JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

FLOW BEHAVIOR ON GROOVED PIPE WITH A CD – NOZZLE USING CFD

[1]Venkatesh Gandhikota, [2]Srikanth S, [3]Dr. B. S. V. Rama Rao
[1]MTech Student, [2]Assistant Professor, [3]Professor & HOD
[1]Department of Mechanical Engineering,
[1]Pragati Engineering College, Surampalem, India

Abstract : A convergent nozzle that has a divergent path promotes a forced flow due to its throat. The CD-shaped protrusions along the internal surface of a pipe have been identified as potential obstacles to the flow of the pipe. The geometry of the pipe is provided with a fluid medium which passes through it. The generated model has been used to evaluate the inlet velocities and the lineation of the pipe. This software simulates the data collected during the experiments. The results of the tests are presented in terms of their correctness and applicability.

***Index Terms* – CD – Nozzle, Grooved Geometries, Contours.**

I. INTRODUCTION

There are various drawings in the field of design and simulations that are used for the study and design of pipes. They can be of various kinds such as straight, grooved, and wall pipes. A pipe is a major component used for transferring a fluid through a distance. It can provide various modifications in terms of velocity and pressure. Various types of pipes are used for different applications. The grooved pipes have an inherent design that provides an increase in the flow sensitivity of the system.

II. LITERATURE REVIEW

Ramadhan A., et al., worked on a two-dimensional numerical prediction for a forced flow through a modified geometry where the depth of the grooves is different is proposed as the best model for the grooved geometry [1]. Sher Afghan Khan., et al., carried out a study was carried out to evaluate the micro-jets' effectiveness in controlling a base in a 2D planar duct. The results were evaluated using various models and techniques [2]. Dirar., et al., considered the refined mesh geometry of the normal and rifled tube was studied. The results indicated that the heat swap function could be enhanced by the removal of the broad wall geometry [3]. P R Chandra and Ahmed T. have studied the heat transfer and friction characteristics of air flow through a rectangular channel with varying number of walls [4]. Ahmed T., et al., worked on the evaluation is carried out under the constant wall heat flux threshold condition. The results indicated that the heat transfer from the pipe was enhanced by the angle of convergence [5].

III. OBJECTIVE OF THE WORK

A pipe section with triangular protrusions is connected to a CD nozzle to contribute to the better flow of a fluid. The modified pipe geometry is then analysed to determine its velocity and pressure distributions.

IV. DETAILED PROCEDURE

A 2D pipe cross section with a radius of 40cm and a width of 295cm was designed with triangular grooves on both sides. The CD attachment was mounted on one end of the pipe.

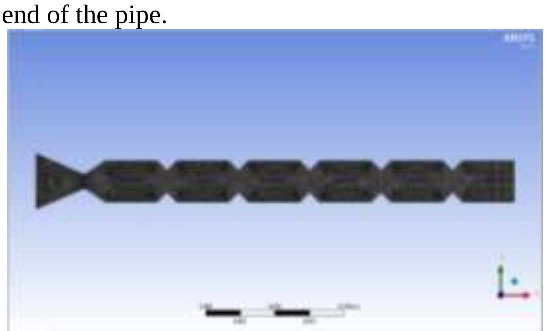

Fig.1: Meshed geometry of the considered model

Table 1: Description for the various models considered

MODEL	DETAIL
M1	Pipe with 1m/s inlet velocity
M2	Pipe with 2m/s inlet velocity
M3	Pipe with 3m/s inlet velocity
M4	Pipe with 4m/s inlet velocity
M5	Pipe with 5m/s inlet velocity

V. METHODS AND STRATEGIES

The concept of the geometry is then computed in the context of the ANSYS – FLUENT software. The resulting geometry is then meshed with the defined metal surface and subjected to various conditions to determine the velocity, pressure, and temperature contours.

4.1 Strategies and Conditions*:*

(a) Continuity equation: according to this equation, the rate at which the fluid enters the geometry is equal to the rate at which the fluid leaves the geometry.

$$m_1 = m_2 \quad (1)$$
$$\frac{dm1}{dt} = \frac{dm2}{dt} \quad (2)$$
$$\rho_1 A_1 U_1 = \rho_2 A_2 U_2 \quad (3)$$
$$A_1 V_1 = A_2 V_2 \quad (4)$$

(b) Momentum and Bernoulli's equation: The momentum equation is related to the second eqn of motion which is Newton's second eqn. It states the difference between the forces acting on the particles and the forces acting on the stream.

F = mass × acceleration

Now,

Consider a small element of the fluid from the entire flow

Let,

dA = cross sectional area of the considered fluid element
dL = length of the fluid element
dW = weight of the fluid element
u = velocity of the fluid element
p = pressure of the fluid element

assume that the fluid is steady, non-viscous, incompressible so that the frictional losses are zero and the density of the fluid is constant

the different forces acting on the fluid are,

a. Pressure force acting in the direction of the flow (PdA)
b. pressure force acting in the opposite direction of the flow [(P+dP)dA]
c. gravity force acting in the opposite direction of the force (dWsin θ).

Therefore,

Total force = gravity force + pressure force

The pressure force in the direction of low

$$F_p = P\,dA - (P + dP) \quad (5)$$

The gravity force in the direction of flow

$$F_g = -dw\, sin\theta \quad (6)$$
$$[W = mg = \rho\, dA\, dL\, g]$$
$$= -\rho g\, dA\, dL\, sin\theta \quad (7)$$
$$[sin\theta = \frac{dZ}{dL}]$$
$$= -\rho g\, dA\, dZ \quad (8)$$

The net force in the direction of flow

$$F = m\,a$$
$$[m = \rho\, dA\, dL] = \rho\, dA\, dLa \quad (9)$$

We have

$$\rho\, dA\, dU = -dP\, dA - \rho\, g\, dA\, dZ \quad (10)$$

{÷ ρ dA}

$$\frac{dP}{\rho} + u\, dU + dZ\, g = 0 \quad (11)$$

(Euler's equation of motion)

By integrating the Euler's equation, we get the Bernoulli's equation

$$\int \frac{dP}{\rho} + \int U\, dU + \int dZ\, g = constant$$

(12)

$$\frac{P}{\rho} + \frac{U_2}{2} + Zg = constant \quad (13)$$
$$\frac{\Delta P}{\rho} + \frac{\Delta U_2}{2} + \Delta Z\, g = 0 \quad (14)$$

(Bernoulli's equation)

(c) kappa – epsilon model:

the kappa – epsilon energy equation is used for the analysis of the turbulent flow. Kolmogorov has first introduced it in the year 1942. Harlow and Nakayama redefined and developed a new set of forced flow equations for the kappa – epsilon energy equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial t}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + p_k + p_b\, \rho\epsilon\, \gamma_k + s_k \quad (15)$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial t}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_1 \frac{\epsilon}{k}(p_k + C_3\, p_b) - C_2 \rho \frac{\epsilon^2}{k} + s_k \quad (16)$$

VI. RESULTS & DISCUSSIONS

The manipulated pipe geometry has been attached with a CD Nozzle at one end. The arrangement is allotted for the analysis of the flow behavior in terms of the velocity and pressure descriptions. The velocity and pressure contours of the manipulated geometry are measured and analyzed respectively. The results are then presented as figures. The variations can be seen in the figures accordingly.

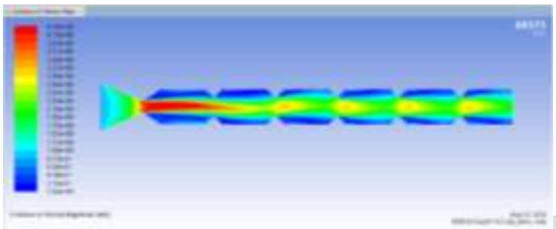

Fig. 2 : Velocity contour at 1m/s inlet velocity

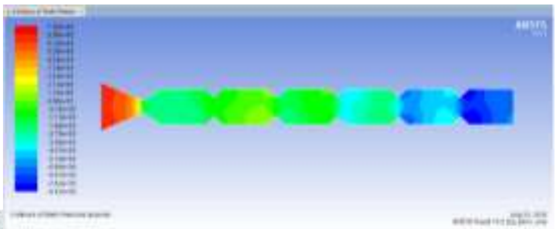

Fig. 3 : Pressure contour at 1m/s inlet velocity

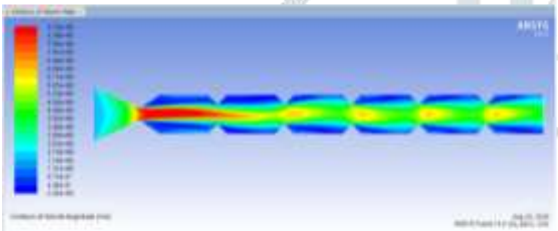

Fig. 4: Velocity contour at 2m/s inlet velocity

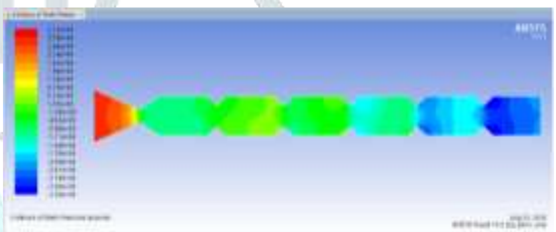

Fig. 5: Pressure contour at 2m/s inlet velocity

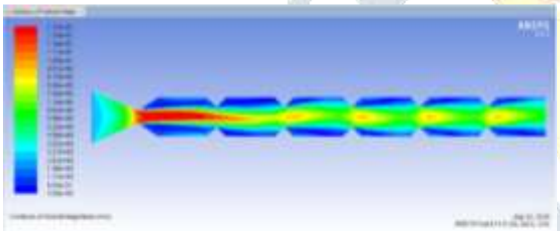

Fig. 6 : Velocity contour at 3m/s inlet velocity

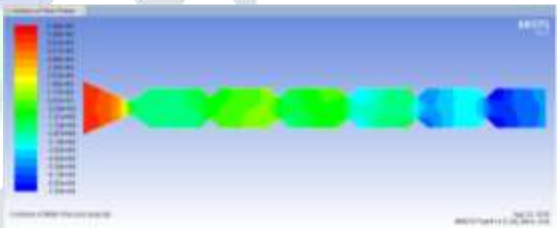

Fig. 7 : Pressure contour at 3m/s inlet velocity

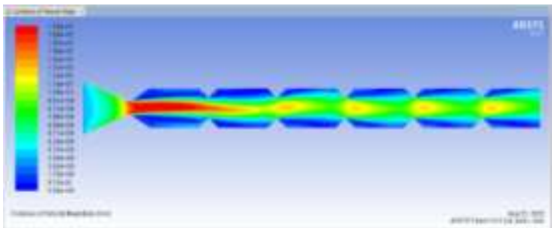

Fig. 8 : Velocity contour at 4m/s inlet velocity

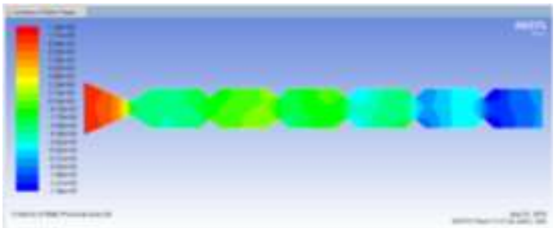

Fig. 9 : Pressure contour at 4m/s inlet velocity

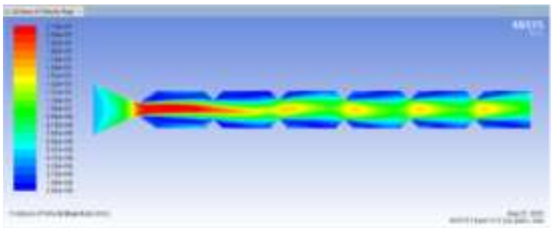

Fig. 10 : Velocity contour at 5m/s inlet velocity

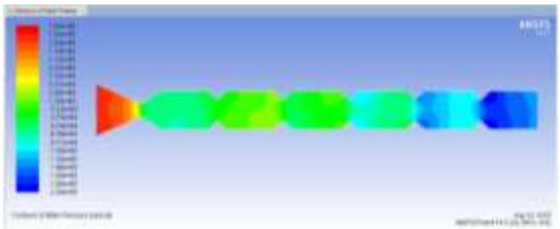

Fig. 11 : Pressure contour at 5m/s inlet velocity

Table 2 : Velocity contours of the considered models.

INLET VELOCITY	OUTLET VELOCITY	
	MIN	MAX
1m/s	0 m/s	4.3632 m/s
2m/s	0 m/s	8.7217 m/s
3m/s	0 m/s	13.0752 m/s
4m/s	0 m/s	17.4695 m/s
5m/s	0 m/s	21.8690 m/s

Table 3: Pressure contours of the considered models.

INLET VELOCITY	OUTLET PRESSURE	
	MIN	MAX
1m/s	-8340.066 Pa	7798.717 Pa
2m/s	-33865.34 Pa	31107.15 Pa
3m/s	-75587.22 Pa	69897.32 Pa
4m/s	-133661.60 Pa	124155.8 Pa
5m/s	-208326.7P0 Pa	193873.9 Pa

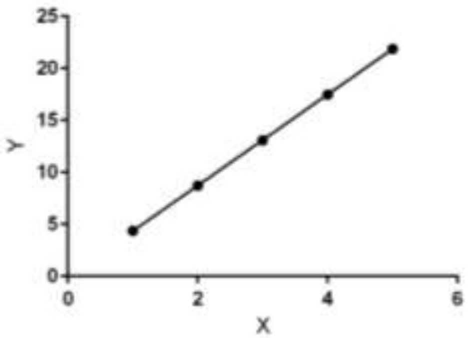

Fig. 12 : Regression plot for the velocity contours

VII. REGRESSION ANALYSIS

The proposed work compares the simulated data with the data collected by means of regression. The data with the derived model is then subjected to the proposed final equation. Simulations are commonly used to generate the maximum velocities for models.

$$\mathbf{y = 4.3759x - 0.0281}$$

Table 4 : Comparison between the simulated data and regression data

INLET VELOCITY	OUTLET VELOCITY	
	SIMULATED DATA	REGRESSION DATA
1m/s	4.3632 m/s	4.3634 m/s
2m/s	8.7217 m/s	8.7277 m/s
3m/s	13.0752 m/s	13.0752 m/s
4m/s	17.4695 m/s	17.4695 m/s
5m/s	21.8690 m/s	21.8690 m/s

Table 5.2.2: R – Sq. details

S	0.0556612
R- Sq.	99.7%
R-Sq. (adj)	98.5%

VIII. CONCLUSIONS

The pipe geometries with the triangular grooves have been attached with a CD nozzle at the end of the pipe. The analysis has been carried out in the ANSYS – FLUENT software for the comparison of velocity and pressure contours. The pipe with the triangular shapes has been attached to a CD nozzle. The analysis revealed that the arrangement of the nozzle and the grooves along the inner surface of the pipe can lead to a substantial increase in the area of the pressure and velocity contours. The velocity contours displayed an increase of 12.91% and pressure contours displayed an increase of 8.15%. The software simulated data when compared to the regression data, depicts the accuracy and the applicability of the present work. The experimental data can be referred for the applications in larger intends.

REFERENCES

[1]. A A Ramadhan., et al., (2012) Groove geometry effects on turbulent heat transfer and fluid flow., Heat Mass Transfer, Springer, pp. 185-195San JY, Huang WC (2006) Heat transfer enhancement of transverse ribs in circular tubes with consideration of entrance effect. Int J Heat Mass Transfer 49(17–18):2965–2971.

[2]. Sher Afghan Khan, Abdul Aabid, Maughal Ahmed Ali Baig (2018). CFD Analysis of CD Nozzle and Effect of Nozzle Pressure Ratio on Pressure and Velocity for Suddenly Expanded Elbows. International Journal of Mechanical and Production Engineering Research and Development. Vol. 8, Issue – 3, pp : 1147 – 1157.

[3]. Dalle Donne M, Meyer L (1977) Turbulent convective heat transfer from rough surfaces with two-dimensional rectangular ribs. Int J Heat Mass Transfer 20:583–620.

[4]. Jaurker AR, Saini JS, Gandhi BK (2006) Heat transfer and friction characteristics of rectangular solar air heater duct using rib– grooved artificial roughness. Sol Energy 80:895–907.

[5]. Lorenz S, Mukomilow D, Leiner W (1995) Distribution of the heat transfer coefficient in a channel with periodic transverse grooves. Exp Thermal Fluid Sci 11:234–242.