ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)

Volume 11, Issue 10, October 2024

Flow Analysis in Restriction Orifice for Various Spacing and Number of Stages to Achieve Maximum Pressure Drop using CFD

[1] Dharmender Tiwari, [2] T Vijaya Kumar, [3] Sheikh Nasiruddin

[1][2] Department of Civil and Environmental Engineering, Delhi Technological University, Delhi, India
[3] Department of Mechanical Engineering, Indian Institute of Technology Jammu, Jammu, India
Email: [1] mr.dharmendertiwari@gmail.com, [2] tvijayakumar@dtu.ac.in, [3] sheikh.nasiruddin@iitjammu.ac.in

Abstract— ***Many processing industries involves handling of very high pressure and after processing very high pressure drop is required before releasing it into environment. However, this pressure drop should be gradual in nature, so that it does not cause any atmospheric and mechanical disturbance. In this regard, restriction plate plays an important role for achieving gradual pressure drop by adopting various arrangements within the pipe. However, limited literature does not give idea about what should be the best arrangement of restriction orifice within pipe in terms of spacing and number of stages. The objective of this study is to perform flow analysis using Ansys software using CFD technique in terms of pressure drop by arranging restriction orifice at fixed and then varying spacing of 1D, 1.5D, 2D, 2.5D, 3D for 5, 6 and 7 number of stages.***

Index Terms— ***CFD, Fluid Flow, Pressure Drop, Restriction Orifice, Spacing of Orifice***

I. INTRODUCTION

In various industrial processes, the pressure within a system often needs to be reduced to meet downstream requirements or to safely release it into the atmosphere. The pressure reduction must occur without causing excessive noise, vibrations, or other adverse effects. Typically, this reduction is achieved using control valves. However, when dealing with significant pressure drops, a single control valve can face challenges such as clogging, excessive noise, and vibrations. These issues can ultimately lead to the valve's structural failure. In such cases, a more gradual pressure reduction is necessary, and one effective solution is the use of restriction orifice assemblies.

Restriction orifices offer several advantages over traditional control valves. They contain no moving parts, making them silent and free from vibration. The number of stages within the orifice assembly can be adjusted to achieve the desired pressure drop. Additionally, restriction orifices are durable and cost-effective, making them a preferred choice in high-pressure applications.

Despite their widespread use, there is a lack of comprehensive literature on the design and optimal arrangement of restriction orifices. To address this gap and promote the development of indigenous expertise in this field, the objective of this study is to perform a flow analysis using ANSYS software and Computational Fluid Dynamics (CFD). The analysis focuses on the pressure drop achieved by arranging restriction orifices at varying spacings of 1D, 1.5D, 2D, 2.5D, and 3D, for configurations with 5, 6, and 7 stages.

The restriction orifice plate assembly consists of multiple orifice plates arranged sequentially within a pipeline or welded onto the pipe's surface. As fluid passes through each orifice, pressure drops gradually occur due to flow interference. The design of the assembly must minimize noise and vibration while ensuring the desired pressure reduction. Understanding the flow behavior through each stage is critical, especially when dealing with gases, where the effects of compression become more pronounced. As the pressure decreases, the flow rate through each stage increases, potentially causing the orifice to clog despite the constant mass flow rate. This means the behavior of multi-stage systems can differ significantly from single-stage systems.

The primary objective of this research is to perform a detailed flow analysis using ANSYS software to identify the optimal arrangement of restriction orifices. The study explores various spacing configurations and stage numbers to maximize pressure drop efficiency.

Several studies have explored different aspects of multi-stage orifice design. Haimin et al. [1] investigated multistage orifices and concluded that these configurations can effectively reduce flow-induced noise and vibrations while maintaining pressure drop efficiency. Sanghani et al. [2] conducted numerical studies highlighting the impact of orifice plate geometry (concentric, eccentric, segmental, and sectoral) on pressure drop. Abdulrazaq et al. [3] used the realizable k-ε eddy viscosity turbulence model to study fluid flow through multistage orifices and found that the position of the vena contracta downstream of the second orifice is influenced by the distance between orifices, with the first orifice diameter playing a key role. Hou et al. [5] developed a numerical model using the RNG k-e turbulent model to study the effects of relative angles between inner and outer porous shrouded holes, orifice plate thickness, and hole diameter. They determined that a 180° relative angle provides the highest decompression with minimal turbulence. They also found that plate thickness has a smaller impact on throttling, while smaller hole diameters lead to a more uniform pressure distribution.

This study builds upon these findings by investigating the effect of varying the spacing and number of stages in restriction orifice assemblies to optimize pressure drop performance.

II. METHODOLOGY

Governing Equations:

The Navier-Stokes equations for steady-state turbulent flows are solved using a numerical solver in their discretized form. To accurately model the flow, it is essential to ensure the conservation of continuity, momentum, and energy. These governing equations describe the conservation of mass and momentum in a steady-state flow and are expressed as follows:

$$\frac{\partial}{\partial x_j}(\rho u_i) = S_m$$

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial P_i}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F$$

where S_m is the source term,
P_i is the static pressure,
ρg_i is the body force due to gravity and
F is the external body force.

In the above equation. The term τ_{ij} represents the stress tensor and is computed as:

$$\tau_{ij} = \mu\left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right] - \frac{2}{3}\mu\delta_{ij}\frac{\partial u_k}{\partial x_k}$$

Turbulent flow can be broken down into two parts: a time-averaged value and a fluctuating part, represented by u=ū+u'. In the Navier-Stokes equation, the instantaneous velocity field is replaced by decomposed quantities and time averaging is applied to obtain the Reynolds-averaged Navier-Stokes equation (RANS). The equation then takes the following form:

$$\frac{\partial}{\partial x_j}(\rho \bar{u}_i \bar{u}_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \bar{u}_i}{\partial x_j}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right)$$

The last term on the right-hand side of the above equation is the Reynolds stress term. This term cannot be solved directly for the closure solution and therefore needs to be modeled. The Boussinesq hypothesis is used to correlate the Reynolds stress with the mean velocity gradient as:

$$-\rho\overline{u_i' u_j'} = \mu_t\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) - \frac{2}{3}\delta_{ij}\left(\rho k + \mu_t\frac{\partial \bar{u}_i}{\partial x_i}\right)$$

Selection of Turbulent model

The selection of an appropriate turbulence model is based on the guidelines provided in the **ANSYS Fluent User's Guide**, Chapter 12 (Modeling Turbulence). After reviewing the fluid dynamics problem and evaluating the criteria outlined in the manual, the **SST k-ω** (Shear Stress Transport) model was identified as the most suitable for this analysis.

The SST k-ω model is particularly well-suited for capturing the behavior of both the boundary layer and free-stream turbulence. It combines the advantages of the k-ε model in the far field with the k-ω model's accuracy near walls, making it the optimal choice for modeling complex flow behavior and pressure drops in multi-stage orifice assemblies.

III. NUMERICAL MODELING

Geometry

Numerical modelling should aim to represent the results as accurately as possible, reflecting the actual flow conditions. In this study, we will examine eccentric orifices with different spacing and different numbers of stages. We will use ANSYS Design Modular to model these orifices.

Design parameters:

1. Diameter of main pipe = 0.0254 m
2. Thickness of restriction orifice = 0.0035m
3. Upstream length =35D
4. Downstream length = 60D
5. Spacing between restriction orifice = 1D, 1.5D, 2D, 2.5D and 3D
6. Number of stages = 5, 6 and 7

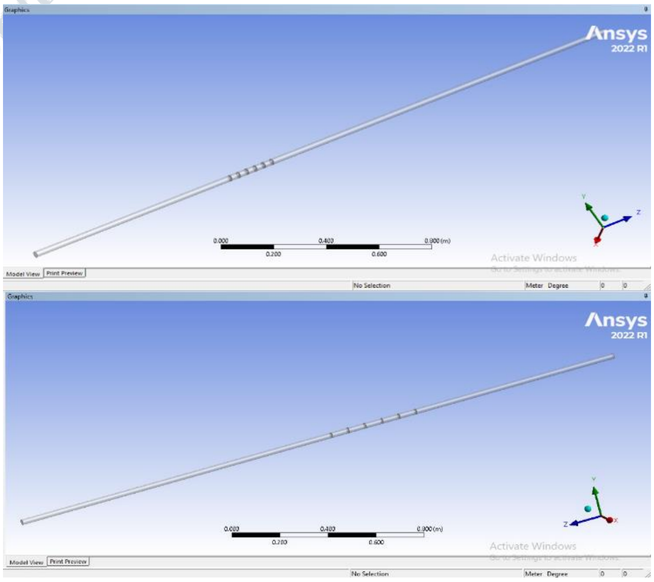

Fig. 1 Geometry of restriction orifice at spacing 1.5D and 3D for 6 Number of stages

Mesh generation

The domain is divided into small discrete cells and the flow conditions within each cell are determined by solving the Navier-Stokes equation. For achieving precise and accurate results, the most crucial factor in mesh generation is sound accuracy. High-quality mesh results in better outcomes, but in the meantime it consumes significant hours of CPU. Nevertheless, use of coarse mesh leads to less precise outcomes. As a result, the optimal mesh size was chosen to minimize CPU usage while still maintaining the accuracy of the results.

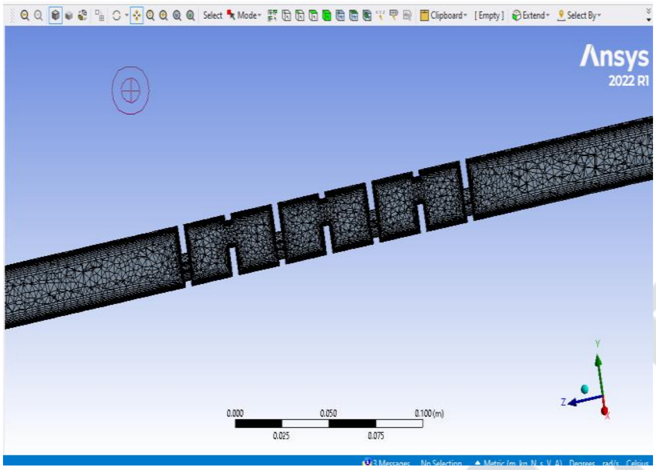

Fig. 2 Tetrahedral meshing section

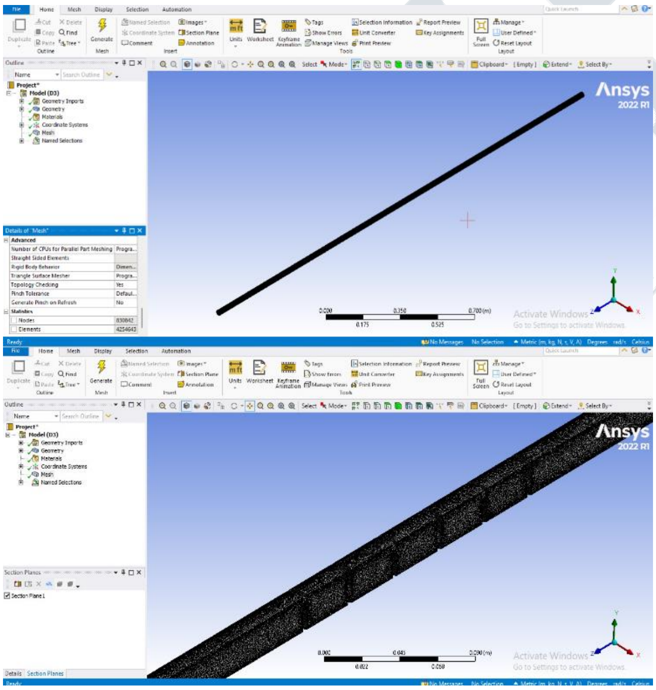

Fig. 3 Tetrahedral meshing

Furthermore, tetrahedral mesh was converted into polyhedral mesh for better accuracy and less CPU use.

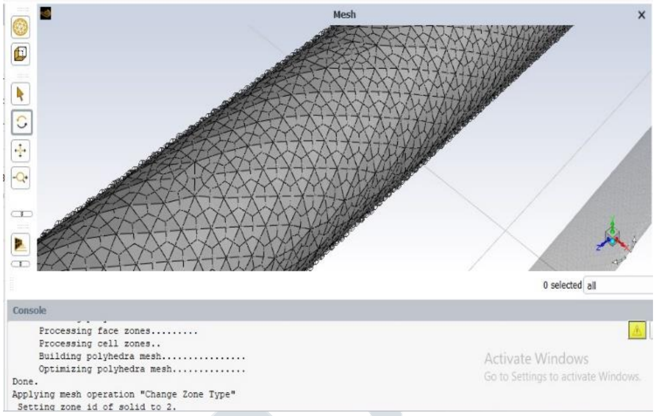

Fig. 4 Polyhedral meshing

Boundary condition

Tackling of fluid problem, boundary conditions are helpful to set limitations. These conditions provide streamline path for fluid to follow from which solution can be obtained. At the inlet, the pressure inlet is specified that is fixed 60 Mpa, and the flow is directed through the annular space between the orifices and the pipe. The selection of hydraulic diameter and turbulent intensity as parameters for specifying turbulent quantities has been made. Because the entire study focuses on pressure analysis, a pressure-based solver with a simple algorithm has been chosen. Air has been utilized as a medium to guide the movement of fluids within pipes. The residuals were set to 10-6 degrees to achieve a convergent solution.

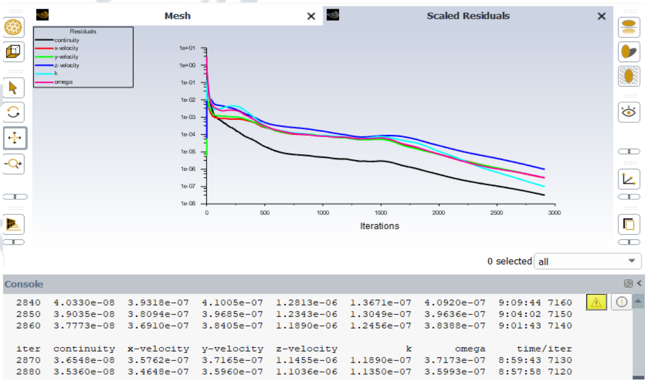

Fig. 5 Residuals

IV. RESULT AND DISCUSSIONS

Based on boundary condition, Post processing module of Ansys has been utilized for tracing pressure contours. As shown in fig 5. Results in terms of pressure have been obtained for 5, 6 and 7 no stages of restriction orifice by varying spacing 1D, 1.5D, 2D, 2.5D and 3D between them.

IFERP
Explore Your Research Journey.

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)

Volume 11, Issue 10, October 2024

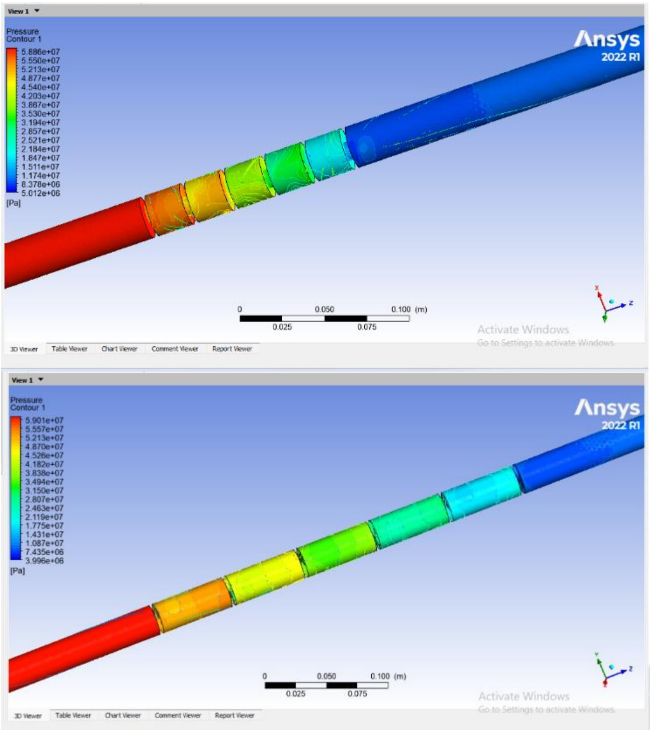

Fig. 6 Pressure contours for 1D and 3D

Results in terms of pressure have been obtained for 5, 6 and 7 no stages of restriction orifice by varying spacing 1D, 1.5D, 2D, 2.5D and 3D between them are as follows:

Table. 1 Pressure distributions of 5 No of stages

Spacing	Orifice 1	Orifice 2	Orifice 3	Orifice 4	Orifice 5/Outlet
1D	51.2	41.59	29.58	19.68	9.88
1.5D	52.13	40.84	29.13	20.08	9.78
2D	51.20	41.11	30.51	20.35	9.86
2.5D	51.54	41.46	30.81	20.48	9.96
3D	51.28	41.00	30.15	20.21	9.74

Table. 2 Pressure distributions of 6 No of stages

Spacing	Orifice 1	Orifice 2	Orifice 3	Orifice 4	Orifice 5	Orifice 6 /Outlet
1D	55.31	47.40	37.97	29.37	19.04	8.03
1.5D	51.51	43.37	35.08	26.43	17.27	8.36
2D	51.71	43.76	35.68	26.73	18.24	8.94
2.5D	52.52	44.97	36.67	27.63	18.35	8.89
3D	51.88	43.17	34.02	25.26	16.77	8.00

Table. 3 Pressure distribution of 6 No of stages

Spacing	Orifice 1	Orifice 2	Orifice 3	Orifice 4	Orifice 5	Orifice 6	Orifice 7 /Outlet
1D	54.30	45.80	39.66	31.20	22.62	13.50	5.99
1.5D	52.13	44.84	37.13	29.08	20.59	11.97	3.69
2D	51.20	45.11	32.51	29.35	20.86	12.67	4.07
2.5D	52.54	45.46	37.81	29.48	20.96	12.84	4.10
3D	52.28	45.00	36.15	28.21	19.74	11.77	3.31

Graphical representaion of obtained results in form of pressure analysis are as follows:

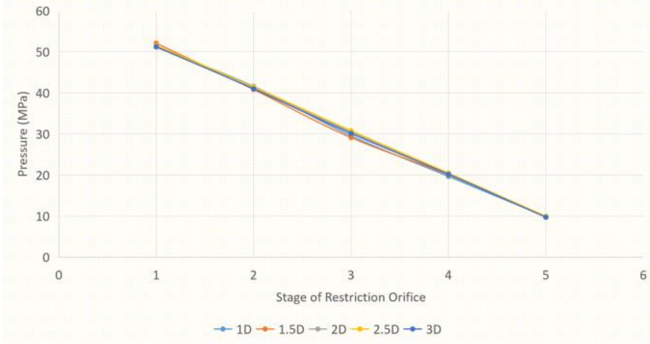

Fig. 7 Pressure analysis for 5 No of stages

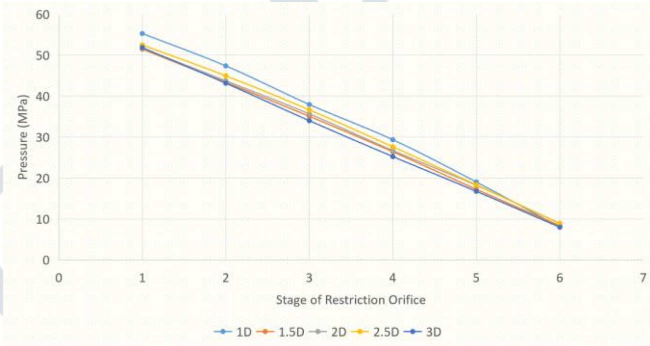

Fig. 8 Pressure analysis for 6 No of stages

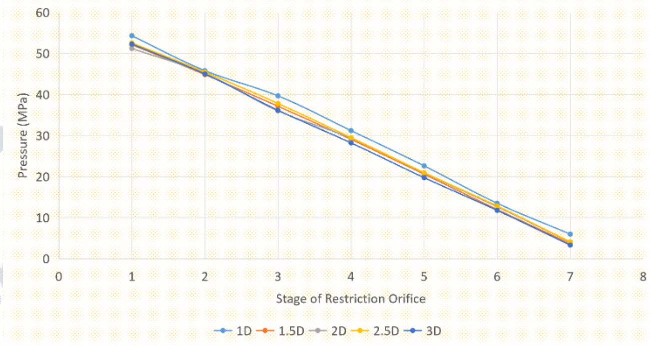

Fig. 9 Pressure analysis for 7 No of stages

Results have been achieved for various distances between the restriction orifices. The pressure inlet remains constant to obtain precise information about pressure changes by adjusting the distance between the restriction orifices for each stage of restriction. It has been noted that flow reversal is prevalent and continues until 750 to 1500 iterations, when orifices are placed at close proximity. However, when the spacing between restrictions kept between 2.5D and 3D the flow reversal decrease and the solution converges easily, but it does not have a significant impact on the pressure drop.

V. CONCLUSION

Based on results obtained, it has been observed that results obtained are comparatively better when restriction orifices are placed at spacing 1.5D with comparatively less degree of turbulence and require less iteration for solution convergence.

It has been observed that flow reversal is comparatively less when orifices are placed at spacing 1.5D.

REFERENCES

[1] Haimin, W., Shujuan, X., Qingyi, S., Caimin, Z., Hao, L., & Eryun, C. (2013), "Experiment study on pressure drop of a multistage letdown orifice tube," Nuclear Engineering and Design,265,633–638, https://doi.org/10.1016/j.nucengdes.2013.09.014

[2] Sanghani, Chirag & Jayani, D. (2016), "Comparative Analysis of Different Orifice Geometries for Pressure Drop," International Journal of Science Technology & Engineering. 2. 2349-784.

[3] Araoye, A. A., Badr, H. M., & Ahmed, W. H. (2016), "Dynamic behaviour of flow through multi-stage restricting orifices," International Conference of Fluid Flow, Heat and Mass Transfer. https://doi.org/10.11159/ffhmt16.161

[4] Zahariea, D. (2016), "Numerical analysis of eccentric orifice plate using ANSYS fluent software," IOP Conference Series: Materials Science and Engineering, 161, 012041. https://doi.org/10.1088/1757-899x/161/1/012041

[5] Hou, C.-wei, Qian, J.-yuan, Chen, F.-qiang, Jiang, W.-kang, & Jin, Z.-jiang. (2018), "Parametric analysis on throttling components of multi-stage high pressure reducing valve," Applied Thermal Engineering, 128, 1238–1248 https://doi.org/10.1016/j.applthermaleng.2017.09.081.

[6] Gao, J., Wu, F., Tang, J., & Geng, Z. (2021), "A method of two-stage pressure control based on multistage orifices,"Applied Sciences, 11(2), 589. https://doi.org/10.3390/app11020589

[7] Vemulapalli, S., & Venkata, S. K. (2022), "Parametric analysis of orifice plates on measurement of flow: A review," Ain Shams Engineering Journal, 13(3).