INVESTIGATION OF TURBULENT FLUID FLOW BEHAVIOR IN A CIRCULAR DUCT WITH MULTIPLE SIDE IMPOSITIONS: A COMPUTATIONAL ANALYSIS

EDAKE VAIBHAV VILAS
Reg No: 241217063
Research Scholar
Mechanical Engineering
Shri JJT University,
Rajasthan.

Dr. S. CHAKRADHAR GOUD
Professor
Mechanical Engineering
Shri JJT University,
Rajasthan.

Dr. CHANDRA SEKHAR REDDY N
Principal & Professor
Mechanical Engineering
Rishi M.S Institute of Engineering and Technology for Women
Hyderabad.

Abstract

In a circular axis symmetric conduit, we numerically explore the mixing of a turbulent fluid injected at 45 and 90 degrees to the main turbulent bulk flow. By adjusting the parameters of the original k-model, we were able to account for the influence of streamline curvature. The effects of varying the swirl number at the duct's input on the resulting turbulent flow pattern were studied. The effect of rim thickness on the discern ability of the transition between the primary and secondary input fluid flows has been studied. The results of changing the rim thickness of a conventional cylindrical tube while keeping the flow velocity ratio between the primary and secondary inlets constant were studied. It has been determined that the size and intensity of the recirculation bubble depend on a number of parameters. Considerations include the velocity-to-distance ratio, the speed of the lateral injection, and the position of the injection point. Coaxial flows and turbulent side-mass jets have been the subject of a great deal of research. The intensity of turbulence, axial temperature change, and axial velocity has all been the subject of much study.

Keywords: *axisymmetrical duct, closed duct, cross-flow, turbulent intensity, Ansys Fluent*

INTRODUCTION

When a fluid is flowing, centrifugal forces cause it to produce secondary motion most obvious example of this is a pair of counter-rotating vortices. To function properly, fluid mechanical systems rely on ducts (inlets, nozzles, diffusers, contractions, elbows, etc.) that must be both highly efficient and very complex in design. Engineers use a mathematical model of the flow field to guide the construction of ducts for such applications.

The process sector has significant challenges related to heat utilization, conservation, and recovery. The efficient utilization of heat in process facilities is a key factor in their cost-effective design and operation. Heat exchanger mass size is increasing as a result of unit power and output. This undertaking has yearly operating and capital expenses in the tens of millions of dollars.

Therefore, it is crucial to discover methods of reducing the heat exchangers' total dimensions.

Innovations in heat exchanger technology have been spurred by the need to reduce costs. The phrase "heat transfer enhancement," often spelt "heat transport augmentation," refers to an umbrella of methods for improving heat transmission.

Pipelines may go either horizontally or vertically. Several models have been developed to examine the two-phase flow and flow patterns in geothermal wells. The observed and predicted pressure drop was compared. By examining the speeds and percentages of vacant space in the horizontal pipes, they concluded that stratified wavy flow was typical. Downhill two-phase flow in pipes of varied diameters was also studied, along with the flow pattern and transitions between flow regimes. The downward flow was really choppy; it was whirling, whirling, and cap-bubbling. Horizontal flows may be classified into many distinct categories, with some examples being pseudo-slug flow, stratified flow, and plug flow. The flow was seen to change from annular to turbulent when the surface velocity of the liquid was increased beyond a critical value. Vertical geothermal, oil, and gas wells' temperature and pressure profiles were predicted using heat transfer and two-phase flow models. The air-water two-phase system generates much more pressure than a conventional single-phase system. Using an innovative heat transfer model, we determined the temperature increase in the production pipe for a two-phase geothermal fluid.

LITERATURE REVIEW

Amjad Ali Pasha [2022]. The stretching rate is shown to rise with the square of the distance from the slot. The conservation of energy equation provides a stable foundation for the mechanical heat transmission process by taking into consideration both thermal radiation and viscous dissipation. We next resort to numerically solving the ensuing problems using Keller's box approach. To adjust the controlling parameters while still understanding the physical circumstances of the stretching sheets, numerical simulations are performed. The velocity field and temperature distributions gave visual representations of the effects and typical behaviors of physical variables.

AliosatEbrahimNejad [2021] Using the sine and cosine transform functions; we provide a novel method for determining governing equation for momentum in Cartesian and Cylindrical coordinates. Due to its symmetry, the duct's cross section only has to be addressed in a very short range (= 0 to /2). The data suggests that the velocity profile would flatten out and the velocity variation will decrease if the cross section is changed from non-circular to circular. When the pressure difference between the two cross sections remains constant, the highest velocity in the circular cross section becomes the lowest velocity in the rectangular or round edge cross section. It is shown that given a fixed pressure gradient, both the average velocity and mass flow increase with m.

Pingnan Huang (2020) concludes that the bionic fractal micro-channel heat sink (MHS) is the best option because of its uniform velocity distribution and high heat transfer efficiency. However, the MHS's fluid flow and heat transfer performance are heavily influenced by the structure of the MHS's variable cross-section. We begin by proposing a tree-shaped MHS (TMHS) with a dynamic cross-section, in light of the analogy between the bionic fractal and a tree. Flow and heat transfer characteristics, pressure drop, and overall performance of TMHSs with cavities and ribs were compared to those of the typical smooth TMHS by numerical analysis using the commercial software Fluent. The results suggest that the TMHS's dynamic cross-section may enhance its heat-transfer capability. Pressure drop was greatest for the TMHS with ribs (TMHS-R), while heat transfer efficiency was best for this design.

Hira Narang. (2019) there are a growing number of scientific and commercial applications that need the capacity to predict heat and mass transfer in capillary porous media. The numerical calculation of the heat and mass flow equations in capillary porous media may be time consuming. In this investigation, a well-established acceleration technique from the area of computer graphics is used to efficiently solve heat and mass transport equations numerically. The CUDA programming paradigm is widely utilized in the gaming industry because of its ability to do parallel processing on graphics processing units. This research shows that the heat and mass transport equations for a cylinder made of a capillary porous radically composite material may be easier to solve under the first kind of boundary conditions.

Turbulence

Turbulence is a feature of fluid flows with extremely high Reynolds numbers. In fluid mechanics, the Reynolds number quantifies the predominance of turbulent flow due to viscous effects over inertial ones. Turbulent flow, in contrast to the smooth lines of laminar flow, is typified by rapid eddy fluctuations. In addition to the momentum and energy transfer already present in the fluid element, whereas the energy of smaller eddies is converted into internal momentum by the viscous forces of the flow.

Characteristics of Turbulent Flow

Once turbulent flow has started, the velocity, viscosity, and linear dimension will all have increased. The random motion of particles inside a fluid is an indicator of its flow. The mobility of particles in a fluid is erratic and choppy. According to this theory, turbulence is less of a deterministic phenomenon and more of a statistical one.

The velocities in both types of pipes are quite similar when the flow is turbulent. This causes the liquid to drip at a constant rate and to fall quite close to the wall. Diffusivity is the property that, during ebb, makes it possible for greater mass, velocity, and energy to be mixed together and transferred.

Turbulence models

The three most well-known methods for simulating fluid mixing processes are the Reynolds average Navier-Stokes (RANS) method, the direct digital simulation (DNS), and the large eddy simulation (LES) method. The latter were designed specifically to measure the turbulent viscosity of the flow (t), an essential factor in turbulence-dependent CFD fire simulations. Finding the medium-scale flow variables and computing the large-scale motions are all that is required by the RANS method for solving Navier-Stokes equations. Due to its ease of use and minimal hardware requirements, RANS has seen increased adoption in industrial CFD applications in recent years.

While the RANS method has found widespread use, it has been widely accepted that it is inadequate for modeling the extremely variable flow that happens during a flood. DNS, on the other hand, aims to resolve edges across several temporal spatial scales in order to solve the full complexity of the underlying equations.

Circular duct with time-periodic boundary conditions

Within a rectangular cavity with two adiabatic walls on the horizontal plane, a uniform and constant temperature on the vertical wall, and a uniform temperature distribution that varies with time according to a sinusoidal law or a uniform heat flux distribution that undergoes periodic square wave time-change, we use numerical techniques to investigate the dynamics of natural convection. In contrast to popular opinion, it is shown that for wall temperature oscillations of sufficiently significant amplitude, there is a critical frequency that corresponds to a maximum of the time-averaged heat flow. For Prandtl values over 0.277, the authors show, the friction factor measured at the wall exhibits resonant behavior. The Prandtl number specifies the resonance frequency of the dimensionless heat flow in any plane parallel to the channel wall when the boundary condition is an oscillation.

Here, we study laminar mixed convection of a Newtonian fluid in a vertical circular conduit during its steady-periodic phase. In order to satisfy the thermal boundary condition, the wall temperature must be held constant while the temperature in the solidified region varies sinusoidal. Analytical methods are used in this inquiry.

Direct numerical modeling of turbulent duct flows

Ducts with a rectangular cross section may be used in a variety of ecological, industrial, and biological contexts. Researching flows in rectangular ducts of varied aspect ratios (ARs; AR = W/H) may provide light on how 3D phenomena affect such design-critical flow parameters as wall shear, pressure drop, and so on.

Primary objectives of the study are: Find out what role a secondary turbulent flow (here interpreted as a cross-flow field) plays in the turbulence close to the walls of a duct and how the turbulence develops. Scaling of near-wall turbulence structures in wall-bounded shear flows is dominated at sufficiently high Reynolds numbers by the viscous length scale = v/u (where is the fluid kinematic viscosity and the friction velocity u = w/ is computed as a function of the wall shear stress w and the fluid density u). These structures are most obvious in the area just near to the wall; however they may also be discernible over the whole overlap layer. Therefore, turbulent duct flows need a degree of large-scale motion.

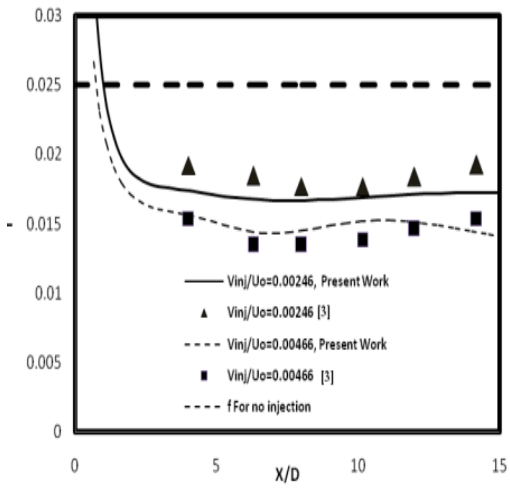
METHODOLOGY

At an angle or perpendicular to the duct's axial course, the side mass flow is injected into the main turbulent bulk flow in a number of industrial operations. The gas temperature of a gas turbine's combustor must be lowered to an acceptable level for the material of the turbine blades, and this is the job of the dilution zone. Combustor performance is very sensitive to the location, size, and injection speed of side jets. Side injection is utilized in primary turbulent fluid flow to improve and manage mixing, which is especially important given the difficulty in maintaining a stable combustion flame due to the recirculation bubble. In order to get a good performance rating and cut down on carbon particles, the fuel in the combustor must be completely burned. There is harm done to the environment and the combustion process by these carbon particles. It is crucial to generate the recirculation zone in the combustor to prevent the flame from being extinguished and destabilized, allowing for effective turbulent mixing and high combustion efficiency.

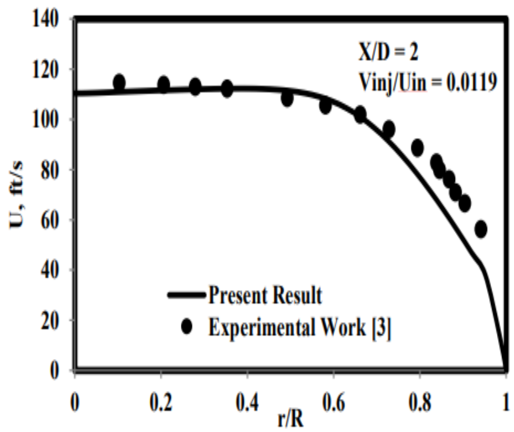
RESULTS

Validation of the Present Numerical Model

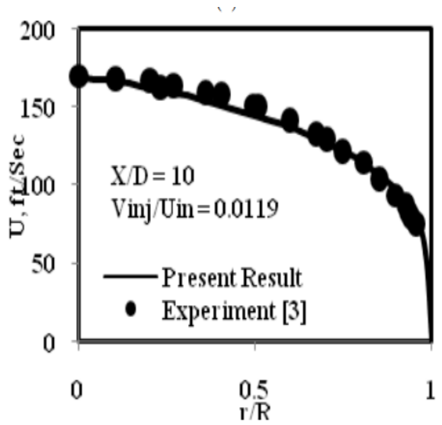
The results have been double-checked by comparing them to the reference implementation, which is considered the gold standard. Experimental results obtained by Olson and Eckert are shown in Graph 6.2, which depicts a turbulent flow in a porous circular tube subjected to homogenous fluid injection along the tube wall. Friction factors determined by numerical analysis (left) and experimental data (right) from Olson and Eckert are compared in Graph 6.2 (a). Figures 3.3 (b) and (c) depict the experimental and current numerical results for the axial velocity profiles at different axial locations. The numerical technique employed in this study to depict the turbulent flow in a circular duct when a mass is injected from the side is supported by the excellent agreement between experimental data and numerical prediction.



(a)



(b)



Graphs: Validation of the present numerical model

CONCLUSION

This numerical analysis included a variety of geometries and boundary conditions. This article summarizes the most important takeaways from the research. The flow pattern of the main flow is drastically altered when the side injection velocity is altered. The intensity of the recirculatory flow and the depth to which it may penetrate may both be enhanced by increasing the velocity of the side injection. As the injection velocity to intake velocity ratio rises, the recirculation bubble grows in both width and depth. There is a strong relationship between the location and size of the injection zone and the resulting flow pattern and recirculation bubble size. By raising the swirl velocity, the input swirl helps decrease recirculation bubble size. When the main injection angle is decreased and the secondary injection angle is increased, a recirculation bubble forms upstream near the axis for angular twin side injection. The area between the main and secondary injection zones, and especially close to the wall, is where the duct's vortices is strongest. If there is a central jet, a recirculation bubble forms near the wall of the duct's intake; if there is an annular jet, the bubble forms on the axis. Due to the jet's lateral motion, less mass is recalculated when the injection speed of an annular jet is raised.

REFERENCE

1. *Ahmed JassimShkarah(2013),"CFD Simulation of Heat Transfer and Turbulent Fluid Flow over a Double Forward-Facing Step",Mathematical Problems in Engineering,ISSNno:1563-5147,Vol.2013,Pages.2013.https://doi.org/10.1155/2013/895374*
2. *A. Karimipour (2012),"Simulation of Fluid Flow and Heat Transfer in the Inclined Enclosure",World Academy of Science, Engineering and Technology,ISSNno:1307-6892,Vol.61,*
3. *Francisco Marcondes (2019),"Modeling and computational simulation of fluid flow, heat transfer and inclusions trajectories in a tundish of a steel continuous casting machine",Journal of Materials Research and Technology,ISSNno:2238-7854,Vol.8,No.5,Pages.4209-4220.https://doi.org/10.1016/j.jmrt.2019.07.029*
4. *M. Brunet (2002),"An experimental and numerical CFD study of turbulence in a tundish container",Applied Mathematical Modeling, ISSNno:0307-904X,Vol.26,Pages.323–336.*
5. *Jesus Patrick E. Nuqui (2022),"Computational Fluid Dynamics (CFD) analysis of the heat transfer and fluid flow of copper (II) oxide-water nano fluid in a shell and tube heat exchanger",Digital Chemical Engineering,ISSNno:2772-5081,Vol.3,https://doi.org/10.1016/j.dche.20 22100014*
6. *Dongsheng Wen (2004),"Experimental investigation into convective heat transfer of NF at the entrance region under laminar flow conditions",International Journal of Heat and Mass Transfer,ISSNno:0017-9310,Vol.47,Issue.24,Pages.5181-5188.https://doi.org/10.1016/j.ijheatmasstransfer.2004.07.012*
7. *Amir Jokar (2013),"Heat Transfer and Fluid Flow Analysis of NF in Corrugated Plate Heat Exchangers Using Computational Fluid Dynamics Simulation",Journal of Thermal Science and Engineering Applications,ISSNno:1948-5093,Vol.5 (1),DOI:10.1115/1.4007777*
8. *Yimin Xuan (2000),"Heat transfer enhancement of NF",International Journal of Heat and Fluid Flow,ISSNno:0142-727X,Vol.21(1),Pages.58–64.DOI:10.1016/S0142-727X(99)00067-3*
9. *Nima Amanifard (2007),"Heat transfer in porous media",Brazilian Journal of Chemical Engineering,ISSNno:1678-4383,Vol.24(2),Pages.223-232.DOI:10.1590/S0 104-66322007000200007*
10. *Abbas Moraveji (2017),"Computational fluid dynamics simulation of heat transfer and fluid flow characteristics in a vortex tube by considering the various parameters", International Journal of Heat and Mass Transfer,ISSNno:0017-9310,Vol.113,Pages.432-443.DOI:10.1016/j.ijheatmasstransfer.2017.05.095*