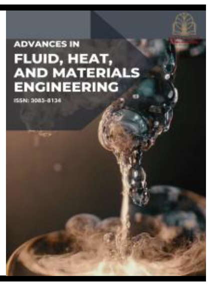
KARYA ILHAM PUBLISHING
CONNECTING MINDS, BRIDGING IDEAS, INSPIRING READERS

Advances in Fluid, Heat and Materials Engineering

Journal homepage:
https://karyailham.com.my/index.php/afhme/index
ISSN: 3083-8134



Analysis of the Jet Impingement Cooling on a Heated Flat Plate

Ngu Pei Kang[1], Ishkrizat Taib[1,*]

[1] Department of Mechanical Engineering, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia

ARTICLE INFO

Article history:
Received 25 April 2026
Received in revised form 10 May 2026
Accepted 3 June 2026
Available online 30 June 2026

ABSTRACT

Jet impingement cooling is frequently used in thermal engineering because it can reach high heat transfer rates. Previous study has reported that mesh quality and resolution have a significant impact on the accuracy of Computational Fluid Dynamics (CFD) simulations especially in regions with large gradients such as the stagnation zone. In this case, improper mesh selection may cause the incorrect prediction of flow behaviour. So, the purpose of this study is to investigate the effect of mesh resolution on the numerical modelling of jet impingement cooling on a heated flat plate. The simulation was conducted in ANSYS Fluent using three-dimensional model consisting of a vertical nozzle and a flat plate. The model then is meshed into three different sizes namely, coarse, medium and fine mesh. The simulation is then run under steady-state conditions with the k-ε turbulence model. The results were analysed for velocity distribution, pressure contours, streamline patterns, and velocity profiles along the jet centerline. From the result, Grid Independence Test shows that maximum velocity increase from 7.355m/s for coarse mesh to 7.439 m/s for medium mesh. Only small difference is observed between medium and fine mesh. Medium mesh contains 114,802 elements is selected as optimal mesh due to the balance between accuracy and computational efficiency. The flow behaviour shows the formation of a stagnation region at the point of impingement then followed by radial spreading of the flow throughout the surface. In conclusion, the medium mesh is sufficient for effectively capturing flow properties while keeping computational efficiency. This study confirms the importance of mesh selection in CFD simulations beside showing the effects on the accuracy of jet impingement cooling analysis.

Keywords:
Jet impingement cooling; computational fluid dynamics; grid independence test; mesh resolution; velocity distribution; stagnation region

1. Introduction

Jet impingement cooling has gained popularity in thermal engineering due to its capability to generate extremely high local heat transfer rates [1,2]. This technique involves projecting a high-velocity fluid jet onto a heated surface. This results in a stagnation zone with the highest heat transfer coefficient [1]. Because of its high cooling capability, jet impingement is frequently used in engineering systems such as electronic cooling, gas turbine blade cooling and industrial thermal management operations [3,4]. Zuckerman and Lior [1] reported that jet impingement is one of the

* *Corresponding author.*
E-mail address: iszat@uthm.edu.my

https://doi.org/10.37934/afhme.9.1.19a

most effective strategies for increasing convective heat transfer due to intense fluid mixing and boundary layer disruption.

Previous research has demonstrated that the performance of jet impingement cooling is influenced by several parameters. This includes Reynolds number, nozzle configuration, and jet-to-surface distance [1-3]. With the evolution of Computational Fluid Dynamics (CFD), numerical simulation has become an effective method for analyzing flow behavior and heat transfer characteristics in such systems. Roache [5] showed that CFD can accurately forecast velocity distribution, pressure contours, and Nusselt number fluctuations in jet impingement systems under a variety of operating situations. Similarly, much numerical research has been done to explore the influence of geometry and flow characteristics on thermal performance. These stress the need for precise modeling methodologies in CFD simulations.

However, one important feature of CFD simulations is the numerical results accuracy and dependability. These are highly dependent on mesh quality and resolution. A coarse mesh may fail to capture critical flow characteristics such as high velocity gradients near the stagnation area. Meanwhile, an extremely fine mesh increases computational costs without significantly improving accuracy [6]. As a result, a mesh independence analysis is required to confirm that grid size has no effect on simulation outcomes. Previous CFD studies emphasized the need for mesh independence techniques to generate reliable and consistent results in fluid flow simulations [7].

Despite its importance, mesh sensitivity analysis is not typically thoroughly studied in jet impingement research. This can lead to potential errors in simulation results [7]. As a result, this study focuses on the numerical analysis of jet impingement cooling on a hot flat plate using CFD, with particular focus on mesh independence analysis. The importance of this study is in determining an appropriate mesh resolution that ensures accurate flow and heat transfer behavior while remaining computationally efficient. The objective of this study is to analyze the impacts of course, medium, and fine mesh sizes on velocity and temperature distributions, and to determine the grid independence of the simulation results.

2. Methodology

2.1 Geometry Description

The current study uses computational fluid dynamics to examine jet impingement cooling on a hot flat plate. The geometry consists of a vertical circular nozzle over a flat plate. A fluid jet is fired from the nozzle and impinges perpendicularly on the heated surface. This creates a stagnation zone where heat transmission is maximized. The computational domain is designed to capture the jet's formation and flow distribution throughout the plate surface. The geometry of the jet nozzle and flat plate used in this study is shown in Figure 1.

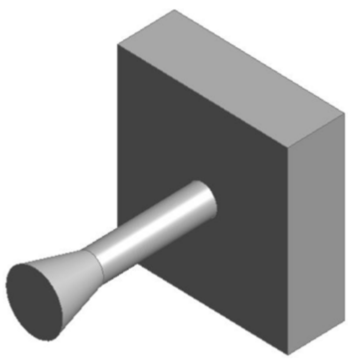
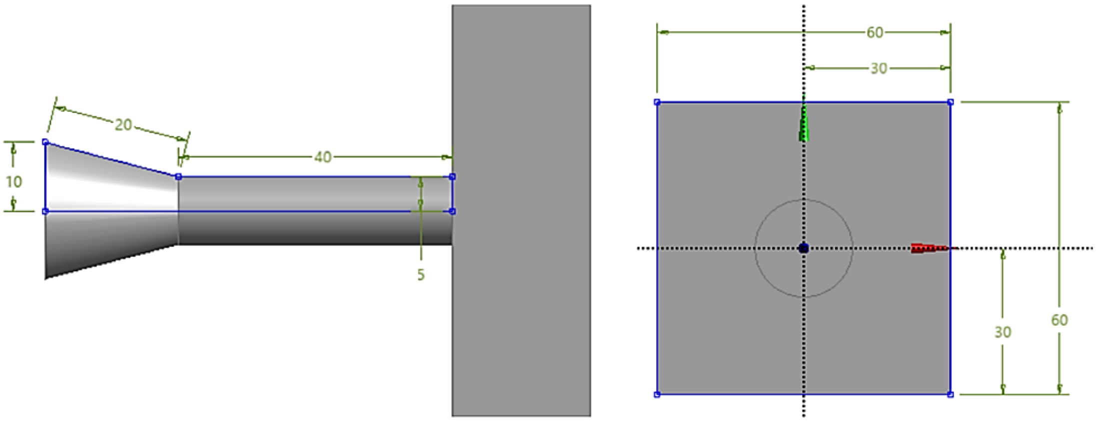


Fig. 1. Geometry shape of the jet nozzle and flat plate

The base plate was built at a dimension of 60 mm x 60 mm. The nozzle length was built at radius of 10 mm and length of 20 mm. The nozzle to plate distance is set at 40 mm. Detailed geometry specification is shown in Figure 2.



(a) (b)

Fig. 2. Detailed geometry specification (a) Side view of nozzle and plate (b) Back view of plate

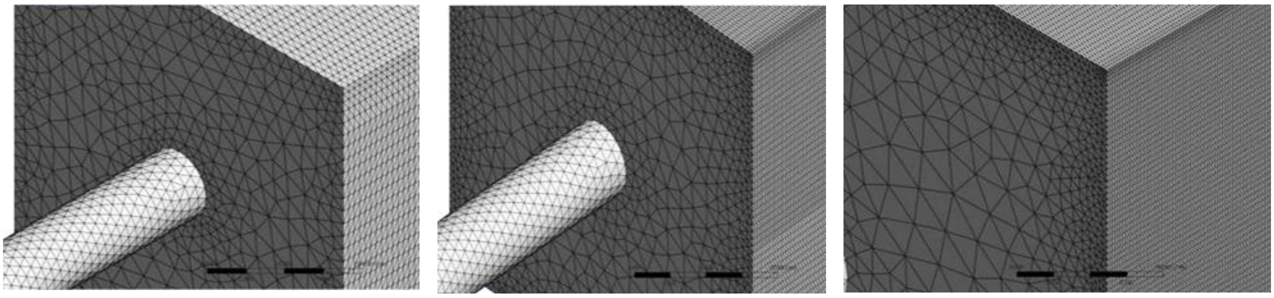
2.2 Mesh Generation

The computational domain was divided into finite volumes using a structured or unstructured mesh. To determine the effect of grid size on simulation accuracy, three different mesh resolutions were generated namely coarse, medium and fine meshes. The mesh was enhanced in areas with large flow gradients, particularly at the jet impingement region and wall limits. This is to better capture velocity and thermal variations. A finer mesh improves flow feature resolution especially in the stagnation area. But this makes a higher computing cost [8,9].

The coarse mesh contains 40,625 elements. It gives a simple representation of the flow domain at a lower computational cost. The medium mesh has 114,802 elements. This allows for improved resolution of velocity and heat gradients. The fine mesh which has 352,921 elements provides the most information. This information includes the stagnation region where significant gradients exist. The mesh refinement was defined based on the number of divisions assigned to key edges of the geometry. The coarse, medium, and fine meshes were generated using 30/20, 60/40, and 120/80 divisions respectively. The first value corresponds to the number of divisions along Edge 1, and the second value represents Edge 2. The mesh statistics for each case are summarized in Table 1. The mesh structures for the coarse, medium, and fine cases are shown in Figure 3. The mesh statistics for each mesh configuration are shown on Figure 4.

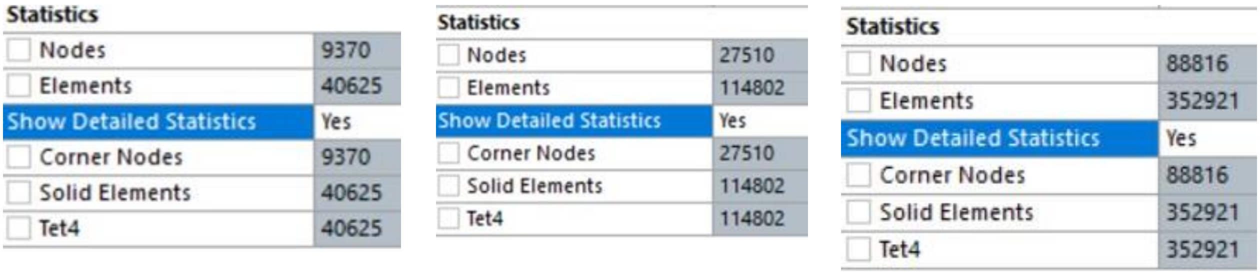
Table 1
Mesh statistics for different grid resolutions

Mesh Type	Edge divisions (edge 1 / edge 2)	Nodes	Elements
Coarse	30/20	9370	40625
Medium	60/40	27510	114802
Fine	120/80	88816	352921



(a) (b) (c)

Fig. 3. Mesh configurations (a) Coarse (b) Medium (c) Fine



Statistics

Nodes	9370
Elements	40625
Show Detailed Statistics	Yes
Corner Nodes	9370
Solid Elements	40625
Tet4	40625

Statistics

Nodes	27510
Elements	114802
Show Detailed Statistics	Yes
Corner Nodes	27510
Solid Elements	114802
Tet4	114802

Statistics

Nodes	88816
Elements	352921
Show Detailed Statistics	Yes
Corner Nodes	88816
Solid Elements	352921
Tet4	352921

(a) (b) (c)

Fig. 4. Mesh statistics (a) Coarse mesh (b) Medium mesh (c) Fine mesh

2.3 Boundary Conditions

The boundary conditions applied in the simulation are summarized in Table 2. The working fluid was assumed to be air with constant thermophysical properties. Turbulent flow conditions were considered based on the Reynolds number of the jet.

Table 2
Boundary conditions applied

Boundary	Type	Value
Inlet	Velocity inlet	40 m/s
Outlet	Pressure outlet	0 Pa
Heated plate	Wall	No-slip
Nozzle wall	Wall	No-slip
Fluid	Air	Standard properties

2.4 Solver Setup

The simulation was carried out using ANSYS Fluent. A steady-state solver was employed to solve the governing equations of fluid flow and heat transfer. The turbulence effects were modeled using k-epsilon (k-ε). The governing equations solved include continuity equation (Eq. (1)), Navier Strokes momentum equations (Eq. (2)) and energy equation (Eq. (3)). Second-order discretization schemes were used to improve solution accuracy. Convergence was achieved when residuals dropped below 10^{-5} for all equations.

$$\nabla.\rho V = 0 \quad (1)$$

$$\rho(V.\nabla)V = -\nabla p + \mu\nabla^2 V \quad (2)$$

$$\rho c_p(V.\nabla T) = k\nabla^2 T \tag{3}$$

2.5 Grid Independence Test

A grid independence test was carried out to demonstrate that mesh size had no effect on the simulation findings. Mesh resolution is important in CFD simulations because it determines the accuracy of numerical results particularly in regions with strong gradients like the stagnation zone of an impinging jet [10]. Three mesh resolutions, which are the coarse, medium, and fine meshes were created and examined. The simulation results were compared using key variables including velocity distribution and surface pressure. Grid independence is considered obtained when further mesh refining results in minor changes to the specified parameters.

3. Results

3.1 Grid Independence Test

To evaluate the effect of mesh resolution on simulation accuracy, a grid independence test was conducted using coarse, medium, and fine meshes. The simulation results were compared based on key parameters which includes maximum velocity and surface pressure. The comparison of results obtained from different mesh resolutions is shown in Table 3. The coarse mesh reports a maximum velocity of 7.354771 m/s and surface pressure of - 1.513535 Pa. When the mesh is adjusted to the medium grid, the maximum velocity increases to 7.439347 m/s and the surface pressure decreases to −1.421866 Pa. This indicates a significant improvement in solution accuracy due to increased resolution of flow gradients [9].

Refinement of the fine mesh resulted in a small change in maximum velocity to 7.378965 m/s and surface pressure to -1.343584 Pa. The difference between medium and fine mesh values is less than that between coarse and medium meshes. This trend is compatible with grid convergence theory, which states that further mesh refinement results in decreasing changes in numerical findings.

The comparatively higher deviation found in the coarse mesh is due to poor resolution of high velocity and pressure gradients in the impinging jet's stagnation zone. Previous research has demonstrated that accurate modelling of flow characteristics in high-gradient locations requires sufficiently precise grids [11]. As a result, the medium mesh is thought to be sufficient for capturing the jet impingement system's critical flow and temperature characteristics. It is chosen as the best mesh for further investigation because it strikes a reasonable balance between computational accuracy and computational expense. This method is often used in CFD research to provide consistent results while remaining computationally efficient [12,13].

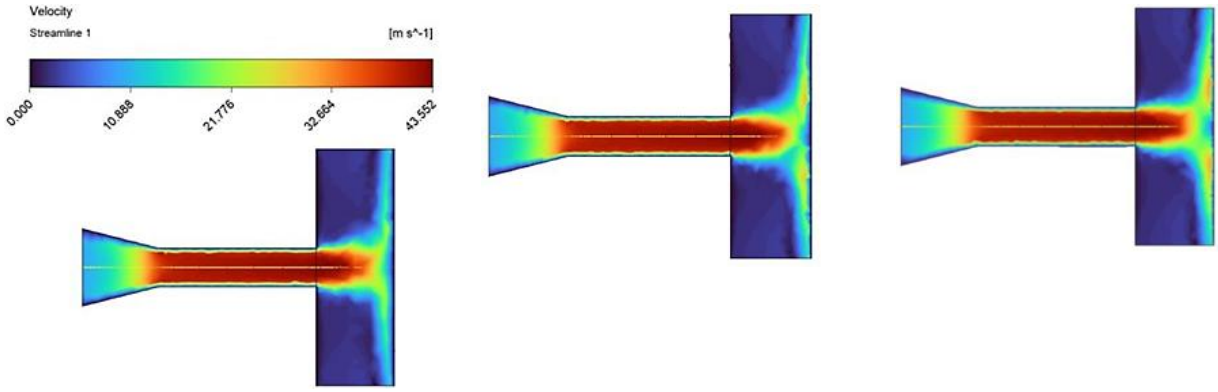
Table 3
Comparison of simulation results for different mesh resolutions

Mesh Type	Elements	Maximum velocity (m/s)	Surface pressure (Pa)
Coarse	40,625	7.354771	-1.513535
Medium	114,802	7.439347	-1.421866
Fine	352,921	7.378965	-1.343584

3.2 Velocity Distribution and Flow Behaviour

The velocity distribution of the jet impingement flow is shown in Figure 5. The results for coarse, medium, and fine mesh resolutions are presented together for comparison. The fluid jet exits the nozzle at high velocity and impinges directly on the flat plate. This results in a stagnation region at the point of impact. In this location, the velocity decreases rapidly due to flow deceleration. The fluid is redirected radially along the surface forming a wall jet [14]. Figure 5 shows that the coarse mesh gives less detailed velocity contours, especially around the stagnation area where sharp velocity gradients exist. The transition from the jet core to the wall jet is not well recorded due to low mesh resolution.

In contrast, the results for medium and fine meshes demonstrate significantly improved flow structure resolution. The high-velocity jet core is well defined, and the flow's radial spread following impingement is more properly portrayed. The formation of the wall jet along the plate surface is visible and constant throughout both meshes. Furthermore, the velocity contours produced from the medium and fine meshes are extremely similar, with just tiny changes in smoothness. This shows that further mesh refinement has no significant impact on the projected flow pattern. This validates the grid independence. Overall, the measured velocity distribution is consistent with fundamental jet impingement flow behaviour, where a high-momentum jet transforms into a radial wall jet with significant velocity gradients near the stagnation region [15-17].

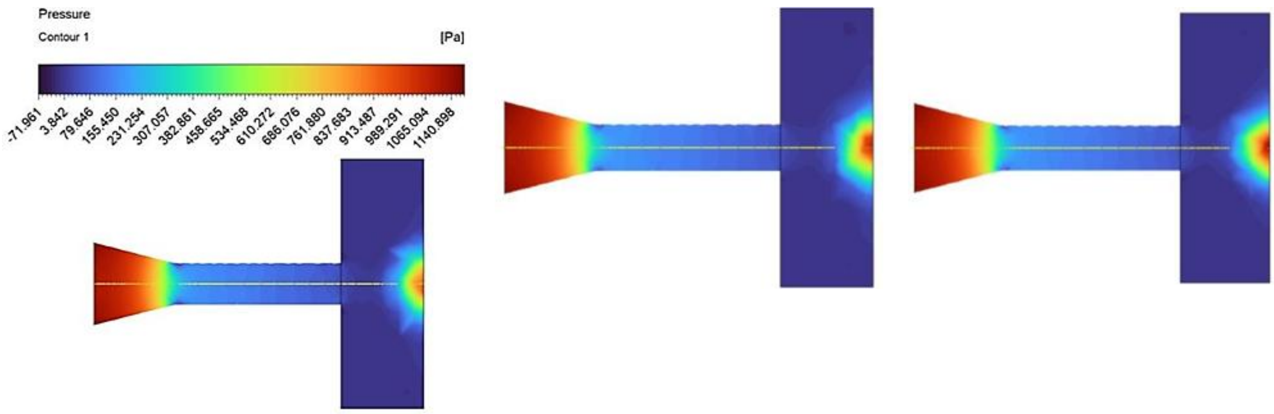


(a) (b) (c)

Fig. 5. Velocity contour (a) Coarse mesh (b) Medium mesh (c) Fine mesh

3.3 Pressure Distribution

Figure 6 shows the pressure distribution for different mesh resolutions. The results for coarse, medium, and fine meshes compared in a single plot. The stagnation area where the jet immediately impinges on the flat plate has the highest pressure. It is because the flow's kinetic energy is converted into pressure energy as it decelerates [18,19]. The coarse mesh results in less precise pressure contours and a less defined high-pressure zone. This is due to low mesh resolution, which reduces pressure gradient prediction [13]. The medium and fine mesh measurements show a more defined pressure distribution, including a distinct stagnation region and smooth pressure variation over the plate. Pressure drops outwards from the stagnation point as the flow accelerates along the surface [20]. The comparability of the medium and fine mesh results suggests that additional mesh refinement has no substantial effect on the pressure field, confirming grid independence.

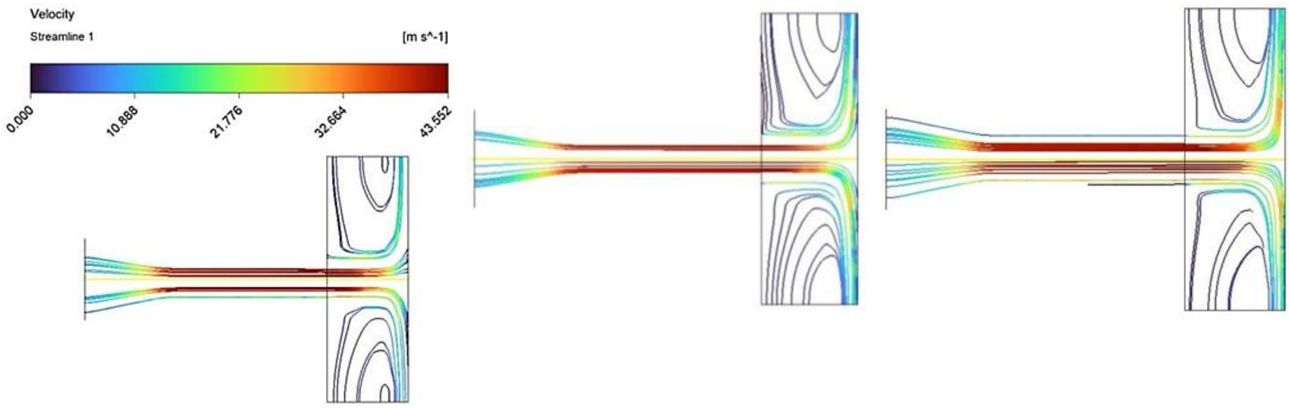


(a) (b) (c)

Fig. 6. Pressure contour (a) Coarse mesh (b) Medium mesh (c) Fine mesh

3.4 Streamline Analysis

Figure 7 shows the streamline patterns for different mesh resolutions. The streamlines shows that the fluid's direction when it exits the nozzle, impinges on the flat plate, and spreads radially outward. From the figures, it can be observed that the flow initially travels in a straight path within the nozzle. Then it forms a high-velocity jet core. Upon impingement, the flow is diverted along the surface. This results in a radial wall jet. This behaviour is consistent with typical jet impingement flow configurations described in the literature [17,18]. The streamline patterns obtained from medium and fine meshes are smooth and continuous. This indicates that the flow field is accurately resolved. In comparison, the coarse mesh shows fewer smooth streamline patterns. This shows a lesser precision in capturing detailed flow dynamics.



(a) (b) (c)

Fig. 7. Streamline (a) Coarse mesh (b) Medium mesh (c) Fine mesh

3.5 Velocity Profile Analysis

Figure 8 shows the velocity variations along the jet centreline at coarse, medium, and fine mesh resolutions. The velocity increases rapidly from the nozzle exit and reaches a maximum value within the jet core region as the flow accelerates. When the flow reaches the impingement surface, its velocity drops significantly. This is due to flow deceleration in the stagnation region. Beyond this point, the velocity decreases as the fluid travels radially across the surface losing momentum [14]. The velocity profiles obtained from the medium and fine meshes are nearly identical. Small variations are detected in the coarse mesh. This suggests that the medium mesh is enough for accurately capturing the velocity variation along the jet centreline. The close relationship between medium and fine mesh findings indicates that the simulation has achieved grid independence. This behaviour is consistent with standard CFD techniques which says greater mesh refinement produces small changes in expected flow variables.

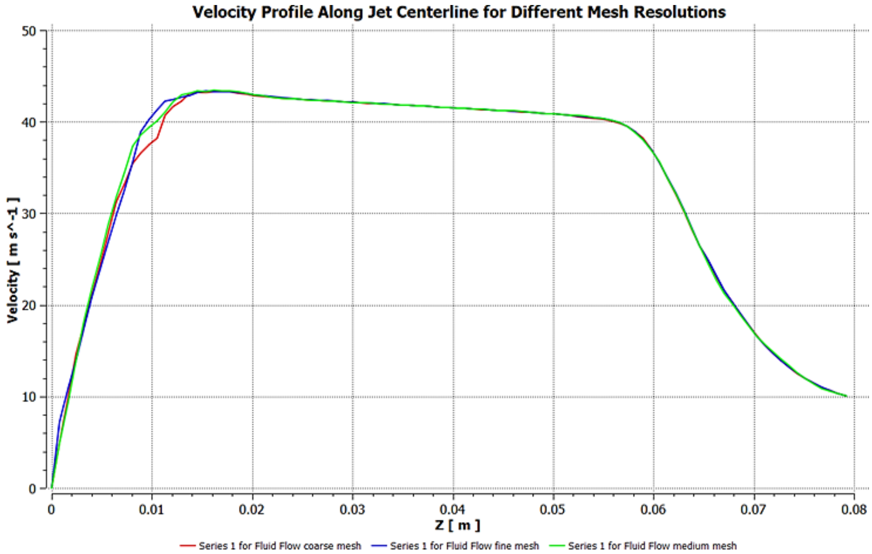


Fig. 8. Velocity profile of three meshes

4. Conclusions

Computational Fluid Dynamics (CFD) simulations were used to analysis the jet impingement cooling behaviour on a heated flat plate. This study focuses on the effect of the mesh size on simulation accuracy. Three different mesh sizes were used in the study namely, coarse, medium and fine meshes. A grid independence test is done in this study. It revealed that the coarse grid resulted in a large change. It is because good resolution of flow gradients including the stagnation region was not obtained. The medium and the fine meshes yielded almost identical velocities and pressures prediction. These result shows that increasing mesh beyond medium mesh does not give a significant increase in accuracy of the simulation result. So, medium mesh was selected as the optimum mesh size due to its reasonable level of accuracy in calculations. The simulation results of velocity contour shows that the fluid jet leaving the nozzle at significant velocity and strikes the flat plate. This leads to a stagnation region of very low velocity. The flow is rerouted along the surface forming a radial wall jet.

For the pressure distribution result, it shows that the highest pressure is in the stagnation area. In this area, the kinetic energy is transformed into pressure energy. The flow behaviour was validated by streamline analysis. The analysis shows a smooth flow redirection in the jet core to well jet zone. The results also show that the velocity profile of the medium and the fine mesh values near the centreline of the jet were almost the same. This results in the solution to be grid independent. The observed trends agree with predictions of the theoretical jet impingement flow. At last, this study shows the necessity of the resolution of the mesh in CFD simulations especially for regions where the gradients are high.

References

[1] Zuckerman, N., and Noam Lior. "Jet impingement heat transfer: physics, correlations, and numerical modeling." *Advances in heat transfer* 39 (2006): 565-631. https://doi.org/10.1016/S0065-2717(06)39006-5

[2] Behnia, M., S. Parneix, Y. Shabany, and P. A. Durbin. "Numerical study of turbulent heat transfer in confined and unconfined impinging jets." *International Journal of Heat and Fluid Flow* 20, no. 1 (1999): 1-9. https://doi.org/10.1016/S0142-727X(98)10040-1

[3] Viskanta, R. "Heat transfer to impinging isothermal gas and flame jets." *Experimental thermal and fluid science* 6, no. 2 (1993): 111-134. https://doi.org/10.1016/0894-1777(93)90022-B

[4] Ferziger, Joel H., Milovan Perić, and Robert L. Street. *Computational methods for fluid dynamics*. Vol. 3. Berlin: springer, 2002. https://doi.org/10.1007/978-3-642-56026-2

[5] Roache, Patrick J. "Perspective: a method for uniform reporting of grid refinement studies." *Journal of Fluids Engineering* 116, no. 3 (1994): 405-413. https://doi.org/10.1115/1.2910291

[6] Purohit, Shantanu, and K. Vasudeva Karanth. "Computational flow and heat transfer study on impingement cooling in a turbine blade leading edge using an innovative convergent nozzle." *Journal of the Brazilian Society of Mechanical Sciences and Engineering* 45, no. 2 (2023): 1-14. https://doi.org/10.1007/s40430-023-04020-4

[7] Rachdi, Zakia, Nidhal Hnaien, Aboulbaba Eladeb, Badr M. Alshammari, Lioua Kolsi, and Hacen Dhahri. "CFD analysis of heat transfer enhancement in impinging jet array by varying number of jets and spacing." *Scientific Reports* 15, no. 1 (2025): 3023. https://doi.org/10.1038/s41598-025-86360-w

[8] Martin, Holger. "Heat and mass transfer between impinging gas jets and solid surfaces." In *Advances in heat transfer*, vol. 13, pp. 1-60. Elsevier, 1977. https://doi.org/10.1016/S0065-2717(08)70221-1

[9] Versteeg, Henk Kaarle. *An introduction to computational fluid dynamics the finite volume method, 2/E*. Pearson Education India, 2007.

[10] Beg, Md Nazmul Azim, Rita F. Carvalho, and Jorge Leandro. "Effect of manhole molds and inlet alignment on the hydraulics of circular manhole at changing surcharge." *Urban Water Journal* 16, no. 1 (2019): 33-44. https://doi.org/10.1080/1573062X.2019.1611887

[11] Blazek, Jiri. *Computational fluid dynamics: principles and applications*. Butterworth-Heinemann, 2015. https://doi.org/10.1016/B978-0-08-099995-1.00012-9

[12] Anderson, John D. *Computational fluid dynamics: the basics with applications*. McGraw-Hill, 1995.

[13] Launder, Brian Edward, and Dudley Brian Spalding. "The numerical computation of turbulent flows." In *Numerical prediction of flow, heat transfer, turbulence and combustion*, pp. 96-116. Pergamon, 1983. https://doi.org/10.1016/B978-0-08-030937-8.50016-7

[14] Incropera, Frank P., David P. DeWitt, Theodore L. Bergman, and Adrienne S. Lavine. *Fundamentals of heat and mass transfer*. Vol. 6. New York: Wiley, 1996.

[15] Schlichting, Hermann, and Klaus Gersten. "Boundary–layer equations in plane flow; plate boundary layer." In *Boundary-layer theory*, pp. 145-164. Berlin, Heidelberg: Springer Berlin Heidelberg, 2016. https://doi.org/10.1007/978-3-662-52919-5_6

[16] Bird, R. Byron, Warren E. Stewart, and Edwin N. Lightfoot. *Transport Phenomena*. 2nd ed. New York: Wiley, 2002.

[17] Fox, Robert W., Alan T. McDonald, and John W. Mitchell. *Fox and McDonald's introduction to fluid mechanics*. John Wiley & Sons, 2020.

[18] Kays, W. M., and A. L. London. *Compact Heat Exchangers*. 3rd ed. New York: McGraw-Hill, 1984.

[19] Han, Je-Chin, and Lesley Wright. *Experimental Methods in Heat Transfer and Fluid Mechanics*. CRC Press, 2020. https://doi.org/10.1201/9781003021179

[20] Cengel, Yunus, and John Cimbala. *Ebook: Fluid mechanics fundamentals and applications (si units)*. McGraw Hill, 2013.