Computational analysis of heat transfer due to turbulent annular jet impingement

Prasun Dutta¹ and Himadri Chattopadhyay²

¹, ²Department of Mechanical Engineering, Jadavpur University, Kolkata–700032, India
e-mail: pd.iiest@gmail.com

Abstract. Present paper deals with the numerical study of fluid flow and heat transfer due to an annular turbulent jet impingement on a flat heated plate. The geometrical configurations of annular jet was selected such a way that the mass and momentum efflux at the annular jet exit will be same as a circular jet having same inner diameter at a particular Reynolds number. Three dimensional turbulent flow field was resolved using Transition SST Model by solving the mass, momentum and energy equation with SIMPLE algorithm. A highly refined mesh was used for numerical computation after validation of present model with established results of same geometrical configuration. The flow structure and heat transfer characteristic of the annular jet was compared with the conventional circular jet and also with an established two dimensional study of same configurations and reported in this paper. It was observed that that two dimensional studies on annular jet under predicts heat transfer characteristics.

1. Introduction

It is well known that, high heat transfer rate can easily obtained by implementation of the jet impingement technique on a hot surface. For that reason, impingement of jet is extensively used in numerous industry. The applications contain moderating of glass, paper drying, chemical vapor removal, cooling of small electronic component to large turbine blade. In the past few decades enormous studies have been done to scrutinize the prime factors of impinging jet analogous to flow structure and heat transfer enhancement [1–3].

In a number of fluid-thermal application, consistency of heat transfer with an annular jet over a broad area are beneficial as an alternative of localised heat transfer near the impingement area with the circular jet. Jet impingement technique using an annular jet probably a prospective method of succeeding more consistency and enrichment in terms of heat transfer than the circular jet impingement method. The flow and geometric parameters for annular jet impingement is mainly categorized by; (a) “blockage ratio $BR = D_i/D_o$”, where $D_i$ and $D_o$ are the inner and outer diameters of the annular jet, (b) “non-dimensional jet exit to impingement plate distance, $H = h/D_o$”, where $h$ is the dimensional distance from jet exit to impingement plate, and (c) “Reynolds number, $Re$”. In earlier effort of Maki and Yabe [4], it was revealed that a possibility of occurrence of the “reverse stagnation point” may develop in the impingement surface subject to the jet height from the surface. They also observed that the Reynolds number does not play a major role on heat transfer at this particular reverse stagnation point where the characteristic length of the jet is defined by the jet width. Meanwhile Ramos [5] reported one dimensional analytical and numerical studies on liquid annular jet. The effect of nozzle arrangement was of subject of interest in some of the previous works of Shuja et al. [6] where they observed higher heat transfer for the conical nozzles as compared to the annular nozzles.
The prime objective of the present study is to estimate consistently the performance of the turbulent annular jet through numerical simulations with three dimensional model and also compare with the circular jet as well as established two dimensional model keeping same mass and momentum efflux at the nozzle exit. Recently, some similar studies on the annular jet was performed by Chattopadhyay \cite{7,8} considering two dimensional axisymmetric case. They suggested to consider the inner diameter of the nozzle as characteristics length to define the Reynolds number \((Re)\) and to choose the outer diameter of the nozzle in a systematic manner so that the same level of mass and momentum efflux can be achieved at the nozzle exit for both annular and circular jet same inner diameter. They also suggested that it would be more meaningful to define \(Re\) with the inner diameter of the nozzle as characteristics length and choose the outer diameter in such a way so that the mass and momentum efflux at the nozzle exit is at the same value as in the case of a circular jet. Recently, Dutta and Chattopadhyay \cite{9} performed numerical simulations on annular jet impinging over a hot moving surface. They observed that, the flow structure over the impinging surface was strongly influenced by the surface velocity which leads to the enhancement of heat transfer.

In the present study, inner and outer the diameter of the annular jet are considered as \(D_i\) and \(\sqrt{2}D_i\) respectively, so that the area under the annular portion is same as the area of a circle with diameter \(D_i\). Utilising that concept, a three dimensional model is created for present study. The schematic diagram of present model is shown in figure 1. It was recognized by the literature that present computational domain is adequate to exclude the end effects \cite{10}. The comparison of performance of the annular jet of inner diameter \(D_i\) was done with a circular jet of same inner diameter \(D = D_i\) and the impingement height \(h = 2D_i\).

2. Problem description and solution procedure

The flow is assumed to be incompressible with constant properties. Air is considered for present study with Prandelt number \((Pr)\) of 0.71. The flow is considered in turbulent regime with Reynolds number \((Re)\) of 20,000 which is defined on the basis of velocity and characteristic length of the jet. To measure the heat transfer characteristic of the jet, local temperature gradient is used to calculate the local Nusselt number \(Nu = -\partial T/\partial Z\). The area averaged Nusselt number is calculated by \(Nu_{avg} = Nu(r)\times \partial r/L\) \((L\) denotes the length of the domain in \(r\) direction).
Figure 1: (a) Axisymmetric schematic diagram of circular and annular jet, (b) validation of present model with published data

The Transition SST model [11] is used in the present study to solve the governing equations; i.e., mass, momentum and energy (Reynolds-averaged Navier-Stokes (RANS) equations) with SIMPLE formulation [12] using the commercial CFD code ANSYS Fluent [13]. The detailed information about RANS equations and Transition SST model with its additional equations for turbulence closure are accessible in ANSYS Fluent user manual [13] and not reported here.

Non-uniform structured mesh is used for simulations. Dense grids are placed near the surface of jet impinging region as well as near the jet exit. For selecting appropriate mesh, a grid independence study has been carried out with surface averaged Nusselt number for different grid size. After getting final mesh which gives grid independent results, present model & simulation setup were validated against published data [8]. The result from validation study shows a good agreement with published data.

3. Results and Discussions
For getting a clear picture about flow structure, streamlines on symmetry mid plane passing the centre of nozzle at H=2D is shown in figure 2 for both circular and annular jet flow. In case of annular jet, first circulation was found just after the jet exit i.e., in the initial merging zone; a secondary circulation was also observed near the flat plate. On the other hand no such recirculation zone was found. This circulation sustained near the plate upto 1D in the radial direction from the center of annular jet which can be clearly seen by the radial velocity distribution plot in figure 3.

Figure 2: Streamlines of jet at symmetry mid plane, (left: circular jet; right: annular jet).
Figure 3: Compression of velocity profiles at symmetry mid plane, (left: circular jet; right: annular jet).

In previous studies [7,8] (2 dimensional cases) it was found that, the averaged Nusselt number of the annular jet is near 20\% lesser than circular jet for same mass & momentum efflux condition. But three dimensionally, a clear picture of the distribution of Nusselt number on the flat impingement surface is needed for a concluding remark. For that purpose distributions of local Nusselt number on the surface of jet impingement from present numerical simulations are presented in figure 4. The peak of Nusselt number is found near the center of the plate for circular jet and it shifted away from the center of plate for annular jet. The values of area-averaged Nu at the jet height of $h = 2.0$, is presented in table 1 along with the previous two dimensional study results [8]. It is observed that similar to the two dimensional study, averaged Nusselt number of annular jet is lower compare to the circular jet. It is also observed that the two dimensional studies on annular jet under predicts heat transfer characteristics.

Figure 4: Compression of Nu$_{avg}$ distribution on flat pale, (left: circular jet; right: annular jet).
Table 1: Heat transfer results

|                     | Re   | Nu<sub>avg</sub> (circular jet) | Nu<sub>avg</sub> (annular jet) | Difference(%) |
|---------------------|------|---------------------------------|---------------------------------|---------------|
| Chattopadhyay 2007  | 20,000 | 59.61                          | 54.85                           | 7.98          |
| (two dimensional)   |      |                                 |                                 |               |
| Present study       | 20,000 | 67.26                          | 60.34                           | 10.29         |
| (three dimensional) |      |                                 |                                 |               |

4. Conclusions

Heat transfer from both circular and annular jet keeping same mass and momentums efflux at a particular Reynolds number in the turbulent regime has been simulated using the Transition SST Model. Compressions between Three dimensional flow fields of circular jet with annular jet in terms of streamline with radial velocity profile plot shows the presences of circulation zones only for annular jet. From the heat transfer results of present study it was found that two dimensional studies on annular jet under predicts the heat transfer characteristic.

References

[1] Jambunathan K, Lai E, Moss M A and Button B L 1992 A review of heat transfer data for single circular jet impingement Int. J. heat fluid flow 13 106–15
[2] Weigand B and Spring S 2011 Multiple jet impingement— a review Heat Transf. Res. 42
[3] Carlomagno G M and Ianiro A 2014 Thermo-fluid-dynamics of submerged jets impinging at short nozzle-to-plate distance: a review Exp. Therm. fluid Sci. 58 15–35
[4] Maki H and Yabe A 1989 Heat transfer by the annular impinging jet Exp. Heat Transf. 2 1–12
[5] Ramos J I 1996 One-dimensional models of steady, inviscid, annular liquid jets Appl. Math. Model. 20 593–607
[6] Shuja S Z, Yilbas B S and Budair M O 2001 Local entropy generation in an impinging jet: minimum entropy concept evaluating various turbulence models Comput. Methods Appl. Mech. Eng. 190 3623–44
[7] Chattopadhyay H 2004 Numerical investigations of heat transfer from impinging annular jet Int. J. Heat Mass Transf. 47 3197–201
[8] Chattopadhyay H 2007 Impinging Heat Transfer Due to a Turbulent Annular Jet. Int. J. Transp. Phenom. 9
[9] Dutta P and Chattopadhyay H 2019 NUMERICAL SIMULATION OF HEAT TRANSFER DUE TO ARRAY OF LAMINAR ANNULAR JETS IMPINGING ON HOT MOVING SURFACE PROCEEDINGS BOOK OF FIFTH INTERNATIONAL CONFERENCE ON ADVANCES IN MECHANICAL ENGINEERING: ICAME 2019 (Istanbul) pp 162–70
[10] Laschefski H, Cziesla T and Mitra N K 1997 Evolution of flow structure in impinging three-dimensional axial and radial jets Int. J. Numer. methods fluids 25 1083–103
[11] Abraham J P, Sparrow E M and Tong J C K 2009 Heat transfer in all pipe flow regimes: laminar, transitional/intermittent, and turbulent Int. J. Heat Mass Transf. 52 557–63
[12] Patankar S 2018 Numerical heat transfer and fluid flow (CRC press)
[13] Fluent A 2016 Fluent 15 users guide Lebanon, USA