International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 06 Issue: 09 | Sep 2019 www.irjet.net p-ISSN: 2395-0072

CFD Simulation and Analysis of Fluid Flow through Concentric Reducer Pipe Fitting

Ganeshprasad Jivani[1], Dr. Kishan Naik[2]

[1]Student(M.Tech), University B.D.T College of Engineering, Davangere.
[2]Assistant Professor, Dept. of Studies in Mechanical Engineering, UBDTCE Davangere. Karnataka, India.

---***---

Abstract - *Pipe fittings are the components which are used to change the direction of flow or to reduce the size of the pipe or to connect the different components or to stop the fluid flow through the pipes. Before installing the pipe fittings in the pipeline knowing the flow property variations of fluid while flowing through the pipe fitting is very important. The wrong selection and installation of pipe fitting in an industry may cause the damage or failure in the system and also it may increase the energy losses also. This work is carried out by taking the Water Treatment Plant of Davangere Sugar Company Limited, Kukkuwada, as the reference. In this work the CFD Simulation and Analysis of Fluid flow through the Reducer type Pipe fittings is carried out for the Fluid flowrate of fluid 200m^3/hr. The fluid flow analysis is carried out for the Pipe fittings Elbow, Tee, Concentric Reducer and Eccentric Reducer and their variants. By using SolidWorks and ANSYS Geometry the 3D Modelling of the Concentric Reducer Pipe fitting is completed and using standard k-ε turbulence Model in the ANSYS Fluent 15.0 solver the analysis is completed. The results of the analysis are drawn in the form of Contours and the Values are Tabulated.*

Key Words: CFD, ANSYS, Concentric Reducer, STEP, WTP, ASME, 2D&3D Modelling.

1. INTRODUCTION

Pipe fittings are the components which are used to change the direction of flow (Ex. Elbows, Tees) or to reduce the size of the pipe (Ex. Reducers) or to connect the different components (Ex. Couplings) or to stop the fluid flow through the pipes (Ex. Caps). There are different types of pipe fittings are used in piping, mainly Elbows, Tees, Reducers, Unions, Couplings, Crosses, Caps, Swage Nipples, Plugs, Bushes, Expansion joints, Adapters, Steam traps, Long radius bends and Valves.

Fittings for the piping systems can be expensive and require a proportionally large labour element to install, therefore correct selection and use is of vital importance to a well installed piping system. Every type of piping material has a range of fittings that can be used with it and some piping materials can have multiple different ranges of fittings that can be used. For example, copper piping systems can be installed by bending the pipe and therefore using no elbows, using soldered copper fittings or compression brass fittings depending on the type of service being transferred in the copper pipe. Fittings are available with ends to match the piping installation therefore the following information will not differentiate between welded, threaded or compression but will concentrate on the orientation and the use of the fitting.

2. METHODOLOGY

The methodology of this work contains Selection of pipe fittings, Modelling of selected pipe fittings in SolidWorks and ANSYS Software, after completing the Modelling the Analysis is carried out in ANSYS Fluent 15.0 Solver.

2.1. Selection of Pipe Fittings

In this work Concentric Reducer and Eccentric Reducer are chosen for the CFD analysis. On the basis of the survey carried out in the Water Treatment Plant (WTP) of Davangere Sugar Company Limited, Kukkuwada, the dimensions are collected.

The Concentric Reducer Pipe Fitting is selected such that, by taking the measured values of Concentric Reducer Pipe Fitting as the reference, are then compared with the ASME B16.9-2001 standards. After the comparison of measured Values and the standards, the dimensions of the Concentric Reducer pipe fitting are chosen from the ASME B16.9-2001 Standards. The thickness of the Concentric Reducer Pipe fittings is specified as 5mm thick.

2.2. Modelling

Mechanical Modelling is the Process of Converting 2D Drawings, Sketches or Concepts into live 3D Models. For converting 2D sketches into 3D Models some Mechanical Design Software will be utilized, for Example SolidWorks, CATIA, Siemens NX, Solid Edge, AutoCAD and so on. In this work the three-dimensional models of Concentric Reducer Pipe fitting are created using the SolidWorks Software. The models are done as per the standard dimensions provided in 2D Drawings of Concentric Reducer Pipe fitting. The 3D Model of the Concentric Reducer Pipe fitting of Height, H=203mm is shown in Fig-1, similarly 3D Models are created for the Heights H=178mm and 330mm. After completing the modelling, the models are then exported into STEP file. Because which is one of the acceptable file formats for transferring the 3D model from SolidWorks to ANSYS.

After completing the SolidWorks Modelling, the exported files of the pipe fittings are then imported to the ANSYS Geometry software in ANSYS Workbench R15.0. then the unwanted Nodes, Edges and unwanted surfaces are removed for reducing the complexity of the analysis of the model this technique is called Geometry Clean-up. After the Geometry Clean-up process surfaces are created at both ends of the pipe fitting by using the option the surfaces from edges for getting an enclosed hollow volume, from that hollow Volume the fluid is extracted from the pipe fitting in the form of solid, by using Fill option with all bodies, because of it two domains will be created such as Solid Domain i.e. Pipe Fitting and Fluid Domain i.e. Extracted Fluid, in these two Domains the solid domain don't place any role in the analysis because it is a static one that's why it will be suppressed and for further Analysis part the Fluid Domain is carried. After that by using face extrusion on both entry and exit faces of Fluid Domain are extruded by 500 mm and 1500 mm by length for the purpose of getting the uniform and Steady flow at the entry and exit of the pipe fitting respectively and which makes easy to study the flow Parameters variation at the entry and exit of the pipe fittings, because at the exit of the pipe fitting we won't get the study flow suddenly, to avoiding this problem more length is provided at the exit of the pipe fitting. The ANSYS Geometry Model is shown in Fig-2.

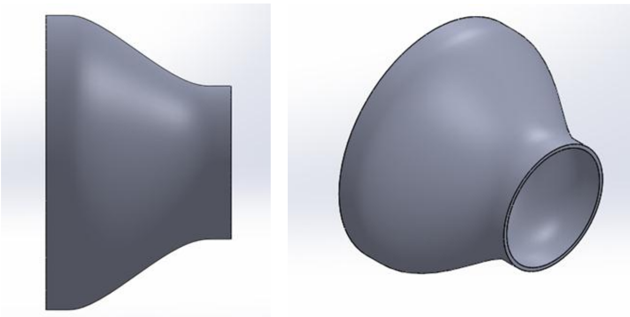

Fig-1: SolidWorks Modelling of Concentric Reducer Pipe Fitting of Height, H=203mm

The extracted fluid model is then divided into different regions by using Named Selection for the purpose of making easy to apply the boundary conditions. After completing this process, the geometry is then imported to Mesh, where CFD Mesh will be generated using ANSYS Mesh software is shown in Fig-3. During generation of Mesh Inflation is provided at the surface of the fluid domain for getting fine mesh at the surface of the fluid domain, this fine mesh will help to study the small variations in the fluid flow properties at the surface of the fluid domain, it means the surface which is in contact with the pipe fitting and also the layers near to it.

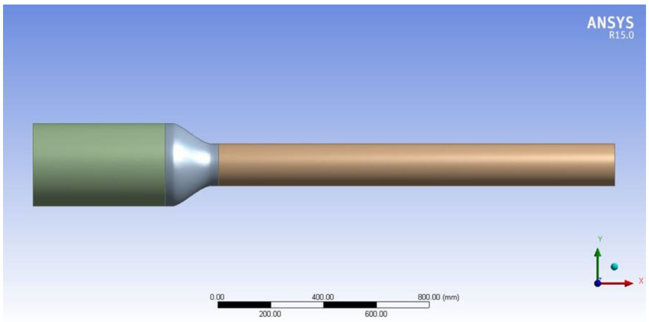

Fig-2: ANSYS Geometry Model of the Concentric Reducer Pipe Fitting of Height, H = 203 mm

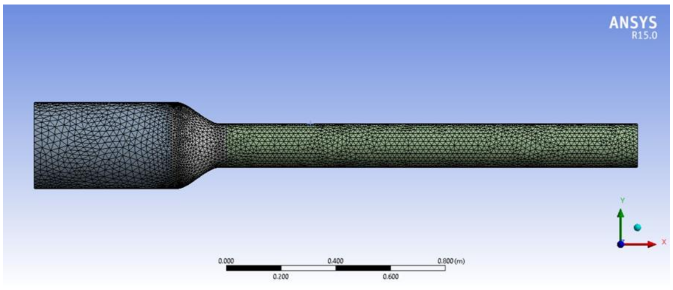

Fig-3: ANSYS Mesh Model of the Concentric Reducer Pipe Fitting of Height, H = 203 mm

3. RESULTS

The analysis of fluid flow through the Concentric Reducer is carried out by varying the height of the reducer 'H' as 178mm, 203mm and 330mm. These standard heights of the Concentric Reducer are taken from the ASME B16.9-2001. The results of this analysis are taken in the form of Contours and values of Static Pressure, Velocity, Turbulent Kinetic Energy and Wall Shear Stress are shown below. The flowrate of 200 m^3/hr of water is used as the inlet mass flow rate.

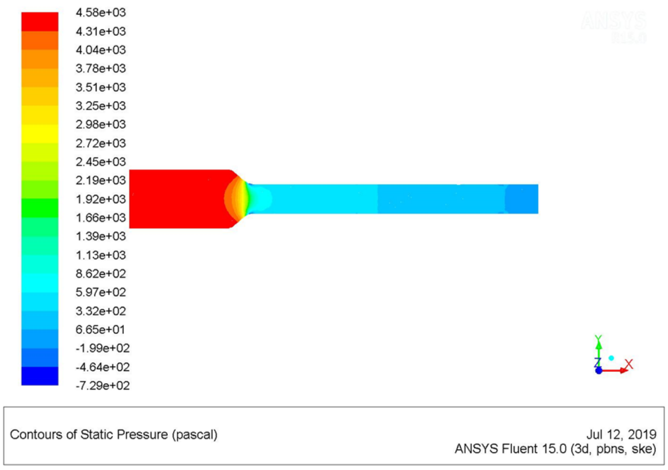

Fig-4: The Sectional View of Static Pressure Distribution in Concentric Reducer Pipe Fitting of Height, H = 178 mm

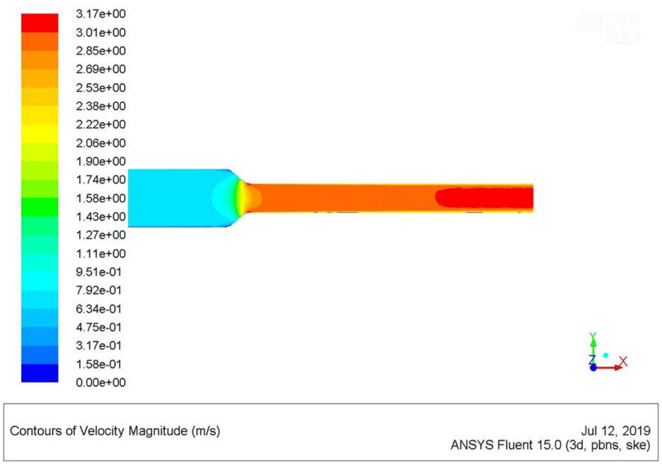

Fig-5: The Sectional View of Velocity Distribution in Concentric Reducer Pipe Fitting of Height, H = 178 mm

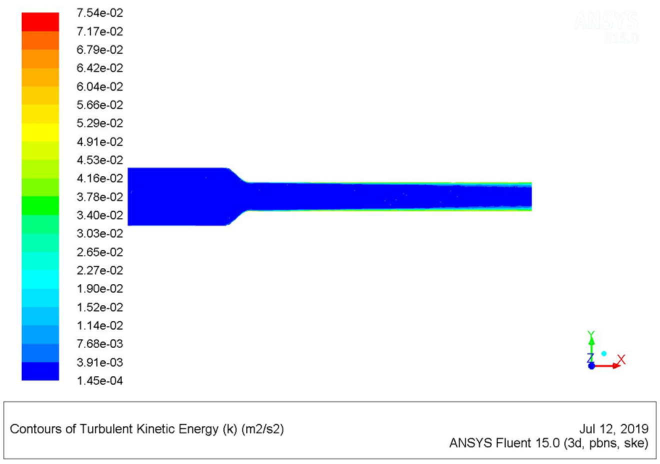

Fig-6: The Sectional View of Turbulent Kinetic Energy Distribution in Concentric Reducer Pipe Fitting of

Height, H = 178 mm

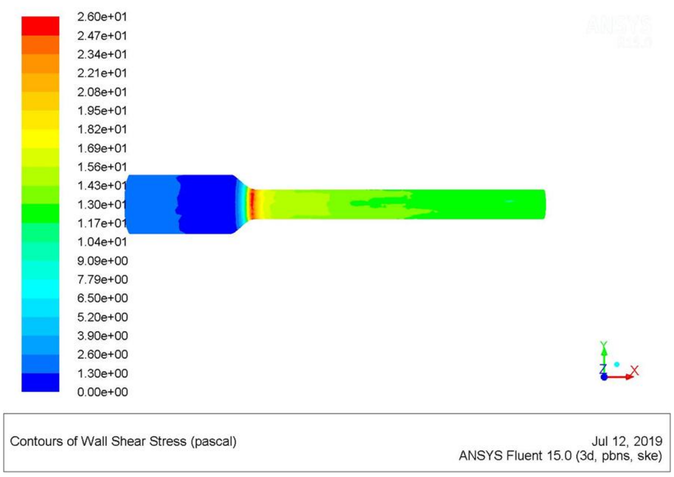

Fig-7: The Surface View of Wall Shear Stress Distribution in Eccentric Reducer Pipe Fitting of Height, H = 178 mm

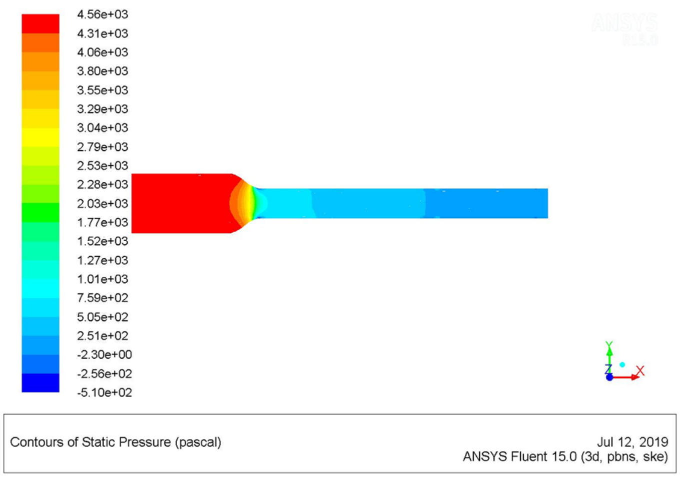

Fig-8: The Sectional View of Static Pressure Distribution in Concentric Reducer Pipe Fitting of Height, H = 203 mm

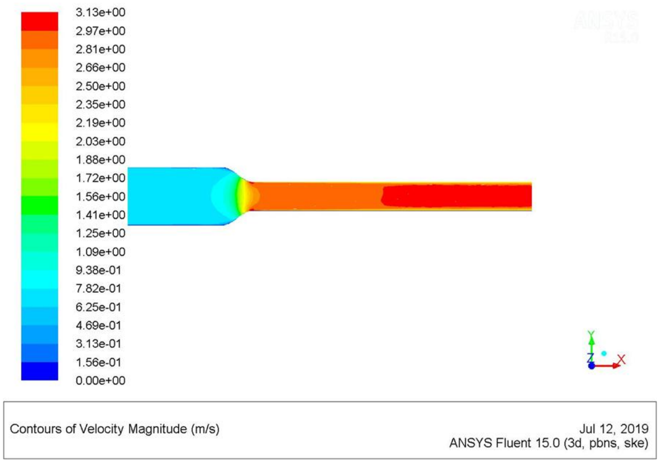

Fig-9: The Sectional View of Velocity Distribution in Concentric Reducer Pipe Fitting of Height, H = 203 mm

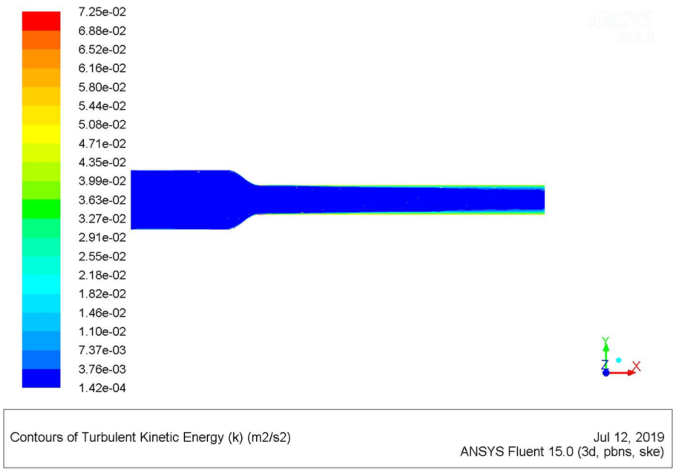

Fig-10: The Sectional View of Turbulent Kinetic Energy Distribution in Concentric Reducer Pipe Fitting of

Height, H = 203 mm

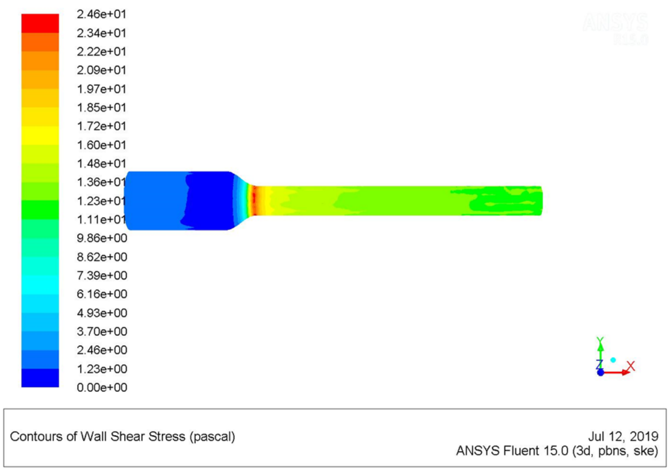

Fig-11: The Surface View of Wall Shear Stress Distribution in Eccentric Reducer Pipe Fitting of Height, H = 203 mm

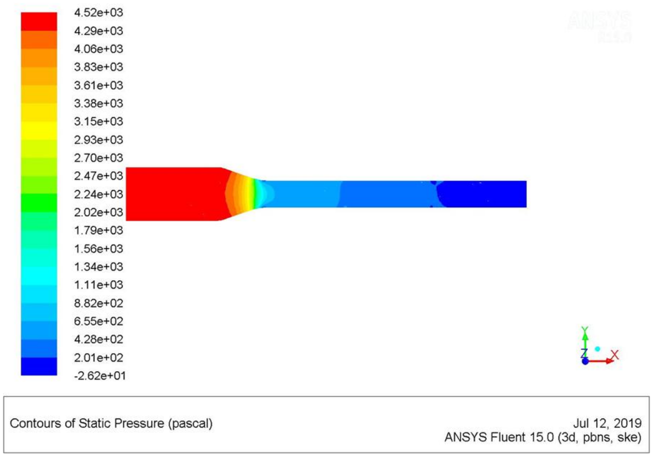

Fig-12: The Sectional View of Static Pressure Distribution in Concentric Reducer Pipe Fitting of Height, H = 330 mm

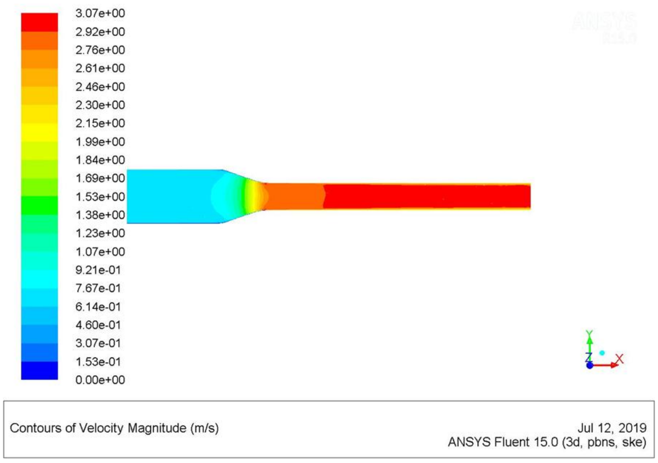

Fig-13: The Sectional View of Velocity Distribution in Concentric Reducer Pipe Fitting of Height, H = 330 mm

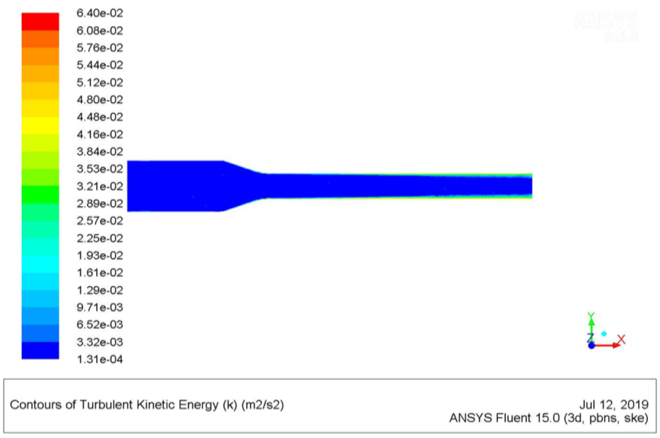

Fig-14: The Sectional View of Turbulent Kinetic Energy Distribution in Concentric Reducer Pipe Fitting of

Height, H = 330 mm

Fig-15: The Surface View of Wall Shear Stress Distribution in Eccentric Reducer Pipe Fitting of Height, H = 330 mm

The Fig-4, Fig-8 and Fig-12 shows the sectional view of the Static Pressure distribution in the fluid which is flowing through the Concentric Reducer of Height, H=178mm, 203mm and 330mm respectively. The cross-sectional area perpendicular to the axis of the reducer at the inlet is greater than the cross-sectional area at the outlet, due to the gradual reduction in the cross-sectional area along the height of the reducer, the pressure at the inlet section will starts to increase, which will be more than that of the pressure at the exit section. The increase in the pressure at inlet is inversely proportional to the height of the Concentric Reducer. The Concentric Reducers Exactly functions like Nozzle. As the height of the reducer increases the pressure at inlet starts to reduce, if it starts to decrease the pressure at the inlet section will starts to rise.

The Fig-5, Fig-9 and Fig-13 shows the distribution of velocity in the fluid flowing through the Concentric Reducer of Height, H=178mm, 203mm and 330mm respectively. The difference in the pressure at inlet and exit of the reducer creates the high velocity at the exit of the reducer as compared to the inlet of the reducer. The high pressure at the inlet of the reducer is utilized to increase the velocity of the fluid which is flowing through the Concentric Reducer. The high velocity of the fluid will cause the velocity difference in between the fluid at the axis of the Reducer and the fluid which near and attached to the wall of the Reducer and pipe. This difference in the velocity starts to generate eddies in the less velocity region this tends to the generation of turbulence effect in the fluid.

The Fig-6, Fig-10 and Fig-14 shows the Turbulent Kinetic Energy Distribution in the fluid flowing through the Concentric Reducer of height, H=178mm, 203mm and 330mm respectively. The high velocity of the fluid and the friction between the fluid and the walls of the pipe and pipe fitting causes the Wall Shear Stress. The Generation of Wall Shear Stress is more at the neck of the Concentric Reducer, because the fluid gets compressed and pressure energy in the fluid gets converted into velocity during this process more shearing action takes place between the fluid and wall of the pipe fitting. In high pressure region the flowrate of the fluid is not that much high in the reducer therefore less shear stress exists at the entry and moderate value exists at the exit is shown in Fig-7, Fig-11 and Fig-14 for the height of the reducer as H=178mm, 203mm and 330mm respectively.

Table-1: Analysis Results of Concentric Reducer Pipe Fitting of Different Height

Concentric Reducer Pipe Fittings of different Height (H)		Parameters			
		Static Pressure (N/m^2)	Velocity (m/s)	Turbulent Kinetic Energy (J/kg)	Wall Shear Stress (N/m^2)
H=178 mm	Inlet	4486.794	0.7183	0.0019348	0
	Outlet	0	2.83009	0.0138362	0
	Wall	2118.584	0	0.0272504	8.31702
	Interior Fluid	2589.365	1.62282	0.0049100	0
	Net	2573.297	1.61403	0.0058017	0.33256
H=203mm	Inlet	4483.281	0.7183	0.0019348	0
	Outlet	0	2.83042	0.0137994	0
	Wall	2136.185	0	0.0270689	8.27345
	Interior Fluid	2559.202	1.70064	0.0048893	0
	Net	2544.847	1.62992	0.0057898	0.33648
H=330mm	Inlet	4477.250	0.7183	0.0019348	0
	Outlet	0	2.82899	0.0138332	0
	Wall	2221.845	0	0.0260263	8.00946
	Interior Fluid	2435.664	1.77133	0.0050242	0
	Net	2429.316	1.68928	0.0059733	0.36271

The above Table-1 shows the analysis results of fluid flowing through the Concentric Reducer of height H = 178 mm, 203 mm and 330 mm. By comparing the results of other two cases on the basis of Static pressure, Velocity, Turbulent Kinetic Energy and Wall Shear Stress, the Concentric Reducer of height H = 203 mm gives the best result. The variation in the height of the reducer causes the change in the fluid flow properties, if the height of the reducer reduces, the slope between inlet and outlet section increases, then the pressure at the inlet section increases but, if the height of the reducer is increases, the slope between inlet and outlet section reduces, then the pressure acting on the wall of the Pipe and pipe fitting increases instead of inlet section.

4. CONCLUSIONS

1. The analysis of fluid flow through Concentric Reducer is carried out in three different cases by keeping diameters of inlet and outlet sections constant and varying the height of the reducer, such as

    a. Concentric Reducer of height, H = 178 mm

    b. Concentric Reducer of height, H = 203 mm

    c. Concentric Reducer of height, H = 330 mm

2. The CFD analysis of all three cases above mentioned is completed and the results are tabulated, with this Contours of Static Pressure, Velocity, Turbulent Kinetic Energy and Wall Shear Stress are drawn.

3. These three cases of analysis play the same role in the application therefore the comparison is done in these three cases. By comparing the first and third case results with the second case result it is clear that the change in the height of the reducer causes the change in the fluid flow properties.

4. If the height of the reducer is more, then the energy losses due to turbulence and the wall shear stress will be more, if it is less, then the energy losses due the generation of inlet pressure and the turbulence will be more.

5. By seeing the second case results we can conclude that Concentric Reducer of Height, H = 203 mm gives the best results in the application, therefore maintaining the height of the reducer as per the standards is important.

In the Concentric Reducers Maintaining the Height of the Reducer Plays the main role in the reduction of the Energy losses during the Flow of Fluid through Concentric Reducer pipe fitting.

REFERENCES

[1]. Balvinder Singh, Harpreet Singh and Satbir Singh Sehgal, "CFD Analysis of Fluid Flow Parameters within a Y-Shaped Branched Pipe", Vol. 2 Issue 2 March 2013.

[2]. Jaroslav Stigler, Roman Klas and Oldrich Sperka, "Characteristics of the T-Junction with the Equal Diameters of all Branches for the Variable Angle of the Adjacent Branch", 2014.

[3]. V. H. Bansode, Achari Swati Nivrutti, Pawar Saurabh Nandlal and Bhardande Nikita Sanjay, "Pressure Drop Analysis of Inlet Pipe with Reducer and Without Reducer Using CFD Analysis", Vol. 4, Issue 3, April - May 2015.

[4]. Prasun Dutta and Nityananda Nandi, "Effect of Bend Curvature on Velocity & Pressure Distribution from Straight to a 90° Pipe Bend - A Numerical Study", Vol:2(4), 2016.

[5]. Mr. R. Satheesh Kumar and Mr. Anil Kumar Pentakota, "Computational Fluid Dynamics Simulations of Pipe Elbow Flow", Volume 9 /Issue 2 / September 2017.

[6]. M. Venkateswararao, P. Kumar Babu and S. Srikanth, "Design and CFD Analysis of Different Pipe Joints used in Water Supply Industries", Volume No. 5, Issue No. 4, June-July 2017

[7]. P. A. Aswatha Narayana and K. N. Seetharamu, "Engineering Fluid Mechanics" Narosa Publishing House, 2012.