International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 04, April-2015

Turbulence Modeling and Simulation on Cutting Fluid Flow through a Sudden Contraction Nozzle

R. Chakraborty
Department Of Mechanical Engineering
Dr. Sudhir Chandra Sur Degree Engineering College
Kolkata, India

S. Saha
Department Of Mechanical Engineering
Dr. Sudhir Chandra Sur Degree Engineering College
Kolkata, India

***Abstract*— This work investigated numerically, the performance of a cutting fluid flow through a sudden contraction nozzle. The two dimensional continuity, momentum and energy equations are solved by using finite volume base upwind scheme for different Reynolds Numbers and for a fixed aspect ratio. Computations have been done with respect to wall static pressure, wall shear stress, wall temperature distribution and stream line contours. In case of nozzle wall shear stress, wall static pressure and stream line contours have a greater importance in nozzle. The numerical result has been shown in details in result and discussion sections by using ANSYS FLUENT 13.0.**

Keywords—Streamline contours, wall shear stress, and wall static pressure.

Nomenclature

Re	Reynolds number
u	Velocity in z-direction (ms^{-1})
v	Velocity in r-direction (ms^{-1})
x, y	Cartesian co-ordinates (m)
ρ	Density ($Kg\ m^{-3}$)
μ	Dynamic viscosity ($Kg\ m^{-1}s^{-1}$)
SC	Sudden Contraction
CR	Contraction Ratio
P_s	Wall Static Pressure (N/m^2)
τ	Wall Shear Stress (N/m^2)
X	Position (m)
T	Temperature (K)
L_i	Inlet Length (m)
L_x	Exit Length (m)
H_i	Width at inlet (m)
H_x	Width at outlet (m)
W	Width of pipe (m)

I. INTRODUCTION

Numerical analysis of cutting fluid flow through a nozzle has become an attractive, demanding and important, research area for many researchers. Cutting fluid is used as a coolant and lubricant for metal cutting and machining processes. There are various kinds of cutting fluids such as oils, oil-water emulsions, pastes, gels, aerosols (mists), and air or other gases. In this work water is used as a cutting fluid. Historically, more than 100 years ago, water was used mainly as a coolant due to its high thermal capacity and availability. The use of cutting fluid permits higher cutting speeds, higher feed rates, greater depths of cut, lengthened tool life, decreased surface roughness, increased dimensional accuracy and reduced power consumption. Cutting fluid protect the machined surface from oxidation.

Literature has been enriched with experimental, numerical and theoretical activities on fluid flow through nozzle. From review of literature, it is observed Shih.T.H.et al.[1] highlighted a new model dissipation rate equation and a new realizable eddy viscosity formulation. The new model dissipation rate equation based on the dynamic equation for fluctuating vortices. Webster et al.[2] have analyzed the limitations of current application system and used fluid mechanics to develop flow conditioners to compensate for bend in prices. Hammed et al. [3] have numerically studied the flow characteristics through axis symmetric sudden contractions nozzle using computational fluid dynamics tools. Man et al. [4] have presented a new method for the design of supersonic nozzle tip for high gas pressure laser cutting. Their design is based on the theory of gas dynamics in that the potential energy of high stagnation pressure is converted totally into effective velocity energy. Dhal et al. [5] worked on benchmark turbulent Backward Facing Step (BFS) airflow in detail through a program of tightly coupled experimental and CFD analysis.Vieira et al. [6] have done an experimental investigation on the performance of an emulsion of mineral oil, semi-synthetic and synthetic cutting fluids when face milling AISI 8640 steel with coated carbide tool. Chakrabarti et al. [7]discuss how the finite volume method is used for volume discretization in low Reynolds Number fluid flow in sudden expansion. The entire computational Domain is divided into a number of control volumes. Ezugwu et al. [8] have done the machining of Inconel 718 with whisker reinforced ceramic tool. They have supplied the coolant at a high pressure up to 15MPa and compared to conventional coolant supplies and found that the use of 15MPa coolant supply tend to improve Tool life. Diniz et al.[9]have described tool wear mechanisms influenced by fluid pressure, flow rate and direction of application in turning of AISI 1045 steel using coated carbide tools. Raisee et al. [10] have experimented and found that a sudden contraction creates a relatively small recirculation bubble immediately downstream of the nozzle contraction. Shuja et al. [11] have considered annular nozzle and jet impingement on to a conical cavity, and examined heat transfer rates from the cavity surfaces for various jet velocities, two outer angles of annular nozzle, and two cavity depths. Majumdar .P. [12] Worked on different turbulent modeling approaches such as Reynolds Averaged Nervier-Stokes (RANS) equations, Large Eddy Simulations (LES) and Direct Numerical Simulation (DNS).Wal I. Aly [13] worked on computational fluid dynamics (CFD) to investigate the 3D turbulent flow and heat transfer of coiled tube-in-tube heat exchangers (CTITHEs). The realizable k-ε model with enhanced wall treatment was used to simulate the turbulent flow and heat transfer in the heat exchangers. Bayrakta [14]In the present study, results obtained from three-dimensional incompressible flow over backward-facing step in a rectangular duct using Realizable k-e turbulence model was reported. Y. Habib et al. [15] they simulated an unsteady flow around an airfoil type NACA0012 using the Fluent software. The objective was to control the code on the one hand and on the other hand the simulation of unsteady flows. H. bhowmik [16] worked on three convection modes, forced, natural or free and mixed, related to laminar and turbulent direct air and liquid cooling channel flow, for application to cooling of electronic systems, and with emphasis on useful correlations. Jackson et al. [17] have described how computational techniques have been used to develop axis-symmetric, straight, sonic-line, minimum length micro nozzles for lasermicro-machiningapplication. Promvonge and Eiamsa-ard[18]have described the experimental study of the influence of Conical-nozzle turbulator inserts on heat transfer and friction characteristics in a circular tube. Balakrishna et al.[19] have studied the change of flow patterns during the simultaneous flow of high viscous oil and water through the sudden contraction and expansion in a horizontal conduit. Alkberdi et al. [20] Threw some lights on the research works devoted to the optimization of usable flow rate in grinding operations. Computational Fluid Dynamics is of great help in optimization of the design of grinding nozzles.

From the comprehensive literature review, it has been observed that, none of them have studied the flow characteristics with respect to stream line contour, wall static pressure and wall shear stress, so far. Hence the main aim of this work is to study two dimensional turbulent fluid flow characteristics, flowing through a sudden contraction nozzle for different Reynolds Number ranging from 4200 to 4800 for a fixed aspect ratio 0.3 with respect to stream line contour, wall shear stress, temperature distribution and wall static pressure. The important results have been presented by using ANSYS fluent 13.0, in details in result and discussion section. The major conclusions are reported in the conclusion section.

II. MATHEMATICAL FORMULATION

1) *Governing Equations*

The key line of this work is turbulent flow of cutting fluid through a sudden contraction nozzle with considering heat transfer. The flow at the entrance is considered to be uniform and at the exit absolute pressure is assumed. A schematic diagram of the computational domain is shown in fig1.In this study the dimensional velocity components pressure and temperature are governed by the mass momentum and energy conservation equations. For the turbulent flow in the nozzle with considering heat transfer the dimensional governing equation along the x, y directions are as follows

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0-------------------------(1)$$

$$\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right)=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{\mu}{\rho}\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)-\rho\frac{\partial(uv)}{\partial y}----(2)$$

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}\right)=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\frac{\mu}{\rho}\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)-\rho\frac{\partial(uv)}{\partial x}----(3)$$

$$\rho c_p\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}\right)=k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right)+\mu\left[2\left(\frac{\partial u}{\partial x}\right)^2+2\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)^2\right]---(4)$$

In the above dimensional equations, ρ is the fluid density, μ is the viscosity.

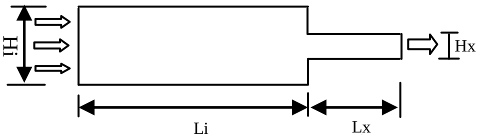

Figure 1 Schematic diagram of the computational domain

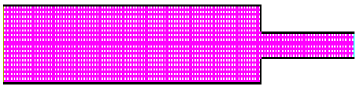

Figure 2 Schematic diagram of the computational nodes

2)*Boundary Conditions*

Three different types of boundary conditions have been applied to the present problem. They are as follows,

i) At the walls: No slip condition, i.e. u = 0, v = 0.

ii) At the inlet: Axial velocity has been specified and the transverse velocity has been set to zero,

i.e. u = specified, v= 0.

iii) At the exit: Constant pressure has been specified.

3)*Numerical Procedure*

The dimensional partial differential continuity and momentum equations (1)-(4) have been solved according to the SIMPLE method in the finite volume formulation by use of a uniform grid in both coordinating directions. The convection terms have been discretized with the help of upwind scheme. Turbulent k- ε model has been selected for simulation. For all calculations, the length, inlet and exit diameter of the nozzle is considered to be 0.136m, 0.018m and 0.005m respectively. During computation, the numerical mesh is considered to be comprising of 136X17 grid nodes along x and y direction respectively. For this simulation Prandtl number can be considered constant. For this problem the value of μ, ρ, k and C_p is equal to 0.001003kg/m-s,998.2kg/m^3,0.6w/m-k, 4182J/kg-k respectively. Putting the above mentioned value of μ, Cp and k into the equation (6) the Prandtl Number becomes 6.99 during computation which is assumed to be constant. Convergence of the iterative scheme is achieved when the normal residuals of mass and momentum equations summed over the entire calculation domain fall below 10^{-5}.The non-dimensional parameters, which have been considered in this work, are

Reynolds number, $\mathrm{Re}=\frac{\rho w u}{\mu}$ ------------------------ (5)

Prandtl number, $\mathrm{Pr}=\frac{\mu c_p}{k}$ ------------------------- (6)

III.RESULTS AND DISCUSSION

In this work the effect of Reynolds number on wall static pressure (WSP),wall shear stress (WSS), streamline contours have been investigated. The parameters during study identified as:

(1) Reynolds number, $4200 \le \mathrm{Re} \le 4800$.
(2) Contraction Ratio, CR=0.3

1) Variation of wall static pressure (WSP)

The pressure of fluid flow through a sudden contraction nozzle is a very important parameter from the metallurgical point of view. The internal pressure generates internal stresses, which increase with increasing pressure in the nozzle. When, the value of internal stress exceeds the value of working stress of the nozzle material, cracks start initiating and propagating. So, pressure is an important parameter for fluid flow. Through a nozzle Turbulent flow has greater impact on wall static pressure and, therefore, in this section an attempt has been made to study the effect of Reynolds number on wall static pressure .Fig (1) shows the variation of wall static pressure for Reynolds number 4200, 4300, 4500, 4700, and 4800 of sudden contraction nozzle for a fixed aspect ratio 0.3. In all the cases wall static pressure gradually decrease along the length of the nozzle from inlet to throat. At the throat of nozzle wall static pressure suddenly decrease due to conversion of pressure to kinetic energy. After the throat wall static pressure again decreases along the length of nozzle from the throat to exit. From fig (1) it has also been observed that wall static pressure increases with increase in Reynolds number.

2) Variation of wall shear stress (WSS)

The study of shear stress distribution in the wall of a nozzle is considered to be another important aspect and it is also related to the scales formation which will decrease the rate of flow on discharge through nozzle. Apart from that, stress distribution in the wall is also considered to be important as the wall shear stress may cause the damage to the wall. The alteration in wall shear stress may lead to further structural changes in the wall of the equipment's. Therefore, the study of the shear stress in the nozzle is important. The wall shear stress distribution for Reynolds number 4200, 4300, 4500, 4700, and 4800 for a fixed aspect ratio 0.3 are presented in the figure (2). In all the cases wall shear stress gradually decrease along the length of the nozzle from inlet to throat. At the throat of nozzle wall shear stress suddenly increases. After the throat wall shear stress again decreases along the length of the nozzle from throat to exit. From figure (2) it has also been observed that wall shear stress increases with increase in Reynolds number.

3) Variation of total temperature

Variations of wall temperature for different Reynolds Number play an important role for flow analysis. Apart from that, temperature distribution in the wall is also considered to be important as it may cause the damage to the wall. Therefore, the study of the wall temperature distributions in the industrial equipment's is important. fig (3) shows the variations of total temperature distribution for different Reynolds Number 4200, 4300, 4500, 4700 and 4800 for a fixed aspect ratio 0.3. It has been observed from the figure that total temperature gradually increases from inlet to throat. At the throat the magnitude of wall temperature suddenly increases and becomes maximum and then suddenly decreases and becomes minimum. Immediately after the throat wall temperature again gradually increases up to the end of the nozzle. It has been also observed that with increasing Reynolds Number wall temperature distribution along the length of the nozzle remain same.

4) Variation of Streamline Contour

Streamlines are a family of curves that are instantaneously tangent to the velocity vector of the flow. These show the direction of a fluid element that will travel at any point of time. A description of the various steady turbulent flow patterns that may be encountered is, perhaps, best rendered by a display of streamline contour for representative conditions. Contours for different Reynolds number 4200, 4300, 4500, 4700 and 4800 for a fixed aspect ratio 0.3 are observed in fig(4). In all the cases recirculating bubbles has been formed at the throat area due to which rate of heat transfer increases which has been shown in the fig (3).

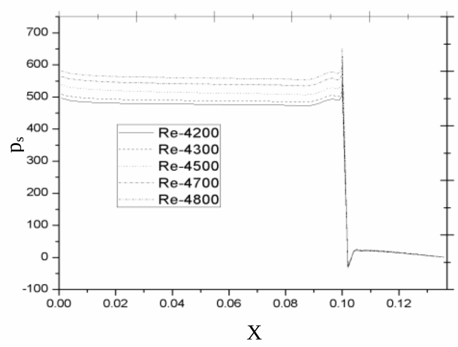

Figure 1. Variation of Wall Static Pressure for different Reynolds number and for a fixed aspect ratio 0.3

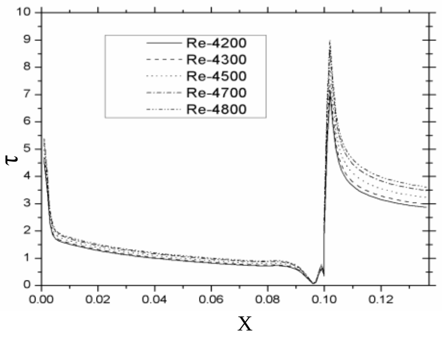

Figure 2. Variation of Wall Shear Stress (WSS) for different Reynolds number and for a fixed aspect ratio 0.3

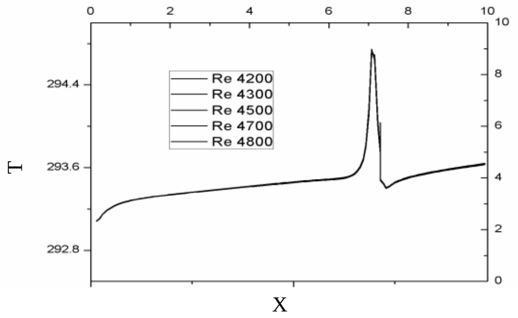

Figure 3. Variation of Total Temperature for different Reynolds number and for a fixed aspect ratio 0.3

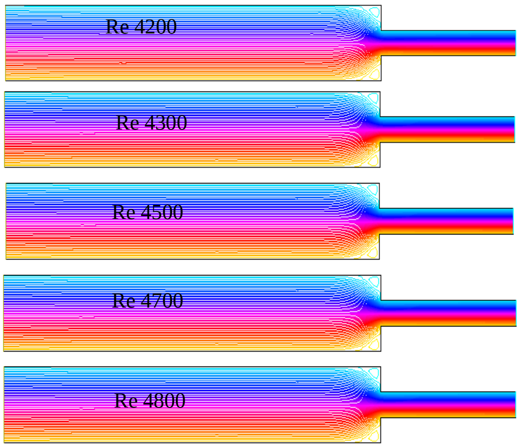

Figure 4. Variation of Stream line contours for Different Reynolds number and for a fixed aspect ratio 0.3

IV.CONCLUSION

In the present work, two dimensional turbulent flow characteristics of a fluid, flowing through a sudden contraction nozzle for different Reynolds number ranging from 4200 to 4800 for a fixed aspect ratio 0.3 have been studied with respect to stream line contour, wall shear stress, wall temperature distribution and wall static pressure. The effect of Reynolds number on important flow parameters has been investigated and this leads to the following conclusions:

1) In all the cases wall static pressure gradually decrease along the length of the nozzle from inlet to throat. At the throat of nozzle wall static pressure suddenly decrease due to conversion of pressure to kinetic energy.

2) Wall shear stress gradually decreases along the length of the nozzle from inlet to throat. At the throat of nozzle wall shear stress suddenly increases. After the throat wall shear stress again decreases along the length of the nozzle from throat to exit.

3) Total temperature gradually increases from inlet to throat. At the throat the magnitude of wall temperature suddenly increases and becomes maximum, and then suddenly decreases and becomes minimum. Immediately after the throat wall temperature again gradually increases up to the end of the nozzle. It has been also observed that with increasing Reynolds Number wall temperature distribution along the length of the nozzle remain same.

4) Streamline Contours have been observed for different Reynolds number (4200-4800) which shows in all the cases recirculating bubbles has been formed at the throat area due to which rate of heat transfer increases.

Thus, it can be concluded that flow characteristics such as wall shear stress and wall static pressure is appreciably affected with the variation of Reynolds Numbers but wall temperature distribution and stream line contours have not been affected at all, with Reynolds Number.

V. REFERENCES

(1) T.H Shih., W.W Liou, A.Shabbir.A,Yang andJ.Zhu., 1994 A New k-_ Eddy Viscosity Model for High Reynolds Number Turbulent Flows-Model Development and Validation, Institute for Computational Mechanics in Propulsion and Center for Modeling of Turbulence and Transition Lewis Research Center Cleveland, Ohio

(2) J. A Webster, C.Cui, Jr B. R. Mindek, . andR.Lindsay, 1995, Grinding Fluid Application System Design, CIRP Annals-Manufacturing Technology, 44, 333 – 338.

(3) J. K. Hammad,.and G. C Vradis, 1996, Creeping Flow of a Bingham Plastic Through Ax symmetric Sudden Contractions with Viscous Dissipation, International Journal of heat and Mass Transfer, 39 (8), 1555 – 1567.

(4) C.H Man, and M.T.Vue, 1997, Design and characteristic Analysis od Supersonic Nozzles for High Gas Pressure Laser Cutting, Materials Processing Technology, 63, 217 – 222.

(5) D.Hall.Stephen, J.Tracie. 2001, A Detailed CFD and Experimental Investigation of a Benchmark Turbulent Backward Facing Step Flow, BarberUniversity of NSW, Sydney, NSW 2052, Australia. sd.hall@unsw.edu.au ,University of NSW, Sydney, NSW 2052, Australia.

(6) J.Vieira, R.A.Machado, andO.E.Ezugwu,, 2001, Performance3 of cutting fluids during face milling of steels, Materials Processing Technology, 116, 244 -251.

(7) S.Chakrabarti , S.Rao and D.K. Maandal ,2010, Numerical simulation of the performance of a sudden expansion with fence viewed as a diffuser in low Reynolds Number regime, J. Engineering for gas Turbine and Power, 132, 114502 – 11.

(8) E.O. Ezugwa,., J.Bonney, D.A.Fadare, and W.F.Sales , 2005, Machining of nickel-base, Inconel 718 alloy with ceramic tools under finishing conditions with various coolant supply pressures, Material Processing Technology, 162 - 163, 609 – 614.

(9) A.E.Diniz, and R.Micaroni, 2007, Influence of the direction and flow rate of the cutting fluid on tool life in turning process of AISI 1045 steel, Machine Tools & Manufacture, 47, 247-254.

(10) M.Raisee, and S.H.Hejazi, 2007, Application of Linear and non-linear Low-Re k-□ Models in Two-Dimensional Predictions of Convective Heat Transfer in Passages with Sudden Contractions, International Journal of heat and Fluid Flow, 28, 429 – 440

(11) S.Z.Shuja, B.S.Yelbas, And S.Khan, 2009, Jet impingement onto a conical cavity : Effects of annular nozzle outer angle and jet velocity on heat transfer and skin friction, Thermal Science, 48, 985 – 997.

(12) P.Majumdar. 2011 Computational Fluid Dynamics Analysis of Turbulent Flow, Department of Mechanical Engineering, Northern Illinois University, Illinois ,U.S.A

(13) Wael.Aly.2014 Computational Fluid Dynamics and Optimization of Flow and Heat Transfer in Coiled Tube-in-Tube Heat Exchangers Under Turbulent Flow Conditions ,J. Thermal Sci. Eng. Appl. 6(3), 031001 (Jan 31, 2014) (10 pages) Paper No: TSEA-13-1084; doi: 10.1115/1.4026120 History: Received April 07, 2013; Revised November 25, 2013

(14) S.Bayraktar. 2013 Numerical Solution of Three-Dimensional Flow over Angled Backward-Facing Step with Raised Upper Wall Yildiz Technical University, Department of Naval Architecture & Marine Engineering, Istanbul, 34349, Turkey (Received July 23, 2013; accepted September 6, 2013)

(15) M. Y Habib..andD.jaaffar 2014 Numerical simulation and modeling of unsteady flow around an air foil Received: 22 March 2014 / Accepted: 22 June 2014 / Published online: 30 June 2014

(16) H.Bhowmik. 2014 Review on convection heat transfer in channel flow with discrete heater arrays for electronic cooling, International Journal of Engineering, Science and Technology Vol. 6, No. 2, 2014, pp. 31-44

(17) M.J.Jackson, Robinson, G. M., Gill, M. D. H. and W.O'Neill ,2005, The effect of nozzle design on laser micro-machining of M2 tool steels, Materials Processing Technology, 160, 198- 212.

(18) Promvonge, Pongjet and Eiamsa-ard, Smith, 2007, Heat transfer and turbulent flow friction in a circular tube fitted with conical-nozzle tabulators, International Communications in Heat and Mass Transfer, 34, 72 – 82.

(19) T. Balakrishna, S.Ghode, G.Das. And P.K.Das, 2010, Oil-water flows through sudden contraction and expansion in a horizontal pipe - Phase distribution and pressure drop, Multiphase Flow, 36, 13-24.

(20) R.Alkberdi, J.A.Sanchez, I.Pombo, N.Ortega, B.S.Izquerdo, and D.Barrenetxea 2011, Strategies for optimal use of fluids in grinding, Machine Tools & Manufacture, 51, 491- 499.