Volume 10, Issue 8, pp. 34-42, August 2026

International Journal of Scientific Engineering and Science
ISSN (Online): 2456-7361

Design of Orifice Plate as Differential Pressure Flow Meter in Anti Scale Rig Test System

Yoga Andreansyah[1], Komarudin[2], Tri Noviyanto[3], Taufik Nur Isman[4], Ir. Bodhimula Satyajati, M.T.[5]
[1]Mechanical Engineering, Dian Nusantara University, Jakarta, Indonesai-11470
[2]Mechanical Engineering, Dian Nusantara University, Jakarta, Indonesai-11470
[3]Mechanical Engineering, Pancasila University, Jakarta, Indonesai-12630
[4]Mechanical Engineering, Dian Nusantara University, Jakarta, Indonesai-11470
[5]Mechanical Engineering, Dian Nusantara University, Jakarta, Indonesai-11470

Abstract— *Accurate fluid flow measurement is required in an anti-scale rig test system to monitor flow characteristics and support scale-formation testing in piping systems. This study aimed to design a concentric orifice plate as the primary element of a differential-pressure flow meter for the anti-scale rig test system at West Sumatra, with reference to ISO 5167. The system operating conditions consisted of an internal pipe diameter of 42.8 mm, an upstream pressure of 14 bar, a fluid temperature of 210°C, a mass flow rate of 2 tonnes per hour, and a differential-pressure transmitter target range of 100 inH$_2$O. The research method included analytical calculations using the Bernoulli equation, continuity equation, Reynolds number, discharge coefficient, flow losses, and mechanical strength analysis of the plate. The resulting geometry was developed using AutoCAD and numerically analysed using ANSYS Fluent. The design resulted in an orifice bore diameter of 12.66 mm, a beta ratio of 0.296, a discharge coefficient of 0.61, a total plate thickness of 6 mm, an edge thickness of 3 mm, and SS 316L as the plate material. The resulting stress was 4.993 MPa, with a safety factor of 20.428, indicating that the plate was mechanically safe under the operating conditions. The analytical calculation produced a differential pressure of 57,526 Pa. Meanwhile, the simulation produced average upstream and downstream pressures of 1,399,805 Pa and 1,342,474 Pa, respectively, resulting in a differential pressure of 57,331.25 Pa. The difference between the analytical and simulation results was 194.75 Pa, corresponding to 0.34%. The maximum flow velocity of approximately 11.58–11.98 m/s occurred near the vena contracta. The proposed design satisfied the basic geometric requirements of ISO 5167 and was mechanically safe. However, the resulting differential pressure of 230.16 inH$_2$O exceeded the transmitter range of 100 inH$_2$O. Therefore, the orifice bore diameter should be further optimised, or a transmitter with a wider measurement range should be selected.*

Keywords— *Orifice plate, differential pressure, ISO 5167, ANSYS Fluent, anti-scale rig test.*

I. INTRODUCTION

Accurate fluid flow measurement plays an important role in various industrial applications, including piping systems, chemical processes, and experimental test facilities. The ability to measure flow characteristics precisely is essential to ensure operational reliability, improve process control, and evaluate fluid behavior under specific operating conditions. One of the most widely applied flow measurement devices is the orifice plate flow meter, which operates based on the differential pressure principle caused by flow restriction through a precisely designed opening installed inside a pipe.

In an anti-scale rig test system, reliable flow measurement is required to monitor fluid characteristics and evaluate the influence of flow conditions on scale formation phenomena within piping systems. The orifice plate is considered a suitable measurement element due to its simple construction, high durability, relatively low manufacturing cost, and compatibility with differential pressure-based measurement systems. However, the performance of an orifice plate strongly depends on its geometric parameters, such as bore diameter, beta ratio, plate thickness, and installation configuration. Improper design parameters may lead to inaccurate differential pressure generation, resulting in unreliable flow measurement and incorrect interpretation of fluid behavior.

The anti-scale rig test system at West Sumatra, experienced limitations related to the existing orifice plate design, where the generated pressure distribution was not fully compatible with the requirements of the ISO 5167 standard. This condition affected the accuracy of differential pressure measurement and reduced the reliability of the instrumentation system in detecting flow variations. Therefore, a properly designed orifice plate is required to ensure that the generated differential pressure signal accurately represents the actual flow conditions within the test system.

Several studies have demonstrated that the ISO 5167 standard provides a standardized approach for designing differential pressure flow meters, particularly concerning beta ratio selection, discharge coefficient determination, and dimensional requirements of concentric orifice plates. However, the application of this standard requires optimization based on specific operating conditions, including fluid properties, pressure conditions, and piping geometry. Therefore, a design evaluation and validation process are necessary to obtain an orifice plate configuration that satisfies both measurement accuracy and mechanical reliability.

This study aims to design a concentric orifice plate as a differential pressure flow meter for the anti-scale rig test system at West Sumatra based on the ISO 5167 standard. The research focuses on determining the optimal design parameters, including bore diameter, beta ratio, plate thickness, and material specification, to produce an accurate

International Journal of Scientific Engineering and Science
Volume 10, Issue 8, pp. 34-42, August 2026 ISSN (Online): 2456-7361

and representative differential pressure response under operating conditions. The proposed design is further evaluated to ensure its capability in measuring flow variations and supporting reliable operation of the anti-scale rig test system.

II. METHODOLOGY

A. Research Procedure

The research procedure was carried out through several stages, starting from data collection, orifice plate design, analytical calculation, numerical simulation, and result validation. The research flow is shown in Fig 1.

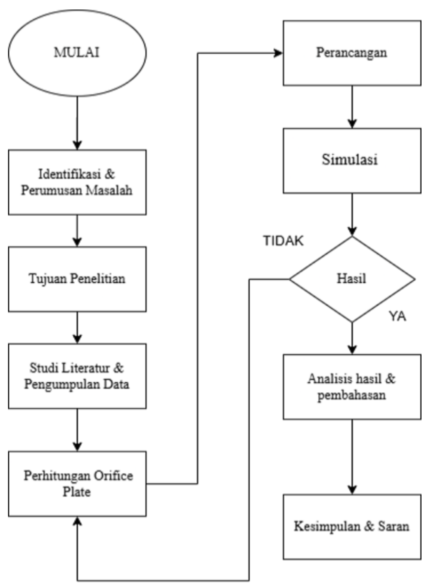

Fig. 1. Research Flowchart

B. Rig test Anti-Scale System Specification

The following data shows the instrumentation piping diagram for the anti-scale test rig, in which an orifice plate will be installed in the center between the flanges and in the upstream section of the piping system, before the fluid enters the orifice plate, with a total upstream run length of 300 mm. For complete specifications of the anti-scale test rig, see Table 1.

TABLE I. Rigtest Anti-Scale System Specifications

Parameters	Value	Unit
Inlet diameter (D)	42,8	mm
upstream (P_1)	14	Bar
Flow	2	Ton/h
Materials *orifice plate*	SS 316	-
Allowable stress SS 316L (σ_{allow})	102	MPa
Viscosity	0,129	cP
Temperature Fluida	210	°C
Specify Gravity (SG)	0,95	-
Differential Range Target	100	InH2O
Expansion Factor	1	Y
Coefficient Discharge	0,61	Cd

As shown in Figure 2, this is a Schematic diagram of one of the pipe lines in the Rig test anti-scale system.

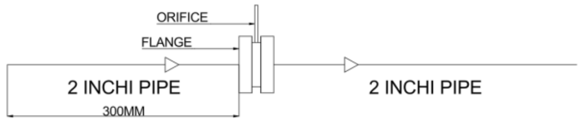

Fig. 2. Schematic diagram

C. Ansys Simulation of Orifice Plate Setup and Design

The orifice plate design was developed based on the calculation results and ISO 5167 standard requirements for differential pressure flow measurement. The designed orifice was applied to the anti-scale rig test system using a concentric orifice plate configuration. The main design parameters included pipe diameter, orifice bore diameter, beta ratio, plate thickness, and material selection.

The orifice plate was designed using 3D CAD modeling software by considering the geometric characteristics of the plate, including the bore diameter, plate thickness, and bevel configuration. The selected plate material was SS316L due to its corrosion resistance and suitability for high-pressure fluid applications. The final design was prepared as a flow restriction component installed between the upstream and downstream pipe sections to generate measurable differential pressure.

The numerical simulation was performed using ANSYS Fluent based on the Computational Fluid Dynamics (CFD) approach to evaluate the flow characteristics through the designed orifice plate. The computational domain consisted of an upstream pipe section, orifice plate region, and downstream pipe section to capture the pressure and velocity changes caused by the flow restriction.

The simulation was conducted under the operating conditions of the anti-scale rig test system. The working fluid properties were defined according to the experimental conditions, with a density of 1045 kg/m³ and the maximum flow rate of 2 ton/h. The inlet boundary condition was specified as a mass flow inlet, while the outlet boundary condition was defined as pressure outlet.

The mesh was generated to obtain an appropriate resolution around the orifice region, where significant changes in velocity and pressure occur. A turbulence model was applied to represent the turbulent flow behavior generated by the sudden contraction and expansion of the flow area. The main parameters evaluated from the simulation were pressure distribution, velocity contour, and differential pressure between the upstream and downstream sections.

The simulation results were then compared with analytical calculations to validate the performance of the designed orifice plate. The comparison focused on the differential pressure value generated by the orifice as the primary parameter for evaluating the accuracy of the flow measurement design.

Design calculations for a concentric orifice plate as a differential flow meter in the anti-scale rig test system, based on the system specification data presented in Chapter B (Table 1) and calculation steps that follow the ISO 5167 standard. The calculations were performed in a step-by-step, iterative manner, beginning with the determination of flow parameters,

IJSES
International Journal of Scientific Engineering and Science
Volume 10, Issue 8, pp. 34-42, August 2026 ISSN (Online): 2456-7361

the diameter ratio (β), the Reynolds number (Re), the discharge coefficient (Cd), the pressure difference (ΔP), and ending with the minimum thickness of the orifice plate.

A. Conversion of Actual Data Units

Initial operating data—including pipe diameter, pressure, mass flow rate, and fluid viscosity—are converted to the International System of Units (SI). This conversion is necessary because all equations for calculating energy losses and flow characteristics use SI units.

Inner pipe diameter :

$$D = 42{,}85\ mm = 0{,}0428\ m$$

Upstream operating pressure :

$$P_1 = 14\ bar = 1400000\ Pa$$

Maximum and minimum mass flow rates :

$$\dot{m} = 2\ ton/jam = 2000kg/jam = 0{,}5556kg/s$$

Viscosity :

$$\mu = 0{,}129\ cP = 0{,}000129Pa.s$$

Differential Pressure Range :

$$100\ inH2O = 24900\ Pa$$

B. Volumetric Flow Rate Calculation

The mass flow rate is then converted to a volumetric flow rate to determine the actual flow capacity through the pipe cross-section. Since the available data is specific gravity, the fluid's density is calculated as follows:

$$\rho = SG\ x\ \rho_{brine}$$

notes:

ρ : density of the fluid (kg/m³)
SG : specific gravity
P_{brine} : 1100kg/m³

$$\rho = 0{,}95\ x\ 1100$$
$$\rho = 1045\ kg/m^3$$

C. Volumetric Flow Rate Calculation

To Calculate Volumetric Flow Rate, the mass flow rate is converted to volumetric flow rate using :

$$Q = \frac{\dot{m}}{\rho}$$

$$Q = \frac{0{,}5556}{1045} = 0{,}0005316\ m^3/s$$

The calculation results are used to determine flow characteristics and serve as key parameters in energy loss calculations.

D. Calculation of Pipe Cross-Section Area

Next, calculate the cross-sectional area of the pipe using the formula :

$$A_1 = \frac{\pi}{4} D^2$$

$$A_1 = \frac{3{,}14}{4} 0{,}0428^2$$

$$A_1 = 0{,}001438\ m^2$$

E. Calculation of Fluid Flow Velocity in a Pipe

because flow velocity is used to determine flow characteristics and serves as the primary parameter in energy loss calculations.

The flow velocity is calculated using the continuity equation:

$$V = \frac{Q}{A_1}$$

Calculating for the flow:

$$V = \frac{Q}{A_1}$$

$$V = \frac{0{,}0005316}{0{,}001438} = 0{,}369\ m/s$$

TABLE II. Results of fluid flow velocity calculations

Parameter	Q	A_1	V = m/s
Flow	0,0005316	0,001438	0,369

F. Reynolds Number Calculation

The Reynolds number is used to determine the type of flow. To calculate the Reynolds number, use the following formula:

$$Re = \frac{\rho V D}{\mu}$$

Calculating based on maximum flow:

$$Re = \frac{2416\ x\ 0{,}369\ x\ 0{,}0428}{0{,}000129}$$

$$Re = 295785$$

Based on the calculation results, a Reynolds number of 295785 was obtained. This value is above the transition threshold of 4,000, so the flow is classified as turbulent. The turbulent conditions indicate that the effects of friction and local disturbances caused by the installation of the orifice must be taken into account in the design.

G. Calculation of the Friction Coefficient

Since the flow is turbulent and the Reynolds number is in the range 4000<Re<10^5, the friction factor is calculated using the Blasius equation to calculate the friction factor.

$$f = \frac{0{,}316}{Re^{0{,}25}}$$

Calculating the coefficient of friction:

$$f = \frac{0{,}316}{295785^{0{,}25}}$$

$$f = 0{,}0761$$

H. Calculation of Major Losses

Major losses are energy losses caused by friction along the pipe walls. In this study, major losses were calculated based on the length of the flow path before the orifice plate was installed. They were calculated using the following formula:

$$h_f = f \frac{L}{D} \frac{V^2}{2g}$$

Note :

L (Pipe length) = 327.5 mm = 0.3275 m

$$h_f = 0{,}0761\ x\ \frac{0{,}3275}{0{,}0428} x \frac{0{,}369^2}{2(9{,}81)}$$

$$h_f = 0{,}004041\ m$$

Pressure was relieved due to major losses :

$$\Delta p\ major = 2416\ x\ 9{,}81\ x\ 0{,}004041$$
$$\Delta p\ major = 95{,}775\ Pa$$

The pressure loss due to friction is relatively small because the length of the pipe being analyzed is short compared to the

Volume 10, Issue 8, pp. 34-42, August 2026

International Journal of Scientific Engineering and Science
ISSN (Online): 2456-7361

pressure loss caused by the geometric changes in the orifice plate.

I. Determination of the Beta Ratio

The beta ratio is the ratio of the orifice diameter to the inner diameter of the pipe. The beta value determines the sensitivity of the differential pressure measurement. A value that is too small results in high permanent pressure loss, while a value that is too large causes low differential pressure, making it difficult for the transmitter to read.

The beta ratio is calculated using the following formula:

$$\beta_0 = \left[1 + \left(Y \cdot \frac{C}{S_M}\right)^2\right]^{-0,25}$$

Before that, let's first find the value of Sm using the formula:

$$S_M = \frac{\dot{m}}{\left(A_1\sqrt{2 \cdot \rho \cdot \Delta P}\right)}$$

$$S_M = \frac{0,5556}{\left(0,001438\sqrt{2 \cdot 1045 \cdot 24900}\right)}$$

$$S_M = 0,05355$$

Let's move on to finding the beta ratio.:

$$\beta_0 = \left[1 + \left(1 \cdot \frac{0,61}{0,053558}\right)^2\right]^{-0,25}$$

$$\beta_0 = \left[1 + \left(1 \cdot \frac{0,61}{0,53558}\right)^2\right]^{-0,25}$$

$$\beta_0 = 0,296$$

Next, determine the orifice bore diameter using the calculated beta ratio data:

$$\beta_0 = 0,296$$

$$d = \frac{\beta}{D}$$

$$d = 0,296 \times 42,8$$

$$d = 12.66\ mm = 0,01266m$$

So, the diameter of the borehole orifice is 12.66 m or 0.01266 m

J. Calculation of minor losses in an orifice plate

Minor losses are calculated from the upstream components downstream of the orifice plate:

$$K_{total} = K_{orifice}$$

The minor loss orifice coefficient based on pipe velocity can be calculated using the following formula:

$$K_{orifice} = \frac{1 - \beta^4}{C_d^2 \beta^4}$$

$$K_{orifice} = \frac{1 - 0,296^4}{0,61^2 x 0,296^4}$$

$$K_{orifice} = 347,4$$

followed by the calculation of minor losses :

$$h_m = K\frac{V^2}{2g}$$

$$h_m = 347,4\frac{0,369^2}{2(9,81)}$$

$$h_m = 2,410\ m$$

Pressure loss due to minor losses:

$$\Delta\rho\ minor = 2416\ x\ 9,81\ x\ 2,410$$

The total system pressure loss is the combination of major losses due to friction and minor losses due to the installation of the orifice plate.

$$\Delta\rho\ minor = 57119\ Pa$$

The results show that pressure loss due to the orifice plate is far more significant than loss due to pipe friction. This is consistent with the characteristics of a differential flow meter, which utilizes pressure changes caused by flow restriction.

K. Calculation total losses

To calculate total losses, the results of major losses and minor losses are combined.

Total head loss :

$$h_{Ltotal} = h_f + h_m$$

$$h_{Ltotal} = 0,004041 + 2,410$$

$$h_{Ltotal} = 2,414\ m$$

Total pressure loss :

$$\Delta\rho_{total} = \Delta\rho_{major} + \Delta\rho_{minor}$$

$$\Delta\rho_{total} = 95,775 + 57119$$

$$\Delta\rho_{total} = 57215\ Pa$$

The discharge coefficient is used to correct for the difference between the theoretical discharge and the actual discharge due to losses caused by flow contraction, friction, and turbulence.

In the previous calculation, Cd = 0.61.

Therefore, the equation for the actual discharge becomes :

$$\Delta P = \frac{\rho}{2}\left(\frac{Q}{C_d A_2}\right)^2 (1 - \beta^4)$$

Since the value of C_d has been used in the differential pressure calculation, the ΔP result for the beta variation is already corrected for the discharge coefficient.

L. Calculation of Differential Pressure

Once the bore diameter and discharge coefficient have been determined, the differential pressure value is calculated using the orifice meter equation to ensure that the design is capable of producing a differential pressure within the transmitter's range.

Continue calculating the cross-sectional area of the bore orifice

Note :

$$\beta = 0,296$$

$$d = 12,66\ mm = 0,01266\ m$$

$$A_2 = \frac{\pi}{4}d^2$$

$$A_2 = \frac{3,14}{4}0,01266^2$$

$$A_2 = 0,0001258\ m^2$$

Since the cross-sectional area of the borehole orifice has already been calculated, we can now proceed to calculate the pressure:

$$\Delta P = \frac{\rho}{2}\left(\frac{Q}{C_d A_2}\right)^2 (1 - \beta^4)$$

Volume 10, Issue 8, pp. 34-42, August 2026
International Journal of Scientific Engineering and Science
ISSN (Online): 2456-7361

$$\Delta P = \frac{2416}{2}\left(\frac{0{,}0005316}{0{,}61 \times 0{,}0001258}\right)^2 (1 - 0{,}296^4)$$
$$\Delta P = 57526\ Pa$$

M. Calculation of Downstream Pressure

The downstream pressure drop is calculated to determine the pressure conditions after passing through the orifice plate and to ensure that the operating pressure remains within the system limits.

$$P_2 = P_1 - \Delta P$$
$$P_2 = 1400000 - 57526$$
$$P_2 = 1342474\ Pa$$
$$P_2 = 13{,}42\ bar$$

N. Calculation of Orifice Plate Thickness

The theoretical wall thickness calculated using R.W. Miller's equation yields a minimum value of 0.5 mm. However, this value only satisfies the hydraulic requirements and does not yet account for the mechanical strength needed to withstand an operating pressure of 15 bar and a temperature of 240°C.

Note :

$$0{,}005D \le e \le 0{,}02D$$
$$D = 42{,}8mm$$
$$e_{min} = 0{,}005 \times 42{,}8$$
$$e_{min} = 0{,}214\ mm$$
$$e_{max} = 0{,}02 \times 42{,}8$$
$$e_{max} = 0{,}856\ mm$$

For the total thickness of the plate:

$$e \le E \le 0{,}05D$$
$$E_{max} = 0{,}05 \times 42{,}8$$
$$E_{max} = 2{,}14\ mm$$

To determine the minimum thickness, use R.W. Miller's equation, which is:

$$\boxed{e \le E \le 2{,}14\ mm}$$

$$t_{min} = D_{pipe} \cdot \sqrt{(0{,}681 - 0{,}651\beta) \cdot \frac{\Delta P}{Y}}$$

To make it more mechanically logical, the pressure used is the system operating pressure of 1400000 bar:

$$\Delta P = Poperasi = 1400000\ Pa$$
$$\beta = 0{,}296$$
$$\Delta P = 1400000\ Pa = 1{,}4\ Mpa$$
$$Y = 102\ MPa$$

$$t_{min} = 42{,}8\sqrt{(0{,}681 - 0{,}651(0{,}296)) \cdot \frac{1{,}4}{102}}$$
$$t_{min} = 3{,}5\ mm$$

However, mechanically speaking, a 3.5 mm thick plate would not be able to withstand a pressure of 14 bar at a temperature of 210°C; therefore, for safety reasons, a 6 mm thick plate is selected to ensure it can withstand a pressure of 14 bar at a temperature of 210°C. Additionally, the edges of the holes through which the fluid passes must be chamfered at an angle of 30°–45°, leaving a straight-edge thickness (e) of only 0.5–3 mm to ensure the plate remains structurally strong, while hydraulically, the flow still treats the plate as thin in accordance with ISO 5167 standards.

Therefore, the thickness selected is 6 mm. Stress Due to Fluid Pressure

Membrane stress analysis :

D : plate diameter (42,8mm = 0,0428m)

T : 6mm = 0,006m

$$\sigma = \frac{1{,}4x0{,}0428}{2\,(0{,}006)}$$
$$\sigma = 4{,}993\ Mpa$$

Comparison of Stress with the Material's Allowable Stress.

Allowable stress for SS316L at 210°C

$$\sigma_{allow} = 102Mpa$$

Safety Factor

$$SF = \frac{\sigma_{allow}}{\sigma}$$
$$SF = \frac{102}{4{,}993}$$
$$SF = 20{,}428$$
$$20{,}428\ MPa < 102MPa$$

Therefore, the stress caused by fluid pressure remains well below the allowable stress limit for SS316L material. With a safety factor of 20,428 Mpa the 6-mm-thick plate is deemed mechanically safe to withstand the system's operating pressure of 14 bar at a temperature of 210°C.

TABLE III. Summary of the Count Results

Parameter	Symbol	Result	Unit
Inlet pipe	D	42,8	mm
Beta Ratio	β	0,296	-
Reynolds Number	Re	295785	-
Diameter *orifice bore*	d	12,66	mm
Coefficient discharge	Cd	0,61	-
Differential pressure	ΔP	57526	Pa
Major losses	ΔPmajor	95,775	Pa
Minor losses after the orifice	ΔPminor	57119	Pa
Total *pressure loss*	ΔPloss,total	57215	Pa
Total plate thickness (E)	t	6	mm
Thickness of the hole side (*e*)	t	3	mm
Materials	-	SS 316L	-
Safety Factor	SF	20,428	-

O. Design of Orifice Plate

This design was created using AutoCAD 3D software based on the data obtained from the calculations above.

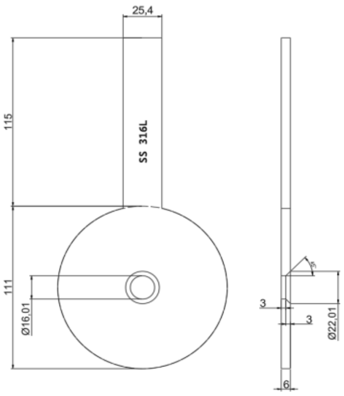

Fig. 3. Design Orifice Plate

International Journal of Scientific Engineering and Science
Volume 10, Issue 8, pp. 34-42, August 2026
ISSN (Online): 2456-7361

III. Simulation Results Using Ansys

a) Numerical Modeling

A three-dimensional Computational Fluid Dynamics (CFD) simulation of steady-state flow was performed using a pressure-based solver. The SST $k-\omega$ turbulence model was selected for its ability to accurately represent pressure gradients, flow separation, and the formation of fluid recirculation in the orifice area. The working fluid is assumed to be brine 1.045kg/m³, 0,000129 kg/(m·s) in a domain without mass sources, a porous model, or reference frame motion. The boundary conditions are set using an inlet total pressure of 1,400,000 Pa and an outlet static pressure of 1,342,474 Pa. The turbulence intensity is set to 5% with a hydraulic diameter of 0.0428 m. Stationary wall and no-slip boundary conditions are applied to all pipe and orifice surfaces. The geometric model used is shown in Figure 4.2, while a complete summary of the simulation parameters is presented in Table 4.

The simulation geometry parameters are shown in the following table IV.

TABLE IV CFD Simulation Geometry Parameters

Parameters	Units
Simulation Dimension	3D
Solver **Type**	Pressure-based
Flow conditions	Steady
Model turbulensi	SST (k-\omega)
Density of a fluid	1.045 kg/m³
Dynamic viscosity	0,000129 kg/(m·s)
total inlet **pressure**	1.400.000 Pa
statis outlet **pressure**	1.342.474 Pa
Intensitas turbulensi	5%
Hydraulic diameter	0,0428 m
Condition of the walls	Stationary, no slip

As shown in Figure 4, this is the geometric model for the orifice plate.

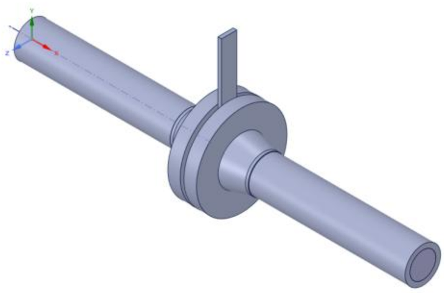

Fig. 4. Geometry of the orifice plate simulation model

b) Mesh Generation Results

The domain discretization (meshing) process produced 1,012,869 cells, 2,221,638 faces, and 292,709 nodes using a mixed-cell element type. Mesh quality was evaluated based on a minimum orthogonal quality value of 0.1103 and a maximum aspect ratio of 91.95 (summarized in Table 5). These parameters indicate that the generated mesh meets computational criteria and is suitable for use. However, elements with the lowest quality near the sharp edges of the orifice require special attention, given that this area has extreme velocity and pressure gradients.

TABLE V Mesh Parameter

Mesh Parameter	***Results***
Cells total	1.012.869
faces **total**	2.221.638
nodes **total**	292.709
Minimum orthogonal quality	0,1103
Maximum aspect ratio	91,95
Elemen type	Mixed cell

c) Numerical Solving Methods

The pressure-velocity interaction was solved using a pseudo-time-based coupled method. The discretization of momentum, turbulent kinetic energy, and specific dissipation rate was performed using a second-order upwind scheme to optimize the accuracy of the velocity and pressure distributions, particularly in the constriction area and downstream of the orifice. Meanwhile, the pressure discretization used a standard scheme. The simulation was run for up to 300 iterations.

d) Simulation Convergence

After 300 iterations, the solutions to the continuity and momentum equations (velocities along the x, y, and z axes) have satisfied the specified convergence criteria 10^{-3}. On the other hand, the residuals for turbulent kinetic energy k and the specific dissipation rate ω are still slightly above the convergence threshold, at 0.002113 and 0.002396, respectively (summarized in Table 6).

TABLE VI Simulation Convergence

Equation	Final Residual	Criteria
Continuity	0,0004548	0,001
X-velocity	0,0003887	0,001
Y-velocity	0,0001324	0,001
Z-velocity	0,0001318	0,001
Turbulent kinetic energy, (k)	0,002113	0,001
Specific dissipation rate, (ω)	0,002396	0,001

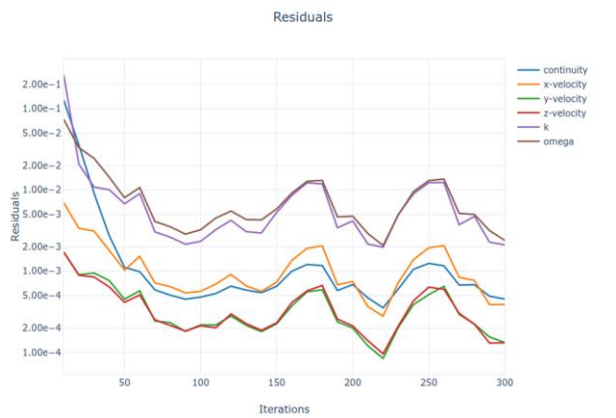

Fig. 5. ANSYS Fluent Residual Convergence Plot

Based on the trend shown in Figure 5, the fluctuations in the residual values of k and ω are a normal phenomenon resulting from flow separation and recirculation in the downstream region after the fluid passes through the orifice. Given that the primary parameters (continuity and momentum)

Volume 10, Issue 8, pp. 34-42, August 2026

International Journal of Scientific Engineering and Science
ISSN (Online): 2456-7361

have achieved good convergence, the results of this simulation can be considered valid overall and suitable for further analysis.

e) Pressure Distribution from the Simulation Results

The area-weighted average results show that the average pressure on the upstream side is:

$$P_{Upstream} = 1.399.805\ Pa$$

Meanwhile, the average pressure on the downstream side is:

$$P_{downstream} = 1.342.474\ Pa$$

For differential pressure, the simulation results are:

$$\Delta P_{CFD} = P_{upstream} - P_{downstream}$$

$$\Delta P_{CFD} = 1.399.805 - 1.342.474$$

$$\Delta P_{CFD} = 57.331,25\ Pa$$

The following is a more concise version adapted to the style of academic journals: Based on the computational results at the 300th iteration, the mass balance value was recorded as very low, namely 0.0001468 kg/s, indicating that the principle of mass conservation is satisfied within the flow domain. The upstream pressure P_1 and downstream pressure P_2 were 1,399,805 Pa and 1,342,474 Pa, respectively. These conditions result in a differential pressure of 57,331.25 Pa (equivalent to 0.5733 bar or 230.16 inH2O, as summarized in Table 10

TABLE VII Mesh Balance Parameters

Parameter	Symbol	Result
upstream	(P_1)	1.399.805 Pa
downstream	(P_2)	1.342.474 Pa
Differential pressure	(ΔP)	57.331,25 Pa
Differential pressure	(ΔP)	0,5733 bar
Differential pressure	(ΔP)	230,16 inH_2O
Mass balance	–	0,0001468 kg/s

Based on the monitoring graph, the ΔP profile fluctuated during the initial phase of the computation. However, the value began to remain constant (stable) at around 57.3 kPa from the 40th to the 50th iteration and remained so until the end of the iteration. The same stability was also exhibited by the downstream pressure profile, which remained constant at 1,342,474 Pa.

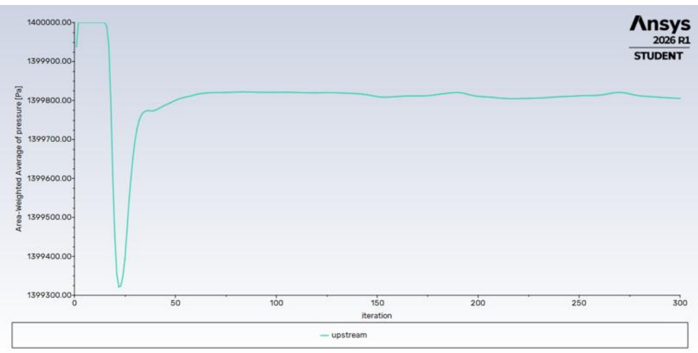

Fig. 6. Upstream Graph

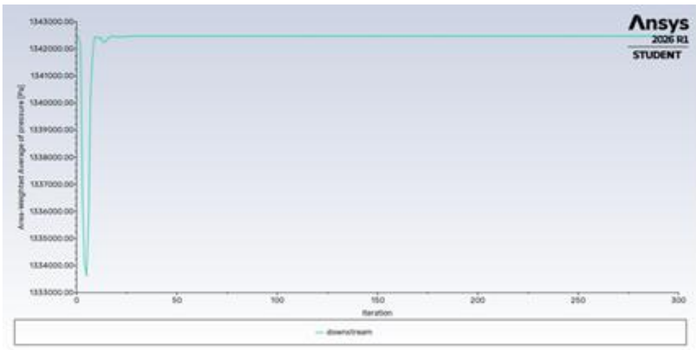

Fig. 7. Downstream Graph

f) Calculation Results and Simulation Results

Previous analytical calculations yielded a differential pressure of:

$$\Delta P_{Perhitungan} = 57.331,25\ Pa$$

Meanwhile, the ANSYS Fluent simulation results showed:

$$\Delta P_{CFD} = 57.331,25\ Pa$$

The difference between the two results is:

$$Selisih = 57.526 - 57.331,25$$

$$Selisih = 194,75\ Pa$$

The percentage difference is:

$$Error = \frac{57331,25 - 57526}{57526} 100\%$$

$$Error = 0,34\%$$

Metode	Differential pressure	Different
Calculatiom	57.526 Pa	–
ANSYS Fluent	57.331,25 Pa	194,75 Pa
Persentase error	–	0,34%

The difference of 0.34% indicates that the simulation results are very close to the analytical results. This small difference may be influenced by the numerical solution, non-uniform velocity distribution, fluid viscosity, flow separation, turbulence, and the three-dimensional shape of the orifice hole.

g) Analysis of Conformity with the Transmitter Range

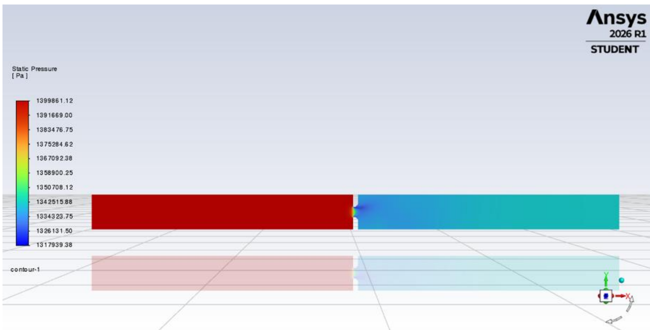

Fig. 8. Static Pressure

Based on the distribution of static pressure contours in Figure 4.6, the fluid pressure appears stable and high on the upstream side, with a maximum value of 1,399,861.12 Pa. As the fluid passes through the orifice, the pressure drops sharply to a minimum of 1,317,939.38 Pa. This drastic drop is caused by the narrowing of the cross-sectional area, which triggers an acceleration in flow velocity.

International Journal of Scientific Engineering and Science
Volume 10, Issue 8, pp. 34-42, August 2026 ISSN (Online): 2456-7361

After passing through the plate, the flow enters a low-pressure zone on the downstream side before gradually recovering pressure as the flow continues. The characteristics of this visual pressure gradient have theoretically validated the operating principle of the orifice plate, namely as a differential-pressure-based flow measurement instrument.

h) Velocity Contour Analysis

Based on the contour and velocity vector visualizations in Figure 4.7, the flow profile on the upstream side appears relatively low. The fluid velocity increases dramatically upon entering the orifice bore due to the narrowing of the cross-sectional area. The velocity profile does not reach its maximum value inside the orifice bore but peaks shortly after passing through it, with a maximum velocity of approximately 11.58 m/s.

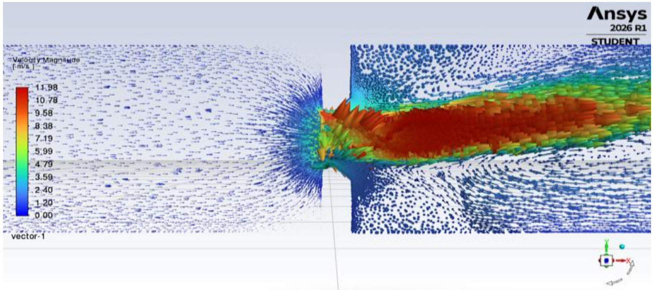

Fig. 9. Velocity Visuals

This concentration of maximum velocity on the downstream side clearly confirms the formation of a flow jet and the vena contract phenomenon. After passing this point, the flow expands to once again fill the cross-sectional area of the main pipe, followed by a gradual decrease in velocity. This velocity profile distribution visually demonstrates the conversion of pressure energy into kinetic energy due to the restriction of the cross-sectional area.

i) Recapitulation Simulation Results

Parameters	Results	Note
Number of cells	1.012.869	Mesh 3D
Minimum orthogonal quality	0,1103	There are some low-quality cells
Maximum aspect ratio	91,95	Please note
Model Turbulence	SST (k-\omega)	Steady
Iterations	300	The simulation has been run
Residual continuity	0,0004548	Konvergen
Residual (k)	0,002113	konvergen
Residual (ώ)	0,002396	konvergen
upstream	1.399.805 Pa	Area-weighted average
Downstream Pressure, Differential Pressure, CFD	1.342.474 Pa	Area-weighted average
Downstream Pressure, Differential Pressure, CFD	57.331,25 Pa	0,5733 bar
Differential pressure CFD	230,16 inH₂O	Exceeding the target transmitter's range
Differential pressure analitis	57.526 Pa	*Results calculatiom*
Error analitis–CFD	0,34%	A small difference
Maximum contour speed	11,5759 m/s	Around the vena contracta
Maximum velocity vector	11,98 m/s	Sekitar downstream orifice
Mass balance	0,0001468 kg/s	Mass difference

IV. CONCLUSION

Based on the results of the design and computational fluid dynamics (CFD) simulations using ANSYS Fluent on a concentric orifice plate in an anti-scale rig test system (ISO 5167 standard), the following conclusions can be drawn:

1. Geometrical and Mechanical Feasibility**:** The orifice plate design (SS 316L material, 6 mm total thickness) with a bore diameter of 12.66 mm yields a beta ratio (β) of 0.296 and a discharge coefficient (C_d) of 0.61, completely fulfilling the ISO 5167 standard. Under extreme operating conditions (14 bar, 210°C), the design is proven to be mechanically safe, generating a maximum stress of only 4.993 MPa and a high safety factor of 20.428
2. Numerical Performance Validation: A highly accurate numerical agreement is achieved between the analytical calculations and the CFD simulation, showing an error margin of only 0.34%. The simulated differential pressure is recorded at 57,331.25 Pa, closely aligning with the analytical result of 57,526 Pa. The pressure drop within the system is predominantly governed by minor losses caused by sudden contraction and turbulence around the orifice region
3. Flow Phenomenon Characteristics: Flow visualizations validate the conversion of pressure energy into kinetic energy, in accordance with Bernoulli's principle. The fluid pressure drops sharply and locally at the orifice area, inversely correlating with the fluid velocity, which surges to 11.58–11.98 m/s and concentrates at the vena contracta region on the downstream side
4. Evaluation and Recommendations: Although the design is theoretically and mechanically valid, the resulting differential pressure (230.16 inH_2O) exceeds the targeted operational capacity of the pressure transmitter (maximum 100 inH_2O). Therefore, further optimization is required by enlarging the orifice bore diameter to reduce the differential pressure value so it falls within the transmitter's measurement range. Additionally, experimental validation is recommended to confirm the numerical computation results.

V. RECOMENDATIONS

Based on the research results and the limitations identified, the following recommendations can be made:

1. In the simulation, attention must be paid to mesh quality, particularly around the sharp edges of the orifice, the vena contracta region, and the downstream side. This is necessary because the minimum orthogonal quality value is still 0.1103 and the maximum aspect ratio is 91.95
2. The number of iterations must be adjusted until the residual turbulent kinetic energy and specific dissipation rate fall below the specified criteria.
3. Fluid property data—specifically the density and viscosity of the brine—must be verified to match the operating temperature of 210°C and used consistently throughout all analytical calculations and simulations.
4. The final design should be experimentally tested on an anti-scale rig. Testing should be conducted at several flow rate variations to compare actual flow rate, differential pressure, discharge coefficient, and simulation results.
5. The manufacturing process must maintain the sharpness of the inlet edge, edge thickness, bevel angle, hole position, and

plate surface in accordance with ISO 5167 specifications so that the actual flow characteristics do not differ significantly from the design results.

ACKNOWLEDGMENT

The authors would like to express their sincere gratitude to the Mechanical Engineering Program, Faculty of Engineering and Informatics, Dian Nusantara University, for the academic guidance and research facilities provided. The authors would also like to thank all parties who contributed to the completion of this research, particularly during the process of designing an orifice plate as a differential pressure flow meter in an anti-scale rig test system.

REFERENCES

[1] Akhyan, A., & Fadhli. (2023). Pengaruh variasi tebal orifice dan bilangan reynolds terhadap penurunan tekanan pada entrance region. Jurnal Quantum, 4(2), 75–82

[2] Annisa, Carina, Prasetyo, & July, D. (2024). Pembelajaran Autocad 2021. vi + 90.

[3] Arman, R., Mahyoedin, Y., Kaidir, K., & Desilpa, N. (2019). Studi Aliran Air Pada Ball Valve Dan Butterfly Valve Menggunakan Metode *Simulation* Computational Fluid Dynamics. Jurnal Kajian Teknik Mesin, 4(1), 38–49.

[4] Chirende, B., Li, J. Q., & Vheremu, W. (2019). Application of finite element analysis in modeling of bionic harrowing discs. Biomimetics, 4(3), 1–11.

[5] Eko Julianto , Febriyan , Gunarto , Fuazen, E. S. (2022). Jurnal Dinamis. Jurnal Dinamis, 19(2), 28–34.

[6] Gerhart, P. M. (2020). Munson's Fluid Mechanics, 8th Edition, Global ed. 716.

[7] Komarudin, Yanuarta Ilham Partama, I. S. (2022). PERANCANGAN POWER UNIT SISTEM HIDROLIK Universitas Dian Nusantara , Jakarta Institut Sains dan Teknologi Nasional , Jakarta terus berkembang . Penerapan teknologi sistem hidrolik yang berkembang saat ini adalah Hydraulic Power Unit (HPU) . Dalam hal ini. Jurnal Tera, 2(1), 34–47.

[8] Mubarok, M. H., Zarrouk, S. J., & Cater, J. E. (n.d.). Pressure Differential Devices for Measuring Total Mass Flow Rate and Enthalpy in Two-phase Geothermal Pipelines.

[9] Noviasari, A. N., Pradana, M. R., & Budi, S. H. (2026). Smart Monitoring Pada Orifice Flow Meter Menggunakan Mikrokontroler. Prosiding Seminar Nasional Teknologi Energi Dan Mineral, 5(1), 1181–1190.

[10] Ritonga, F. S., Purba, A. S., Simanjuntak, J. K. R. S., Fadilah, N., & Amalia, M. (2025). Dimensional check of weld neck flange based on ASME B16.5. International Journal of Mechanical Computational and Manufacturing Research, 14(2), 49–56.

[11] Sherza, J. S. (2024). Theoretical Investigation of The Major and Minor Losses in Pipes and Fittings. Babylonian Journal of Mechanical Engineering, 2024, 12–18.