International Journal of Civil, Mechanical and Energy Science, 6(4)
Jul-Aug, 2020 | Available: http://ijcmes.com/

Numerical Simulation and Flow Analysis of a 90-Degree Elbow

Prof. Ali Alabdalah, Engineer Lubna Wnos*

Al-Baath University –Syria

*Corresponding Author: lubnaa2803@gmail.com

***Abstract*— *Enhancing the heat transfer by using of curved has a great importance now adays, because the fluid dynamics inside curved tubes offer certain advantages over the straight tubes. The curved shape of tube causes the flowing fluid to experience centrifugal force which depends on its local axial velocity of the fluid particle. Due to boundary layer, the fluid particles flowing close to the tube wall have a lower velocity with respect to the fluid flowing in the core of the tube thus they are subject to a lower centrifugal force. This causes the fluid from the core region is pushed outwards forming a pair of recirculating counter-rotating vortices called secondary flow which enhances fluid mixing and thus heat transfer when compared to that in a straight tube. The aim of this work is to simulate the fluid flow in a 90-degree elbow with CFD analysis using NX-10 software. The temperature contours, static pressure contours and velocity vectors were plotted using NX-10. This contours have been analyzed and discussed to find out when the stream passing from the straight to the curved portion of the pipe is accompanied by an increase of the pressure at the outer wall and a decrease at the inner wall, and by a corresponding a decrease of the stream velocity at the outer wall and an increase at the inner wall.*

***Keywords*— secondary flow, 90-degree elbow, CFD.**

I. INTRODUCTION

A 90-degree elbow is commonly used in many piping systems to connect different components. It is well know that during the flow of fluid in a pipe with elbow, a secondary motion of the flow is developed and it is superimposed on its primary stream flow .The curvature of the 90-degree elbow induces centrifugal force , causing the development of the secondary flow .The intensity of the secondary flow is dependent on the radius of the bend curvature (Rc) and Reynolds number (Re) based on the pipe diameter (D) and velocity (U).

II. LITERATURE REVIEW

[1] Zhi et al. (2008) put forward a swirling flow theory model based on N-S equation numerical solution, which made an improvement on the theoretical research of elbow flow based on traditional free swirl theory. But the disadvantage of this theory is that many differential equations need to be solved with the help of the computer before we get an accurate flow coefficient.

[2] When the non-dimensional radius of curvature of the bend is greater than 1.5 (i.e. *Rc/D* > 1.5), the secondary flow consisting of a pair of counter-rotating vortices are generated. At the same time, the velocity profile of the primary streamwise flow IS distorted and shifted away from the center of the curvature of the elbow. If *Rc/D* is smaller than 1.5, the flow becomes unsteady because of a flow separation occurring immediately downstream of the bend (Weske , 1948; Hellstrom et al., 2011).

[3] The flow in the downstream region of a 90-degree bend is important for the primary and secondary cooling systems of nuclear power plants, where many bends are used to interconnect the components. The distortion in the streamwise velocity profile of the coolant by an elbow can affect the distribution of heat transfer rates. In the case of a bubbly two-phase flow which may occur during an accident in a nuclear power plant, the distribution of the bubbles in the pipe with an elbow is very different compared to that in a pipe without an elbow. Due to the inherent difference in density of the two phases, the gas bubbles are driven toward the centers of the two counter-rotating swirls generated by the elbow (Yadav, 2013).

[4] In the past, several researchers have investigated turbulent flows in pipes with elbows by means of theoretical, experimental, and numerical methods. Weske (1948) experimentally investigated the velocity distribution at the outlet of elbows with various shapes of cross-sections including round, elliptical, square, and rectangular cross sections at Reynolds numbers ranging from 0.2 to 0.6 × 106 for the design of an aircraft duc.

[5] Al-Rafai et al. (1990) performed an experimental and numerical study of a turbulent air flow in circular pipe bends to investigate the influence of *Rc/D* on the flow. During the experiments, two bends with *Rc/D* = 3.49 and *Rc/D* = 6.975 were used, and the flow had the Reynolds number of 34,132.

III. NUMERICAL MODELLING AND SIMULATION

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, and mass transfer, chemical and related phenomena by solving the mathematical equations which govern these processes using a numerical process.

3.1. Governing Equations: The continuity equation, energy equation and the momentum equation govern the flow of the fluid in the curve tubes.

The continuity equation:

$$\frac{\partial \rho}{\partial \tau} + \nabla.(\rho\vec{v}) = 0$$

The momentum equation:

$$\rho\left[\frac{\partial \bar{u}_i}{\partial \tau} + \frac{\partial(\overline{u_j u_i})}{\partial x_j}\right] = -\frac{\partial p}{\partial x_i} + \frac{\partial \mathrm{T}_{ij}}{\partial \mathrm{x}_j} + B_T.g.(T - T_\infty) + \mathrm{F}$$

F: centrifugal forces, and $B_T.g.(T - T_\infty)$: fluid buoyancy forces

Energy equation:

$$\frac{\delta\rho c_{\mathrm{P}}\mathrm{T}}{\delta\tau} + \frac{\delta\rho c_P U_j \mathrm{T}}{\delta x_j} = \frac{\delta\left[k\frac{\delta T}{\delta x_j} - \rho c_P \overline{u_j \tau}\right]}{\delta x_j} + \phi_{\mathrm{i}}$$

3.2. Boundary condition for CFD Analysis:

The simulation of the 3-dimentional flow within elbow was carried out under steady state conditions, so for turbulent flow (Re= 50945 , Ma= 0.024, u_{max} =12.91 m.s^{-1}, and u=10.46 m.s^{-1}) , the Standard *k*-epsilon model is taken and Superheated water steam at 229.5° C and 1.2 Mpa $_{(gage)}$ was used as the working fluid , For this fluid we can deal with it as ideal gas which defined as one for which both the volume of molecules and forces between the molecules are so small that they have no effect on the behavior of the gas .

Table 2 Boundary condition for CFD Analysis

	Type	Value
	Volume Flow	**0.001611 m^3/s**
	Density of steam	**5.94504 kg/m^3**
	Specific volume of steam	**0.168207 m^3/kg**
	Specific heat of steam c_v	**1721.42 j/kg .k**
	Specific heat of steam c_p	**2386.33 j/kg .k**
	Speed of sound	**533.407m/s**
	Dynamic viscosity of steam	**1.71145×10^{-5} Pa.s**
	Isentropic coefficient	**1.29835**
	Critical point properties	
	T_C	**647.3 K**
	P_c	**22.09Mpa**
	Z= PV/RT	**0.94**
Outlet	**Pressure rise**	**0.00002175 Mpa**

3.3. CFD METHODOLOGY

The simulation procedure consists of following steps [6]:

1. Geometry
2. Mesh
3. Solution

1. Geometry:

A 90-degree elbow model have been developed using NX Siemens as seen in figure 1. The detailed geometry parameter is shown in table 1.

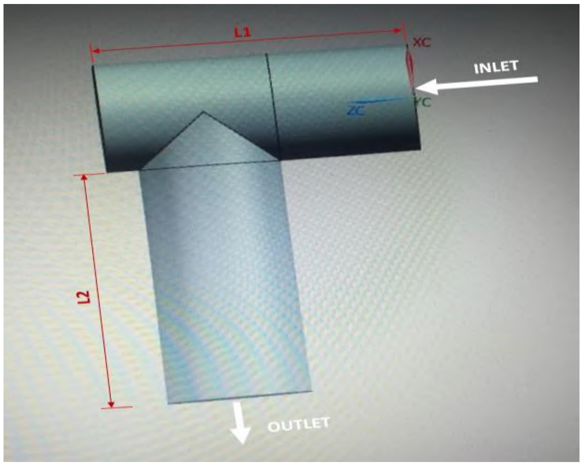

Fig 1: The geometry model of the elbow structure

Table 1: The geometry parameters of the structure model

Dimensional Parameters	**Values**
Inner pipe diameter	14 mm
Outer pipe diameter	22 mm
Inlet pipe length (L1)	30 mm
Outlet pipe length (L2)	36 mm

2. *Mesh:*

Meshing is the discretization of a body into small parts. The grid designates the cells or elements on which the flow is solved. This mesh contains hierarchical elements having both triangular and quadrilateral faces at the boundaries as seen in figure 2.

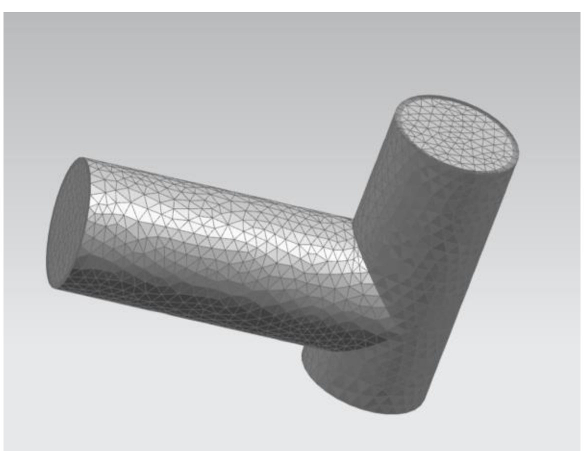

Fig 2 Meshing applied on a 90-degree elbow fluid domain

The mesh details view gave us the following information:

Nodes: 13830

Elements: 41034

3. *Solution:*

In computational fluid dynamics (CFD) software, using a numerical procedure the governing equations are solved simultaneously. Now, the domain is divided into number of elements and the partial differential equations (PDEs) are then applied to each element. Therefore, each element now becomes a domain and PDEs are discretized and applied to that element. The approximate solution of differential equations is obtained by truncated Taylor series expansions in this study:

Flow solver iteration limit: 1000

Flow converged when RMS residual: <0.0002

Flow solver physical time step: 0.5 s

Flow solver heat imbalance: 0.02

Table 3 shows the solution data

y+ information			
	Min y+	Max y+	Ave y+
ADIABATIC FACES	4.39×10^{-2}	2.40	1.99×10^{-1}

IV. RESULTS AND DISSCUSSION

The results are the last section for the CFD NX-10 analysis.

4.1. **The velocity distribution in the elbow**

Figure 3, due to the action of centrifugal forces which pushed the fluid particles from the core region towards the outer wall and originated a biased velocity distribution profiles are varies along the elbow.

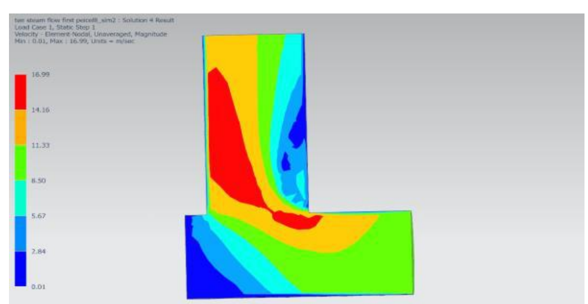

Fig 3 Contours of Velocity in a 90-degree elbow

4.2. The pressure distribution in the elbow

Figure 4 shows the static pressure distribution in the domain as opposite effect of velocity profiles distribution .

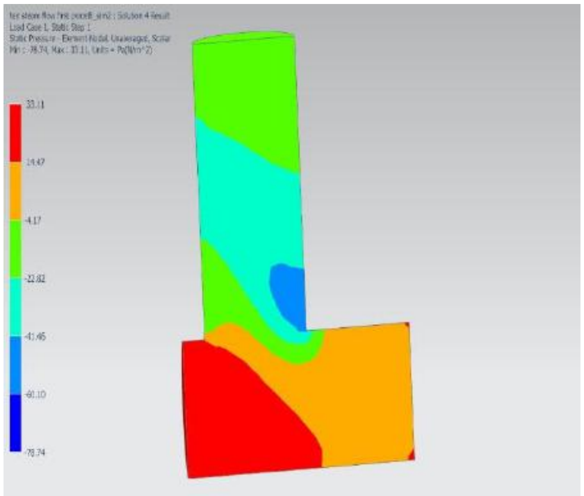

Fig.4 Contours of Static pressure in a 90-degree elbow

4.3 The temperature distribution in the elbow

Figure 5 as a result of this isentropic flow the temperature remain nearly constant.

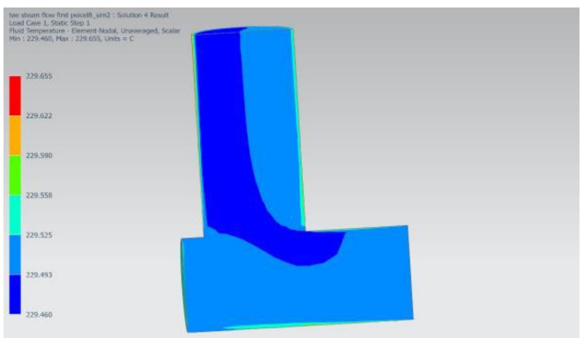

Fig.5 Contours of Temperature in a 90-degree elbow.

V. CONCLUSIONS

An investigation of the flow characteristics of a 90-degree elbow was carried out numerically by using *k*-epsilon turbulence model .The simulation results show that the fluid's motion in the elbow is under the control of the centrifugal forces .The static pressure near the extrados is higher than near the intrados. The velocity in the near intrados region is higher than in the near extrados, which is opposite to the pressure distribution trend. Under the action of the centrifugal forces , the secondary flow emerges in the latter half of the elbow and complicates the flow field by generating two vortexes which rotate in a different direction .The elbow can cause biased velocity distribution and larg velocity fluctuation with low frequency.

REFERENCES

[1] Al-Rafai WN, Tridimas Y D, Woolley NH (1990). A study of turbulent flows in pipe bends. *Journal of Mechanical Engineering Science* 204:399-408.

[2] Hellstrom LHO, Sinha A, Smits AJ (2011). Visualizing the very-large-scale motions in turbulent pipe flow. *Physics of Fluids* 23:011703.

[3] Weske JR, (1948). *Experimental Investigation of Velocity Distributions Downstream of Single Duct Bends*. NACA TN-1471, USA.

[4] Yadav MS (2013). *Interfacial Area Transport across Vertical Elbows in Air-water Two-phase Flow*. Ph. D thesis, PSU, USA

[5] L. Zhi, M. Xianju, and L. Shaofeng (2008). Theoretic research on elbow ducts based on the N-S equation. Journal of Hebei Polytechnic University, vol. 30, no. 1, pp. 41–45.

[6] Hoyas.S.C , and Alabdalah.A , Erasmus Mundus Partnership - Action 2 Lot -Syria "integrated studies for Syrian and European universities" grant agreement nuber 2014-0845/001 - 001, CFD Simulations of burners 23\08\2015-20\06\2016