DYNAMIC PERFORMANCE OF COMPACT ORIFICE RADIAL FLOW SIMULATION BY USING FEM

Seelam Jamala Reddy
Registration No: 22516084
Research Scholar
Mechanical Engineering
Shri JJT University
Rajasthan

Dr. Venkateswarlu Ganta
Co-Supervisor
Professor & Principal
Mechanical Engineering
Sree Chaitanya College of Engineering

Dr. S. Chakradhar Goud
Supervisor
Dept of Mechanical Engineering
Shri JJT University
Rajasthan.

Abstract

The numerical analysis of fluids passing through small orifices with respect to radial flow is a complex problem that requires the use of advanced computational techniques. In this study, we have analyzed the flow of different fluids, including water, oil, and gas, through small orifices with respect to radial flow. The numerical simulations were conducted using the finite element method (FEM), which is a powerful computational tool for solving complex fluid flow problems. The fluid flow paths of this proposed valve consist of annular flow channel, a single radial flow channel and an orifice flow channel through structural design, using ANSYS, including achieving optimal magnetic field distribution and yield stress in the annular flow path and radial flow path, respectively. Fluids in liquid phase are incompressible whereas fluids in gaseous phase are compressible. Liquid occupies the same volume at different pressures whereas gases occupy different volumes at different pressures.

Keywords: *Small orifices, finite element method, fluids, radial flow, ANSYS.*

Introduction

Radial flow of fluids between two parallel disks is pertinent to a number of engineering applications such as radial diffusers, air-bearings, heat exchangers and drilling fluid losses in fractures. Radial flow of non-Newtonian fluids has been extensively studied. Radial flow of Power-Law fluids has been investigated by several researchers and compared with experimental data. Several models for viscoelastic fluids have been developed in prior investigation. Recently, there has been increased interest in the flow behavior of Yield-Power-Law (YPL) fluids which simultaneously describe yield stress and shear-thinning/thickening effect of the fluid. To our knowledge, nobody has ever studied the radial flow of YPL fluids. Drilling fluids are usually non-Newtonian and there is a lack of agreement on the way the theology of the drilling fluids influences mud losses in fractures. The fracture-well configuration in naturally fractured formations imposes radial flow geometry. The Yield-Power-Law model has been checked experimentally to accurately describe the rheological behavior of various drilling fluids. The velocity reducing nature of radial diverging flow induces a considerable change in shear rate and apparent viscosity of non-Newtonian fluids in radial geometry.

Radial Flow Pumps function by forcing water towards the outer edge of a rotating impeller, where the discharge is captured by the pump casing and the kinetic energy is converted to pressure energy before leaving the pump. When this happens, a negative pressure zone is created at the inlet of the pump chamber, which in turn draws water into the pump. These pumps can be driven by electricity (grid or solar) or directly by diesel/petrol engines, and they can be situated at ground level (suction pumps) or submersible. Radial Flow Pumps can operate over a range of depths up to around 400 metres, with flow rates up to 280,000 L/hour at lower heads. In general, they are good for higher flow requirements, as their mechanical efficiency increases with higher flows. For borehole pumps, a non-return valve is generally installed after the impellers. An important design consideration of velocity pumps is that the water flow can vary significantly with differences in head, meaning that careful design is needed to meet flow requirements. This entails creating a system curve based on the total elevation to which water must be transported plus any additional energy (frictional) losses in the pipe at different pumping velocities. Gravity can be used as an energy source for transporting water by taking advantage of differences in elevation to move water (usually via pipelines). This can occur either from elevated water sources to storage tanks and treatment facilities or directly from elevated storage facilities to supply points. It can be used in many different stages in a water system and in all phases of an emergency.

Literature Review

Zhuangguo Zhu [2022] In existing magnetorheological dampers (MRD) under impact load of launching devices, there is a contradiction between output damping force and adjustable coefficient of damping force, which is hard to cope with the harsh working conditions. To solve this problem, a new structure of three-channel MRD is proposed. The multi-physical field of the damper is simulated, and damping performance of the damper under different excitation currents and single coil control is simulated. The influence of the radial position of the damping channels on the damping performance of the damper is investigated. It is demonstrated from the results that the closer of the damping channels is, the larger the output damping force of the damper is, and the maximum output damping force is 1881 N, which is 5.69 % higher than that of the existing three-channel damper. This work provides a new idea for making full use of multi-coil and multi-channel MRD structure and improving damping performance of MRD.

Guoliang Hu [2021] A compact annular-radial-orifice flow magnetorheological (MR) valve with variable radial damping gaps was proposed, and its structure and working principle were also described. Firstly, a mathematical model of pressure drop was established as well to evaluate the dynamic performance of the proposed MR valve. Further, the experimental test rig was setup to explore the pressure drop performance and the response characteristic of the MR valve and to investigate dynamic performance of the valve-controlled cylinder system under different radial damping gaps. In addition, the damping force of the proposed MR valve-controlled cylinder system decrease with the increase of the radial damping gap. The maximum damping force can reach about 4.72 kN at the applied current of 2 A and the radial damping gap of 0.5 mm. Meanwhile, the minimum damping force can reach about 0.67 kN at the applied current of 0 A and the radial damping gap of 1.5 mm. This study clearly demonstrates that the radial damping gap of the MR valve is the key parameter which directly affects the dynamic characteristics of the valve-controlled cylinder system, and the proposed MR valve can meet the requirements of different working conditions by changing the radial damping gaps.

Radial Flow

Radial flows between parallel and concentric disks are of great interest in engineering and science. Liquid flowing radially from, or to, an inlet (or outlet) located at the center of the tray, to (or from) down comers (or inlets) at the tray periphery. Some examples of technological applications related to radial flows are air thrust bearing, aerosol impactors and electro-discharge machining. From the fundamental point of view radial flows present some unresolved issues. The concept of fully developed flow, when applied to radial diffusers, despite its use, is still not completely explored and leads to physical inconsistencies. Also unresolved are the different patterns that the flow may assume as the Reynolds number is increased. Some authors believe that from a parallel configuration the flow separates at a given critical Reynolds number and that separation occurs periodically and alternately from the diffuser walls causing the formation of a vortex street. Other works have reported flow separation under steady state regime. In the present work the attention is focused on radial flow with axial feeding. For this configuration the diffuser is fed axially through an existing orifice in one of the disks. The main motivation for this research is to understand the heat transfer in reed-type valves usually encountered in hermetic compressors. The valve system of this type of compressor comprises two valves: the suction and the discharge valve. Both valves operate according to the same principle. As the piston goes up and down the valves are submitted to different pressures on their sides causing the valves to open or to close. For a detailed study on the pressure distribution along valve reeds of hermetic compressors reference is made to.

Application Areas of Fluid Mechanics

Fluid mechanics is widely used both in everyday activities and in the design of modern engineering systems from vacuum cleaners to supersonic aircraft. Therefore, it is important to develop a good understanding of the basic principles of fluid mechanics. To begin with, fluid mechanics plays a vital role in the human body. The heart is constantly pumping blood to all parts of the human body through the arteries and veins, and the lungs are the sites of airflow in alternating directions. Needless to say, all artificial hearts, breathing machines, and dialysis systems are designed using fluid dynamics. An ordinary house is, in some respects, an exhibition hall filled with applications of fluid mechanics. The piping systems for cold water, natural gas, and sewage for an individual house and the entire city are designed primarily on the basis of fluid mechanics. The same is also true for the piping and ducting network of heating and air-conditioning systems. A refrigerator involves tubes through which the refrigerant flows, a compressor that pressurizes the refrigerant, and two heat exchangers where the refrigerant absorbs and rejects heat. Fluid mechanics plays a major role in the design of all these components. Even the operation of ordinary faucets is based on fluid mechanics. We can also see numerous applications of fluid mechanics in an automobile. All components associated with the transportation of the fuel from the fuel tank to the cylinders—the fuel line, fuel pump, fuel injectors, or carburetors—

as well as the mixing of the fuel and the air in the cylinders and the purging of combustion gases in exhaust pipes are analyzed using fluid mechanics. Fluid mechanics is also used in the design of the heating and air-conditioning system, the hydraulic brakes, the power steering, automatic transmission, and lubrication systems, the cooling system of the engine block including the radiator and the water pump, and even the tires. The sleek streamlined shape of recent model cars is the result of efforts to minimize drag by using extensive analysis of flow over surfaces.

On a broader scale, fluid mechanics plays a major part in the design and analysis of aircraft, boats, submarines, rockets, jet engines, wind turbines, biomedical devices, the cooling of electronic components, and the transportation of water, crude oil, and natural gas. It is also considered in the design of buildings, bridges, and even billboards to make sure that the structures can withstand wind loading. Numerous natural phenomena such as the rain cycle, weather patterns, the rise of ground water to the top of trees, winds, ocean waves, and currents in large water bodies are also governed by the principles of fluid mechanics

Orifices

Orifice is defined as the small opening on side or bottom of a tank through which any kind of fluid is flowing. The opening can be of circular, triangular or rectangular in cross section and they are named on the basis of shape accordingly. Orifices are mainly used for measuring the rate of fluid flow. An orifice is a plate that is inserted in a line and typically has a round hole in its centre. Orifices are used as fixed throttles that generate head loss. The head loss caused by an orifice can be used to determine the volume or mass rate of flow during flow metering. Orifices used in flow metering are preferably designed as a standard orifice. Orifices can be fitted to lines branching from piping for throttling purposes so that volumetric flow can be distributed as required.

Methodology

ANSYS FLUENT

Fluent has been used as solver for numerical experimentation work. Three popular solvers are available in ANSYS FLUENT. These are given below-

- Pressure based segregated solver (PBSS)
- Pressure based coupled solver (PBCS)
- Density based coupled solver (DBCS)

As per the ANSYS manual, among above three, first one (PBSS) has been used due to following reasons –

- It is suitable for incompressible fluid as assumption is made.
- It solves three momentum equations, continuity and energy equations. Itupdates velocity also.
- Pressure based segregated solver is applicable to wide variety of problems.
- It requires less memory or lower RAM requirements.

This solver is also quick than other solvers. As pressure based segregated solver is chosen for pressure velocity coupling, default algorithm in FLUENT, Semi Implicit Method for Pressure Linked Equation (SIMPLE) will be used. It is most robust algorithm among others available in FLUENT like SIMPLEC, PISO and coupled.

FLOW STEPS USED FOR SOLUTION OF PROBLEM USING FLUENT TOOL OF ANSYS

For the solution of problem related to engineering, FLUENT tool of ANSYS software can be used and following are the steps necessary for solution:

- Analysis at the primary stage
- Mathematical design
- Creating a mesh
- Formation of the setup physically
- Arithmetic result
- Proof & authentication

Pre-Analysis: In pre analysis as per the problem or components design requirements observe the real physical conditions and finalize the boundary condition. The selections of boundary conditions are very important step to achieve an accurate simulation results.

Geometry: As the volumetric analysis has to be done that is why 3D model was imported for simulation. ANSYS design modeler software i.e. ANSYS WORKBENCH may use to create 3D geometry.

Meshing: Outcome of Simulation is dependent upon meshing. It is an important part of simulation. The quality of mesh decides the outcome of simulation. The divergence in outcome of simulation may occur due to poor quality of meshing.

Physical setup: In physical setup step, user can give inputs and boundary conditions such as physical and mechanical properties and select the procedure as per problems of simulations in the ANSYS software.

Numerical solutions: As per the equation of solver of ANSYS software, it run the simulations and after sometimes it gives results in the form of graphs and tables.

Verification and validation: After getting results, observer can run video and can verify and validate the results with actual problems and compare with theoretical simulation results with actuals experiments. The complete process of simulation has also been explained with the help of flow Chart given in figure:

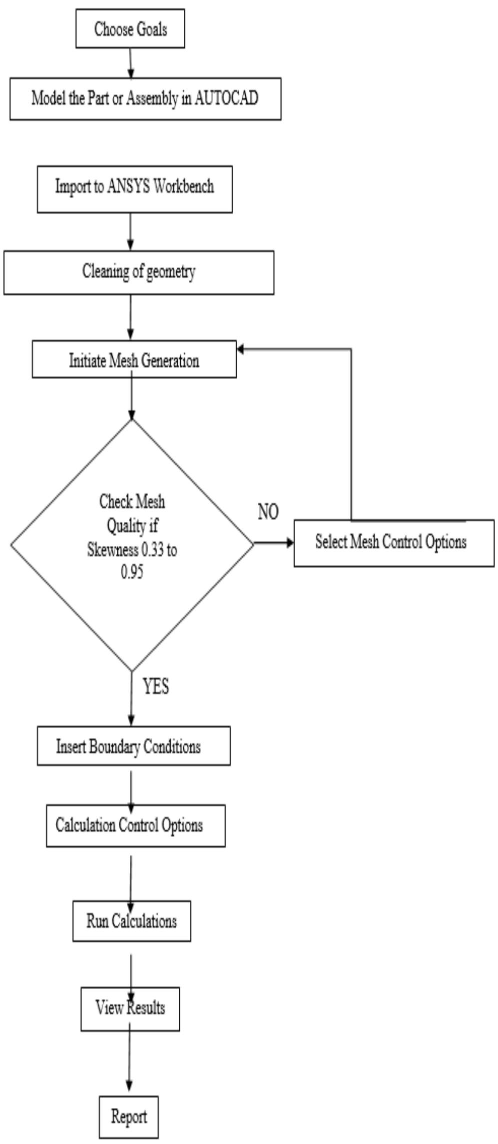


Figure: Flow Chart for fluid flow analysis

Results and Discussions

SIMULATION FOR SQUARE INLET PLENUM SHAPE OF MICRO CHANNEL

To create a simulation for thermal flow and flow of fluid ANSYS software is used.

Consider a steady state fluid flowing through a square inlet shape of micro channel of uniform transverse section as shown in Figure(a). The rectangular micro channel has a length 35 mm and width 0.8 mm in the flow direction and a height of 3.2 mm, by varying width of square inlet plenum of side 10, 20 and 30 mm respectively. The fluid enters into the channel through plenum from centre at varying pressure, temperature and velocity according radial channel conditions. Figure(b) shows the bottom view where constant heat flux is provided.

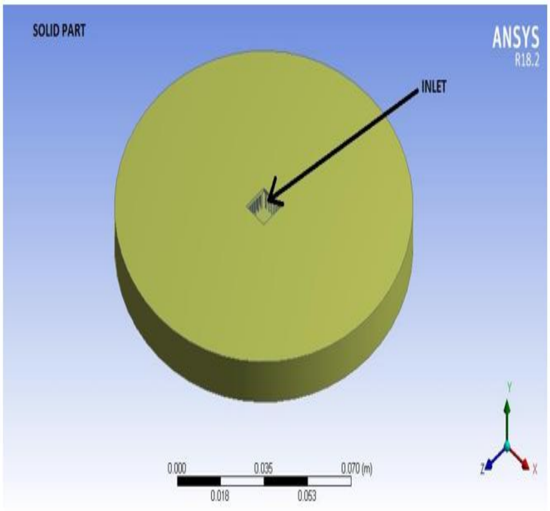
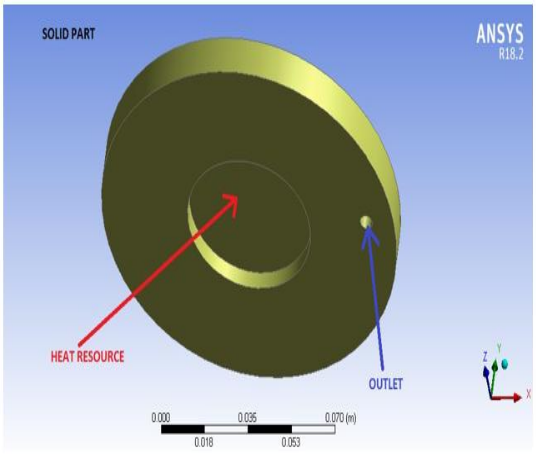


Figure (a): Top side of micro channel shows rectangular inlet shape



Figure(b): Bottom side of micro channel shows circular outlet shape

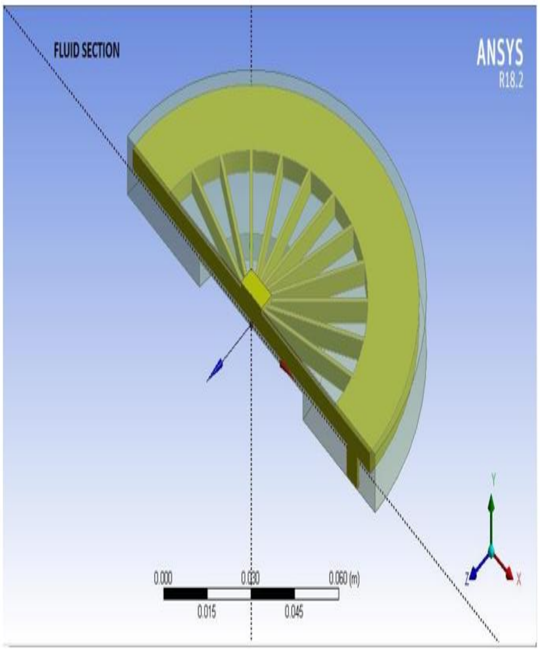


Figure: Sectional for square inlet plenum view

Figure shows the sectional view of the microchannel with square inlet plenum.

SIMULATIONS FOR CIRCULAR INLET PLENUM SHAPE OF MICRO CHANNEL

ANSYS software is used for flow of fluid and simulation of heat flow in micro channel having the shape of inlet plenum as circular. Consider a steady state flow of fluid through a rectangular inlet shape of micro channel of uniform cross section as presented in Figure (a and b) The rectangular micro channel has a length 35 mm and width 0.8 mm in the flow direction and a height of 3.2 mm. Radial micro channel has varying diameter of circular inlet plenum of diameter 10, 20 and 30 mm respectively. The fluid exhausts into the radiator chamber at varying pressure, temperature and velocity according to radial channel conditions. Figure depicts the half sectional view of the microchannel with circular inlet plenum.

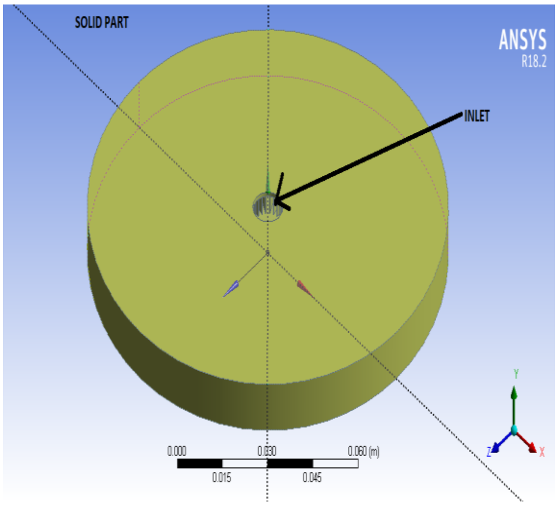
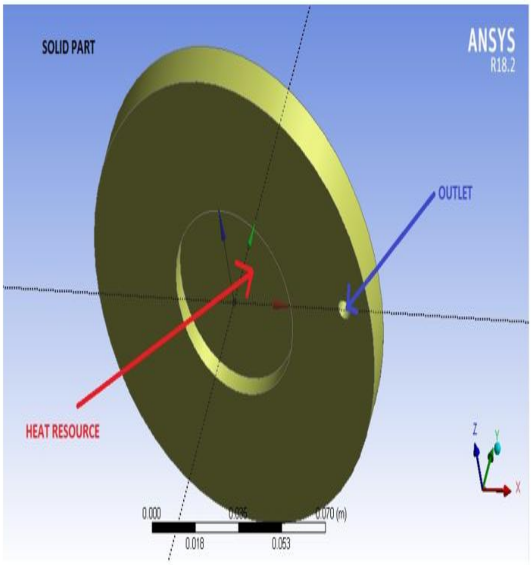


Figure (a): Top side of micro



channel shows circular inlet shape

Figure (b): Bottom side of micro channel shows circular outlet shape

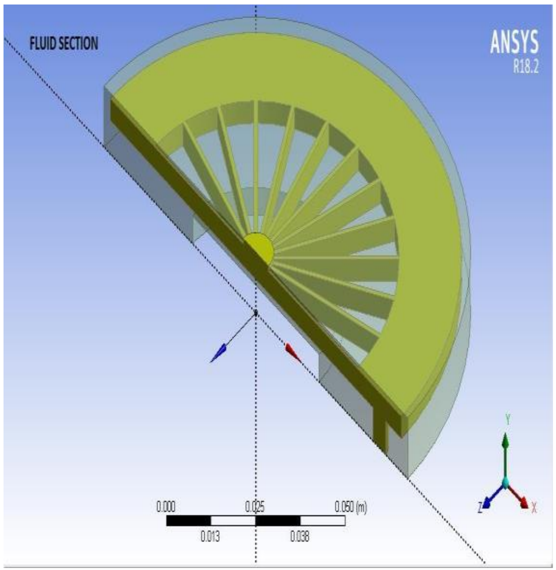


Figure : Sectional view of Circular inlet plenum

Conclusion

The flow modelling study conducted confirms the appropriateness of the structural approach to expanding the application of the directly controlled valves to the high flow rate operational domains. Simulation based analysis shows small pressure losses in the suggested valve design along with low steady flow torques. The nonlinear opening area promises to improve controllability in actuator speed control. The developed valve design presents the feasible and sound structure of control valves for high flow rate power hydraulic systems. Its construction ensures improved controllability and maintainability due to the utilization of a stepper motor and cartridge assembly. Furthermore, grouping four proposed valves in the independent metering arrangement also opens the opportunity to advance the efficiency of the fluid power systems by the implementation of energy regeneration and recuperation in the regulation of motors. All in all, noted factors prove the effectiveness of the proposed approach for designing of unconventional throttling orifices in flow control valves of high flow rate fluid power systems

References

1. *Guoliang Hu (2021),"Dynamic Performance Analysis of a Compact Annular-Radial-Orifice Flow Magnetorheological Valve and Its Application in the Valve Controlled Cylinder System", Actuators, ISSNno:2076-0825, Vol.10(5), Pages.104. DOI:10.3390/a ct10050104.*
2. *Xu, Peng (2008),"An analysis of the radial flow in the heterogeneous porous media based on fractal and constructal tree networks," Physica A: Statistical Mechanics and its Applications, Elsevier, ISSNno:0378-4371,Vol.387(26), pages6471-6483. DOI:10.1 016/j.physa.2008.08.021.*
3. *Zhuangguo Zhu (2022),"Damping performance optimization of the magnetorheo-logical damper with three parallel channels", Structures, ISSNno:2352-0124, Vol.44, Pages.1962-1973. https://doi.org/10.1016/j.istruc.2022.08.110.*
4. *Roland Glowinski (2011),"On the Numerical Simulation of Viscoplastic Fluid Flow", Handbook of Numerical Analysis, ISSN no:1875-5445, Vol.16, DOI:10.1016/B978-0-444-53047-9.00006-x.*
5. *Sabine Attinger (2021),"The extended generalized radial flow model and effective conductivity for truncated power law variograms", Advances in Water Resources, ISSN no: 0309-1708, Vol.156, https://doi.org/10.1016/j.advwatres.2021.104027.*
6. *Yang Zhou (2019),"Analytical solution for one-dimensional radial flow caused by line source in porous medium with threshold pressure gradient", Applied Mathematical Modelling, ISSNno:0307-904X, Vol.67, Pages.151-158. https://doi.org/1 0.1016/j.apm.2018.10.024.*
7. *Yang Zhou (2019),"Analytical solution for one-dimensional radial flow caused by line source in porous medium with threshold pressure gradient", Applied Mathematical Modelling, ISSNno:0307-904X, Vol.67, Pages.151-158. https://doi.org/1 0.1016/j.apm.2018.10.024.*
8. *Yehia A. Eldrainy (2009),"Investigation of Radial Swirler Effect on Flow Pattern inside a Gas Turbine Combustor", Modern Applied Science, Canadian Center of Science and Education, ISSNno:1913-1852,Vol.3(5), Pages 1-21.*
9. *Yue Hao (2017),"Energy Performance and Radial Force of a Mixed-Flow Pump with Symmetrical and Unsymmetrical Tip Clearances", Energies, ISSNno:1996-1073, vol. 10(1), pages 1-13.*
10. *Zhu, Sipeng (2015)," "Modeling and extrapolating mass flow characteristics of a radial turbocharger turbine," Energy, Elsevier, ISSNno:0360-5442, Vol. 87(C), pages 628-637. DOI: 10.1016/j.energy.2015.05.032.*