A review of convective heat transfer in cavity-channel assemblies

Ahmed Kadhim Hussein[1] , Amaal Abdul Razaq Abdul Hussein [2], Awatef Abidi [3], Muneer A. Ismael[*4,5], Ahmed B. Mahdi [6], Bashar S. Bashar[7], Farhan Lafta Rashid[8], Raad Z. Homod[9], ObaiYounis[10,11], Lioua Kolsi [12,13] and Ali J. Chamkha[14]

[1] College of Engineering -Mechanical Engineering Department - University of Babylon - Babylon City – Hilla – Iraq.

E-mail: ahmedkadhim7474@gmail.com

[2] Babylon Governorate - Directorate of Babylon Sewerage- Hilla - Iraq

E-mail: amal.hussein@student.uobabylon.edu.iq

[3] Physics Department, College of Sciences Abha, King Khalid University, Saudi Arabia.

Email: amabedei@kku.edu.sa

[4] Mechanical Engineering, Department, Engineering College, University of Basrah, Basrah 61004–Iraq

[5] College of Engineering, University of Warith Al-Anbiyaa, Karbala, Iraq

Email: muneer.ismael@uobasrah.edu.iq

[6] Anesthesia Techniques Department, Al-Mustaqbal University College, Babylon, Iraq

ahmed.baseem@mustaqbal-college.edu.iq

[7]Al-Nisour University College – Baghdad - Iraq

bashar.s.eng@nuc.edu.iq

[8] Petroleum Engineering Department, College of Engineering, University of Kerbala, Karbala 56001, Iraq

farhan.lefta@uokerbala.edu.iq

[9] Department of Oil and Gas Engineering, Basrah University for Oil and Gas, Iraq

raadahmood@yahoo.com

[10] Department of Mechanical Engineering, college of engineering in Wadi Addwasir, Prince Sattam Bin Abdulaziz university, Al-Kharj 11942 , Saudi Arabia.

[11] Department of Mechanical Engineering, Faculty of Engineering, University of Khartoum, Khartoum, Sudan.

Email: oubeytaha@hotmail.com

[12] Department of Mechanical Engineering, College of Engineering, University of Ha'il, Ha'il City 2440, Saudi Arabia.

E-mail: lioua_enim@yahoo.fr

[13] Laboratory of Metrology and Energy Systems, Department of Energy Engineering, University of Monastir, Monastir 5000, Tunisia

[14] Faculty of Engineering, Kuwait College of Science and Technology, Doha District, 35004 Kuwait

Email: a.chamkha@kcst.edu.kw

[*]Corresponding author, mobile: Mobile:

Abstract: Convective heat transfer induced in open cavities is one of the main pillars that the topic of energy saving relies on. This article reviews and categorizes the results of researches on mixed convection in open cavity connected with a channel and highlights the gap that should be filled in future works. It is found that the best heat and mass transfer is attained when the source of heat and/or species is located at a vertical wall of the cavity where it opposites the flow direction. The review has revealed that the experimental studies are relatively scare where it 10% of the total reviewed studies, while those dealing with nanofluids and porous media are 9% for each. It is found that the

process of injection or aspiration of the flow have received very few studies despite its promised improvement of the heat and mass transfer. Furthermore, few researches have studied the contamination removing from the cavity.

***Keywords**: Mixed convection; Open cavity; Channel; Porous medium; Partially layered;*

Nomenclature			
A	velocity amplitude ratio	Ri	Richardson number
AR	aspect ratio (width/height)	Sc	Schmidt number
Bi	Biot number	Sh	Sherwood number
Ca	Cauchy number	Str	Strouhal number
Da	Darcy number	***Abbreviations***	
Gr	Grashof number	MP	Multi-Processing
Ha	Hartman number	TR	Time resolved
J	Joule heating parameter	PIV	Particle image velocimetry
K	conductivity ratio	POD	Proper Orthogonal Decomposition
N	buoyancy ratio	PFV	Parallel finite volume
n	power low index	FEM	Finite element method
Nu	average Nusselt number	FVM	Finite volume method
Le	Lewis number	FDM	Finite difference method
Pr	Prandtl number	***Greek Symbols***	
q''	heat flux	ϕ	Nanoparticles volume fraction
Re	Reynolds number	γ	Inclination angle of the assembly
Rer	ratio of Reynolds numbers		

1. Introduction

Convective heat transfer induced in open cavities is one of the main pillars that the topic of energy saving relies on. This importance stems from its wide range of technical processes such as heat exchange for cooling and heating fluid as in nuclear reactors [1], materials removing from cavities [2], controlling of fire in buildings [3], thermal management of electronic components and solar energy [4], heat exchangers [5] and food industry. The mixed convection is recalled when the natural convection is insufficient to transfer heat. Several mechanisms induce the mixed convection like driving one or more walls in contact with the fluid [6], rotating cylinder(s) immersed within the fluid [7-8] or forcing the fluid itself through a heated boundary. Mixed convection heat transfer in a channel with open cavities is reviewed in this article. Better understanding of such a phenomenon will assist in suitable thermal design and improving the heat transfer rate. Observations and conclusions of lots of numerical articles, which have been published in the last decades contribute in a better understanding of the phenomenon. On the other hand, experimental setups are more efficient and pertinent [9]. Nevertheless, several aspects remain unclear and continue to be the object of active research throughout the world.

The impacts of geometrical and physical parameters on the convective exchanges in the cavity-channel are presented. Among the most important ones, the aspect ratio, the inclination, the nature of the coolant fluid, the thermal boundary conditions and the characteristics of the fluid and of the enclosure boundaries.

2. Numerical solution and /or experimental technique approaches

Mixed convective phenomena are treated numerically and/or experimentally. The experimental approaches give trusted and stable results and not limited in a specified range such as compressible, laminar or Newtonian fluids. The numerical approach requires the solution of non-linear partial differential equation systems consistent with fluid mechanics and thermodynamics conservation principles. The numerical solution of the problem depends on the characteristics of the resulting flow. In each case, an appropriate complex methods must be used to obtain a reliable solution. The continuously improving numerical methods and computer technology facilitate deeper analysis of certain aspects of mixed convection such as artificial neural network (ANN) [10] and Lattice-Boltzmann method (LBM) [11].

3. Mixed convection in the cavity–channel assembly

Several investigations considered the mixed convection in clear, porous or multilayer cavities connected with a channel in different cases such as filling with fluids or nanofluids. The shape of the cavity may be square, rectangular, triangular, trapezoidal and wavy. The thermal and/or solutal boundary conditions also accounts the case of the mixed convection, for example if the active wall of the cavity is prescribed opposing to the flow direction in the cavity, then the case is called opposing and, contrary, the case is called assisting as shown in Fig. 1. Depending on the shape of the cavity, several configurations will be treated in the present review and will be devoted to two categories, depended on the shape of the cavity, regular and complex.

3.1 Mixed convection in the clear cavity–channel assembly

The convective exchanges in confined domains strongly depend on the cavity geometry. Accordingly, several configurations have been treated. Manca et al. [1] in 2003 studied numerically the effect of the heated wall location on the mixed convection of air in a channel with a U-shaped open cavity. One of the cavity walls was heated by a uniform heat flux, while the other walls were thermally insulated. Three different heating modes were considered (assisting, opposing, and heating from below). The effects of Richardson number Ri, Reynolds number *Re* and the ratio of the width of the inflow and outflow ports to the heat source length (H/D) were investigated. They concluded that, when *Re* and *Ri* numbers increase, the maximum temperature decreases. Also, it was found that the opposing forced flow had the best thermal performance. Fang [2], 2003, focused his study on the effect of mixed convection on transient hydrodynamic removal of contaminants from a rectangular cavity located below a horizontal channel. The cavity was subjected to a constant heat flux at its bottom wall. The results showed that the change in *Gr* had a significant effect on the flow field orientation and cleaning efficiency. Leong et al. [3], 2005, performed computation of the combined convection in a channel with a cavity heated from below. Air was pumped into the channel at a uniform velocity and temperature. They concluded that the flow field was controlled by *Re* and *Gr* and the *AR* had a significant effect on the flow field orientation. Andreozzi et al. [4], 2005, adjusted the left sidewall of the cavity at uniform heat flux, while the top adiabatic wall of the channel was considered moving at a constant velocity in two different directions. They suggested correlations of Nusselt number in terms of *Ri* and *Re*. Carozza et al. [5], 2005, explored the mixed convection in open cavity which was located under a channel.

It was found that the *Nu* presents maximum values at the ratio between the channel and cavity height *H/D* = 0.1. Brown and Lai [12], 2005, inspected the double diffusive mixed convection of air in a horizontal channel with an underneath open cavity. The bottom wall of the cavity was maintained at high temperature and concentration. Fresh air was pumped into the channel at a uniform velocity and a cold temperature and concentration. They suggested correlations for the natural, forced and the mixed convection based on the scale analysis. Buonomo et al. [13], 2008, analyzed the steady combined convection of air in an open cavities filled with a fluid-saturated porous medium. Two different geometrical configurations were considered namely (horizontal channel with a cavity at the lower wall (U-shaped cavity) and a vertical channel with a cavity at its right sidewall (C-shaped cavity). The wall of the cavity on the inflow side was heated at uniform heat flux. They concluded that the two considered configurations exhibited similar behaviors for low values of *Ri* and *Re* numbers. Also, in all cases, the maximum wall temperature value for U-shaped cavity was lower than that noted in C-shaped one. Aminossadati and Ghasemi [14], 2009, addressed an interesting numerical data regarding the mixed convection of air entered the horizontal channel integrated with an open rectangular cavity. The cavity was subjected to a discrete heat source at three different locations (left, right and bottom walls). It was found that, the heat transfer was enhanced with increasing the aspect ratio (*AR)* of the cavity. The mixed convection assisting air flow in 3D horizontal channel-square cavity assembly was considered numerically by Stiriba [15], 2008. Both the left and right side walls of the cavity were kept at an isothermal hot and cold temperatures, respectively. It was found that *Nu* increases with increasing *Ri* for all considered ranges of *Re.*

Wong and Saeid [16], 2009, considered the opposed mixed convection from a jet impingement cooling in an open cavity inside the channel filled with a porous medium. They deduced that *Nu* augments with the increase in Rayleigh number *Ra*. Rahman et al. [17], 2012, explored the mixed convection in a horizontal channel with an open rectangular cavity numerically. A heated hollow cylinder was located inside the cavity. It was found that the *Nu* at the heated surface enhances as *Ra* and thermal conductivity ratio *K* increased and it was irrespective to *Pr*. Rahman et al. [18], 2012, performed a numerical computations of the combined laminar and steady forced and natural convection in 2D channel with an open triangular heated cavity. The magnetic field and Joule heating effects were investigated also. It was found that the heat transfer increases by increasing *Re* and *Pr*. Contrarily, it was decreased by increasing *Ha* and *J*. Selimefendigil and Yurddas [19], 2012, considered the impact of the pulsating flow on the mixed convection of air inside a horizontal channel with a square cavity heated from below and a left vertical side. Their main conclusions were that the heat transfer enhancement is increased with *Str* for all considered values of *Re* and Ri = 100. Selimefendigil. [20], 2013, described numerically the two dimensional laminar mixed convection of air in a square cavity, which is linked to a horizontal channel. It was found that, at Re = 800, the *Nu* at Ri = 10 is smaller compared with that found at Ri = 5. Mehrez et al. [21], 2013, considered the entropy production along with the convective heat transfer of the nanofluids flowing in a horizontal channel with an open cavity. Heat transfer and entropy generation analysis were investigated for various types of nanoparticles (Cu, Al_2O_3, CuO and TiO_2). They concluded that the heat transfer and the entropy generation increase with the increase of *Re*, *Ri*, ϕ and varied with *AR* and nanoparticle types. Also, it was deduced that the maximum enhancement of heat transfer is found for Cu-water nanofluid. Stiriba et al. [22], 2013, discussed the 3D laminar mixed convection of air in an open cavity located underneath a channel. It was found that, at low values of *Ri* and *Re* the flow becomes steady, while the increase in *Re* and *Ri* brings the flow to be transient. Rahman et al. [23], 2014, presented computations of conjugate mixed convection in a channel with a thick walled cavity. They concluded that the heat transfer is an increasing function of *K* ratio. Also, the *Nu* was increased with increasing of *Ri* except at D = 0.5 where it was decreased with increasing of *D*. Sidik and Jahanshaloo [24], 2014, performed a study on the possibility of removing contaminants from a cavity located under a horizontal channel. They concluded that, the increase of *Gr* rises the rate of contaminant removal at the considered value of Reynolds number (*Re* = 50). A numerical study of the entropy generation in the assisting mixed convection of Cu-water nanofluid in an inclined open cavity connected to a horizontal channel was presented by Mehrez et al. [25], 2015. The results showed for various value of *Re*, *Ri*, ϕ and the inclination angle γ. They deduced that, the entropy generation is affected by the inclination angle γ and this depends on *Re* and ϕ. Sharma et al. [26], 2015 solved the mixed convection in a channel grooved by a cavity with a baffle downward towards the core of the cavity. They solved the problem using an in-house CFD code depending on the finite volume method with SIMPLE algorithm. The position of the hot wall was altered according to get opposing and assisting mixed convection. The positive role of the baffle was one of the main conclusions. Timuralp and Altaç [27], 2016, examined the fluid flow and heat transfer in a 3D cavity located at the bottom of a horizontal square duct. The effects of the *AR* of the cavity, duct height to cavity height ratio, *Ri* and *Re* were studied and discussed. They concluded that, the *Nu* increased by increasing the heights ratio between duct and cavity. Similar numerical investigation to [19] and [20] was carried out by Selimefendigil [28], 2016. He used the same geometry and the boundary condition of both the channel and cavity. The distinct aspect was that the mixed convection was pulsating flow for varies range of *Ri* at *Re* = 50. It was found that the heat transfer enhances with an increase in the velocity amplitude. Burgos et al. [29], 2016, studied transient steady laminar mixed convection of air in a square cavity located at the bottom of a channel. The results computed by Lattice Boltzmann Method (LBM) and compared with ANSYS-FLUENT for validation. They deduced that at $Ri \geq 1$, a clear enhancement in the heat transfer rate was noticed. Also, it was found that the flow is unsteady for $Re \geq 500$ and $Ri = 10$, whereas the buoyancy force becomes negligible with $Ri \leq 0.1$ at any value of *Re*. A numerical study of the generation of entropy was presented by Zamzari et al. [30], 2017, where the mixed convection in an open cavity connected to a horizontal channel was inspected. It was found that the heat transfer decreases by the increase of the cavity aspect ratio. Hussain et al. [31], 2017, investigated the entropy generation in unsteady MHD mixed convection inside a horizontal channel containing an adiabatic square obstacle with an open cavity heated from below. The cavity filled with Al_2O_3-Cu-water hybrid nanofluid. Three different vertical locations of the obstacle were considered. They deduced that the increase in *Ri*, *Re* and ϕ boosts the rate of heat transfer and entropy generation. Hussain et al. [32], 2017, considered the entropy generation of the mixed convection in an inclined channel with a cavity filled with alumina-water nanofluid saturated in a porous medium. It was observed, that the heat transfer is enhanced by increasing *Da*, *Re* and the porosity. Also it was found that *Be* increases with increasing *Re* and porosity, while it declines with increasing *Ri*. Yasin et al. [33], 2018, conducted a numerical and experimental research about the effect of a vertical unheated baffle on the mixed convection of air in an open square cavity attached to a square duct. It was observed that the maximum Nusselt number has occurred at the highest length of the baffle. Sivasankaran et al. [34], 2018, portrayed the effect of an adiabatic vertical baffle on the mixed convection in an open. Two different modes of heating (i.e. linearly heating and sinusoidal heating) were imposed. It was found that the sinusoidal heating provides more heat transfer rate than linearly heating. Also it was shown that by increasing the baffle length, the averaged energy transport inside the channel-cavity assembly is increased. Sabbar et al. [35], 2018, analyzed numerically the transient mixed convection in a cavity rectangular–channel assembly due to the interaction between the fluid flow and the elastic walls. They concluded that the heat transfer rate is enhanced by about 17% by the existence of the elastic wall(s) compared with rigid walls of the cavity. The numerical investigation of 2D transient mixed convection of air in an open cavity located at the bottom of a channel with different aspect ratios was performed by Carozza [36], 2018. It was found that the assisting configuration was more efficient compared with the opposing one. Abd Al-Hassan and Ismael [37], 2019, included the unsteady double diffusive mixed convection in an open cavity located in the bottom of a horizontal channel. One of the vertical sides of the cavity was kept at a constant temperature and concentration, while all other walls were considered impermeable and adiabatic. A porous layer was included in the lower part of the cavity. It was found that for the assisting case *Nu* and Sherwood number *Sh* decreases with increasing H_p, while their behavior were different with H_p for the opposing case. Contreras et al. [38], 2019, carried out experiments of 3D opposing combined convection in an open cubical cavity located at the mid-section of a vertical square channel. A stereoscopic time-resolved particle image velocimetry (TR-PIV) was used to assess the thermal effects on the flow. They concluded that the increase in *Re* lowers the critical Richardson number above which the flow was no longer encapsulated. Ahmadi and Farsani [39], 2020, involved the non-Newtonian fluid in a 2D laminar mixed convection in a cavity linked with a horizontal channel. A non-Newtonian two phase fluid flow was entered the channel with the cavity. The polymer solution and water were selected as two-phase flow. The results revealed that the changes in the velocity, pressure and volume fraction of fluids inside the channel and cavity were more sensible to variation in the *Re* instead of changing the rheological index *n*.

Table 1 summarizes some of the regular cavities opened to a rectangular channel. The table includes short portray of the tools used in the investigation, the overall geometry, ranges of the studied parameter, fluid type and the main conclusions.

3.2 Mixed convection in complex cavity–channel assembly

For the sake of promoting the convective heat exchange, sophisticated geometries are studied, these including doubling the cavities, driving one or more of the cavity walls, inserting an oscillating baffles, injecting a secondary flow into the cavity and using discrete heating. As an attempt to improve the heat transfer of a grooved channel, Biswas et al. [40] 2015 suggested a novel strategy that is dividing the total flow into two flows. The first is the main flow which is flow regularly through the channel while the other is injected from an opening made at the base of the groove. With the aid of an in-house

CFD FVM code, they addressed a pronounced augmentation by the injection of the fluid reaches to 141%. After the results achieved by the injection strategy, several geometries were investigated considering natural aspiration (venting fluid by no power) such as; aspiration with the aid of the moving walls [41]; aspiring a cavity involving porous layers [42– 43] or protruding a heating element [44] and different configurations of surface heating elements [45]. García et al. [46], 2019, performed a numerical 2D transient mixed convection in an inclined rectangular channel containing two facing identical open cavities. They concluded that the heat transfer rate is increased by increasing the *Re* for fixed values of the *Ri*. The transient laminar opposing flow mixed convection in an inclined square water-filled channel, which included two symmetric open cubic cavities subjected to a constant wall heat flux was examined experimentally by Cardenas et al. [47], 2019. Particle image velocimetry (PIV) was used for the measurements. It was found that the overall heat transfer was a nonlinear function of the channel inclination angle. Laouira et al. [48], 2020, presented a numerical simulation about the effects of heat source length on the combined convection in a channel with an open trapezoidal cavity. They concluded that *Nu* was increased with increasing the length of the heat source. Yaseen and Ismael [49], 2020, considered the role of the non-Newtonian FSI in mixed convection in a trapezoidal cavity opened to a horizontal channel. An elastic baffle was fixed at the upper wall of the channel and downwards to the open cavity. They concluded that for low *Ri*, the shear-thickening fluid manifests higher Nusselt number while for high *Ri* number, the shear-thinning fluid has the higher values. Yaseen and Ismael [50], 2020, extended their work ([49]) by studying various locations of the baffle (left, center and right) with respect to the cavity center. They found that the location of the elastic baffle near the channel inlet gives a maximum *Nu* number. Al-Farhany et al. [51], 2020, showed the effect of discrete heat source on the combined convection inside a channel with an open complex cavity. A magnetic field was applied to the assembly from its right side. They deduced that *Nu* was increased with the increase of *Ri* and decreases with *Ha*. Ismael et al. [52], 2020, performed a study of the combined convection of air in a horizontal channel with a heated trapezoidal cavity of moving walls. Four different cases were studied depending on the movement of the cavity sidewalls. They studied the effects of *Ri* and Reynolds number ratio Re_r on the flow and thermal fields. They concluded that, the *Nu* increases with the increase of the *Ri* and Re_r. Mebarek-Oudina et al. [53], 2022 developed the study of Ismael et al [52] by considering two discrete heater rather than one on the base of the trapezoidal cavity. The notation of the four cases were also adopted exactly. The sole result they found are that the role of Heater 2 is more pronounced in the case of single moving wall, while Heater 1 gests active when two sidewalls are in a movement, Case II. Sáchica, et al. [54], 2020, dealt with the transient MHD mixed convection and entropy generation of Al_2O_3 –water nanofluid in a rectangular vertical channel with two facing identical open cubic cavities with discrete heating. Their conclusions were that the increase in the nanoparticle volume fraction ϕ, the entropy generation decreases with the magnetic field for all ranges of Ri and Ha. Based on the feasibility of implementing flexible baffles in augmenting the heat transfer, Yaseen and Ismael [55], 2023, focused on the stress and the deflection in a baffle used in promoting the heat transfer in a trapezoidal cavity opened to a rectangular channel considering a power law fluid flow. Their results highlighted the decrease of von Misses stresses with *Re* for fluids of power low index *n* less than unity, while for $n > 1$, von

Misses stresses increase with *Re*. In addition, they added that the flexible baffle exhibits lower stresses than rigid baffle. As an extension to their work, Abd-Al-Hassan and Ismael [56], 2022, promoted their previous work [37] by inclining the porous layer. They indicated that the heat and mass transfers of the inclined porous layer could be 30% and 32%, respectively greater than the horizontal layer.

Some of the reviewed works in this section are summarized in Table 2.

3.3 Numerical methods adopted in mixed convection

The non-linearity of the equations that govern the mixed convection becomes more influential due to the convective terms in the momentum and energy conservation equations, (and the mass conservation equation if the advection is considered). Therefore, attention should be focused on the stability of the numerical solution of such equations. The review has discovered more than one numerical strategies are available with mixed convection. The most strategy used is the finite volume method (FVM) where about 56% of the numerical studies have followed the FVM. This is because such a method is built on the basis of integrating the conservation equations. Therefore, the convergence of the solution is mostly guaranteed while the pressure gradient term is successfully corrected using the SIMPLE algorithm. The finite element method is also followed but because its complexity and the required storage capacity needed, the studies using FEM is about 29% of the total numerical studies. The FEM is recalled when there is an interaction between the fluid and the structure. On the other hand, the finite difference method (FDM) and the Lattice Boltzmann method (LBM) are rarely used in channel-cavity assembly because these methods fail in representing the complex boundaries. Proper Orthogonal

Decomposition (POD) method is one of the rarely used method. The percentage of studies dealing with FDM, LBM and POD are 5% for each. Figure 2 portrays the percentage of papers that uses each numerical method.

3.4 Experimental techniques used in mixed convection

The experimental studies of mixed convection in a cavity-channel assembly are scare compared with the numerical papers. This is mainly because of the experimental investigations deals with real cases rather than assumptions. For example, to preserve a steady flow condition, the test section should be provided by liquid from a constant-head tank which in turn is fed by fluid from another large tank [38]. This cycle is attained by a suitable pump. If the fluid is gas, care must be taken in designing the ducting to avoid the vortices created when air enters to sudden duct. Moreover, flow straightener are necessary prior to the test section to bring uniform steady flow. Another example of the real complication of the experimental work is the effort of maintaining a constant surface temperature, the case which need extra heat exchanger and a circulating pump and a heat transfer fluid to maintain constant wall temperature. Furthermore, in numerical simulation, the results are displayed by several commercial software with a wide range of facilities and presentation, whereas the experimental results are collected using an expensive instrumentations like the PIV.

4. Conclusions

The current paper reviews and categorizes some results of researches conducted on mixed convection in an open cavities connected with rectangular channel. This compilation serves a broad spectrum of scientific and engineering fields where the knowledge of mixed convection in enclosures is applied. The reviewed works have been classified into two main categories, regular and complex geometries. With “complex geometry” we put the double faced open cavity, baffled cavities, wall(s) driven cavities and partially layered open cavities.

4.1 Summary of the findings

It is found that the best geometry that gives maximum heat and mass transfer is that when the source of heat and species is located at a vertical wall of the cavity where it opposites the direction of the flow in the channel. The key parameter in such a geometry is the Reynolds number which notably augments the mixed convection in all conditions. It is found that as shown in Fig. 3, the complex geometries comprises 20% of the reviewed papers. The review has revealed that (as shown in Fig. 4) the experimental studies are relatively scare where it comprises only 9% of the total reviewed studies. This can be returned to the intricacy of the experimental work. On the other hand, published studies dealing with nanofluids and porous media filling cavities are 9% for each. The nanofluids studies, however, are also few compared with the overall reviewed papers. Eventually, Fig. 5 portrays the number of published papers arranged chronologically. The figure emerges that the topic under study has been started two decays ago. In the second decay, the works are increased notably. However, the maximum number of published papers is recorded within 2019 to 2020.

4.2 Future topics

Although the topic of mixed convection in channel-cavity assembly is interesting and widely investigated, the following topics are not found in the open literature, thus it is suitable to be addressed in the future works.

(i) Studying the contamination removing from the cavity i.e. studying the equation of mass transfer.

(ii) The process of injection or aspiration of the flow from the bottom or sides of the cavity, which gives a promised improvement of heat and mass transfer.

Acknowledgments

The authors extend their appreciation to the Deanship of Scientific Research at King Khalid University, Abha, Saudi Arabia for funding this work through the Project of Review Article under grant number (110/ 43)

Conflict of interest

The authors declare that they have no conflict of interest.

Authors Contribution

Conceptualization: [Ahmed Kadhim Hussein]; Methodology: [Muneer A. Ismael]; Formal analysis and investigation: [Amaal Abdul Razaq Abdul Hussein]; Writing - original draft preparation: [Amaal Abdul Razaq Abdul Hussein]; Writing - review and editing: [Raad Z. Homod], [Obai Younis], [Farhan Lafta Rashid]; Funding acquisition: [Awatef Abidi]; Resources: [Ahmed B. Mahdi], [Bashar S. Bashar]; Supervision: [Lioua Kolsi]; Supervision and revision: [Ali J. Chamkha].

References

[1] Manca, O., Nardini, S., Khanafer, K., et al. "Effect of heated wall position on mixed convection in a channel with an open cavity", *Numer. Heat Transf.; A: Appl*., **43**(3), pp. 259-282 (2003).

[2] Fang, L-C. "Effect of mixed convection on transient hydrodynamic removal of a contaminant from a cavity", *Int. J. Heat Mass Transfer*, **46**(11), pp. 2039–2049 (2003).

[3] Leong, J.C., Brown, N.M., and Lai, F.C. "Mixed convection from an open cavity in a horizontal channel", *Int. Commun. Heat Mass Transfer*, **32**(5), pp. 583-592 (2005).

[4] Andreozzi, A., Manca, O., Bianco, N., et al. "Mixed convection in air in an open ended cavity with a moving plate parallel to the cavity open surface", *ASME Summer Heat Transfer Conference*, San Francisco, California USA, 17-22 July, HT2005-72511: pp. 1-13 (2005).

[5] Carozza, A., Manca, O., Nardini, S., et al. "Numerical investigation of mixed convection in open cavities with vertical isothermal walls", XXIII ITU National Congress on Heat Transmission, Parma, 20-22 June, Conference Paper: pp. 1-6 (2005).

[6] Al-Rashed, A.A., Shahsavar, A., Akbari, M., et al. "Finite volume simulation of mixed convection in an inclined lid-driven cavity filled with nanofluids: effects of a hot elliptical centric cylinder, cavity angle and volume fraction of nanoparticles", *Phys. A: Stat. Mech. Appl*. **527,** p. 121122 (2019).

[7] Li, Z., Barnoon, P., Toghraie, D., Dehkordi, R.B., et al. "Mixed convection of non-Newtonian nanofluid in an H-shaped cavity with cooler and heater cylinders filled by a

porous material: Two phase approach", *Adv. Powder Technol.*, **30**(11), pp. 2666-2685 (2019).

[8] Barnoon, P., Toghraie, D., Dehkordi, R.B., et al. " MHD mixed convection and entropy generation in a lid-driven cavity with rotating cylinders filled by a nanofluid using two phase mixture model", *J. Magn. Magn. Mater.*, **483**, pp. 224-248 (2019).

[9] Biswas, N., Roy, P.C., Manna, N.K., et al. "Experimental studies of flow through radial channels using PIV technique", *J. Vi.*, **17**(3), pp. 221-233 (2014).

[10] Faridzadeh, M., Toghraie, D.S., and Niroomand, A. "Analysis of laminar mixed convection in an inclined square lid-driven cavity with a nanofluid by using an artificial neural network", *Heat Transf. Res.*, **45**(4), pp. 361-390 (2014).

[11] Balootaki, A.A., Karimipour, A., and Toghraie, D. "Nano scale lattice Boltzmann method to simulate the mixed convection heat transfer of air in a lid-driven cavity with an endothermic obstacle inside", *Phys. A: Stat. Mech. Appl*., **508**, pp. 681-701 (2018).

[12] Brown, N.M. and Lai, F.C. "Correlations for combined heat and mass transfer from an open cavity in a horizontal channel", *Int. Commun. Heat Mass Transfer,* **32**(8)**,** pp. 1000 –1008 (2005).

[13] Buonomo, B., Foglia, G., Manca, O., et al. "Numerical study on mixed convection in a channel with an open cavity filled with porous media", *In 5th European Thermal-Sciences Conference*, pp. 1-8 (2008).

[14] Aminossadati, S.M. and Ghasemi, B. "A numerical study of mixed convection in a horizontal channel with a discrete heat source in an open cavity", *Eur. J. Mech. B Fluids*, **28**(4), pp. 590-598 (2009).

[15] Stiriba, Y. "Analysis of the flow and heat transfer characteristics for assisting incompressible laminar flow past an open cavity", *Int. Commun. Heat Mass Transfer*, **35**(8), pp. 901-907 (2008).

[16] Wong, K.C. and Saeid, N.H. "Numerical study of mixed convection on jet impingement cooling in an open cavity filled with porous medium", *Int. Commun. Heat Mass Transfer*, **36**(2), pp. 155-160 (2009).

[17] Rahman, M.M., Parvin, S., Rahim, N.A., et al. "Simulation of mixed convection heat transfer in a horizontal channel with an open cavity containing a heated hollow cylinder" *Heat Transfer—Asian Research*, **41**(4), pp. 339-353 (2012).

[18] Rahman, M.M., Öztop, H.F., Ahsan, A., et al. "MHD Mixed convection in a channel with a triangular cavity", *Numer. Heat Transf.; A: Appl.*, **61**(4), pp. 268-282 (2012).

[19] Selimefendigil, F. and Yurddas, A. "Numerical analysis of mixed convection heat transfer in pulsating flow for a horizontal channel with a cavity heated from vertical side and below", *Heat Transf. Res.*, **43**(6), pp. 509–525 (2012).

[20] Selimefendigil, F. "Numerical analysis and pod based interpolation of mixed convection heat transfer in horizontal channel with cavity heated from below", *Eng. Appl. Comput. Fluid Mech.*, **7**(2), pp. 261-271 (2013).

[21] Mehrez, Z., Bouterra, M., El Cafsi, A., et al. "Heat transfer and entropy generation analysis of nanofluids flow in an open cavity" *Comput. Fluids*, **88**, pp. 363-73 (2013).

[22] Stiriba, Y.F., Ferré, J.A., and Grau, F.X. "Heat transfer and fluid flow characteristics of laminar flow past an open cavity with heating from below" *Int. Commun. Heat Mass Transfer*, **43**, pp. 8-15 (2013).

[23] Rahman, M.M., Oztop, H., Mekhilef, S., et al. "A finite element analysis on combined convection and conduction in a channel with a thick walled cavity", *Int. J. Numer. Methods Heat Fluid Flow*, **24**(8), pp. 1888-1905 (2014).

[24] Sidik, N.A. and Jahanshaloo, L. "The use of lattice Boltzmann numerical scheme for contaminant removal from a heated cavity in horizontal channel", *CFD Lett.*, **6**(3)**,** pp. 94-100 (2014).

[25] Mehrez, Z., El Cafsi, A., Belghith, A., et al. "The entropy generation analysis in the mixed convective assisting flow of Cu–water nanofluid in an inclined open cavity", *Adv. Powder Technol.*, **26**(5)**,** pp. 1442-1451 (2015).

[26] Sharma, A.K., Mahapatra, P.S., Manna, N.K., et al. "Mixed convection heat transfer in a grooved channel in the presence of a baffle", *Numer. Heat Transf.; A: Appl.*, **67**(10), pp. 1097-1118, (2015).

[27] Timuralp, Ç. and Altaç, Z. "Investigation of fluid flow and heat transfer in a channel with an open cavity heated from bottom side", *Mugla J. Sci. Technol.*, **2**(1), pp. 55-59, (2016).

[28] Selimefendigil, F. “Numerical analysis of mixed convection in pulsating flow for a horizontal channel with a cavity heated from below”, *Therm. Sci.*, **20**(1), pp. 35-44 (2016).

[29] Burgos, J., Cuesta, I., and Salueña, C. “Numerical study of laminar mixed convection in a square open cavity”, *Int. J. Heat Mass Transfer***, 99**, pp. 599-612 (2016).

[30] Zamzari, F., Mehrez, Z., and El Cafsi, A. “Numerical investigation of entropy generation and heat transfer of pulsating flow in a horizontal channel with an open cavity” *J. Hydrodyn.*, **29**(4)**,** pp. 632-646 (2017).

[31] Hussain, S., Ahmed, S.E., and Akbar, T. “Entropy generation analysis in MHD mixed convection of hybrid nanofluid in an open cavity with a horizontal channel containing an adiabatic obstacle” *Int. J. Heat Mass Transfer,* **114**, pp. 1054-1066 (2017).

[32] Hussain, S., Mehmood, K., Sagheer, M., et al. “Entropy generation analysis of mixed convective flow in an inclined channel with cavity with Al_2O_3-water nanofluid in porous medium”, *Int. Commun. Heat Mass Transfer*, **89**, pp. 198-210 (2017).

[33] Yasin, N.J., Jehhef, K.A., and Shaker, A.M. “Assessment of the baffle effects on the mixed convection in open cavity”, *Int. J. Mech. Mechatron. Eng*., **18**, pp. 1-14 (2018).

[34] Sivasankaran, S., Alzaharani, F.H., and Alshomrani, A.S. “Effect of baffle size and thermal boundary conditions on mixed convection flow in a channel with cavity”, *In Journal of Physics: Conference Serie*s, **1139**, p. 012088 (2018).

[35] Sabbar, W.A., Ismael, M.A., and Almudhaffar, M. "Fluid-structure interaction of mixed convection in a cavity-channel assembly of flexible wall", *Int. J. Mech. Sci.*, **149**, pp. 73-83 (2018).

[36] Carozza, A. "Numerical study on mixed convection in ventilated cavities with different aspect ratios", *Fluids*, **3**(1), pp. 1-18 (2018).

[37] Abd Al-Hassan, A.Q. and Ismael, M.A. "Numerical study of double diffusive mixed convection in horizontal channel with composite open porous cavity", *Spec. Top. Rev. Porous Media*, **10**(4), pp. 401–419 (2019).

[38] Contreras, H., Trevino, C., Lizardi, J., et al. "Stereoscopic TR-PIV measurements of mixed convection flow in a vertical channel with an open cavity with discrete heating", *Int. J. Mech. Sci.*, **150**, pp. 427-444 (2019).

[39] Ahmadi, M. and Khosravi, F.A. "Computational fluid simulation of non-Newtonian two-phase fluid flow through a channel with a cavity", *Therm. Sci.*, **24**(2), pp. 1045-1054 (2020).

[40] Biswas, N., Mahapatra, P.S., and Manna, N.K. "Mixed convection heat transfer in a grooved channel with injection", *Numer. Heat Transf.; A: Appl*., **68**(6), pp. 663-685 (2015).

[41] Biswas, N. and Manna, N.K., "Enhanced convective heat transfer in lid-driven porous cavity with aspiration" *In. J. Heat Mass Transfer*, **114**, pp. 430-52 (2017).

[42] Biswas, N., Manna, N.K., and Datta, P. “Analysis of heat transfer and pumping power for bottom-heated porous cavity saturated with Cu-water nanofluid”, *Powder Technol.*, **326**, pp. 356-369 (2018).

[43] Chakravarty, A., Biswas, N., Ghosh, K., et al. “Impact of side injection on heat removal from truncated conical heat-generating porous bed: thermal non-equilibrium approach” *J. Therm. Anal. Calorim.*, **143**, pp. 3741-3760 (2021).

[44] Biswas, N., Manna, N.K., Datta, A., et al. “Role of aspiration to enhance MHD convection in protruded heater cavity”, *Prog. Comput. Fluid Dyn.*, **20**(6), pp. 363-378 (2020).

[45] Biswas, N., Chamkha, A.J., and Manna, N.K. “Energy-saving method of heat transfer enhancement during magneto-thermal convection in typical thermal cavities adopting aspiration”, *SN Appl. Sci.*, **2**, pp. 1-25 (2020).

[46] García, F., Treviño, C., and Lizardi, J. “Numerical study of buoyancy and inclination effects on transient mixed convection in a channel with two facing cavities with discrete heating”, *Int. J. Mech. Sci.*, **155**, pp. 295-314 (2019).

[47] Cárdenas, V., Treviño, C., Rosas, I.Y., et al. “Experimental study of buoyancy and inclination effects on transient mixed convection heat transfer in a channel with two symmetric open cubic cavities with prescribed heat flux”, *Int. J. Therm. Sci.*, **140**, pp. 71-86 (2019).

[48] Laouira, H., Mebarek- Oudina, F., Hussein, A.K., et al. "Heat transfer inside a horizontal channel with an open trapezoidal enclosure subjected to a heat source of different lengths", *Heat Transfer—Asian Research*, **49**(1)**,** pp. 406-423 (2020).

[49] Yaseen, D.T. and Ismael, M.A. "Analysis of power law fluid-structure interaction in an open trapezoidal cavity", *Int. J. Mech. Sci.*, **174**, p. 105481 (2020).

[50] Yaseen, D.T. and Muneer, M.A. "Effect of deformable baffle on the mixed convection of non-Newtonian Fluids in a Channel-Cavity", *Basrah J. Eng. Sci.*, **20**, pp. 18-26 (2020).

[51] Al-Farhany, K., Alomari, M.A., and Faisal, A.E. "Magnetohydrodynamics mixed convection effects on the open enclosure in a horizontal channel heated partially from the bottom", *In IOP Conference Series: Materials Science and Engineering,* Vol. **870**, p. 012174 (2020).

[52] Ismael, M.A., Hussein, A.K., Mebarek-Oudina, et al. "Effect of driven sidewalls on mixed convection in an open trapezoidal cavity with a channel", *J. Heat Transfer*, **142**(8), pp. (2020).

[53] Mebarek-Oudina, F., Laouira, H., Hussein, A.K., et al. "Mixed convection inside a duct with an open trapezoidal cavity equipped with two discrete heat sources and moving walls", *Mathematics*, **10**(6), p. 929 (2022).

[54] Sáchica, D., Treviño, C., and Martínez-Suástegui, L. "Numerical study of magnetohydrodynamic mixed convection and entropy generation of Al_2O_3-water

nanofluid in a channel with two facing cavities with discrete heating", *Int. J. Heat Fluid Flow,* **86**, p. 108713 (2020).

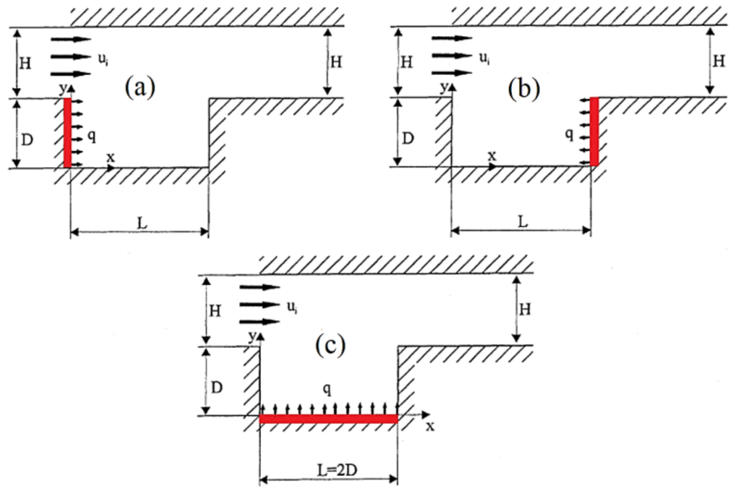
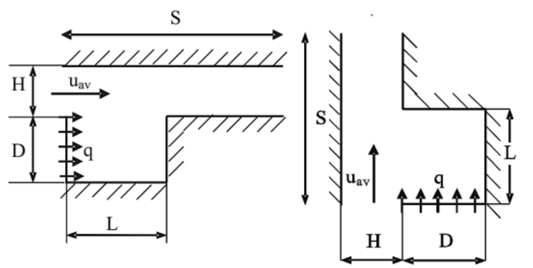
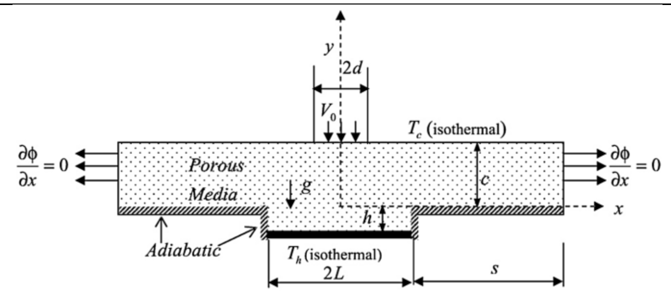
[55] Yaseen, D.T. and Ismael, M.A. "Structural mechanics of flexible baffle used in enhancing heat transfer of power law fluids in channel-trapezoidal cavity", *Exp. Tech.,* **47**(1), pp. 37-46 (2023).

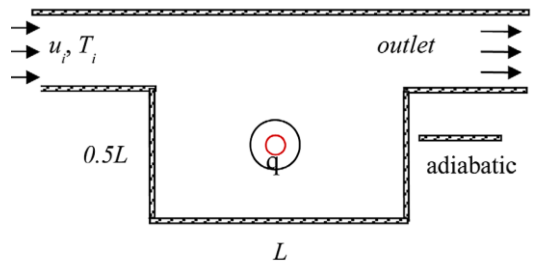
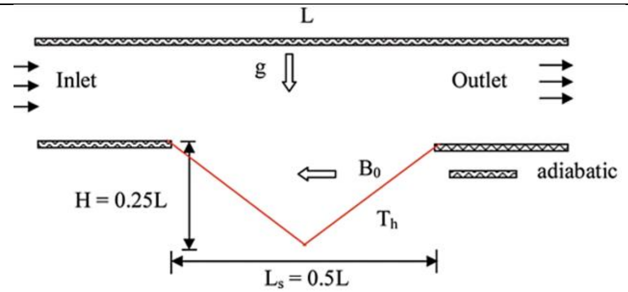
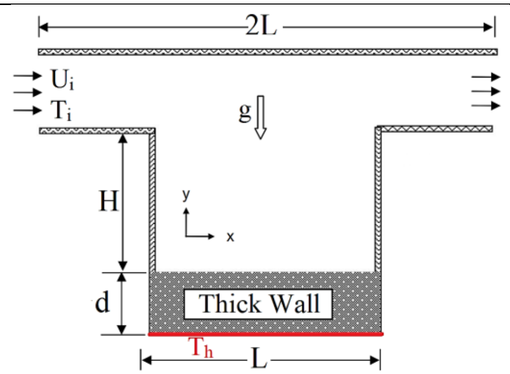
[56] Al- Hassan, A.Q.A. and Ismael M.A. "Effect of triangular porous layer on the transfer of heat and species in a channel- open cavity", *Heat Transfer*, **52**(2), pp. 1300-1323 (2022).

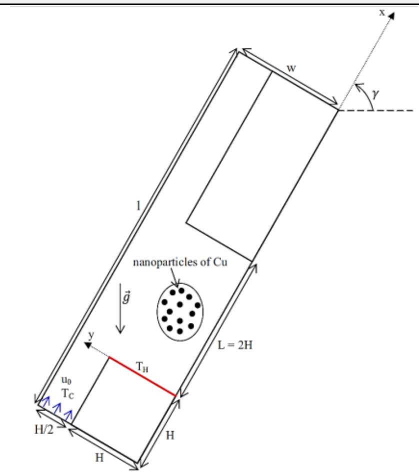
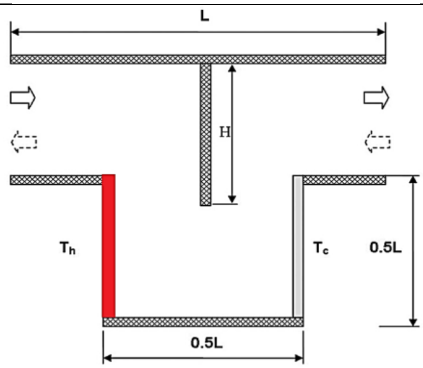
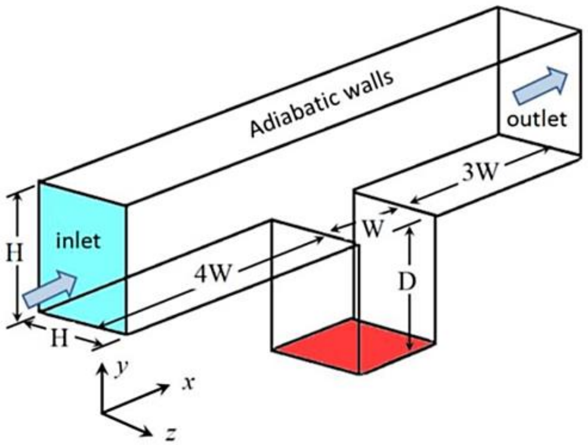
List of Table Captions:

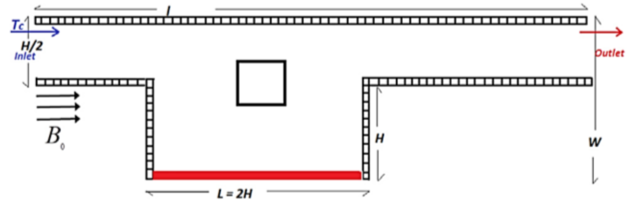
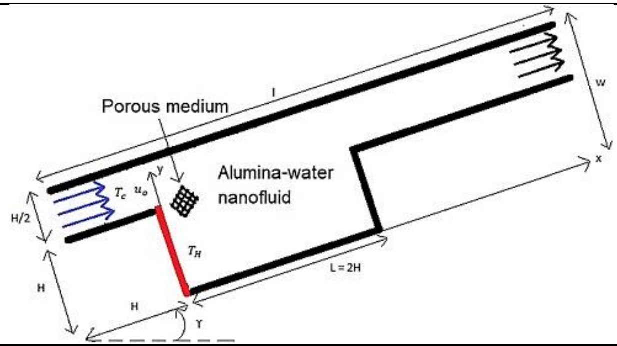
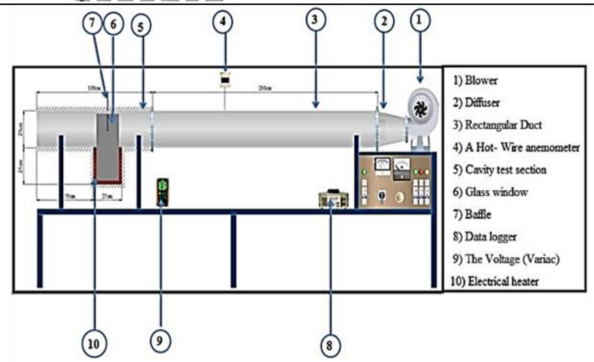
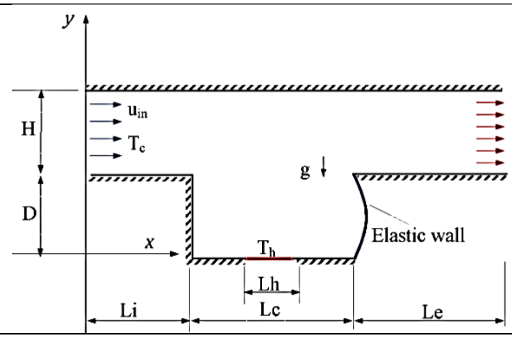
List of Figure Captions:

Table 1. Summary of classical cavity-channel geometries given in the present review

Author & Year	Model & Method	Geometry	Ranges of variable & Working fluid	Conclusions & Correlations
Manca et al. [1], 2003	FEM		$100 \le Re \le 1000$ $0.1 \le Ri \le 100$ $0.1 \le (H/D) \le 1.5$ $AR = 2$ Air, $Pr = 0.71$	. Assisting case: $Nu = 1.6854 - 0.2164(H/D) - 1.933(H/D)^2 + 6.0347(H/D)^3 - 7.057(H/D)^4 + 3.7348(H/D)^5 - 0.74775(H/D)^6$ Opposing case: $Nu = 1.1093 + 5.0147(H/D) - 10.391(H/D)^2 + 6.9637(H/D)^3 + 1.5976(H/D)^4 - 3.5819(H/D)^5 + 1.0624(H/D)^6$
Buonomo et al. [13], 2008	FVM		$Da = 0.01$ $100 \le Re \le 1000$ $0.1 \le Ri \le 100$ $H/D = 1$ $AR = 2$ $Pr = 0.71$ Air Saturated with a porous medium	- The two considered configurations exhibited very similar behaviors for low values of the Ri and Re numbers.
Wong and Saeid [16], 2009	FVM		$50 \le Ra \le 150$ $1 \le Pe \le 1000$ $0 \le h/H \le 0.4$ Air saturated with a porous foam $Pr = 0.71$	- Nu increases with the increase of Ra.

Author & Year	Model & Method	Geometry	Ranges of variable & Working fluid	Conclusions & Correlations
Rahman et al. [17], 2012	FEM		$10^3 \le Ra \le 10^5$ $0.2 \le K \le 50$ $AR = 2$ Different fluids, $0.7 \le Pr \le 7$ $Re = 100$	-The maximum bulk temperature reduces with increasing the conductivity ratio K and Pr.
Rahman et al. [18], 2012	FVM		$100 \le Re \le 2000$ $10^3 \le Ra \le 10^5$ $10 \le Ha \le 100$ $0 \le J \le 5$ Different fluids, $1 \le Pr \le 10$	-The heat transfer increases by increasing Re and Pr.
Rahman et al. [23], 2014	FEM		$0.1 \le Ri \le 10$ $0.01 \le K \le 10$ $0.1 \le D \le 0.5$ Water, $Pr = 7$	The Nu increases with Ri except at D = 0.5, while it decreases with increasing of D.

<table>
<tr><th>Author & Year</th><th>Model & Method</th><th>Geometry</th><th>Ranges of variable & Working fluid</th><th>Conclusions & Correlations</th></tr>
<tr><td>Mehrez et al. [25], 2015</td><td>FVM</td><td>
</td><td>$1 \leq Ri \leq 0.04$
$Gr = 10^4$
$100 \leq Re \leq 500$
$0° \leq \gamma \leq 360°$
$0 \leq \phi \leq 0.06$
Cu-Water nanofluid, $Pr = 6.2$</td><td>- The entropy generation is influenced by the inclination angle γ, Re and ϕ .</td></tr>
<tr><td>Sharma et al. [26], 2015</td><td>FVM</td><td>
</td><td>$0.01 \leq Ri \leq 1000$
$10 \leq Re \leq 200$
Air, $Pr = 0.71$</td><td>- With the aid of the baffle, the Nusselt number at Re = 200 is enhanced by 25–175%.</td></tr>
<tr><td>Timuralp and Altaç [27], 2016</td><td>FVM</td><td>
</td><td>$0.1 \leq Ri \leq 10$
$10 \leq Re \leq 200$
$0.5 \leq H / D \leq 2$
$0.5 \leq AR \leq 1$
Air, $Pr = 0.71$</td><td>- The forced convection becomes dominant at high value of Re.</td></tr>
</table>

Author & Year	Model & Method	Geometry	Ranges of variable & Working fluid	Conclusions & Correlations
Hussain et al. [31], 2017	FEM		$1 \le Re \le 200$ $0.01 \le Ri \le 20$ $0 \le Ha \le 100$ $0.0 \le \phi \le 0.04$ Hybrid nanofluid, Al_2O_3-Cu-water	- The flow is deviated to the channel when Ha number increased.
Hussain. et al. [32], 2017	FEM		$10 \le Re \le 200$ $0.01 \le Ri \le 20$ $0.0 \le \phi \le 0.04$ $10^{-6} \le Da \le 10^{-3}$ $0° \le \gamma \le 360°$ Al_2O_3-Water $Pr = 6.2$ $Porosity = 0.2 - 0.8$	- Bejan number increases with increasing Re and porosity, while it decreases with increasing Ri.
Yasin et al. [33], 2018	Experimental, ANSYS-FLUENT		$300 < Re < 1000$ $1.2 \times 10^7 < Gr < 8.3 \times 10^7$ $250 < Re < 1400$ $1.0 < Ri < 700.$ $q'' = 300$ and $500\ W/m^2$ Air, $Pr = 0.7$	The maximum Nu is recorded at the highest length of the baffle.
Sabbar et al. [35], 2018	(ALE) with FEM		$10^{-5} \le Ca \le 10^{-3}$ aspect ratio, $D/H = 1$ cavity width $Lc/H = 2$ $Le/H = 3$ $0.1 \le Ri \le 100$ $50 \le Re \le 250$ $Air, Pr = 0.71$	- The heat transfer is enhanced by about 17% by the existence of the elastic wall.

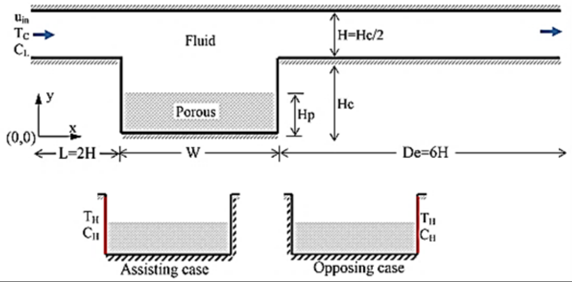
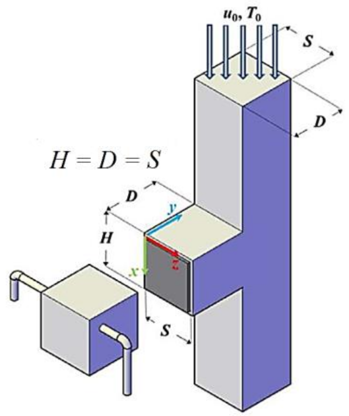
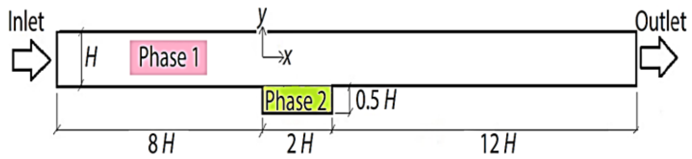
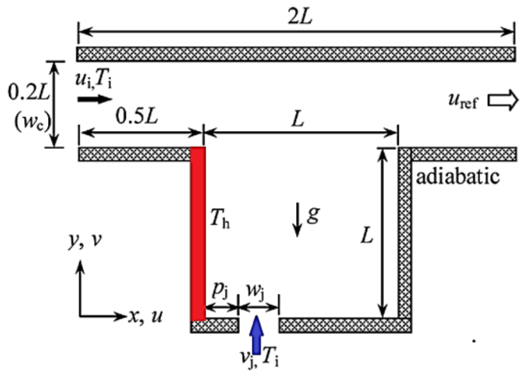
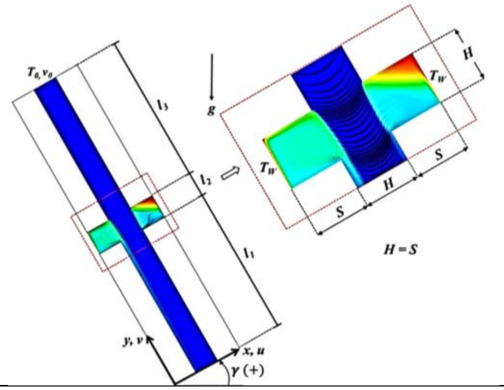
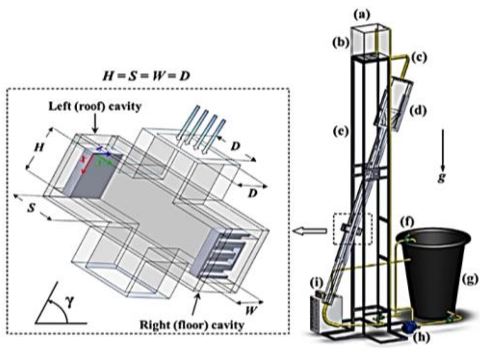
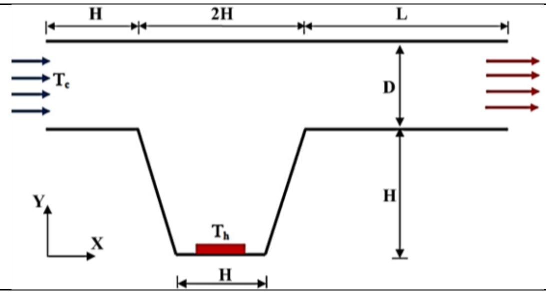
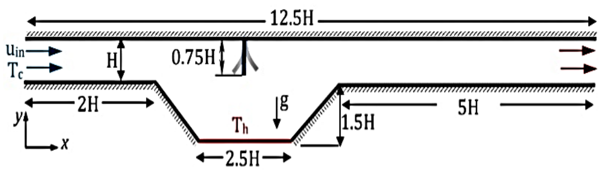
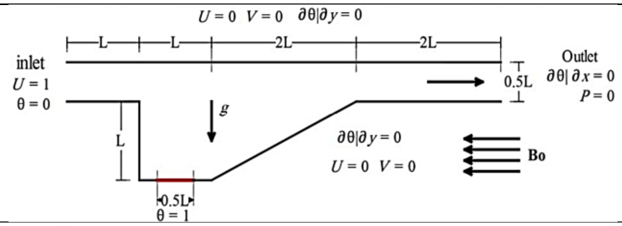
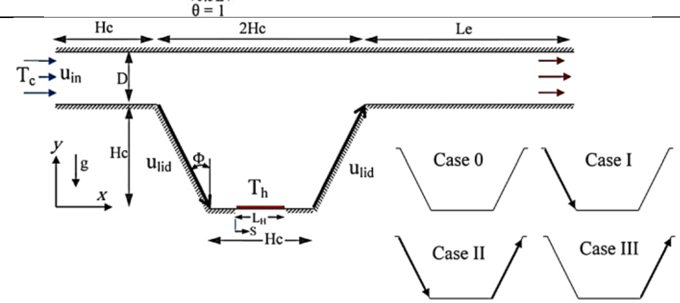
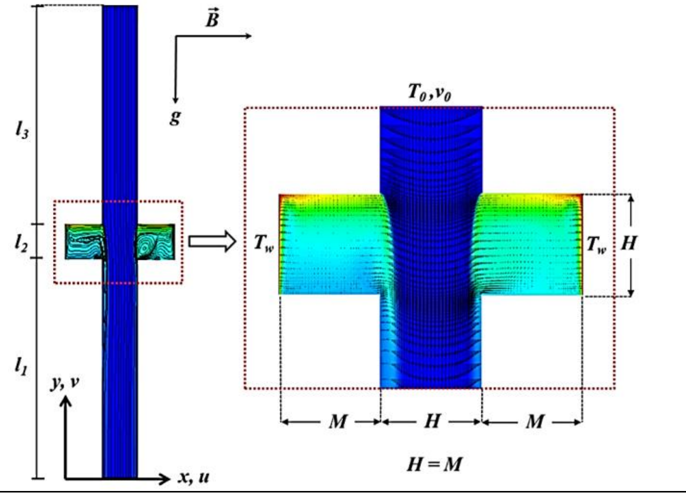
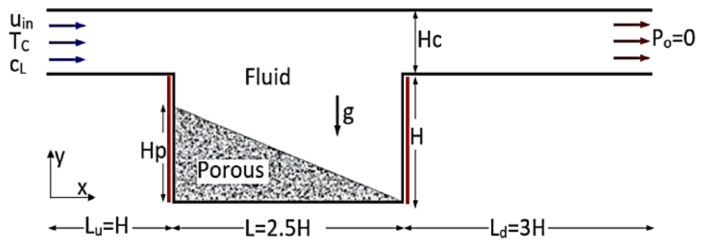
Author & Year	Model & Method	Geometry	Ranges of variable & Working fluid	Conclusions & Correlations
Abd Al-Hassan and Ismael [37], 2019	FEM		$0.01 \le Ri \le 100$ $10 \le Re \le 200$ $0.25 \le Hp \le 1$. $Da = 10^{-3}, N = 0.5, Le = 20$ Water Saturated with a porous medium $Pr = 6.26$	- For the assisting case, Nu and Sh numbers decrease with increasing H_p.
Contreras et al. [38], 2019	Experimental and Numerical (POD)		$Re = 1500$ *and* 4500 $0 \le Ri \le 20$ $Gr = 4.05\times10^{6}$ *and* 4.5×10^{7} Water, $Pr = 7$	- The increase in Re decreases the critical value of Ri, above which the flow is no longer encapsulated.
Ahmadi and Farsani [39], 2019	FVM		$500 \le Re \le 1500$ ρ=800 kgm^{-3}, λ = 0.036 s μ_0 = 0.00345 $kgm^{-1}s^{-1}$ μ_∞=0.056 $kgm^{-1}s^{-1}$ $AR = 4$ non-Newtonian two-phase fluid-flow $0.7 \le n \le 1$	- The changes in the velocity, pressure and volume fraction of fluids inside the channel and cavity are more sensible to Re.

Table 2. Summary of complex cavity-channel geometries given in the present review

Author & Year	Model & Method	Geometry	Ranges of variable & Working fluid	Conclusions & Correlations
Biswas et al. [40], 2015	FVM		$0.1 \le Ri \le 10$ $50 \le Re \le 200$ $0 \le Pj\ (jet\ position) \le 0.95$ $0.01 \le Wj\ (injection width) \le 0.2$ $10\% \le qj\ (injection\ ratio) \le 50\%$ $Air, Pr = 0.71$	- Dividing the total flow to main and injected flows augments Nu by 218%. - The injection becomesore effective when it is located close to the hot wall. - The width of the injection acts adversely on the Nusselt number.
García et al. [46], 2019	FVM using Open-MP		$100 \le Re \le 1000$ $0^o \le \gamma \le 90^o$ $0.25 \le AR \le 1$ $Water, Pr = 7$	-The heat transfer rate increases by increasing the Re.
Cárdenas et al. [47], 2019	Experimental		$32.17 \le Ri \le 300.77$ $0∘ \le \gamma \le 90∘$ $500 \le Re \le 1500$ $H = S = W = D = 5cm$ $Length\ (T - section) = 180cm,$ $Water, Pr = 7$	-The overall heat transfer was a nonlinear function of the inclination angle. - For left cavity wall (Roof) at Re = 1000: $Nu = 0.63736 Gr^{0.21111}$ - For right cavity wall (floor) at Re = 1000:

Author & Year	Model & Method	Geometry	Ranges of variable & Working fluid	Conclusions & Correlations
				$Nu = 0.7763 Gr^{0.20138}$
Laouira et al. [48], 2019	ANSYS		$0.16 \le Heat\ source\ length \le 1$ $0.1 \le Ri \le 100$ $Re = 100$ $Air, Pr = 0.71$	- Nu increases with increasing the length of the heat source.
Yaseen and Ismael [49], 2020	FEM		$10^{-20} \le Ca \le 10^{-3}$ $100 \le Re \le 300$ $0.01 \le Ri \le 10$ Power law Fluid $0.5 \le n \le 1.5$	-For low Ri, the shear-thickening fluid manifests higher Nu number while for high Ri number, the shear-thinning fluid has the higher values.
Al-Farhany et al. [51], 2020	COMSOL Multiphysics		$Re = 100 and 150$ $0.1 \le Ri \le 10$ $0 \le Ha \le 15,$ $Air, Pr = 0.707$.	- Nu increases with the increase of Ri and the decrease of Ha.
Ismael et al. [52], 2020	FEM		$Re = 100$ $0 < Rer < 5$ $0.1 < Ri < 100$ $Air, Pr = 0.71$	The maximum Nu occurs with case I.

Author & Year	Model & Method	Geometry	Ranges of variable & Working fluid	Conclusions & Correlations
Sáchica et al. [54], 2020	CVM		$0 \le Ha \le 0$ $-1 \le Ri \le 5$ $0.0 \le \phi \le 0.2$ $100 \le Re \le 700$ Al_2O_3 –water nanofluid	- Nu decreases with the increase of Ha for all values of Re. -
Abd Al-Hassan and Ismael [56], 2022	FEM		$50 \le Re \le 250$ $0.1 \le Ri \le 100$ $0.25 \le Hp \le 1$ $Da = 10^{-3}, Le = 20, and\ N = 0.5$ Porous medium, $Pr = 6.24$	- The opposing case is better than the assisting case - The heat and mass transfers of the inclined porous layer are 30% greater than the horizontal layer. - Assistance case: $Nu = \begin{cases} 0.187Re^{0.645}Ri^{0.232} \\ 0.159Re^{0.632}Hp^{-0.162} \end{cases}$ - Opposing case: $Nu = \begin{cases} 0.451Re^{0.553}Ri^{0.22} \\ 0.875Re^{0.359}Hp^{0.033} \end{cases}$

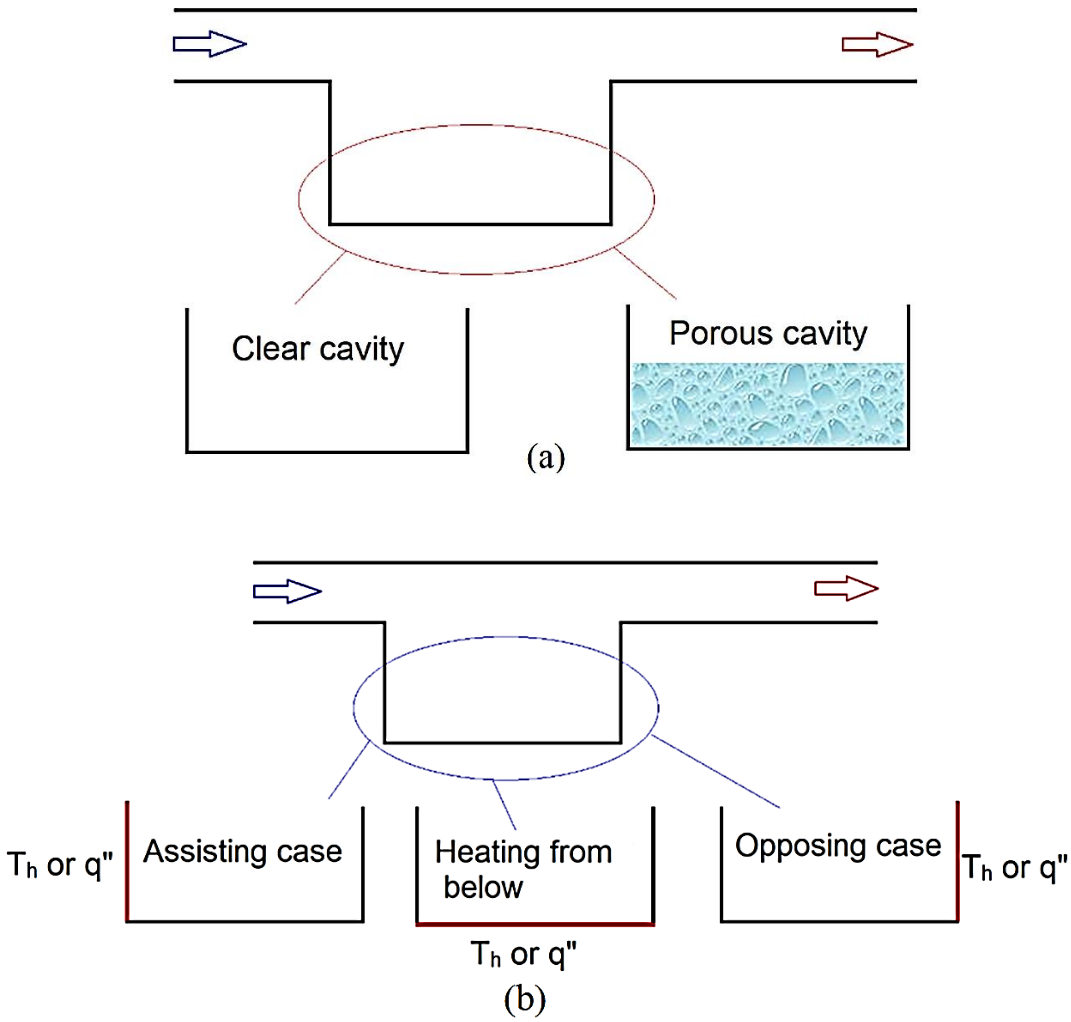


Figure 1. Various cases of cavity-connecting channel, (a) Clear and porous cavities, (b) Cases depending on the boundary condition

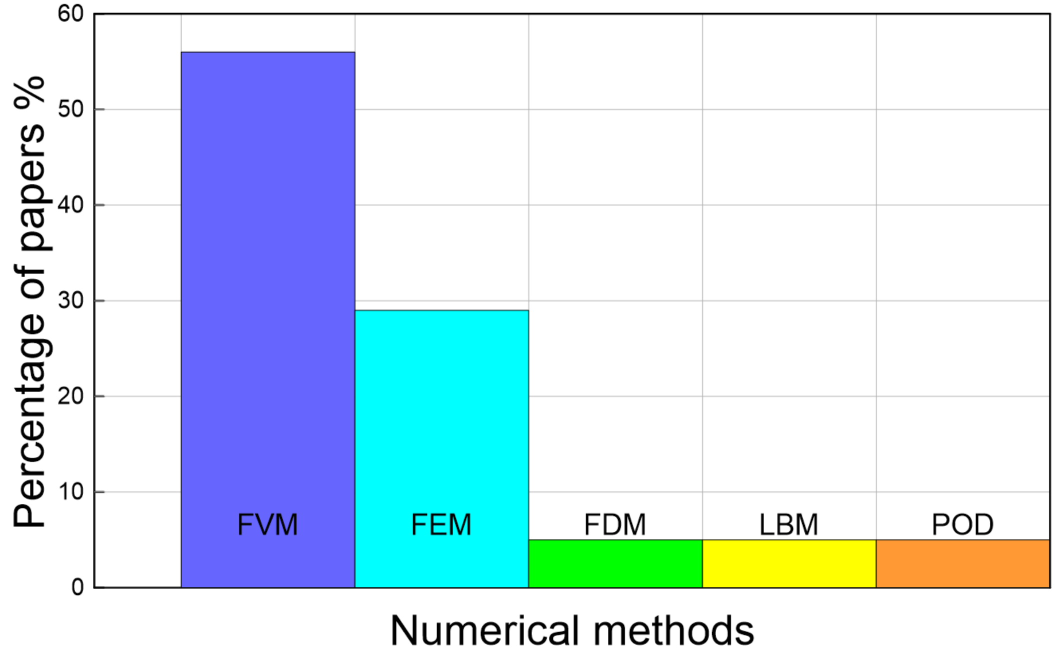


Figure 2. Percentage of numerical works against the numerical method used

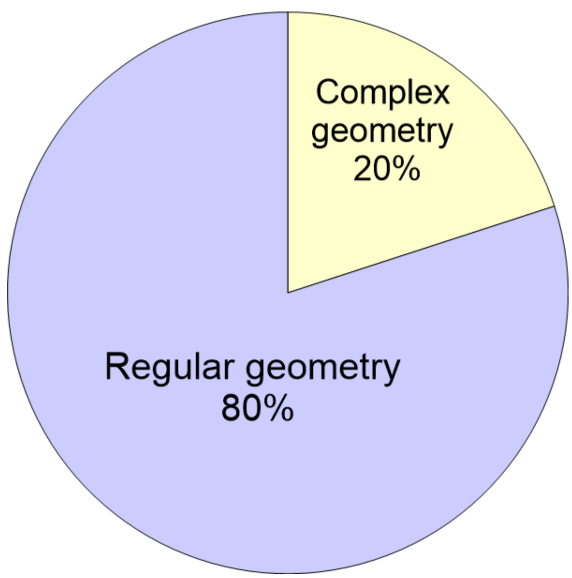


Figure 3. Percentages of the regular and complex geometries of the cavity-channel assembly

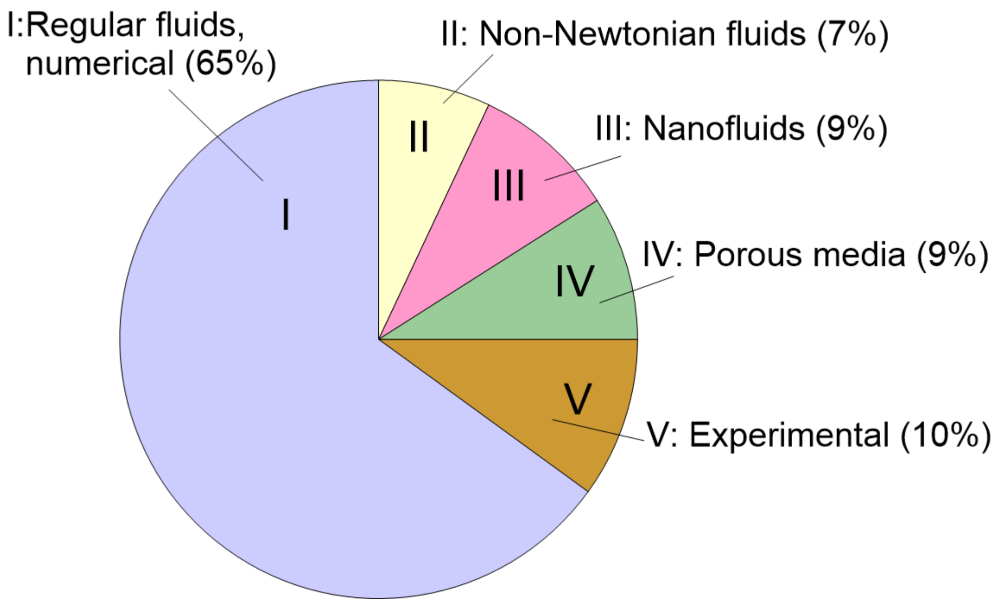


Figure 4. Percentages of different studies

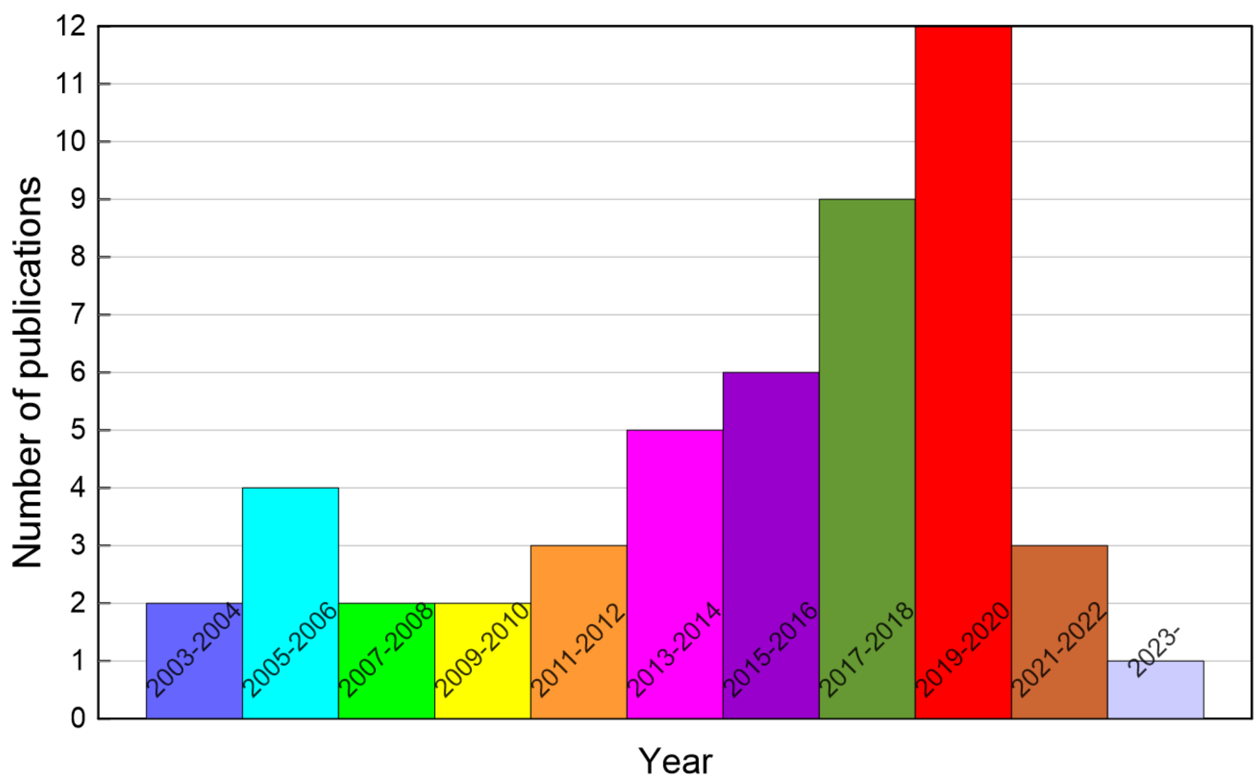


Figure 5. Number of published papers arranged chronologically

Prof. Dr. Ahmed Kadhim Hussein is as a professor in the Mechanical Engineering Department at the University of Babylon, Babylon City, Iraq. He received his Ph.D. in Mechanical Engineering from University of Al-Mustansiriya - Iraq in 2006. Meanwhile, he is a reviewer of many local and international journals and conferences. He is an editor of more than twenty international journals. Also, he is a member of many international scientific societies in Austria, Singapore, United States of America, Japan and United Arab Emirates. His research work concerns numerical modelling of various areas of heat transfer, CFD, aerodynamics, nanotechnology, clean energy and extended irreversible thermodynamics. He has published about 149 and awarded in 2018 the scientific excellence medal - First rank on Science and Technology from Ministry of Higher Education and Scientific Research – Iraq.

Mrs. Amaal Abdul Razaq is a mechanical Engineer graduated in Bachelor of Science Mechanical Engineering at the University of Babylon (2007), where she getting her M.Sc. in Thermo-Fluids Engineering from the University of Babylon (2022). She has a 10 years of experience in hiring and employee management. She worked as assistant manager with five years of Design Department on Directorate of Babylon Sewerage in Babylon Governorate-Hilla-Iraq.

Dr. Awatef Abidi is an Assistant Professor in the Physics Department, College of sciences at King Khalid University Kingdom of Saudi Arabia. She received her Ph.D. degree in Energetic Engineering in 2011 and obtained her Habilitation for Supervising Doctoral Research (HDR) from National Engineering School of Monastir at University of Monastir, Tunisia. Her research interests cover computational fluid dynamics, micropolar fluid, nanofluid, hybrid nanofluid and heat and mass transfer in microchannel, nanofluid and solar energy, phase change materials and porous media heat transfer.

Muneer Ismael is a professor at the University of Basrah, college of engineering where he got his B.Sc., M.Sc. and PhD. In 1996, 1998, and 2007, respectively. He started his research field with the electromagnetic flow measurement, then his passion turned into the CFD, FSI, convective heat transfer of porous media and nanofluids. He published about 60 papers in this field and got five prizes of distinct researcher, two of them from Al-Eyen university award and three from University of Basrah. Currently he works at the Department of Mechanical Engineering, University of Basrah. At the same time, Muneer Ismael is visiting research fellow at University of Warith Al-Anbiyaa, Karbala, Iraq.

Ahmed B. Mahdi is a researcher works now in Anesthesia Techniques Department, Al-Mustaqbal University College that is a private university located in Babylon, Iraq. He is interested in many fields in science and technology such as nanotechnology and pollutants removal from wastewater.

Bashar S. Bashar is a researcher works in Al-Nisour University College. He is a PhD student at the Universiti Tenaga Nasional in Malaysia, born 1989, obtained Master's degree in 2014 from the Middle Technical University, Technical College of Electrical Engineering, Baghdad, Iraq. He published seven papers in Scopus and ISI. He is interested in electronics and communication engineering, microwave engineering and antenna engineering

Farhan Lafta Rashid completed his B.Sc. in 1994 on Nuclear Engineering from University of Baghdad Iraq and his M.Sc. in 2002 on Mechanical Engineering/ Thermo-Fluid Mechanics from University of Technology, and his Ph.D. in 2016 on Mechanical Engineering/ Thermo-Fluid Mechanics from University of Technology Iraq. He started his job journey as a researcher in the Iraqi Atomic Energy Commission. He later moved on to the University of Kerbala as a Lecturer of Fluid Mechanics and Thermodynamics in Petroleum Engineering Department. He has more than 198 published papers in respected journals. His research interest in analysis of fluid flow, CFD, modeling and simulation. He has five patents.

Prof. Dr. Raad Z. Homod, after his graduation in mechanical engineering (with distinction) in 1991 from the University of Basrah, worked as a project engineer on the development of HVAC systems control in Libya, he is also a senior lecturer at the Sebha Centre for vocational rehabilitation. He later completed his M.Sc. from UM and PhD from UNITEN, Kuala Lumpur, Malaysia (both with distinction). He is the author/co-author of more than 80 research papers published in international journals, has three patents, awarded 6 prizes and is the author of 4 books. His current research interests are in artificial intelligence (Reinforcement Learning), nonlinear systems analysis, neural networks, energy efficiency with building automation and control systems, Buildings model and modelling sensors with Fuzzy logic and optimization of control systems.

Dr. Obai Younis is assistant prof. of Mechanical Engineering. He obtained his PhD in 2009 from Universitat Rovira I Virgili in Spain with major in Thermofluids. He worked in different Universities in Sudan, Malaysia and KSA. Currently he is assistant prof. in Department of Mechanical Engineering, College of Engineering in Wadi Addwasir-KSA. He supervised many final year projects, more than 10 Master students and 3 PhD students. He is currently working on collaborative projects with researchers from Iraq, Malaysia, Algeria and KSA.

Lioua Kolsi is currently a full Professor in the mechanical engineering department, University of Ha'il Saudi, Arabia. His current research projects focus on interdisciplinary applications of nanotechnology and Heat and fluid flow. Kolsi's areas of expertise include Thermodynamics, nanofluids applications, CFD, heat and mass transfer, flow visualisation, MHD. Dr Kolsi has co-authored numerous (more than 250) highly cited journal publications, conference articles and book chapters in the aforementioned topics. Dr Kolsi's h-index is 36 according to Web of Science and he is ranked among the top 2% researchers according to Stanford university ranking.

Prof. Ali J. Chamkha is a Distinguished Professor of Mechanical Engineering and Dean of Engineering at Kuwait College of Science and Technology. He earned his Ph.D. in Mechanical Engineering from Tennessee Technological University, USA, in 1989. His research interests include multiphase fluid-particle dynamics, nanofluids dynamics, and fluid flow in porous media, heat and mass transfer, magnetohydrodynamics and fluid-particle separation. He is currently the Editor-in-Chief for the Journal of Nanofluids and has served as an Editor, Associate Editor or a member of the editorial board for many journals such as ASME Journal of Thermal Science and Engineering Applications, ASME Journal of Nuclear Engineering and Radiation Science, International Journal of Numerical Method for Heat and Fluid Flow, Journal of Thermal Analysis and Calorimetry, Thermal Science journal, Scientia Iranica, Special Topics & Reviews in Porous Media, Journal of Porous Media, Journal of Thermal Engineering, Recent Patents on Mechanical Engineering, Journal of Applied Fluid Mechanics, International Journal of Fluids and Thermal Sciences, Journal of Heat and Mass Transfer Research, International Journal for Microscale and Nanoscale Thermal and Fluid Transport Phenomena, International Journal of Industrial Mathematics and many others. He has authored and co-authored over 1100 publications in archival international journals and conferences. His current h-index is 121 and total citations is 48,454. Professor Chamkha was included in the World's Top 2% Scientists 2020, 2021 and 2022 lists (by Stanford University) with a Global Rank #21, #20 and #23 out of a total of 92,645, 109,724, and 121,447, respectively and Rank #1 at the Arab World level in Mechanical Engineering and Transports category for all these years.