

FLOW FIELD SIMULATION AND OPTIMISATION STUDIES FOR IMPINGING JET FLOW

Burak Pehlivan¹
Hacettepe University
Ankara, Turkey

Özgür Ekici²
Hacettepe University
Ankara, Turkey

ABSTRACT

In this study, a two-dimensional fluid dynamics code was developed to investigate single and double impinging jet flow. Flow and thermal field simulations have been carried out for compressible flow with significant temperature variations. Temperature and velocity fields, the change of average Nusselt number near impingement wall and the stagnation Nusselt number have been studied for various Reynolds numbers and domain aspect ratios. Obtained results are compared with those of existing studies in the literature whenever available and a good conformity is found. An improved model to account for the variation of Nusselt number with Reynolds number has been proposed. In addition, optimization studies have been performed where the injection size, Reynolds number, impingement height and wall temperature at the confinement (injection) wall are taken as optimisation parameters and stagnation Nusselt number is maximized.

INTRODUCTION

Impinging jet flow is a type of flow phenomenon that is important especially for cooling problems and associated complex flow fields. Cooling of turbine blades, annealing of metal and plastic sheets, the tempering of glass are some of its important industrial applications.

The problem has been studied for both laminar and turbulent flows and analytical solutions were developed for the laminar incompressible case [Glauert, 1956].

A corresponding solution was developed to account for the compressibility effects in laminar radial wall jets with stream function formulation utilising boundary layer formulation. The viscosity has been assumed to be directly proportional to temperature in the solution. The solution is not fully applicable for the general viscous case [Riley, 1958].

Stream function formulation with finite difference discretisation to solve full set of Navier Stokes equations including energy equation in the laminar flow region has also been used in the literature [Heiningen et al., 1976].

Three-dimensional flow structure and the heat transfer in laminar rectangular impinging jets for incompressible flow have been studied in the literature. Staggered meshing and finite volume method is used. The convection terms are computed using QUICK scheme. SIMPLE

¹ PhD Student in the Department of Mechanical Engineering, Email: burakpehlivn@gmail.com

² Assist. Prof. in the Department of Mechanical Engineering, Email: ozgur.ekici@hacettepe.edu.tr

algorithm has been used. A 101x101x51 grid system has been used. Calculations have been performed up to the maximum Reynolds number of 500 [Sezai and Mohammad, 1999].

As a variation, the heat transfer due to double laminar slot jets impingement has been studied in the literature. Incompressible form of the Navier Stokes and energy equations are discretized with a finite volume procedure on a non-staggered grid arrangement using SIMPLER (SIMPLE-Modified) algorithm. The effect of the jet Reynolds number, the jet-isothermal bottom wall spacing, and the distance between two jets on heat transfer and flow field is examined. The power-law difference scheme is used to discretize the convective terms and central differencing for the diffusion terms. Reynolds numbers of 250, 500, and 750 have been investigated. Left and top walls are taken as adiabatic while the bottom wall and the right boundary are taken as isothermal. Non-dimensionalised forms of the variables have been used in the solution [Dagtekin and Oztop, 2007].

Laminar compressible impinging jet problem for flat and curved plates has also been studied in the literature numerically. Domain height to nozzle width ratio of 2 ($H/b=H/W=2$) is used for the analyses. It is noted that Nusselt number varies approximately linearly with $Re^{0.5}$ for laminar flows [Tahsini and Mousavi, 2012].

In this study, finite volume method has been utilised for laminar, compressible and unsteady flows. A 2-D computational in-house code was developed to investigate the flow and thermal fields in addition to the variation of Nusselt number as a function of Reynolds number.

METHOD AND EQUATIONS SOLVED

The numerically solved equations are given below. Here, su , sv and sT are the source terms. The compressible form of the Navier Stokes equations are needed for high speed flows as well as problems where high temperature and high temperature gradients are present. The equations given below are solved simultaneously.

Continuity:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \quad \text{Eqn. 1}$$

u-momentum:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho uu)}{\partial x} + \frac{\partial(\rho vu)}{\partial y} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left(\mu \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu \frac{\partial u}{\partial y} \right) + su \quad \text{Eqn. 2}$$

v-momentum:

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho vv)}{\partial y} = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left(\mu \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu \frac{\partial v}{\partial y} \right) + sv \quad \text{Eqn. 3}$$

Energy:

$$\frac{\partial(\rho T)}{\partial t} + \frac{\partial(\rho uT)}{\partial x} + \frac{\partial(\rho vT)}{\partial y} = \frac{\partial}{\partial x} \left(\frac{k}{C_p} \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(\frac{k}{C_p} \frac{\partial T}{\partial y} \right) + sT \quad \text{Eqn. 4}$$

Equation of state:

$$\rho = \frac{P}{RT} \quad \text{Eqn. 5}$$

Modified SIMPLE method for compressible flows has been used to obtain pressure values [Darwish et al., 2016]. Here, velocity correction equations for the original and modified SIMPLE are represented by the following equation.

$$v = v_0 + \frac{dx}{a_{pv}} * (P'_s - P'_n) + v * P'(P + P_0) \quad \text{Eqn. 6}$$

In the above equation, original SIMPLE correction is given by the second term on the right hand side (RHS). The third term on the RHS is for the compressible flows. Due to the nature of the second term, the SIMPLE procedure is able to solve subsonic and supersonic flow problems with high temperature gradients. Density correction is represented by the following equation:

$$\rho' = \left(\frac{1}{RT}\right) P' \quad \text{Eqn. 7}$$

The pressure correction equation is represented by the following equation.

$$p' = \left(\frac{1}{a_p}\right) * (a_{pw} * P'_W + a_{pe} * P'_E + a_{ps} * P'_S + a_{pn} * P'_N + b_{corr} - P'_{conv}) \quad \text{Eqn. 8}$$

In Eqn. 8, the term b_{corr} represents a term due to mass residual and a term due to change in density. The term b_{corr} is thus a source like term in the equation. F_i represent the face fluxes.

$$b_{corr} = F_w - F_e + F_s - F_n - \rho' * dx * dy / dt \quad \text{Eqn. 9}$$

P'_{conv} term in Eqn.8 arises in the compressible form of the SIMPLE equations. The presence of the convection like term (P'_{conv}) is essential for the use of SIMPLE procedure for high Ma flows.

$$P'_{conv} = \frac{F_e * P'_e}{P + P_0} - \frac{F_w * P'_w}{P + P_0} + \frac{F_n * P'_n}{P + P_0} - \frac{F_s * P'_s}{(P + P_0)} \quad \text{Eqn. 10}$$

In comparison with the incompressible SIMPLE scheme, the coefficient a_p is modified with a transient term for the compressible SIMPLE scheme.

$$a_p = a_{pw} + a_{pe} + a_{ps} + a_{pn} + C_p * V / \Delta t \quad \text{Eqn. 11}$$

Further details of the modified SIMPLE scheme is given in the literature [Darwish et al., 2016]. Main steps of the SIMPLE compressible approach are also given below:

1. Set the boundary conditions.
2. Initialize, u , v , P , ρ , T .
3. Solve the discretized momentum equation to compute the intermediate velocity field.
4. Compute the uncorrected mass fluxes at faces.
5. Solve the energy equation.
6. Solve the pressure correction equation to produce cell values of the pressure correction, p'' .
7. Update the pressure field: $p_{k+1} = p_k + \alpha_{a_p} * p''$, where α_{a_p} is the under-relaxation factor for pressure.
8. Update velocity field: $(u, v)_{k+1} = (u, v)_k + (\alpha_{a_u}, \alpha_{a_v}) * (u', v')$, where α_{a_u} and α_{a_v} are the under-relaxation factors for velocities, and u' , v' are velocity corrections.
9. Update density field using $\rho_{k+1} = \rho_k + (\alpha_{a_{\rho}}) * (1/RT) * p''$.
10. Loop until mass continuity residual at each cell and other residuals drop below a specified tolerance.

In this study, collocated grid arrangement along with momentum interpolation has been utilised [Choi, 1999]. This approach handles unsteady flows better than the Rhie and Chow's original momentum interpolation technique in that it significantly reduces time step size and under-relaxation dependency.

Laminar flow has been assumed, so that in conformity with this assumption, low Mach number flows have been studied. Diffusion terms have been discretised with second order central differencing and convective terms have been discretised with the hybrid scheme. Details related to formulas implemented for the discretization are given in the literature [Malalesekara and Versteeg, 2010].

ANALYSIS

Single Slot Impinging Jet Flow

Single slot impinging flow has been studied using compressible form of Navier Stokes equations. Mass flow inlet and pressure outlet boundary conditions have been imposed. Domain geometry is shown schematically in Figure 1.

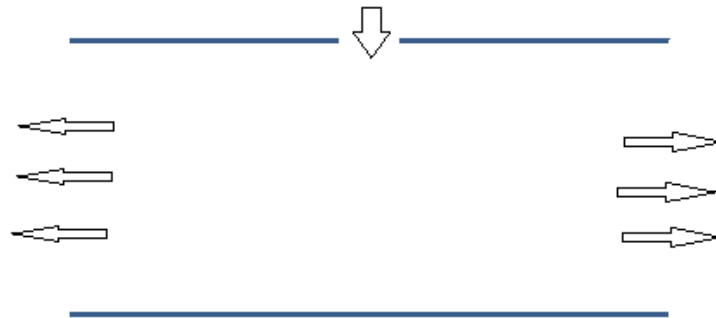


Figure 1. Domain geometry for the solution

Solution parameters for the single slot impinging flow are given in Table 1.

Table 1. Modelling parameters used for single impinging jet flow

Variable name	Symbol	Value
Width of nozzle	W	0.1
Domain height to nozzle width ratio	H/W	2
Domain length to nozzle width ratio	L/W	20
Prandtl number	Pr	0.71
Pressure coefficient	C_p	1000
Dynamic viscosity	μ	2×10^{-5}
Number of nodes in x direction	N_x	102
Number of nodes in y direction	N_y	42 to 102
Cell horizontal dimension	dx	1/50
Cell vertical dimension	dy	1/100 to 1/500
Reynolds number	Re	100 to 1000
Time increment for the unsteady simulation	dt	dy/v_{jet}
Freestream temperature	T	293 K
Lower boundary (impingement plate) temperature	T	393 K
Outflow static pressure	$P_{outflow}$	0
Time limit for unsteady simulations	t_{final}	50 s

Variation of Nusselt number is illustrated in Figure 2 for the Reynolds number of 100. As can be seen from the figure, heat transfer and Nusselt number is maximised at the stagnation point directly under the impingement nozzle. Nusselt number decays close to outer boundary.

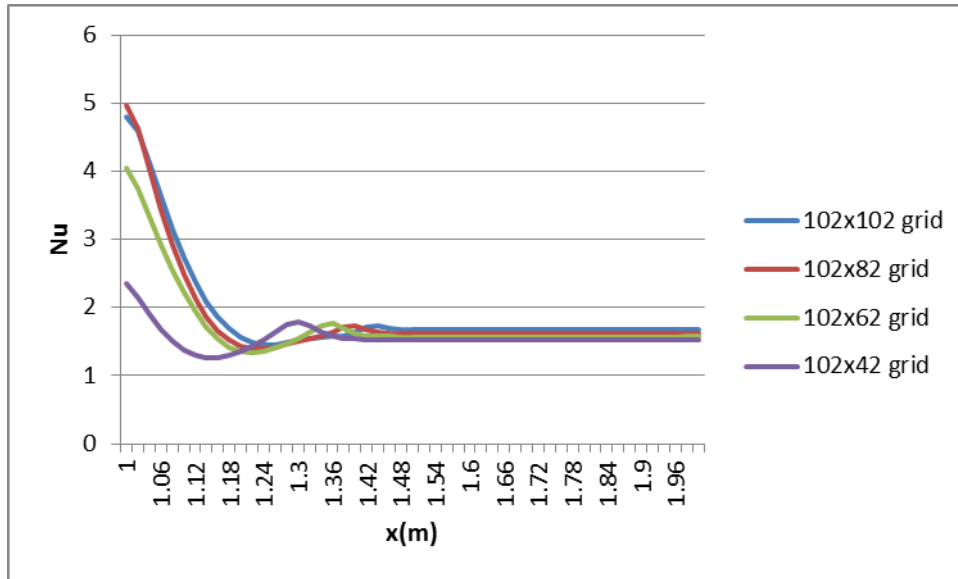


Figure 2. Variation of Nusselt number for Re=100

For the variation of Nusselt number with Reynolds number, the following figure is provided for $HW=2$ [Lee et al., 2008].

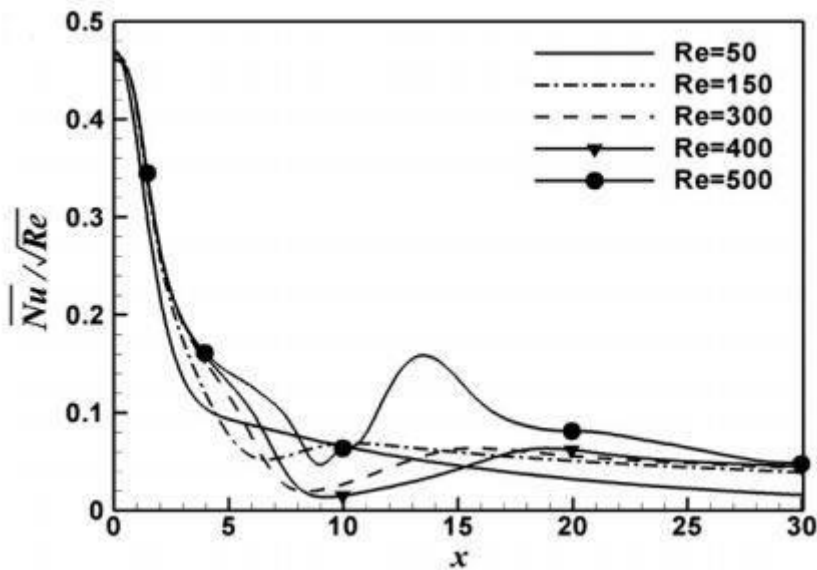
Figure 3. Variation of Nu with $Re^{0.5}$ [Lee et al., 2008]

Figure 3 illustrates that stagnation Nusselt number to $Re^{0.5}$ ratio is almost constant for laminar flows at different Reynolds numbers. Since all the solutions converge to a nearly single point at the stagnation, this can be represented in equation form as:

$$Nu_s/Re^{0.5} = \text{constant}$$

Eqn. 12

The constant is close to 0.47 for $HW=2$.

Figure 4 illustrates the variation of Nusselt number with Reynolds number for $Re=300$ [Tahsini, 2012].

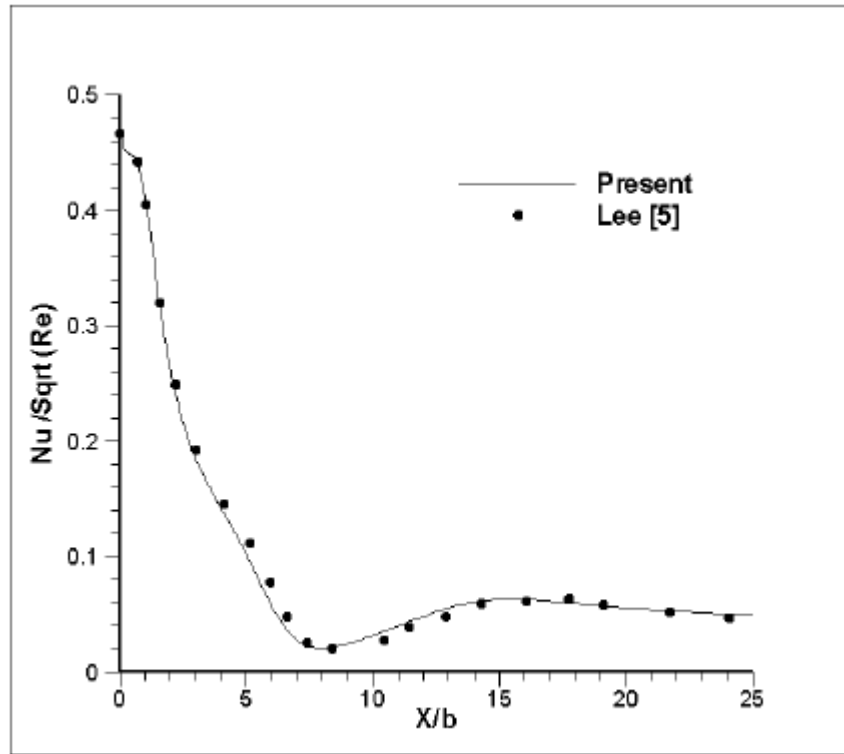


Figure 4. Variation of Nu with $Re^{0.5}$ [Tahsini, 2012]

For $Re=300$, different grid resolutions from 102×42 to 102×102 has been tested to illustrate the validity of results in this study, solve the vortex structures in the flow domain and understand the dependence of results on the grid resolution. Figure 5 illustrates Nusselt number variation at the impingement wall for different grid resolutions. As illustrated in Figure 5, it is found that there are notable differences between coarse grid and fine grid solutions in terms of Nusselt number variation.

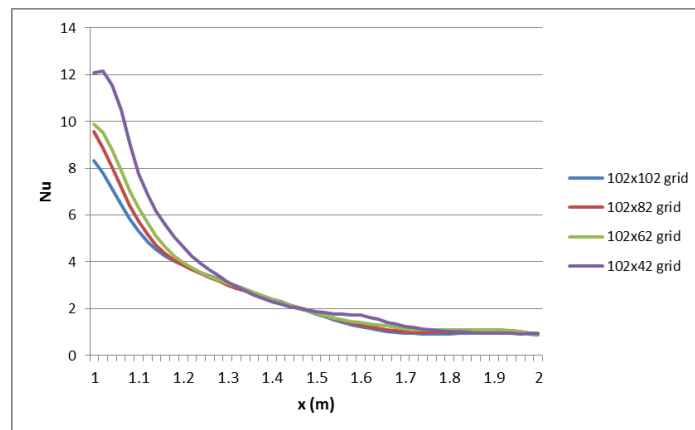


Figure 5. Variation of Nusselt number for $Re=300$

$Nu_s=8.3142$ has been obtained for 102×102 nodes grid system. $Nu_s=8.1406$ is given in the literature [Tahsini and Mousavi, 2012]. Hence the discrepancy is 2.13%. In other words, the result for Nusselt number variation is in accord with the results in the literature.

Next, the solution has been repeated with $Re=500, 600, 1000$.

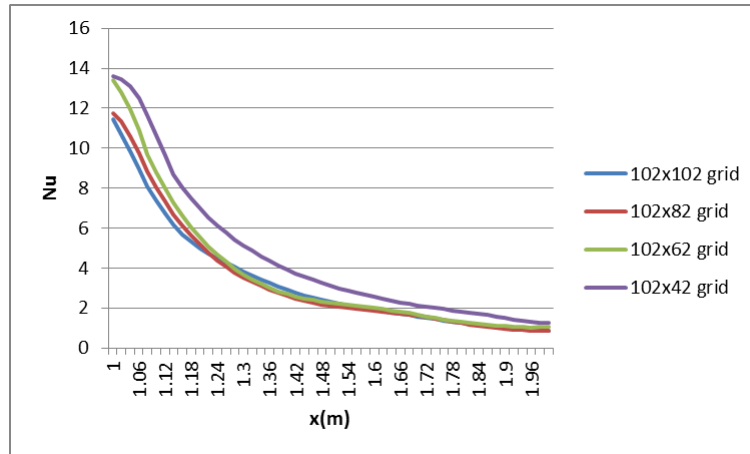


Figure 6. Variation of Nusselt number for Re=500

The Nusselt number is plotted for various Reynolds numbers in Figure 7 for the finest grid solution (102x102).

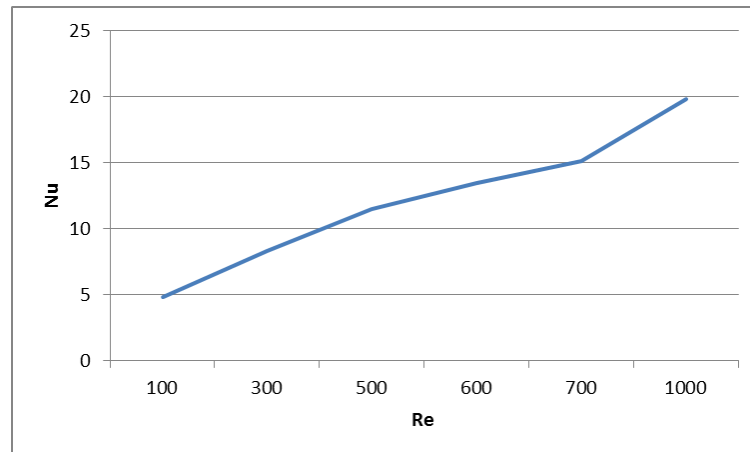


Figure 7. Variation of Nusselt number with Reynolds number

Instead of relating Nu_s with $Re^{0.5}$ by a constant, a new model can be proposed as illustrated in Eqn. 13. This model accounts for the increased convection at higher Reynolds numbers which further increases Nusselt number. This brings a slight increase in mathematical complexity.

$$Nu_s = (0.47 + (Re/10000)) * Re^{(0.5 + (Re/100000))}$$

Eqn. 13

Figure 8 summarises the results. The newly proposed model matches well with the results obtained from numerical studies.

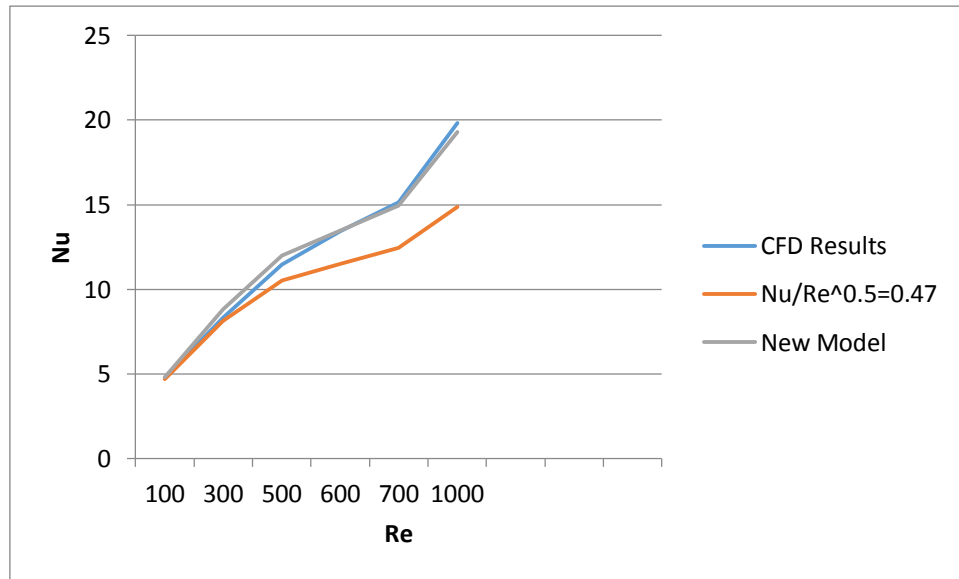


Figure 8. Variation of stagnation Nusselt number with Reynolds number

The variation of temperature is illustrated in-Figure 9 for the finest grid solution of 102x102 for Re=300. Underneath the jet flow, cooling effect is more evident as expected. Next, the density variation has been studied. Figure 10 illustrates the density variation for the 102x102 nodes grid system. It is seen that the density isolines follow a similar but not exactly equal pattern to temperature lines.

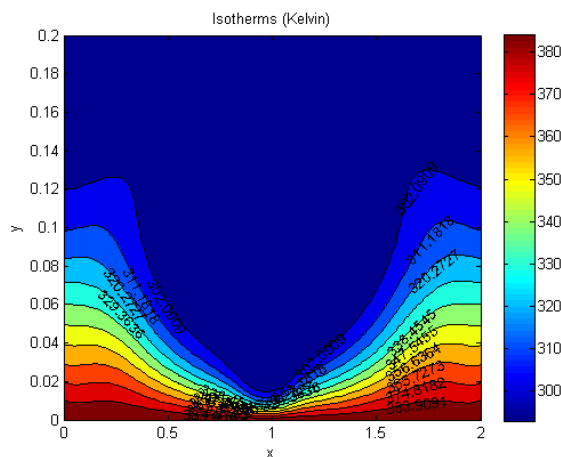


Figure 9. Variation of temperature for single impinging jet flow, Re=300, 102x102 grid

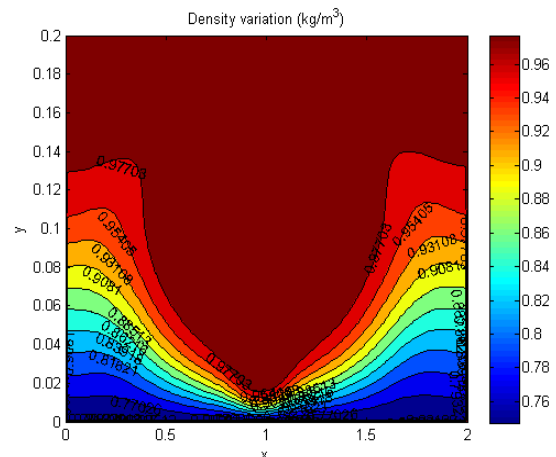


Figure 10. Variation of density for single impinging jet flow, Re=300, 102x102 grid

The velocity field has also been studied for the finest grid. The results show the primary vortex is clearly visible to the right and left of impingement nozzle exit at Re=300 as illustrated in Figure 11.

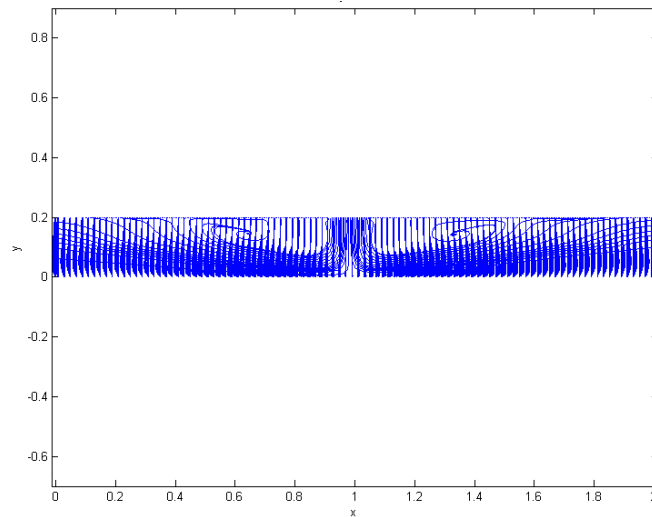


Figure 11. Velocity field for single impinging jet flow, $Re=300$, 102×102 grid

Double Slot Impinging Jet Flow

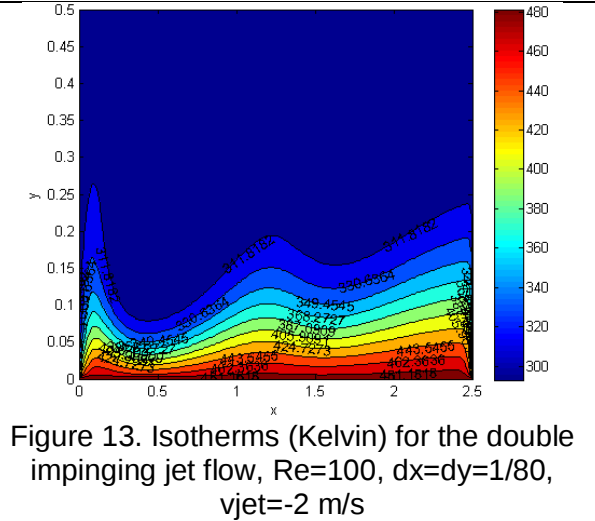
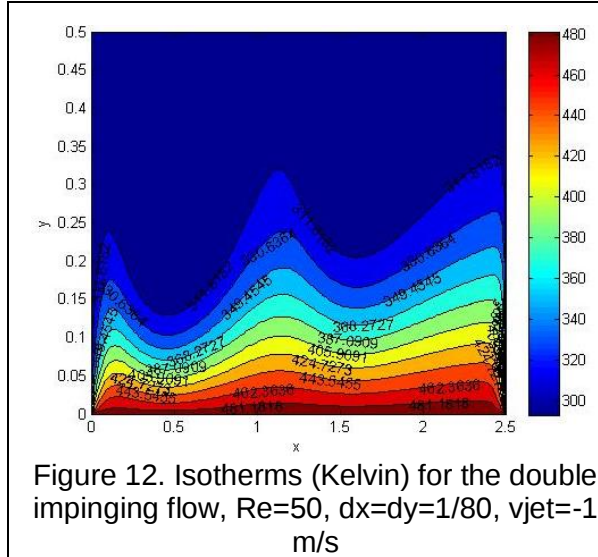
A double slot impinging flow structure has been studied. Domain size of 2.5×0.5 m (LxH) has been used. The slots are located between 0.25 - 0.5 m and 1.5 - 1.75 m from the top left edge. Isothermal boundary conditions have been imposed. The bottom face is maintained at 500 K while the other boundaries are kept at 293 K. Mass flow inlet and pressure outlet boundary conditions have been imposed.

Unsteady compressible Navier Stokes equations have been solved. Due to the temperature variation, energy equation has also been solved and density has been obtained in the domain. Momentum interpolation has been used to obtain face velocities [Choi, 1999]. Different slot injection speeds have been used and its effect on the overall solution has been studied. Results have been obtained for two different grids. Variables used are summarised in Table 2.

Table 2. Modelling parameters used for double slot impinging jet flow

Variable name	Variable symbol	Value
Width of nozzle 1	W	0.25
Width of nozzle 2	W	0.25
Domain height to nozzle width ratio	H/W	2
Domain length to nozzle width ratio	L/W	10
Prandtl number	Pr	1
Dynamic viscosity	μ	$1/100$
Conductivity coefficient	k	1
Pressure coefficient	C_p	100
Number of nodes in x direction	N_x	102 to 202
Number of nodes in y direction	N_y	22 to 42
Cell horizontal dimension	dx	$1/40$ to $1/80$
Cell vertical dimension	dy	$1/40$ to $1/80$
Reynolds number	Re	50 to 100
Injection velocity	V_{jet}	-1 to -2 m/s
Time increment for the unsteady simulation	dt	dy/v_{jet}
Freestream temperature	T	293 K
Lower boundary temperature	T	500 K
Outflow static pressure	$P_{outflow}$	0
Time limit for unsteady simulations	t_{final}	50 s

Due to the impinging flow, cooling in the domain is realized. The cooling is more evident underneath the regions where slots are located. To increase the effect of cooling, impinging jet flow velocity has been increased by a factor of 2 and the solution has been repeated. The results confirm a stronger cooling effect compared to the previous case where impinging jet velocity was lower. Predicted temperature values are lower in this case.



OPTIMISATION STUDIES

In this part, genetic algorithm, which is an evolutionary optimisation technique, has been implemented. The genetic algorithm is particularly suited to nonlinear problems with multiple variables where the possibility of the existence of local minimums and maximums is present.

Optimisation Case

Single impinging jet flow problem has been studied with unsteady simulation involving also the solution of energy equation. Unsteady simulation has been run for 50 seconds. Impinging jet optimisation problem has been studied with fluid and geometric variables apart from thermal variables. A total of six parameters (bottom wall temperature, top wall temperature, impinging jet domain height, impinging jet domain length, Reynolds number, injection nozzle width) have been selected as optimisation variables. The selected intervals for these variables are listed in Table 3.

Table 3. Impinging jet optimisation parameters

Top wall temperature	295-330 K
Bottom wall temperature	400-500 K
Injection nozzle width	1-3
Height of impinging jet domain	1-4
Length of impinging jet domain	15-25
Reynolds number	100-1000

Objective function has been defined as:

Objective function = $-Nu_s$

The optimization problem has been run with genetic algorithm for 40 generation with 35 population. Optimum result (highest stagnation point Nusselt number) is obtained with the set of values for the optimization variables given in Table 4.

Table 4. Second impinging jet optimisation parameter values

Top wall temperature	295
Bottom wall temperature	459
Injection nozzle width	3
Height of impinging jet domain	1
Length of impinging jet domain	15
Reynolds number	1000

Decreasing the height of impinging jet domain, length of domain, and the top wall temperature while increasing injection nozzle width maximise the stagnation point Nusselt number. Due to the thermal fluid interaction, stagnation point Nusselt number is not maximised when bottom wall is at the maximum allowable temperature. Rather, a value approximately at the middle of the selected interval (400-500 K) optimises the stagnation point Nusselt number.

CONCLUSION

An in-house pressure based Navier Stokes solver based on collocated grids have been developed. For the solution of compressible flows, a modified version of SIMPLE procedure has been implemented. The developed code has been applied to impinging jet flow problem

and results obtained have been compared with the results found in the literature. This comparison has shown a good conformity between the results. Stability of the solver and convergence characteristics has been assessed by using different grid densities. An improved model to evaluate the variation of Nusselt number with Reynolds number is proposed. Additionally, optimisation study has been performed. Genetic algorithm code has been matched with the in-house CFD solver to optimise stagnation point Nusselt number for unsteady single impinging jet flow.

References

1. Choi, S. K. (1999) Note on the use of momentum interpolation method for unsteady flows. *Numerical Heat Transfer: Part A: Applications*, vol. 36, pp. 545–550, 1999
2. Dagtekin, I., Oztop, H. F. (2008), ., Heat transfer due to double laminar slot jets impingement onto an isothermal wall within one side closed long duct, *International Communications in Heat and Mass Transfer*, Vol. 35, pp. 65-75
3. Darwish, M., Mangani, L., Moukalled, F. (2016) *The Finite Volume Method In Computational Fluid Dynamics*, Springer International Publishing.
4. Glauert, M. (1956). The Wall Jet, *Journal of Fluid Mechanics*, vol. 1, pp. 625-643.
5. Heiningen, A. R. P., Mujumdar, A. S., Douglas, W. J. M. (1976) Numerical Prediction of the Flow Field and Impingement Heat Transfer Caused by a Laminar Slot Jet, *Journal of Heat Transfer*, *Transactions of ASME*, pp. 654-658
6. Lee, H.G., Yoon, H. S., Ha, M. Y. (2008) A numerical investigation on the fluid flow and heat transfer in the confined impinging slot jet in the low Reynolds number region for different channel heights, *International Journal of Heat and Mass Transfer*, vol. 51, pp. 4055–4068.
7. Rhie, C. M. and Chow, W. L. (1983) Numerical study of the turbulent flow past an airfoil with trailing edge separation, *AIAA Journal*, vol. 21, pp. 1525-1532.
8. Riley, N. (1958) Effects of compressibility on a laminar wall jet, *Journal of Fluid Mechanics*, vol. 4 (6), pp. 615-628.
9. Sezai, I., Mohamad, A. A. (1999) Three-Dimensional Simulation of Laminar Rectangular Impinging Jets, Flow Structure, and Heat Transfer, *Journal of Heat Transfer*, *Transactions of ASME*, pp. 51-56
10. Tahsini, A. M., Mousavi, S. T. (2012) Laminar Impinging Jet Heat Transfer for Curved Plates, *International Journal of Aerospace and Mechanical Engineering*, vol. 6 (12), pp. 2788-2793
11. Versteeg, H., Malalasekera, W. (2010) *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, Pearson Prentice Hall, 2nd edition.