waves, to take into account only a given range of wave frequencies to predict with sufficient precision the recoverable energy and to define an optimal dimensioning of the WEC.

This paper is organized as follows: the first section is devoted to the wave mathematical models. Section two presents the mathematical model of the considered systems. The third section presents the results and their discussion.

2. Wave Mathematical Model

To describe the characteristics of real sea conditions of irregular waves, an approach can be made through considering the wave as the linear superposition of a sum of sinusoidal progressing wave [8] as follow:

$$\eta(t) = \sum_{n=1}^k A_{m_n} \cos(\omega_n t - k_n x(t)) \quad (1)$$

in which $\eta(t)$ is the wave surface elevation, k is total number of components, A_{m_n} is the wave amplitude, ω_n is the angular frequency and k_n is the wave number. Each of the sinusoidal wave in the Eq. (1) will satisfy the properties of linear wave theory [9]. This approach allows using filtering function by removing some frequencies. Moreover, many theoretical methods on the prediction and modeling of irregular could be found in Ref. [10].

3. Mathematical Formulation

The considered system is a floating WEC which consists of two rigid bodies S_1 and S_2 (Fig. 1). Each component carries a cylinder of center O_i ($i=1$ for cylinder 1 and $i=2$ for cylinder 2) and a floater of center j ($j = A$ for floater 1 and $j = B$ for floater 2), the two bodies are paired by a linking bar. The floaters are modeled by cylinders of radius r_j and length H . It should be noted that this geometry corresponds to a pre-model of a DEXA wave.

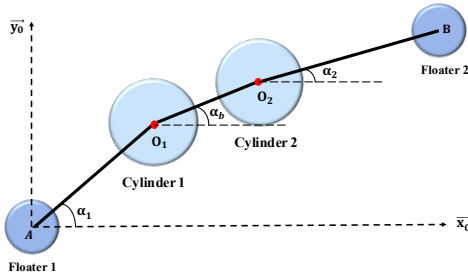


Fig. 1. Scheme 2D of the floating WEC

Taking into account the connections between parts of the system as shown in Fig.1, and since only plane movements of the WEC are considered, we introduce five degrees of freedom for the mechanical system which are the surge (x_A), the heave (y_A), and the pitch (α_1) for solid 1, the pitch (α_2) for solid 2 and the angle (α_b) for the linking bar. (x_A) and (y_A) are the coordinate of the center of gravity of floater 1, such as:

$$\vec{OA} = x_A \vec{x}_0 + y_A \vec{y}_0 \quad (2)$$

In a non-inertial reference frame $\mathcal{R}(O, \vec{x}_0, \vec{y}_0, \vec{z}_0)$ where O is an arbitrary origin taken at a given position of the moving free surface of the fluid and \vec{y}_0 is the upward vertical. It should be noted that the mathematical formulation will be established in $\mathcal{R}(O, \vec{x}_0, \vec{y}_0, \vec{z}_0)$.

Newton's second law of motion applied to each part of the system separately [solid 1, solid 2, linking bar] is expressed for solid i ($i = 1$ for solid 1 and $i = 2$ for solid 2) as follows:

$$[\tau_{Di}] = [\tau_{pi}] + [\tau_{Mi}] + [\tau_{Ai}] + [\tau_{Loi}] + [\tau_{pto}] - [\tau_{ei}] \quad (3)$$

where $[\tau_{Di}]$ is the dynamic torsor, $[\tau_{pi}]$ represents the gravity force torsor, $[\tau_{Mi}]$ is the Morison force torsor representing the inertia forces and viscous forces exerted by the fluid on the system, $[\tau_{Ai}]$ is the Archimedes thrust torsor, $[\tau_{Loi}]$ represents the reactions torsor at the connection between the solid i and the linking bar, $[\tau_{pto}]$ is the forces torsor for the power take off system of the WEC and $[\tau_{ei}]$ is the inertia force torsor related to the non-inertial character of the considered reference frame. The torsor elements are applied for S_1 in G_1 (the gravity center of solid 1), and for S_2 in G_2 (the gravity center of solid 2).

For the linking bar, Newton's second law of motion is written as:

$$[\tau_{Db}] = [\tau_{pb}] - [\tau_{LG_1}] - [\tau_{LG_2}] - [\tau_{eb}] \quad (4)$$

where $[\tau_{Db}]$, $[\tau_{pb}]$, $[\tau_{LG_1}]$, $[\tau_{LG_2}]$ and $[\tau_{eb}]$ represent respectively the dynamic torsor, the gravity force torsor, the reactions torsor between the solid 1 and the linking bar, the reactions torsor between the solid 2 and the linking bar, and the inertia force torsor related to the non-inertial character of the considered reference frame.

The insertion of the torsors expressions in Eqs. (3) and (4) led to obtaining the following system of five coupled

differential equations for the five degrees of freedom x_A , y_A , α_b , α_1 and α_2 :

$$M_T \ddot{x}_A - C_7 \sin(\alpha_1) \ddot{\alpha}_1 - M_{T2} C_2 \sin(\alpha_2) \ddot{\alpha}_2 + F_{m_{A1x}} + F_{m_{A2x}} - C_7 \cos(\alpha_1) \dot{\alpha}_1^2 - \left(M_{T2} + \frac{m_b}{2}\right) L \cos(\alpha_b) \dot{\alpha}_b^2 + F_{m_{Cy1x}} + F_{m_{Cy2x}} = 0 \quad (5)$$

$$M_T \ddot{y}_A + C_7 \cos(\alpha_1) \ddot{\alpha}_1 + M_{T2} C_2 \cos(\alpha_2) \ddot{\alpha}_2 + M_T g + F_{m_{A1y}} + F_{m_{Cy1y}} + F_{m_{A2y}} + F_{m_{Cy2y}} - C_7 \sin(\alpha_1) \dot{\alpha}_1^2 + \left(M_{T2} + \frac{m_b}{2}\right) L \cos(\alpha_b) \ddot{\alpha}_b - M_{T2} C_2 \sin(\alpha_2) \dot{\alpha}_2^2 - F_{arA1} - F_{arCy1} - M_T \sum_{n=1}^k \omega_n^2 A_{m_n} \cos(\omega_n t) - \left(M_{T2} + \frac{m_b}{2}\right) \sin(\alpha_b) L \dot{\alpha}_b^2 - F_{arA2} - F_{arCy2} = 0 \quad (6)$$

$$C_8 \sin(\alpha_1) \ddot{x}_A - C_8 \cos(\alpha_1) \ddot{y}_A - C_9 \ddot{\alpha}_1 + C_4 \cos(\alpha_1) \dot{x}_A \dot{\alpha}_1 + C_4 \sin(\alpha_1) \dot{y}_A \dot{\alpha}_1 - g M_{T1} (C_1 - l_1) \cos(\alpha_1) - \beta (\dot{\alpha}_1 - \dot{\alpha}_2) + (C_1 - l_1) \cos(\alpha_1) [-F_{m_{A1y}} - F_{m_{Cy1y}} + F_{arA1} + F_{arCy1}] + (C_1 - l_1) \sin(\alpha_1) [F_{m_{A1x}} + F_{m_{Cy1x}}] + (C_1 - l_1) \cos(\alpha_1) M_{T1} \sum_{n=1}^k \omega_n^2 A_{m_n} \cos(\omega_n t) = 0 \quad (7)$$

$$C_{10} \sin(\alpha_2) \ddot{x}_A - C_{10} \cos(\alpha_2) \ddot{y}_A - C_{10} l_1 \cos(\alpha_1 - \alpha_2) \ddot{\alpha}_1 + [C_5 + M_{T2} C_2^2] \ddot{\alpha}_2 - C_{10} L \cos(\alpha_2 - \alpha_b) \ddot{\alpha}_b + C_{10} l_1 \sin(\alpha_1 - \alpha_2) \dot{\alpha}_1^2 - C_{10} L \sin(\alpha_2 - \alpha_b) \dot{\alpha}_b^2 - C_6 l_1 \sin(\alpha_1 - \alpha_2) \dot{\alpha}_1 \dot{\alpha}_2 + C_6 L \sin(\alpha_2 - \alpha_b) \dot{\alpha}_b \dot{\alpha}_2 - C_2 \sin(\alpha_2) [F_{m_{A2x}} + F_{m_{Cy2x}}] - M_{T2} C_2 \sum_{n=1}^k \omega_n^2 A_{m_n} \cos(\omega_n t) \cos(\alpha_2) + g M_{T2} C_2 \cos(\alpha_2) - C_2 \cos(\alpha_2) [-F_{m_{A2y}} - F_{m_{Cy2y}} + F_{arA2} + F_{arCy2}] = 0 \quad (8)$$

$$[M_{T2} - M_{T1}] \sin(\alpha_b) \ddot{x}_A - [M_{T2} - M_{T1}] \cos(\alpha_b) \ddot{y}_A - [M_{T2} l_1 - M_{T1} C_1] \cos(\alpha_1 - \alpha_b) \ddot{\alpha}_1 - M_{T2} C_2 \cos(\alpha_2 - \alpha_b) \ddot{\alpha}_2 + L \left[\frac{m_b}{6} - M_{T2}\right] \ddot{\alpha}_b + [M_{T2} l_1 - M_{T1} C_1] \sin(\alpha_1 - \alpha_b) \dot{\alpha}_1^2 + M_{T2} C_2 \sin(\alpha_2 - \alpha_b) \dot{\alpha}_2^2 - \sin(\alpha_b) [F_{m_{A1x}} + F_{m_{Cy1x}} - F_{m_{A2x}} - F_{m_{Cy2x}}] - \cos(\alpha_b) [F_{arA1} + F_{arCy1} + F_{m_{A2y}} + F_{m_{Cy2y}} - F_{arA2} - F_{arCy2} - F_{m_{A1y}} - F_{m_{Cy1y}}] + \cos(\alpha_b) [M_{T2} - M_{T1}] \sum_{n=1}^k \omega_n^2 A_{m_n} \cos(\omega_n t) - [M_{T2} - M_{T1}] g \cos(\alpha_b) = 0 \quad (9)$$

where M_T is the total mass of the system, M_{Ti} is the mass of the solid i , m_b is the mass of the linking bar, L is the length of the linking bar, \ddot{x}_A and \ddot{y}_A are the two accelerations of the floater 1 along \vec{Ox}_0 and \vec{Oy}_0 axis respectively, $\ddot{\alpha}_i$ is the angular acceleration of the solid i , $\ddot{\alpha}_b$ is the angular acceleration of the linking bar and g represent gravity acceleration. $F_{m_{Aix}}$, $F_{m_{Aiy}}$, $F_{m_{Cyix}}$ and $F_{m_{Cyi}}$ are given by Morison equation respectively and are written as:

$$F_{m_{Aix}} = \rho_e C_m V_i \ddot{x}_j + \frac{1}{2} \rho_e C_d S_j \dot{x}_j |\dot{x}_j|, F_{m_{Aiy}} = \rho_e C_m V_i \ddot{y}_j + \frac{1}{2} \rho_e C_d S_j \dot{y}_j |\dot{y}_j| \quad (\text{with } j=A, B \text{ where } A \text{ is the gravity center of floater 1 and } B \text{ is the gravity center of floater 2}),$$

$$F_{m_{Cyix}} = \rho_e C_m V_i \ddot{x}_{oi} + \frac{1}{2} \rho_e C_d S_i \dot{x}_{oi} |\dot{x}_{oi}|,$$

$$F_{m_{Cyi}} = \rho_e C_m V_i \ddot{y}_{oi} + \frac{1}{2} \rho_e C_d S_i \dot{y}_{oi} |\dot{y}_{oi}| \quad \text{where } \rho_e \text{ is the fluid density, } C_m \text{ is the added mass coefficient, } C_d \text{ is defined as drag coefficient, } S_j = r_i H \arccos\left(\frac{y_j}{r_i}\right) \text{ and } S_i = R_i H \arccos\left(\frac{y_{oi}}{R_i}\right) \text{ are respectively the wetted cross-}$$

section area of the floater j and the cylinder i and H is the length of the cylinder i . The Archimedes forces F_{arAix} , F_{arAiy} , F_{arCyix} and F_{arCyiy} along the $\overrightarrow{Ox_0}$ and $\overrightarrow{Oy_0}$ axis respectively, are defined by $-\rho_e V_{im} g$, where V_{im} is the immersed volume. The coefficient β is related to the power take off device, $\dot{\alpha}_i$ is the angular velocity of the cylinder i .

The following notations were introduced $M_{T1} = m_{cy1} + m_{b1} + m_{a1}$, $M_{T2} = m_{cy2} + m_{b2} + m_{a2}$, $C_1 = \frac{l_1}{M_{T1}} \left(\frac{m_{b1}}{2} + m_{cy1} \right)$, $C_2 = \frac{l_2}{M_{T2}} \left(\frac{m_{b2}}{2} + m_{a2} \right)$, $C_3 = m_{a1} \frac{r_1^2}{2} + m_{cy1} \frac{R_1^2}{2} + m_{b1} \frac{l_1^2}{12} + m_{cy1} l_1 (l_1 - C_1) + m_{b1} \frac{l_1}{2} \left(\frac{l_1}{2} - C_1 \right)$, $C_4 = l_1 \left(\frac{m_{b1}}{2} + m_{cy1} \right) - C_1 M_{T1}$, $C_5 = m_{a2} \frac{r_2^2}{2} + m_{cy2} \frac{R_2^2}{2} + m_{b2} \frac{l_2^2}{12} + m_{a2} l_2 (l_2 - C_2) + m_{b2} \frac{l_2}{2} \left(\frac{l_2}{2} - C_2 \right)$, $C_6 = C_2 M_{T2} - l_2 \left(\frac{m_{b2}}{2} + m_{a2} \right)$, $C_7 = C_1 M_{T1} + l_1 M_{T2} + m_b l_1$, $C_8 = C_4 + M_{T1} (C_1 - l_1)$, $C_9 = C_3 + M_{T1} (C_1 - l_1) C_1$, $C_{10} = C_6 - M_{T2} C_2$, where R_i is the radius of cylinder i ($i = 1$ for cylinder 1 and $i = 2$ for cylinder 2), l_i is the length of the linking bar between cylinder i and floater i , m_{bi} is the mass of the linking bar between cylinder i and floater i , m_{ai} is the mass of the floater i , m_{cyi} is the mass of the cylinder i , m_{bi} is the mass of the linking bar between cylinders i . The 4th order Runge-Kutta method is used to solve the system of coupled differential equations (5)-(9).

4. Results and discussion

In the following we consider a DEXA wave where $H = 1m$, $L = 1,3m$, $r_1 = r_2 = 0,6m$, $R_1 = R_2 = 0,6m$, $l_1 = l_2 = 1,8m$, $m_{a1} = m_{a2} = m_{cy1} = m_{cy2} = 579kg$, $m_{b1} = m_{b2} = 1kg$, $\beta = 700Ns/rad$ and $A_m = 0.04m$. It should be noted that the geometry and PTO parameters do not result from an optimization process. The energy recovered by the WEC for a fixed time duration of 15s, is calculated for 4 cases of waves by considering for each case waves of 4 harmonics of the same amplitude. Case 1 includes harmonics with frequencies (0,4,0,6,0,7,0,75 rad/s), case 2 (0,8,0,9,1,1,2 rad/s), case 3 (1,4,1,8,2,2,2 rad/s) and case 4 (2,4,2,8,3,3,2 rad/s). Fig. 2 shows the energy recovered in each case with the four harmonics (first histogram) then that obtained by removing one harmonic each time (2nd, 3rd, 4th and 5th histogram). It is noticed that the WEC is sensitive to the suppression of harmonics for frequencies range lower than 1 but weakly respond to the suppression of harmonics at higher frequencies. This observation allows, in the case of random and irregular waves to focus the identification of the sea state on a range of frequencies at which there is a high sensitivity of the WEC and a significant impact on the energy recovery process.

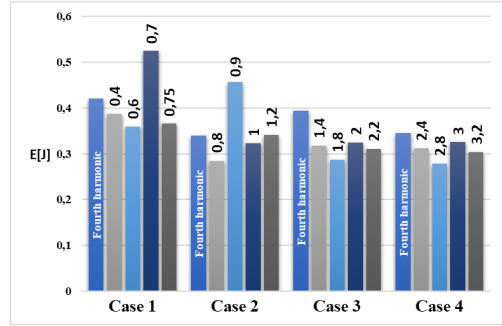


Fig. 2. Recovered energy by DEXA wave in each case

5. Conclusion

This paper investigated the influence of different bandwidths of wave frequencies on the wave energy conversion by a DEXA wave device. The obtained results show that there is a range of frequencies at which there is a high sensitivity of the WEC and a significant impact on the energy recovery process. This observation allows, in the case of random and irregular waves to focus the identification of the sea state on a specific bandwidth for incoming waves.

6. References

- [1] T. Aderinto and H. Li, *Review on Power Performance and Efficiency of Wave Energy Converters*, Energies 12 (2019) 1-24.
- [2] Garcia-Teruel and D.I.M. Forehand, *A review of geometry optimisation of wave energy converters*, Renewable and Sustainable Energy Reviews 139 (2021) 110593.
- [3] X. Zhang and J. Yang, *Power capture performance of an oscillating-body WEC with nonlinear snap through PTO systems in irregular waves*, Applied Ocean Research 52 (2015) 261-273.
- [4] X. Ji, E. Al Shami, J. Monty and X. Wang, *Modelling of linear and non-linear two-body wave energy converters under regular and irregular wave conditions*, Renewable Energy 147 (2020) 487-501.
- [5] B. Zanuttigh, E. Angelelli and J. P. Kofoed, *Effects of mooring systems on the performance of a wave activated body energy converter*, Renewable Energy 57 (2013) 422-431.
- [6] J.R. Morison, M.P. O'Brien, J.W Johnson and S.A. Schaaf, *The forces exerted by surface waves on piles*, Petroleum Transactions of AIME 189 (1950) 149-157.
- [7] W.H. Press, S.A. Teukolsky, W.T. Vetterling and B.P. Flannery, *Numerical recipes in fortran 77*, Cambridge University Press, 1992.
- [8] L. Wang and J. Isberg, *Nonlinear Passive Control of a Wave Energy Converter Subject to Constraints in Irregular Waves*, Energies 8 (2015) 6528-6542.
- [9] J.N. Newman, *Marine Hydrodynamics*, M.I.T. Department of Naval Architecture and Marine Engineering, M.I.T Press, 1999.
- [10] Li. Yong and L. Mian, *Regular and irregular wave impacts on floating body*, Ocean Engineering 42 (2012) 93-101.

Optimisation de la forme du bulbe d'étrave d'un bateau dépollueur

Abdelkader LAHLALI^{1*}, Zakaria EL MASKAOUI¹, Lahbib BOUSSHINE¹

¹Laboratoire de Mécanique d'Ingénierie et d'Innovation LM2I, ENSEM, Université Hassan II de Casablanca.

kader_lali@yahoo.fr, z.elmaskaoui@ensem.ac.ma, lbousshine@yahoo.fr

Résumé :

L'objectif de cet article est d'optimiser la forme du bulbe d'un bateau dépollueur multi-missions pour agir rapidement et efficacement face à une pollution marine de grande ampleur. Néanmoins, les navires dépollueurs existants ne répondent point à certains critères de sécurité, capacités de stockage et délais d'intervention. Notre idée innovante est de permettre de satisfaire toutes ces exigences en procédant à un ratissage global et rapide de la nappe afin de récupérer la totalité des hydrocarbures versés en toute sécurité. A cet effet, nous allons adopter les caractéristiques principales requises et optimiser la forme du bulbe d'étrave propre à ce navire pendant son déplacement afin d'aplatir les vagues d'accompagnement formées autour de la coque, en vue d'uniformiser l'écoulement en amont des ouvertures latérales de récupération des hydrocarbures pour permettre leur collecte à grande vitesse.

Mots clés : *Bulbe d'étrave, navire dépollueur, marée noire, vagues d'accompagnement, résistance à l'avancement.*

1. Introduction

Dans notre planète, la superficie totale des mers et océans représente 71 % de sa surface, soit une surface de 360700000 km². Ces océans sont d'une importance primordiale dans l'équilibre social, économique et environnemental au profit de toute l'humanité. Par ailleurs, le transport maritime représente près d'un tiers du commerce mondial et les pétroliers ont vu leur capacité augmenter de 73% depuis l'an 2000 [1]. En revanche, ces derniers sont confrontés à plusieurs risques, tels que l'échouage, l'incendie, le naufrage, ou le chavirement. Suite à ces incidents, des marées noires peuvent survenir à cause de déversement accidentel d'une grande quantité de pétrole ramenée vers la côte par l'effet des marées, des vents ou des courants [2]. Dans ce contexte, les bateaux dépollueurs sont conçus pour préserver un environnement marin sain dans le cas d'une catastrophe écologique du genre [3]. A cet effet, pour améliorer leur performance lors d'une intervention, ils doivent avoir, entre autres, un bulbe d'étrave de forme optimisée, en vue de diminuer la résistance totale à l'avancement en général et aplatir les vagues d'accompagnement formées autour de la coque pour faciliter la collecte des hydrocarbures en particulier [6].

2. La résistance des vagues sur les carènes

Lors d'un déplacement d'un navire, nous observons la formation des vagues d'accompagnement transversales et divergentes (Fig. 1).

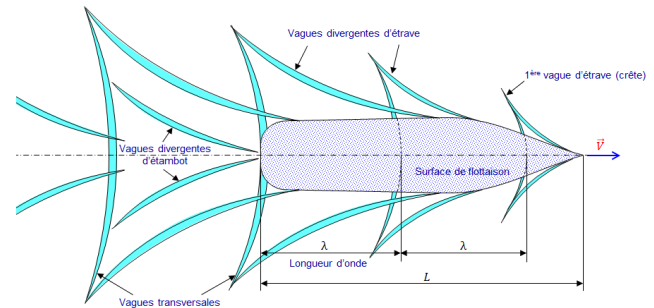


Fig. 2 : Les champs de vagues d'accompagnement.

Pendant l'écoulement autour de la coque, le bulbe d'étrave joue le rôle d'un obstacle qui favorise la création d'une vague (3) déphasée par rapport à celle créée par l'étrave (4).

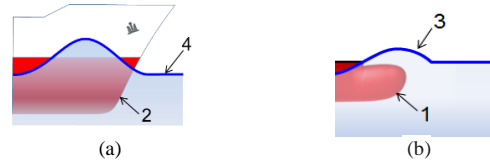


Fig. 3 : Les vagues générées par l'étrave (2) et le bulbe (1)

Toutefois, les deux vagues issues respectivement par le bulbe et l'étrave se trouvent en opposition de phase le long de la coque. La superposition de ces deux ondulations annule, par conséquent, leur effet le long de la coque [6].

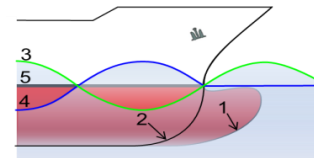


Fig. 3: La superposition des vagues (3) et (4) en opposition de phases.

Pour que la superposition donne une flottaison horizontale sans aucune ondulation (Fig. 4), il faut que l'opposition de phase des deux vagues soit parfaite.

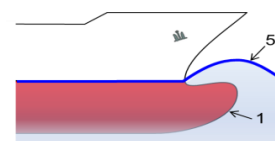


Fig. 4: La superposition des vagues (3) et (4) annule leur effet en donnant (5).

3. Optimisation du bulbe d'étrave

Depuis l'adoption des bulbes d'étrave afin d'améliorer la vitesse et réduire la résistance à l'avancement, le concepteur ne cesse d'améliorer sa forme. Cette forme s'est évoluée depuis la nuit des temps d'une forme d'un éperon à une forme hydrodynamique la plus adaptée à nos jours. Certes, la forme la plus appropriée actuellement est celle en Col de Cygne [4] (Fig. 5). En revanche, ses dimensions, qui la caractérisent, nécessitent une optimisation en fonction des dimensions principales de la coque attribuée.

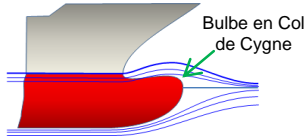


Fig. 5 : Ecoulement autour d'un Bulbe en Col de Cygne optimisé.

En effet, la forme du bulbe en Col de Cygne constitue la forme d'obstacle la plus appropriée qui facilite l'amorçage de la vague décalée par rapport à celle créée par l'étrave. Par conséquent, si le bulbe n'est pas optimisé, les deux vagues issues respectivement de bulbe et l'étrave ne se propagent pas en parfaite opposition de phase (Fig. 6) tout en donnant lieu à une vague résultante (5) non horizontale.

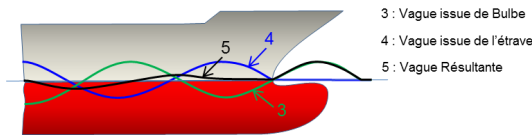


Fig. 6 : Ecoulement autour d'un Bulbe en Col de Cygne non optimisé.

Actuellement, il n'y a aucune approche analytique permettant la conception d'un bulbe d'étrave parfait, vu la complication des interactions hydrodynamiques entre le bulbe et l'étrave. En revanche, pour prédire les meilleures qualités hydrodynamiques, l'expérience et la simulation s'avèrent fondamentales pour aboutir à des résultats plus performants (Fig. 7) [5].

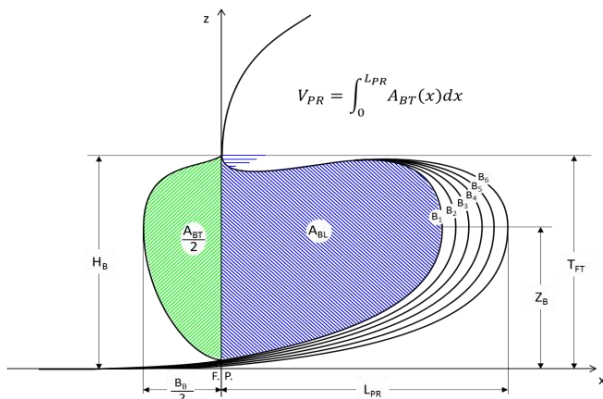


Fig. 7 : Optimisation de la longueur L_{PR} de bulbe en col de cygne.

Dans le tableau ci-dessous, nous avons rapporté les caractéristiques numériques des différents bulbes. Ainsi, cette optimisation sera validée à l'aide d'un calcul hydrodynamique numérique itératif.

Tab. 1 : Dimensions de bulbes en col de cygne.

Bulbe	N° 1	N° 2	N° 3	N° 4	N° 5	N° 6
L_{PR} (m)	5,00	5,30	5,60	5,90	6,20	6,50
B_B (m)	3,00	3,00	3,00	3,00	3,00	3,00
Z_B (m)	3,60	3,60	3,60	3,60	3,60	3,60
H_B (m)	5,40	5,40	5,40	5,40	5,40	5,40
A_{BT} (m ²)	11,54	11,54	11,54	11,54	11,54	11,54
A_{BL} (m ²)	19,71	21,23	22,79	24,44	26,12	27,89
V_{PR} (m ³)	40,40	45,20	50,02	54,94	59,94	65,04

4. Stratégie d'optimisation

Cette stratégie consiste à valider les six modèles de bulbe les plus optimaux par itérations successives. Ensuite, nous allons choisir le meilleur dans le but de minimiser la résistance dues aux vagues d'accompagnement.

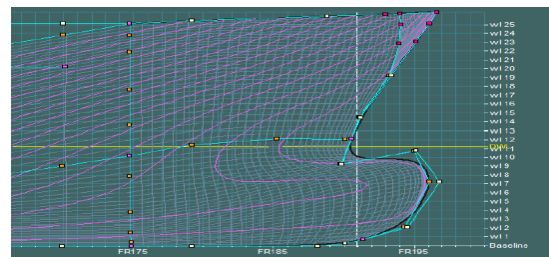


Fig. 8 : Simulation de bulbe d'étrave sur Maxsurf.

Pendant cette étude, l'objectif de cette optimisation vise à minimiser la résistance des vagues. En fixant la section transversale du bulbe, nous avons modifié la section longitudinale, pour avoir 6 formes de bulbes optimisés au préalable (Fig. 8). L'interface de calcul hydrodynamique réalisée a été utilisée pour tracer les courbes des résistances dues aux vagues des différents bulbes, en fonction de la vitesse.

5. Résultats

Le but de ce calcul est de choisir un bulbe parmi les six, qui génère des vagues en opposition de phase sans décalage par rapport aux celles générées par la coque. Automatiquement, la résultante de ces deux vagues opposées se converti à l'horizontal. Par conséquent, pendant l'intervention dans la nappe, les bras latéraux en position ouverte ne seront pas perturbés par les masses d'eau dues aux vagues d'accompagnement et les écoulements des hydrocarbures à travers les ouvertures latérales vers l'intérieur se font d'une manière harmonieuse et laminaire (Fig. 9).

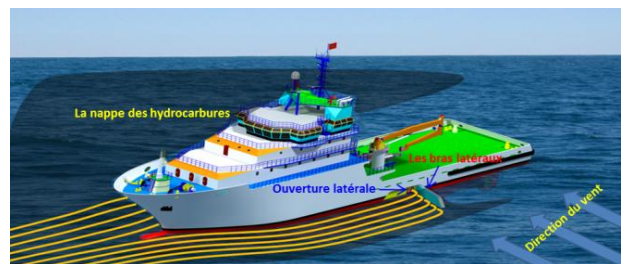


Fig. 9 : Navire dépollueur pendant sa collecte des hydrocarbures.

Vu que le calcul hydrodynamique classique nécessite un test sur un modèle réduit dans un bassin des carènes pour calculer la résistance R_W , nous allons utiliser les équations empiriques entrepris par J. Holtrop et G.G.I. Mennen objet de la référence [5] de cette communication. Cette méthode a été développée grâce à une analyse de régression basée sur les résultats des expériences sur 334 modèles réduits, réalisées au Bassin des Carènes Néerlandais dont la résistance totale à l'avancement a été subdivisée en :

$$R_T = R_F(I+k_I) + R_{APP} + R_W + R_B + R_{TR} + R_A$$

- R_F : Résistance au frottement selon la conférence ITTC 1957 [6] ;
- $I + k_I$: Facteur de forme décrivant la résistance visqueuse du type de coque par rapport au R_F ;
- R_{APP} : Résistance des appendices ;
- R_W : Résistance due aux vagues d'accompagnement ;
- R_B : Résistance à la pression supplémentaire de bulbe d'étrave près de la surface de l'eau ;
- R_{TR} : Résistance à la pression supplémentaire de la partie du tableau arrière immergé ;
- R_A : Résistance de corrélation modèle-réel.

A l'aide d'un code de calcul sur MATLAB, nous allons calculer toutes ces résistances. Ensuite, nous allons représenter dans un même graphe les trois résistances prédominantes : visqueuse $R_V = R_F(I+k_I)$, des vagues R_W et totale R_T (Fig. 10).

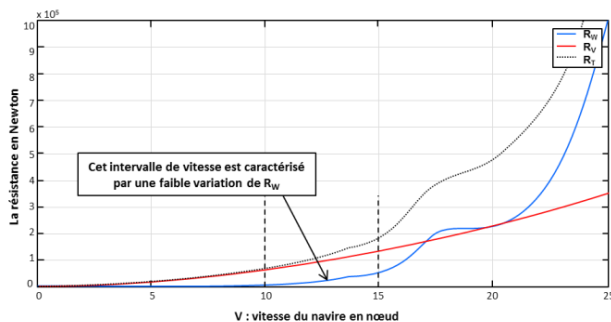


Fig. 10 : R_V , R_W et R_T en fonction de la vitesse.

A cet effet, il est nécessaire de simuler graphiquement la résistance des vagues R_W pour les six bulbes d'étrave. Le problème revient donc à déterminer la résistance R_W minimum pour le bulbe le plus optimum (Fig. 10).

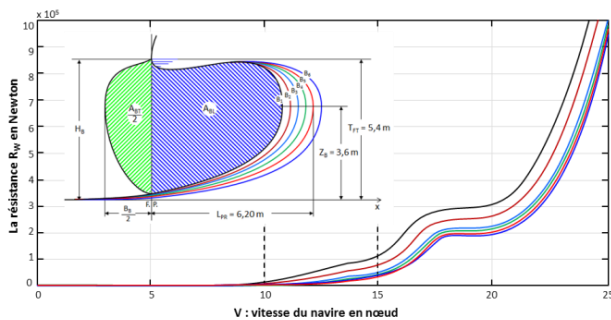


Fig.11 : R_W pour les 6 bulbes en fonction de la vitesse.

Pour la gamme de vitesse d'intervention entre 10 et 15 nœuds, les résultats obtenus les plus recherchés sont ceux du bulbe N°5.

Conclusion

En guise de conclusion, les déversements d'hydrocarbures en mer ont été extrêmement préoccupants en raison de leur impact négatif sur les systèmes économiques et écologiques. Ainsi, le développement d'un navire dépollueur innovant, autonome et multi-missions s'avère inéluctable dans la mesure où les navires dépollueurs existants sont incapables d'accomplir avec succès toute mission de dépollution marine. Certes, ces navires dotés de technologies classiques ne répondent point à certains critères de sécurité, capacités de stockage et délais d'intervention. Dans cette communication, nous avons décrit les performances d'un nouveau concept d'un navire dépollueur capable de satisfaire toutes ces exigences tout en procédant à une récupération totale et rapide des hydrocarbures versés en toute sécurité.

Aussi, pour accomplir cette mission spéciale, nous allons choisir judicieusement les caractéristiques principales nécessaires de ce navire et optimiser son bulbe d'étrave en *Col Cygne*, le bulbe le plus performant jusqu'à présent, afin de faciliter la collecte des hydrocarbures versés en grande vitesse. A cet effet, cette optimisation consiste à ajuster la longueur appropriée de son bulbe d'étrave afin d'aplatir à l'horizontal les vagues d'accompagnement constituées autour de la coque, en vue de diminuer la résistance totale à l'avancement d'une part, et uniformiser le flux en amont des ouvertures latérales de récupération des hydrocarbures d'autre part.

Les résultats obtenus de notre optimisation, certes sont efficaces. En revanche, ils représentent toujours une approche théorique qui aide à la décision. Toutefois, les essais dans les bassins des carènes demeurent indispensables afin de valider et ajuster les résultats obtenus par simulation.

Références

- [1] FAOF fisheries and aquaculture department, "the state of world fisheries and aquaculture", Food and Agriculture Organization of the United Nations, Rome, 2007.
- [2] S. Binkley Kanbace "Ocean technology and Interdisciplinary Coordination", National Science Foundation, February 25, 2019.
- [3] G. Chantelave, "Evaluation des risques et réglementation de la sécurité : cas du secteur maritime-tendances et applications". L'institut National des sciences Appliquées de Lyon, Soutenu le 11 Janvier 2006.
- [4] J.-W. Yu, Y.-G. Lee, and K.-L. Jeong, 'A Study on the Resistance Performance of the Goose Neck Bulbous Bow by Numerical Simulation Method', J. Soc. Nav. Archit. Korea, vol. 47, no. 5, pp. 689-696, Oct. 2010, doi:
- [5] Holtrop, J. and Mennen, G.G.J., 'An approximate power prediction method', International Shipbuilding Progress, Vol. 29, July 1982.10.3744/SNAK.2010.47.5.689.
- [6] Ship Resistance and Propulsion: Practical Estimation of Ship Propulsive Power 2nd Edition. By Anthony F. Molland (Author), Stephen R. Turnock (Author) and Dominic A. Hudson (Author).

Investigation numérique de l'interaction houle-plaque-courant par la méthode des éléments frontières

H. AKARNI^{1*}, L. EL AARABI¹, S. MORDANE¹

1. Laboratoire de Physique des Polymères, Sciences Mécanique et Matériaux (LPPSMM). Faculté des Sciences Ben M'sick, Université Hassan II de Casablanca
*hasnaakarni97@gmail.com

Abstract :

The objective of this work is the numerical study of the interaction of a wave-horizontal plate totally immersed in a flat-bottomed tank with a uniform current flowing in the same direction as the incident wave. The numerical method used in this study is the Boundary Element Method (BEM) and the results obtained will be compared with analytical results and experimental measures existing in the literature.

Résumé :

L'objectif de ce travail, est l'étude numérique de l'interaction houle-plaque horizontale totalement immergée dans un canal à fond plat, en présence d'un courant uniforme de même sens que celui de la houle incidente. La méthode numérique abordée dans cette étude est celle des éléments frontières (BEM) et les résultats obtenus seront confrontés aux résultats analytiques et aux mesures expérimentales existants dans la littérature.

Keywords : wave, plate, current, boundary element method (BEM), reflection coefficient, free surface elevation.

Mots clés : houle, plaque, courant, méthode des éléments frontières (BEM), coefficient de la réflexion, élévation de la surface libre.

1 Introduction

Parmi les problèmes qui se posent lors de la construction des structures maritime, leurs protections contre la propagation de la houle. Pour réduire l'impact de cette dernière sur lesdites structures, on utilise des obstacles immergés qui représentent les solutions les plus efficaces pour protéger les littorales et les ports, sans affecter l'esthétique de la vue côtière. Dans l'environnement océanique, la houle est généralement accompagnée par des différents courants qui peuvent être générés par les vents, les vagues, les marées et les débits des rivières. Ce sujet très vaste a fait l'objet de nombreuses recherches dans le domaine hydrodynamique maritime, l'ingénierie océanique, l'architecture navale et d'autres disciplines.

Plusieurs études expérimentales, analytiques et numériques ont été consacrées à l'interaction houle-obstacle avec et sans courant. En 2011, Rey et al. [1] ont étudié expérimentalement le problème de l'interaction houle-plaque en présence du courant pour des houle régulière et irrégulière. En 2016, Errifaï et al. [2] ont mené une étude analytique de l'interaction houle-plaque-courant, cette étude est basée sur la théorie poten-

tielle linéarisée avec le modèle des modes évanescents. En 2019, Naasse al. [3] ont introduits l'influence des paramètres géométriques sur le coefficient de réflexion [3]. En 2017, Bai et al. [4] ont étudié les forces exercées sur un cylindre horizontal lors du passage d'une houle en présence d'un courant cisailé. En 2019, Ning et al. [5] sont basé sur la théorie potentielle pour estimer les forces et le moment sur une plaque immergée induits par l'interaction houle-courant. En 2021, Akarni et al. [6] ont étudié l'effet de la géométrie de la plaque sur la variation du coefficient de réflexion ainsi que l'élévation de la surface libre en présence du courant.

Dans ce travail, on va présenter une étude numérique de l'interaction houle-plaque-courant en l'utilisation de la méthode des éléments frontières (BEM), afin de déterminer l'élévation de la surface libre et le coefficient de réflexion numérique. Les résultats obtenus seront comparés avec des résultats analytiques ainsi que des mesures expérimentales issus de la littérature.

2 Formulation mathématique

On considère une houle incidente monochromatique, de faible amplitude se propageant dans un canal à fond plat de longueur L et de profondeur H en présence d'un courant uniforme de vitesse horizontale U de même sens que celui de la propagation de la houle incidente, et en présence d'une plaque de longueur l et d'épaisseur e . La plaque est immergée à une profondeur h au-dessous de la position prise par la surface libre au repos (figure 1).

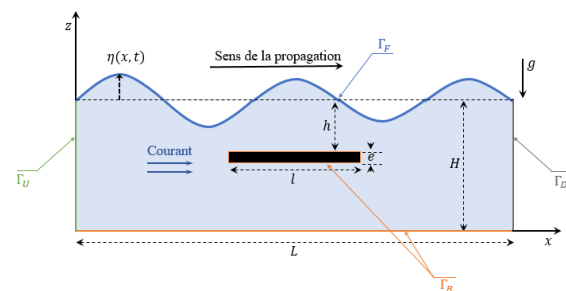


Figure 1: Schéma descriptif du domaine d'étude

On suppose que le fluide est parfait, homogène, pesant et incompressible. L'écoulement est bidimensionnel et irrotationnel dans le plan (O, x, z) . Le problème bidimensionnel de la houle linéaire se ramène à déterminer le potentiel des vitesses $\Phi(x, z, t)$ et l'élévation de la surface libre $\eta(x, t)$. Ces deux variables seront supposées complexes et leur dépendance par rapport au temps est harmonique. Le potentiel des vitesses $\Phi(x, z, t)$ peut s'écrire

sous la forme :

$$\Phi(x, z, t) = \Phi_c(x) + \Phi_h(x, z, t) \quad (1)$$

Où $\Phi_c(x) = U.x$ désigne le potentiel associé au courant et $\Phi_h(x, z, t) = \phi_h(x, z)exp(i\omega t)$ est le potentiel des vitesses associé à la houle. U , i , ω et t sont respectivement la vitesse du courant, le nombre complexe, la pulsation de l'onde incidente et le temps. Le problème de l'interaction houle-plaque en présence du courant se ramène à la résolution du système d'équations suivant:

$$\left\{ \begin{array}{ll} \Delta\phi_h = 0 & \text{dans } D \\ \frac{\partial\phi_h}{\partial n} = -ik^-\phi_h + i(k^- + k^+)f(z) & \text{sur } \Gamma_U \\ \frac{\partial\phi_h}{\partial n} = 0 & \text{sur } \Gamma_B \\ \frac{\partial\phi_h}{\partial n} = \frac{\omega c_0^2(2Uk - \omega)}{g(U^2 - c_0^2)}\phi_h & \text{sur } \Gamma_F \\ \frac{\partial\phi_h}{\partial n} = -ik^+\phi_h & \text{sur } \Gamma_D \\ \eta = -\frac{i(\omega - k^+U)}{g}\Phi_h - \frac{U^2}{2g} & \text{Élévation de} \\ & \text{la surface libre} \end{array} \right. \quad (2)$$

Où n est la normale extérieure sur la frontière ($\partial D = \Gamma_U \cup \Gamma_B \cup \Gamma_D \cup \Gamma_F$) qui délimite le domaine (D), g est l'accélération de la pesanteur, k est le nombre d'onde de la houle incidente en absence du courant, k^+ et k^- sont respectivement, le nombre d'onde de l'onde incidente et de l'onde réfléchiée en présence du courant et c_0 est la célérité ou la vitesse de propagation de la houle incidente en absence du courant, égale à :

$$c_0 = \sqrt{\frac{gH.tanh(kH)}{kH}} \quad (3)$$

$f(z)$ est une fonction liée aux caractéristiques de la houle tel que :

$$f(z) = \frac{a_i g}{(\omega - k^+U) \cosh(k^+H)} \cosh(k^+z) \quad (4)$$

Où a_i est l'amplitude de l'houle incidente en présence du courant, k^+ et k^- sont les solution de l'équation de dispersion en présence d'un courant uniforme U . En tenant compte de ce dernier, l'équation de dispersion est définie par :

$$(\omega \mp Uk^\pm)^2 = gk^\pm tanh(k^\pm H) \quad (5)$$

On remarque que l'ensemble des conditions aux limites s'exprime comme des relations entre le potentiel ϕ_h et sa dérivée normale $\frac{\partial\phi_h}{\partial n}$. Ceci justifie le choix de la méthode des éléments aux frontières.

3 Formulation numérique

L'application de la seconde identité de Green permet d'exprimer ϕ sous forme intégrale de la manière suivante:

$$c\phi_h(x, z) = \int_{\partial D} \left[\frac{\partial\phi_h(x', z')}{\partial n} G(r) - \frac{\partial G(r)}{\partial n} \phi_h(x', z') \right] ds \quad (6)$$

Où $c = 0$ si les points de coordonnées $(x, z) \notin D \cup \partial D$, $c = 0.5$ si les points de coordonnées $(x, z) \in \partial D$ et $c = 1$ si les points de coordonnées $(x, z) \in D$.

$G(r) = -\frac{1}{2\pi} \ln(r)$ est la fonction de Green avec $r = \sqrt{(x - x')^2 + (z - z')^2}$ est la distance entre un point du domaine et un point de la frontière.

Dans la formulation numérique, la frontière est subdivisée en un nombre fini de N segments. On peut réécrire la relation (6) sous la forme matricielle suivante :

$$\{\phi_h\} = [K] \left\{ \frac{\partial\phi_h}{\partial n} \right\} \quad (7)$$

Ce système est de N équations à $2N$ inconnues, qui sont respectivement le potentiel et sa dérivée normale sur les frontières du domaine d'étude. Les N équations supplémentaires s'obtiennent par l'écriture des conditions aux limites. Ces conditions aux limites peuvent être écrite sous la forme matricielle suivante:

$$\left\{ \frac{\partial\phi_h}{\partial n} \right\} = [F] \{\phi_h\} + \{S\} \quad (8)$$

En injectant ces conditions aux limites(8) dans la relation (7), la détermination du potentiel des vitesses associé à la houle ϕ_h se ramène donc à la résolution du système suivant :

$$[L] \{\phi_h\} = \{SN\} \quad (9)$$

Où $[L]$ est la matrice tangente et $\{SN\}$ le seconde membre.

4 Résultats

4.1 Validation du code numérique : (interaction houle-courant)

Dans cette partie, on va valider l'approche numérique proposée à résoudre le problème de l'interaction houle linéaire- courant uniforme, en comparant l'élévation de la surface libre analytique et numérique. Pour cela, on considère une houle incidente monochromatique d'amplitude $a = 0.01 m$ et de longueur d'onde $\lambda = 15 m$, se propage dans un canal à houle numérique de longueur $L = 30 m$ et de profondeur $H = 2.5 m$ en présence du courant de vitesse $U = 0.1m/s$.

La figure 2 présente une comparaison entre notre solution numérique obtenue par la BEM et la solution analytique (équation10) en fonction de la longueur du canal à l'instant $t = T$. D'après cette figure, on remarque que plus le maillage est raffiné plus la solution numérique converge vers la solution analytique ce qui permet de valider l'approche numérique utilisée.

Où la solution analytique s'écrit comme suit:

$$\eta = \frac{a\omega(\omega - k^+U)}{g} \sin(\omega - k^+U) - \frac{U^2}{2g} \quad (10)$$

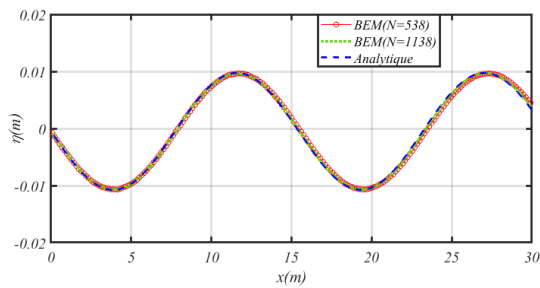


Figure 2: Elévation de la surface libre numérique et analytique en fonction de la longueur du canal pour $t=T$

4.2 Interaction houle-plaque-courant

Après avoir testé la validité de l'approche numérique proposée, on présente dans la figure 3 une comparaison de nos résultats numériques du coefficient de réflexion avec ceux d'un modèle analytique d'onde plane corrigé [3] et des mesures expérimentales de Rey et Touboul [1], en traçant les variations du coefficient de réflexion en fonction de la période T où les calculs ont été faits en présence d'un courant de vitesse $U=0.3\text{m/s}$.

D'après les courbes présentées dans cette figure, On montre que nos résultats numériques approchent mieux les mesures expérimentales.

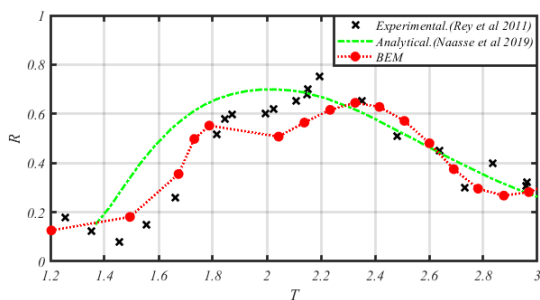


Figure 3: Coefficient de réflexion R en fonction de la période T

La figure 4 présente l'élévation de la surface libre en fonction de la longueur du canal pour différents instants. Les calculs numériques ont été effectués pour une plaque d'épaisseur $e = 5\text{ cm}$ et de longueur $l = 30\text{ cm}$ immergée à $h = 1.25\text{ m}$. D'après les figures 4-a et 4-b, on remarque une création des nœuds et des ventres en amont de la plaque qui sont dus à la réflexion. De plus on observe une augmentation d'amplitude et de la largeur des ventres ainsi qu'un décalage des maximums et des minimums vers le bas quand la vitesse du courant augmente.

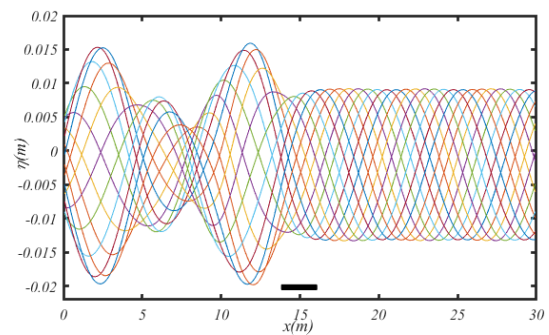
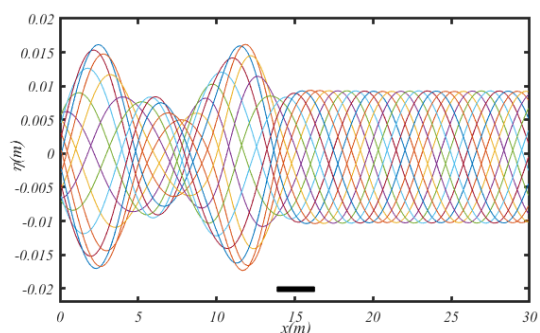


Figure 4: Elévation de la surface libre en fonction de la longueur du canal pour différents instants et $\lambda = 8\text{ m}$. (a): $U = 0.1\text{ m/s}$ (b): $U = 0.2\text{ m/s}$

5 Conclusion

Dans ce travail, nous avons étudié numériquement le problème de l'interaction houle-plaque-courant, l'approche adoptée consiste en une modélisation numérique basée sur la méthode des éléments aux frontières (BEM). En effet, le code de calcul numérique développé s'avère efficace pour calculer d'une part le coefficient de réflexion et d'autre part l'élévation de la surface libre lors du passage d'une houle monochromatique sur un obstacle en présence d'un courant uniforme de même sens que celui de la propagation de la houle incidente.

References

- [1] V. Rey, J. Touboul *Forces and moment on a horizontal plate due to regular and irregular waves in the presence of current*, Applied Ocean Research 33 (2011) 88-99.
- [2] M. Errifaiy, S. Naasse, C. Chahine *Analytical determination of the reflection coefficient by the evanescent modes model during the wave-current-horizontal plate interaction*, Comptes Rendus Mécanique 344(7) (2016) 479-486.
- [3] S. Naasse, M. Errifaiy, C. Chahine *Analytical study of the effect of the geometrical parameters during the interaction of regular wave-horizontal plate-current*, Acta Oceanologica Sinica 38 (2019) 10-20.
- [4] J. Bai, N. Ma, X. Gu *Study of interaction between wave-current and the horizontal cylinder located near the free surface*, Applied Ocean Research 67 (2017) 44-58.
- [5] D. Ning, L. Chen, H. Lin, Q. Zou, B. Teng *Interaction mechanisms among waves, currents and a submerged plate*, Applied Ocean Research 91 (2019) 101911.
- [6] H. Akarni, L. El Arabi, L. Mouakkir, S. Mordane *Numerical Modeling of the Wave-Plate-Current Interaction by the Boundary Element Method*, Fluids 6 (2021) 12:435.

Detection of erosion at the fluid/soil interface during the Hole Erosion Test

Kissi Benaissa^{1*}, Chafik Guemimi¹, M. Angel Parron Verra²

1 University of Hassan II, National High School for the Arts and Professions, LISPSII, Casablanca, Morocco,
2 High polytechnic School of Algeciras, Ramon Puyol s/n, 11202 Algeciras, Spain

Benaissa.kissi@gmail.com

Abstract

Many dam ruptures events have occurred throughout the world, some of them were reported by Foster et al. [1]. Then main cause was piping phenomenon that occurred in the foundation soil or in the dam structure. Serviceability of hydraulic infrastructures needs considering vulnerability of soil to internal erosion under the action of a seepage flow, [2,3]. Understanding the underlying mechanisms and quantifying the effects of pertinent variables that affect this phenomenon is of great importance in order to prevent such catastrophes. Erosion due to liquid flow discharge can be modeled by different approaches. These use some parameters that are identified from laboratory tests stating when erosion starts and what is the expected erosion rate. Among the most important tests that are used for this purpose, one finds the Hole Erosion Test (HET). A simplified one-dimensional model for interpreting the HET with a constant pressure drop was developed by Bonelli and Brivois [4,5]. This model yielded a characteristic erosion time which was found to be depending on the initial hydraulic gradient and the soil coefficient of erosion.

Keywords: *Erosion, Piping, Hole Erosion Test, Computational fluid dynamics, Renormalization group $k - \varepsilon$ model*

1. Introduction

Internal erosion is one of the most common causes of failure of earth dams. There may be no external evidence, or only subtle evidence, that it is taking place. A dam may breach within a few hours after evidence of the internal erosion becomes obvious. Internal erosion may develop the first time water is impounded behind a dam, or it may develop slowly over many years.

Higher water surface elevations and pressure may exacerbate or initiate internal erosion.

You cannot assume that your dam is safe against internal erosion just because it has performed satisfactorily for many years.

Internal erosion failures are often associated with "penetrations" of dams, such as outlet pipes buried in the

embankment, rodent activity, and concrete spillways that cross the embankment. An experienced dam engineer may be able to detect the subtle signs of internal erosion during routine periodic inspections, but you should be aware of what signs to look for between inspections, it is a consequence of forces imposed by groundwater flow, and it's a process that develops concentration of seepage flows and in time may develop large cavities that may produce embankment dam failure with a catastrophic and uncontrolled release of the reservoir. Internal erosion takes several forms, including heave, piping, concentrated leak erosion, contact erosion, and suffusion [International Commission on Large Dams. The effects of these various internal erosion mechanisms have been reported to account for approximately half the dam and levee failures and incidents throughout the world .

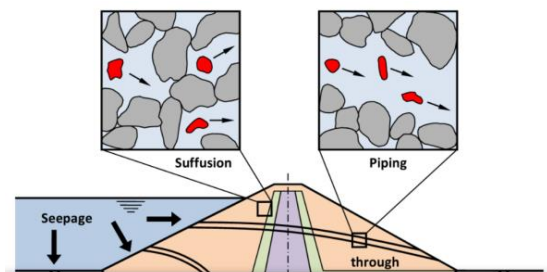


Fig. 1. Erosion and piping process in earth dams [3].

2. Results and discussion

Additional aspects due to two-dimensional features of the HET were observed during tests. For instance, the inlet side of the sample hole undergoes generally much more erosion than the outlet side. But, one-dimensional modeling of this test could not predict this eroded shape since it yields uniform erosion at the whole fluid/soil interface inside the soil sample hole.

The aim of this study is to describe the biphasic turbulent flow at the origin of erosion taking place inside the porous soil sample by considering the influence of variation of the concentration of clay contained in the flowing fluid. A Computational Fluid Dynamics (CFD) approach is used to investigate the shear stress that

develops at the water/soil interface and which represents the main mechanical action that causes surface erosion.

The turbulence modeling of water flowing inside the hole of the HET sample was achieved by means of Fluent software package. This general purpose CFD code is especially appropriate for considering mixtures by modeling each fluid phase independently or as a homogenized medium, [6]. In this work, the modeling is performed by means of an improved version of the standard $k-\varepsilon$ model which is obtained by Renormalization Group Theory (RNG) [7,8]. In comparison with the standard model, RNG based $k-\varepsilon$ model includes refinements which significantly improve the accuracy for rapidly strained flow. The RNG theory provides an analytically derived differential formula for effective viscosity that accounts for low Reynolds number effects when an appropriate treatment of the near-wall region is performed. These advantages make the RNG based $k-\varepsilon$ model more accurate and reliable for a wider class of flows than the standard $k-\varepsilon$ model.

When the shear stress is calculated by means of Fluent, the classical linear erosion law is used to estimate erosion rate. This law gives erosion rate, considered to be the amount of mass departure due to erosion per unit time and by unit surface area, by $\dot{\varepsilon}_{er} = c_{er}(\tau - \tau_{cr})$ where c_{er} and τ_{cr} are constants depending on the considered soil material. For a cylindrical hole, the rate $\dot{\varepsilon}_{er}$ can be related to time variation of local radius by $\dot{\varepsilon}_{er} = \rho_d dR/dt$ where ρ_d is the dry density of soil and R is hole radius. The erosion law yields that $\dot{\varepsilon}_{er}$ is proportional to the amount of shear exceeding the critical shear τ_{cr} for which erosion begins.

The standard HET is such that, the fluid domain which is assumed to be axisymmetric extends over 117 mm in the axial z -direction and 3 mm in the radial r -direction. The domain is oriented such that the inlet section is at left and the outlet section is at right, figure 1.

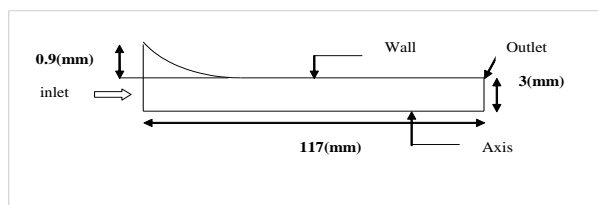


Fig. 2. Geometry of the HET tube

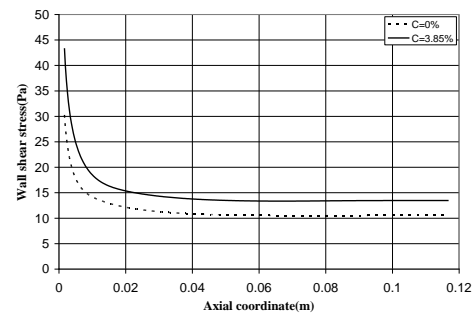


Fig. 3. Wall-shear stress obtained for $P=3726 Pa$ as function of clay concentration

Figure 3 gives the obtained wall-shear stress for inlet pressure $P = 3726 Pa$ and for two clay concentration.

The results show that clay concentration has a significant effect on shear stress and that this last is not uniform along the hole length. Erosion which is proportional to shear stress is also not uniform. This enables to understand why the erosion pattern is not uniform as observed by examining soil samples after HET testing for which the inlet side is usually more eroded than the outlet side.

3. Conclusions

A two-dimensional modelling of the Hole Erosion Test was carried out in this work. Unlike the early models which are essentially one-dimensional, the two-dimensional modeling had shown that the wall-shear stress is not uniform along the hole wall of the soil sample. It was possible then through using a linear erosion law to predict non uniform erosion along the hole length.

Studying the effect of clay concentration has shown that it has not a negligible effect on the wall-shear stress and thus would affect in its turn surface erosion that develops at the fluid soil sample interface, particularly at the outlet extremity of the hole where it is maximal. This enabled qualitatively understanding why the eroded profile of the hole wall as observed during experiment is not uniform.

References

- [1] Foster, R. Fell, M. Spannangle. The statistics of embankment dam failures and accidents. Canadian geotechnical Journal 37(2000), 1000-1024.
- [2] E. Fjar, R.M. Holt, P. Horsrud, A.M. Raaen, R. Risnes. Petroleum related rock mechanics. Revised edition Elsevier, Amsterdam, 2004.
- [3] S. Bonelli, O. Brivois, R. Borghi, N. Benahmed. On the modelling of piping erosion. Comptes Rendus de Mécanique 334 (2006), 555-559.
- [4] D. Lachouette, F. Golay, S. Bonelli. One dimensional modelling of piping flow erosion.

- Comptes Rendus de Mécanique 336 (2008), 731-736.
- [5] S. Bonelli, O. Brivois. The scaling law in the hole erosion test with a constant pressure drop, *International Journal of Numerical Methods in Engineering* 32 (2008), 1573-1595.
 - [6] Fieldview reference Manual, Software Release Version 10, Intelligent Light, 2004.
 - [7] Yakhot V., Orszag S.A. Renormalization Group Analysis of Turbulence: I-Basic Theory. *Journal of Scientific Computing*, 1(1986), 1-51.
 - [8] Fluent 6.2 Users Guide. Fluent Inc., 2005.

Compliant Coaxial Flow Waves: Distensibility Modulation and Fluid Sheets

Analogy

Ondes fluides dans un Coaxial Souple: Modulation de la Distensibilité et

Analogie avec les Films Fluides

P. CATHALIFAUD¹, M. ZAGZOULE²

1. Institut de Mécanique des Fluides de Toulouse (IMFT), catalifo@imft.fr

2. Institut de Mécanique des Fluides de Toulouse (IMFT), mokhtar.zagzoule@imft.fr

Abstract:

In their most simple configuration, i.e. a distensible tube into a rigid one, 1D coaxial fluid-filled models need a *tube law* which *must* distinguish between the rheological and the configurational aspects. This law includes a so-called distensibility parameter which is a *function of cross-sectional area ratio* α , even at zero transmural pressure. In Carpenter's team model [1, 2], this distensibility is assumed *constant*. We will show that this hypothesis leads to an *erroneous non-dimensional wave velocity*, and thus to non-physical phenomenon. When both tubes are compliant, we show that the fast and low velocity waves have symmetrical and antisymmetrical shapes similar to those studied in thin sheets of fluids by G.I. Taylor in 1959 [5].

Résumé:

Dans sa configuration la plus simple, i.e. un tube souple à l'intérieur d'un tube rigide, le modèle 1D coaxial rempli de fluide, nécessite une *loi du tube* qui doit distinguer les aspects rhéologiques des configurationnels. Cette loi comporte un paramètre dit de distensibilité qui est *fonction du rapport de section* α , même à pression transmurale nulle. Dans le modèle de l'équipe de Carpenter [1, 2], cette distensibilité est supposée *constante*. Nous montrons que cette hypothèse entraîne que leur *vitesse d'onde adimensionnelle est erronée*. Ensuite, nous considérons deux tubes souples et montrons que les ondes rapides et lentes ont des formes respectivement symétriques et antisymétriques, similaires à celles étudiées par G.I. Taylor en 1959 [5] dans les fluides en couches minces.

Keywords: Coaxial tubes, distensibility, CSF flow, compliant annular flow waves, thin fluid sheets

Mots clés: Tubes coaxiaux, distensibilité, Écoulement LCS, ondes de tube annulaire souple, films fluides minces

1 Introduction

Wave propagation models of fluid filled concentric distensible tubes is commonly used to investigate physiolog-

ical and physio-pathological situations in the spinal part of the cerebrospinal compartment [6]. As for all wave propagation phenomena the determination of the phase velocity, the speed at which the waves propagate, is crucial. When the wave equation is obtained, by identification the wave velocity is deduced. But one can succeed in finding the right wave velocity expression in its dimensional form and ends up with a wrong *non-dimensional expression*. That's what happened in the model proposed by the Carpenter's team in their 2003 articles (Berkouk et al. [1], Carpenter et al. [2]). Although the physical consequence, the so called "Elastic Jump Hypothesis" (EJH), of their model has been discussed [3, 4], this error has *never been pointed out*. In the following we will show the nature of the error in their model. Hereafter we will refer to Berkouk et al. model of 2003 [1] as the *BCL Model*.

2 1D coaxial model with a rigid and a compliant tube: the BCL model

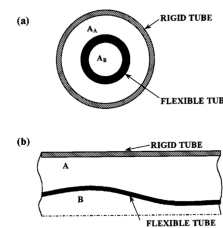


Figure 1: From BCL [1]: A schematic sketch of the theoretical model based on co-axial flexible tubes: (a) end view; (b) side view.

The BCL model is based on one dimensional (1D) modelling of two fluid-filled concentric tubes. The external tube is rigid, with cross-sectional area equal to A_T , and the internal one, A_B , is distensible (cf figure 1). To express the fluid/structure interaction they used the following *tube law*:

$$\frac{A_A - A_{A0}}{A_T} = -D\Delta P \quad (1)$$

where $\Delta P = P_B - P_A$ is the transmural pressure, D being a distensibility parameter given by the following expressions:

$$D = \frac{1}{A_T} \left. \frac{dA_B}{d\Delta P} \right|_{\Delta P=0} = -\frac{1}{A_T} \left. \frac{dA_A}{d\Delta P} \right|_{\Delta P=0} \quad (2)$$

where the suffix 0 denotes values when $\Delta P = 0$ and $A_T = A_A + A_B$ remains fixed.

Using continuity and linear momentum equations for each component of their configuration, they obtained four coupled equations in terms of the cross sections A_A and A_B , the mean velocities U_A and U_B , the internal pressures P_A and P_B , and the cross-sectional area ratio $\alpha = A_A/A_T = 1 - A_B/A_T$.

They then considered the case of small amplitude waves propagating into an initially undisturbed state where the fluid is stationary. This led to a linear version in the form of a classic wave equation :

$$\frac{\partial^2}{\partial x^2} (\Delta p) - \rho D \left(\frac{1}{\alpha} + \frac{1}{1-\alpha} \right) \frac{\partial^2}{\partial t^2} (\Delta p) = 0 \quad (3)$$

and by identification to the following wave speed:

$$c = \sqrt{\alpha(1-\alpha)/(\rho D)} \quad (4)$$

At this level BCL considers that D is *constant* essentially for "mathematical consistency" and defines the product $c\sqrt{\rho D}$ as a *non-dimensional wave velocity*. Their plot of this *velocity* as a function of α is shown in figure 2.

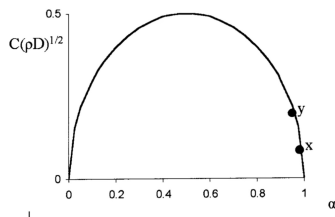


Fig. 2 The variation of dimensionless wave speed with area ratio. Note that the wave speed doubles in value when the area ratio changes from α_x to α_y , thereby illustrating the sensitivity of wave speed to relatively small displacements of the flexible tube wall.

Figure 2: Wave speed from BCL [2].

They also paid special attention to the asymptotic cases $\alpha \rightarrow 0$ and $\alpha \rightarrow 1$, where their model predicts shock waves. Therefore, they conjectured an *Elastic Jump Hypothesis* (EJH) as a possible phenomenon leading to Syringomyelia (syrinx formation within spinal cord). We will show that their finding is mathematically and physically impossible.

First their definition of distensibility does not characterize the specific rheology and dynamics of the internal distensible tube irrespective of it being confined or not in a rigid tube. It is clear that D includes the shape factor α ,

since:

$$\frac{1}{A_T} \frac{dA_B}{d\Delta P} = \frac{1}{A_B} \frac{dA_B}{d\Delta P} \frac{A_B}{A_T} = D_B(1-\alpha) \quad (5)$$

where $D_B = \frac{1}{A_B} \frac{dA_B}{d\Delta P}$ is the single distensibility of the internal tube A_B , and is constant when α varies. If one realise that D already incorporates α whatever Δp is (zero or not) then their "classic wave equation" will have a non dimensional wave speed $c = \sqrt{\alpha/(\rho D_B)}$ (cf figure 3). As we can see, the sign of $\frac{dc}{d\alpha}$ does not change, and is much weaker when α tends to 1 as predicted by their model. Two considerations that led them to postulate their EJH. Thus BCL have normalised the wave speed with a α dependent parameter while they assumed it as a *constant*.

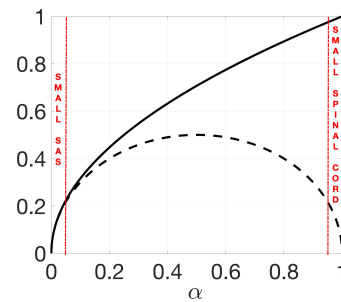


Figure 3: Corrected dimensionless wave speed (straight line) compared to BCL result (dashed line).

D is for them a *known property of the system*, while it should be the result of the interaction between the different compartments of the system. And therefore it should be one of the main objectives of the modelling. This cast serious doubts in their findings and an EJH. Secondly, the limit cases $\alpha \rightarrow 0$ and $\alpha \rightarrow 1$, where the variation of c is high, physically correspond respectively to very small subarachnoid space (SAS) and very small spinal cord. Those 2 cases are not relevant physiologically [6].

3 1D coaxial model with 2 compliant tubes

It can be shown that the inviscid compliant coaxial linear 1D model (see Figure 4) can be expressed as follows:

$$\frac{\partial \mathbf{q}}{\partial t} = M \frac{\partial \mathbf{q}}{\partial z} \quad (6)$$

where $\mathbf{q} = (A_1, A_2, U_1, U_2)^T$ is a vector of the 4 variables of the system, and M a matrix 4×4 :

$$M = \begin{pmatrix} 0 & -A_{10} & 0 & 0 \\ -\frac{c_1^2}{A_{10}} & 0 & -\frac{c_2^2}{A_{20}} & 0 \\ 0 & -A_{10} & 0 & -(A_{20} - A_{10}) \\ 0 & 0 & -\frac{c_2^2}{A_{20}} & 0 \end{pmatrix}$$

whose eigenvalues represent the wave speeds and verify:

$$c_{S,F}^2 = \frac{c_1^2 + c_2^2}{2} \mp \sqrt{\frac{c_1^2 + c_2^2}{2} - c_1^2 c_2^2 (1 - \lambda^2)} \quad (7)$$

where c_1 and c_2 are the wave speeds in each tube taken separately, and $\lambda = \sqrt{\frac{A_{10}}{A_{20}}}$ is a parameter defining the angular confinement at rest.

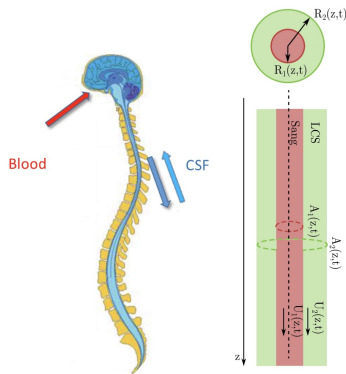


Figure 4: Spinal cavity configuration and our 1D annular model.

We therefore have 2 kinds of wave, slow ones of velocity c_S , and fast ones of velocity c_F . The structure of those 2 waves are determined by the eigenvectors, \hat{q} , of M . We can easily show that there is a relationship between the 2 cross-section areas components, \hat{A}_1 and \hat{A}_2 , through the eigenvectors:

$$\hat{A}_1 = \left[\frac{c_1^2 - c_2^2}{2c_1^2} \mp \sqrt{\left(\frac{c_1^2 + c_2^2}{2c_1^2}\right)^2 - \frac{(1 - \lambda^2)c_2^2}{c_1^2}} \right] \hat{A}_2$$

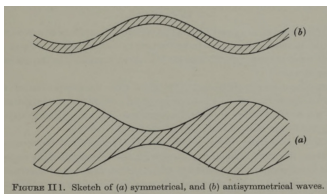


Figure 5: Symmetric and antisymmetric waves in thin sheets of fluid from G.I. Taylor [5].

When $c_1 = c_2$ (i.e. identical wall elastances) this relation becomes $\hat{A}_1 = -\lambda \hat{A}_2$ for slow waves, and $\hat{A}_1 = \lambda \hat{A}_2$ for fast waves. The cross-sectional area perturbation is therefore symmetric for slow waves (\hat{A}_1 and \hat{A}_2 of opposite sign), and antisymmetric for fast waves (\hat{A}_1 and \hat{A}_2 of same sign). The Figure 6 illustrates, in this simple case, the shapes of these 2 waves as they propagate after an initial section perturbation in the central tube at the middle of the coaxial configuration. These waves are similar in shape to those studied by G.I. Taylor [5] in thin sheets of fluids (see figure 5). This is not surprising since a sheet of

fluid has two free surface boundaries just as the coaxial configuration has 2 compliant walls as boundaries. This expand the analogy of shallow water with single compliant tubes.

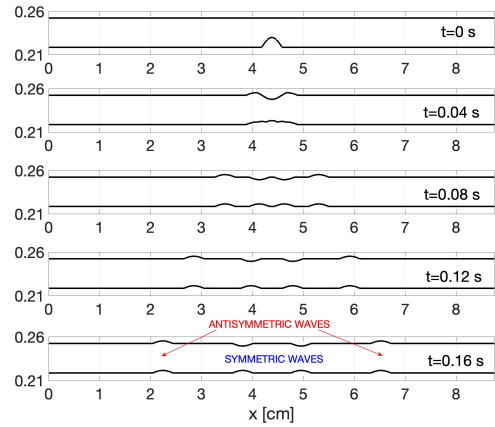


Figure 6: Symmetric (slow) and antisymmetric (fast) wave propagation in the annular cavity.

References

- [1] K. Berkouk, P. W. Carpenter and A. D. Lucey, *Pressure Wave Propagation in Fluid-Filled Co-Axial Elastic Tubes Part 1: Basic Theory*, J Biomech Eng. 125(6) (2003) 852-6.
- [2] P. W. Carpenter, K. Berkouk and A. D. Lucey, *Pressure Wave Propagation in Fluid-Filled Co-Axial Elastic Tubes Part 2: Mechanisms for the Pathogenesis of Syringomyelia*, J Biomech Eng. 125(6) (2003) 857-63.
- [3] C.D. Bertram, A.R. Brodbelt, and M.A. Stoodley, *The origins of syringomyelia: numerical models of fluid/structure interactions in the spinal cord*, Journal of Biomechanical Engineering 127 (2005) 1099-1109.
- [4] Elliott, N.S.J., Lockerby, D.A., Brodbelt, A.R., *The pathogenesis of syringomyelia: a re-evaluation of the elastic-jump hypothesis*, Journal of Biomechanical Engineering 131 (2009).
- [5] G.I. Taylor, *The dynamics of thin sheets of fluid II. Waves on fluid sheets*, Proc. Soc. A 253 (1959) 296-312.
- [6] P. Cathalifaud, M. Maher and M. Zagzoule, *A one-dimensional model of wave propagation within the co-axial viscous fluid filled spinal cavity*, 3rd International CSF Dynamics Symposium, 9-10 July 2015 (Amiens, France).

Index des Auteurs-Tome II

A	
ABABOU R.	8
ABDELBAKI A.	212, 215, 218, 221
ABDENNADHER A.	172
ABOURIDA B.	133
ACHAK N.	33
AGOUNOUN R.	178, 233
AGOUZOUL M.	51
AGRISS A.	51
AIT FARIA M.	190
AIT HRA A.	24
AITCHFI Z.	160
AKARNI H.	255
ALCORTA I.	39
ALLOUHI A.	242
ALLOUHI H.	242
AMAHMID A.	118, 121, 154
AMATOUL F.Z.	209
ANGEL PARRON VERRA M.	258
ANISS S.	5, 57, 69, 72, 75, 81, 84, 96, 102, 105
ARAUD Z.	139
ARROUB I.	136
ASKOUR O.	42
ASSELMAN A.	139
ASSOUL M.	57, 69, 102
ATLASSI K.	15
B	
BAHLAOUI A.	136
BAHRAR B.	18, 30, 33
BANDADI Y.	90, 224
BELARCHE L.	133
BELHOUIDEG S.	136
BELOUAGGADIA N.	127, 148, 151
BEN HADID H.	87, 206
BEN NEJMA F.	139
BENDAOU O.	139
BERASATEGI J.	36, 39
BERGEON A.	172
BORJI S.	252
BOTTON V.	87, 206
BOU-ALI M. M.	36, 39, 166, 169
BOUCHAALA A.	157
BOUCHGL J.	72
BOUFKRI M.	154
BOUKENDIL M.	212, 215, 218, 221
BOUSSHINE L.	252
BOUSSOUF A.	236
BOUZID K.	133
BRAIKAT B.	42, 78
C	
CATHALIFAUD P.	261
CHARAF A.	200, 203
CHARQUI Z.	212, 215, 218, 221
CHARRAOU A.	187
CHARRIER-MOJTABI M.-C.	172
CHOUKRI S.	48
D	
CORDIER L.	66
DAGOIS-BOHY S.	87
DAHANI Y.	118, 121
DAIZ A.	136
DAMIL N.	42
DARIF M.	130, 245
DAYA A.	105
DLIMI M.	203, 233
E	
ECHCHADLI M.	5, 57
ECHCHELH. A	11
EL AARABI L.	255
EL ALAMI M.	187
EL GHANI N.	206
EL HADY Z.	181
EI HAMMA M.	227
EL HARFI H.	184
EL HOU M.	197
EL JAOUAHIRY A.	57
EL KHLIFI M.	15, 21, 45, 54, 60, 63, 66, 99
EL MANSOURI A.	154
EL MASKAOUI Z.	252
EL MOUTAOUAKIL L.	212, 215, 218, 221
EL OMARI A.	66
ELALAOUI S.	163
F	
FADIL R.	78
FARAJI M.	109, 112
FEDDAOUI M.	200, 236
FERTAH S.D.	178
FETIAN H.	124
FILAH I.	121
G	
GÓMEZ A.	36, 39
GOMEZ-ARENAS T.	39
GUEMIMI C.	258
GUERAOUI K.	30, 33, 227
H	
HADDOUT Y.	245
HAKIM M.	48
HAMZA H.	181, 190, 194
HASNAOUI M.	118, 121, 136, 154, 209
HASNAOUI S.	118, 209
HAYANI CHOUJAA M.	81, 84, 105
HDIDOU F.	24
HENRY D.	87, 206
HIDKI R.	212, 215, 218, 221
HIFDI A.	75
HISSOUF M.	200
I	
IDMOUSSA R.	142, 194
IGOZAL M.	160
IKEN O.	178
IMGHORE O.	148
J	
JABRALI A.	249
JAMIL A.	242
JAOUHARI M.	5
JENNAH S.	127
K	
KADDIRI M.	145, 175, 184
KADIRI I.	178
KAHOT Z.	2
KARDOUDI M.	45
KHALED BENKAHLA Y.	233
KHALID NACIRI J.	81, 84, 112, 246
KHATYR R.	112, 246
KISSI B.	255
KOURTA A.	21
L	
LABSI N.	233
LACHHEB A.	200, 221
LAHJOMRI J.	127, 139, 160, 178, 187, 191, 242
LAHLALI A.	249
LAHLAOUI M.L.	121
LAHLOU N.	90, 93
LAHLOU S.	236
LAMINE M.	75
LAMSAADI M.	142, 172, 181
LBIBB R.	124, 145
M	
MADI M.	27
MANSOUR A.	115, 118
MANSOURI K.	148
MARCELIN S.	203
MEHDARI A.	51
MERROUN O.	154
MESMOUDI S.	42, 78
MIALDUN A.	163
MIKDAM A.	154
MILLET S.	87
MIRALLES S.	203
MOJTABI A.	169
MORDANE S.	24, 194, 252
MOUAKKIR L.	194
MOUDA M.	54
MOUNIR BOU-ALI M.	166
MOURID A.	184
N	
NABHANI M.	15, 45, 54, 60, 96
NAJIM M.	197

NGADI Z.	121	SANTAMARIA C.	166
NIKCHI M.	139	SBAI K.	175
O		SBAIBI A.	221
OUAZZANI TOUHAMI M.	81, 84, 93	SEHAQUI R.	236
OUBARRA A.	127, 139, 160, 178, 187, 191, 242	SELLAMI K.	233
OUBENMOH S.	184	SERRAR A.	21
OU DAOUI K.	106	SGHIOURI EL IDRISSI H.	227
OUHAIBI S.	145	SHAIMI M.	112
OUHIMMOU S.	227	SHEVTSOVA V.	163
OUZI M.	18	SIADI A.	130
R		SIOUD K.	169
RACHID H.	90, 93, 224	SOBHI S.	60
RACHIK A.	99	SOUHAR K.	27
RAFIKI A.	27	T	
RAHMOUNE M.	175, 184, 197, 221, 230	TAGHAVI S.M.	27
RAJI A.	133	TAIBI M.	224, 227
RAMMANE M.	42, 78	TAMANI M.	18
REDOINE F.	148	TAOUZER C.	69
RIAH I M.	81, 84, 102	TAZI R.	11
RKIBI W.	33	TER-OVANESSION B.	203
RKIBI O.	30	TIZAKAST Y.	142, 172
RTIBI A.	224	TOUAZIT A.	157
S		TOUIL M.	200, 221, 230
SAADANI R.	175, 184, 197, 200, 221, 230	TRI A.	42, 78
SAHLAOUI Z.	24	TRIBAK F.	136
SAKIM A.	63, 154	Y	
SAKINE M.	102	YOUNSI Z.	145
SAMAOUALI A.	227	Z	
SAMRI H.	18	ZAGZOULE M.	258
SANJUAN A.	166	ZNIBER K.	127, 139, 160, 187
		ZRIKEM Z.	209, 212, 215, 218

Avec le soutien de:



Le Groupement
d'Assurances
du Supérieur

