# TWO PHASE FLOW SIMULATION IN "Y" JOINT DUCT USED IN PIPELINE APPLICATIONS

**[1] JALA HARIKA [2] Mr. M.PRASAD**

[1] PG Scholar, Department of MECH, **Methodist College of Engineering & Technology.**

Abids, Hyderabad – 500 001.

[2] Assistant Professor, Department of MECH, **Methodist College of Engineering & Technology.**

Abids, Hyderabad – 500 001.

## Abstract

Flow diversion through Y branch and T junction ducts are extensively used in pipeline applications for uniform distribution of fluids. In order to understand flow uniformity, typical geometrical situations of different configurations are developed for CAD models using Altair Hyper mesh. The of curvatures in the duct geometry flow takes sharp turns leading to severe turbulent fluctuations whose consequent effects give rises to unequal flow at its exit location. In many physical situations, these ducts are subjected to more than one fluid is called two phase air-dust particles or water and sand particles. In order to understand flow mechanism, geometrical models generated are further subjected computational mesh on which conservation equations are iterated to obtain successful solution with inlet velocity and exit pressure conditions. After successful flow simulation discrete phase module is activated to track particle trajectories along with impact parameters. From the solution, several parameters through graphical forms are visualized and interpreted. In order to improve velocity and flow path in computational domain, several simulations are repeated with refinements in the mesh elements in terms of type and density. It is expected that two phase flow analysis dealing with fluid-particles will provide several insights for design of pipeline layout for proper flow distribution.

## 1. Introduction

Pipe networks are very common in industries, where fluid or gases to be transported from one location to the other. The pressure loss may vary depending on the type of components coming across in the network, material of the pipe, the fluid that is being transported through the network and pipe fitting. The analysis of pipe network is very important in engineering point of view. A lot of engineering problem dealt with it. Due to rigorous engineering application and implications the analysis is important. Fittings are used in pipe and plumbing systems to connect straight pipe or tubing sections, to adapt to different sizes or shapes, and for other purposes, such as regulating or measuring fluid flow. A wye branch (as the name implies, this fitting device is "Y" shaped) allows splitting a branch line equally in two directions. It is a fitting with three openings and is used to create branch lines. A standard wye allows one pipe to be joined to another at a 45 degree angle. Wyes are similar to this except that the branch line is angled to reduce friction that could hamper the flow and that the connection is typically at a 45-degree angle rather than a 90-degree angle . Economy wyes are often spot welded together; industrial wyes have a continuous weld at each seam. In the present work, fluid dynamic analysis for different Y- shape connections will be done, for laminar. The effect of bend angle, pipe diameter, pipe length, Reynolds number on the resistance coefficient is studied and. It

was observed that resistance coefficient vary with the change in flow.

Multiphase flow in reciprocating internal combustion engines is widely used for prediction of thermodynamic parameters of gases in the branched ducts of inlet and outlet systems. These ducts are connected to the branch of "Y" and "T" type, where the gas is transferred from supplier pipes to the outflow pipes in varying of changing in the time gas properties. In the engine ducts the gas parameters are a function of temperature and chemical composition. Taking into account the dynamics of the gases with incident and reflected pressure waves the filling and emptying of the cylinder depends on the flow in the branches particularly in multi-cylinder engines. The concept of calculations, based on the balance of mass, energy, and the values of the pressure loss between the pipes oriented relative to each other, has to determine the thermodynamic parameters of gas and the mass fraction of each component in the branch node. Although the model refers to the three branched ducts, and test results are considered to "T" type of branched pipes, one can apply it to the layout consisting of more pipes. This allows a definition of incoming or outgoing pressure waveforms in the ducts from the valves or the ports of individual cylinders in internal combustion engines. Unsteady gas flow in the engine ducts influences on the mass filling ratio of the gas in each cylinder. In the simulation of any engine, where its performance is related to the mass trapped in the cylinder and pressure wave motion through the branched pipes was considered .

Flexible duct junction boxes are commonly used to split airflows in residential heating, ventilation, and air conditioning (HVAC) systems of insulated flexible ducts. Junction boxes typically are made of fiberglass duct board or insulated sheet metal. Junction boxes are popular and, properly designed, can result in an HVAC system that performs well. However, junction boxes that are poorly designed can have high pressure losses relative to other duct fittings. That is, each junction box creates a higher resistance (e.g., duct friction) that may impede the ability of the air handler unit (AHU) to move the air through the ducts, which in turn may result in occupant comfort complaints. With any duct runout, balancing dampers are needed to adjust the pressure of airflow through the outlets because self-balancing systems are not realistic. Junction boxes are not necessarily the most energy-efficient option for an HVAC system. It is important to weigh their benefits as a low first-cost and easily installed option versus the benefits of using a trunk and branch system that will provide more certainty to Effective Length values information provided  other calculation tools and  information  provided  by  manufacturers' performance data. This measure guideline assumes that designers have an understanding of load calculations for system and room airflow requirements and duct run attributes such as airflow rate, velocity, and friction rate.

## 1.2. Problem definition

The transport of fluids in branched duct generated pressure losses and turbulence due to different turns around sharp corners.The curvatures in the duct geometry   flow takes sharp turns leading to severe turbulent fluctuations whose consequent effects give rises to unequal flow at its exit location. Performance degradations of ducts will be more significant if the working fluid is of particulate laden nature.

Besides thermal gradients on wall surfaces, structurally ducts are subjected to large deformations altering principal stresses.

In order to understand multiphase flow mechanism in the branched duct, detailed numerical studies are essential under three dimensional environments.

## 3. EXPERIMENTAL METHODOLOGY

### 3.1 Overview of Altair-Hyperworks

HyperWorks Desktop is an integrated user environment for modeling and visualization. It can be used to pre-and post-process finite element and multi-body dynamics simulations as well as to manage and visualize simulation and test data.

### 3.1.2 Construction of Geometrical Model

When Altair-Hypermesh is activated on the desktop/laptop, graphical user display opens as shown in Figure 3.1 for development of geometrical model using bottom up approach.
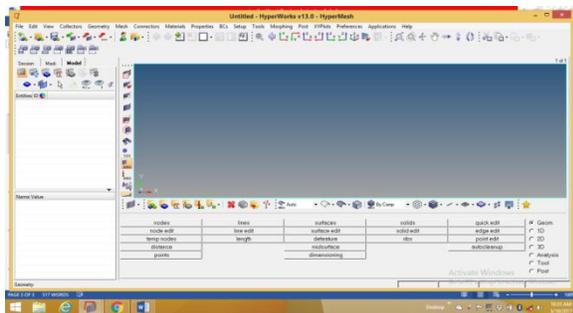


Figure: 3.1 Development of Geometrical Model Using Bottom up Approach

With the options available in 2dimensional options, the line drag command is used to generate the cylindrical tube Surface in xy direction. In the line drag options, one must ensure that drag geometrical option – line to be selected and type should be toggled to surface.

With these options, the geometry of cylindrical tube in xy plane will be generated. Using tools, the lines drawn can be deleted.   The topologically, the cylindrical tube is not closed and hence shows its ends are in red colour as shown in Figure 3.2

Using windowing options undesired surfaces will be deleted. The resulting error free closed surface of y branch is shown in figure.3.3
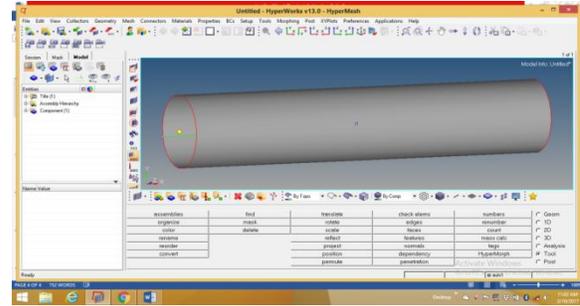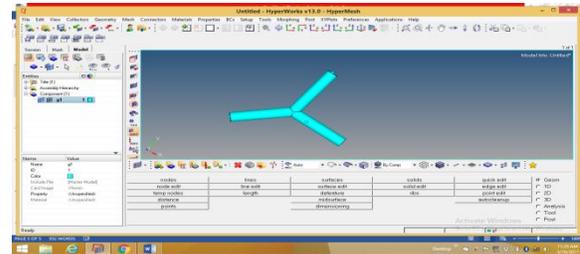


Figure: 3.2 Geometry of Cylindrical Tube



Figure3.3: selecting all surfaces and executing trim option

The last step left for completion of geometry is generation of volume using the surfaces of geometrical model. The software provides generation of solid part option once activated it expects error free closed surfaces. The moment all surfaces are selected and execute the create operation it generates solid part of the model as shown in Figure 3.4
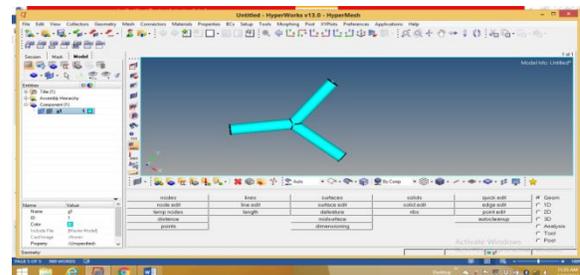


Figure: 3.4 Generation of volume using the surface of geometric model

In order to extend the geometry for structural analysis, thickness of 2 units above inside cylinders are added and associated surfaces at junction are trimmed as shown in the Figure 3.5

After checking the surfaces through topology the volumes of cylindrical y duct with thickness has been created as shown in the Figure 3:6
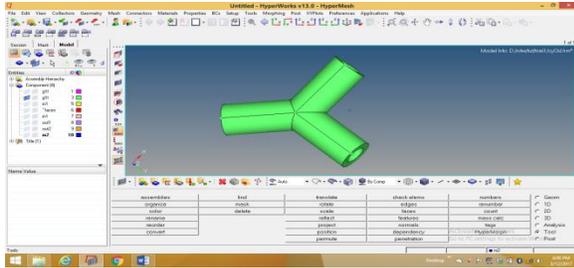
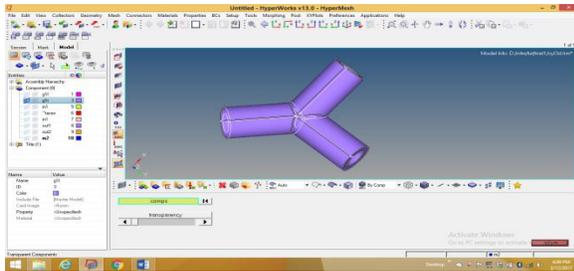FIGURE 3.5 Geometry for structural Analysis


FIGURE 3.6 Volume of cylindrical Y duct with thickness

## 3.2 Computational Grid Generation:

At the core of CFD, computational grid is central element, which often considered as most important and time consuming part in simulation projects. The quality of the grid plays a direct role on the quantification of flow results, regardless of the flow solver used for simulation. The mesh generation concepts were comprehensively discussed and reviewed has resulted to several commercial software's which are being extensively used for several industrial components was shown in

Figure 3-7.

The geometrical model generated with volume generated in previous chapter is discretized with volume tetrahedral elements using 3d panel tool as shown in Figure 3.8.
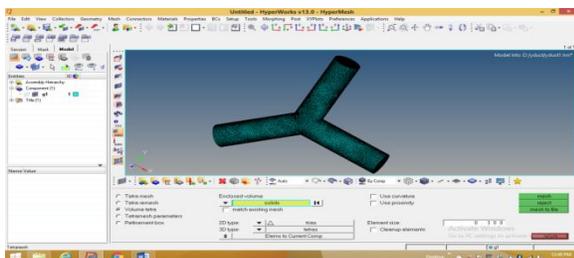

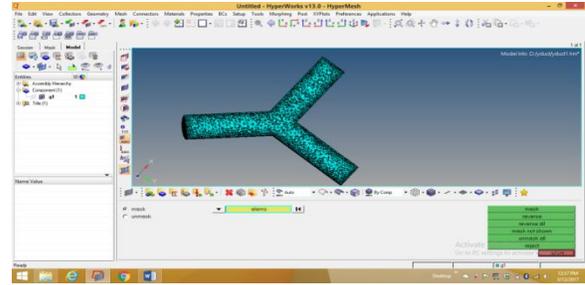FIGURE 3.7 volume tetrahedral elements using 3d panel tool


FIGURE 3.8 selection elements under tools panel and mask option

The element size by default will be 4 with which the grid density is very coarse and the same has been gradually decreased to the value 0.3 so that the mesh size 36805 nodes.

Using Tools panel face option by selection of elements, surface grid is generated and shown in figure 3.9
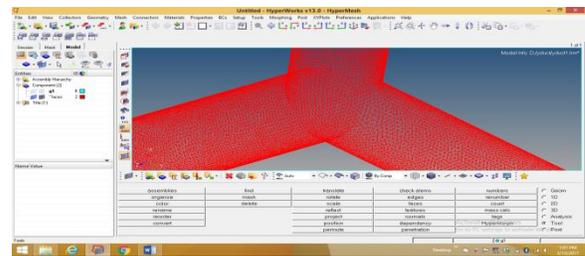

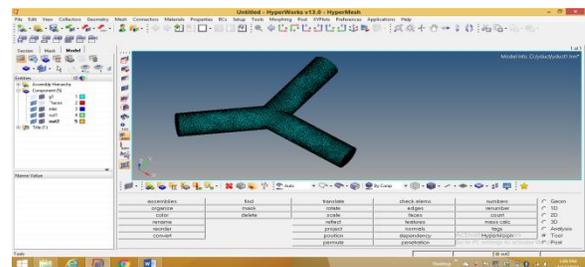FIGURE 3.9Face option by selection of elements


FIGURE 3.10 Creating boundary conditions


FIGURE 3.11 Selecting solver desks CFD

After generating the volume for thickness of y duct cylinders, the tetrahedral mesh has been generated by

providing the suitable element size as shown in Figure: 3.12.
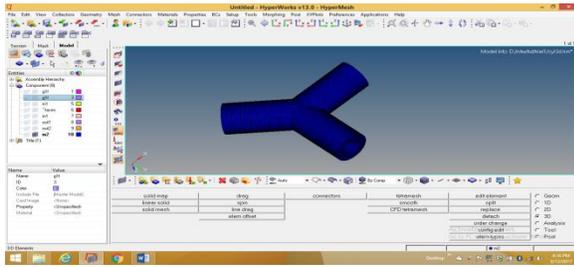


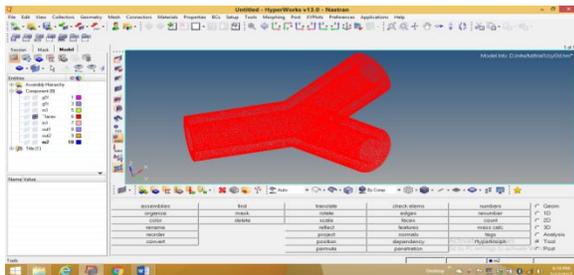FIGURE 3.12 Tetrahedral meshes



FIGURE 3.13 Distortion of elements specially at junction location

After examining the mesh visually, the grid is imported by selecting the NASTRAN Solver Deck by using long format as.bdf file which can be read in static structural analysis solver via finite element modeler of Ansys Products.

### 3.3 Finite element Analysis of Solids

Finite element analysis is a computer based numerical technique for calculating the strength and behavior of engineering structures. It can be used to calculate deflection, stress, vibration, buckling behavior and many other phenomena. It can analyze elastic deformation or "permanently bent out of shape" deformation.

### 3.4 Selection of Flow Solver – Ansys Fluent:

ANSYS Fluent is a state-of-the-art computer program for modeling fluid flow, heat transfer, and chemical reactions in complex geometries

However the input to the solver must be grid file saved as .cas file. The software can be activated by double clicking the .case file in the relevant directory.

It reads geometrical information and checks the grid quality for negative volume displays the mesh along with boundary conditions region Figure 3.21
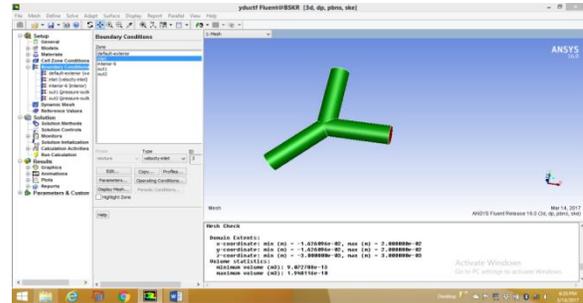


FIGURE 3.21 Checks the grid quality for negative volume displays the mesh along with boundary conditions region

However, in most flow simulation cases, two equation RANS model can be used to account the turbulent losses in the computational domain.
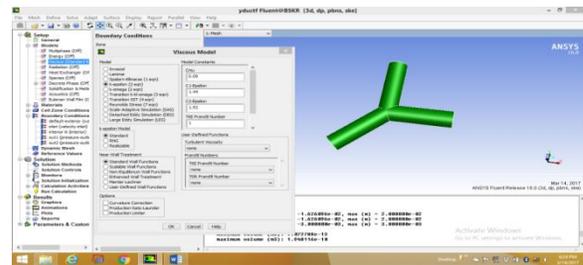


FIGURE 3.22 Viscous models

The material of working fluid by default can be air, depending upon requirements; the user can add different fluid from its data base like water or other gases. Next physics set up is the boundary condition whose values are highly influences the equation residuals. When the grid from third party software is imported, the boundary conditions like inlet, exit, shows as wall type and needs to be changed to respective option like inlet/pressure outlet. The values of velocity in m/sec for most case works fine with default values of turbulent length scale and viscosity. Some this gives trouble; user may change velocity option to mass flow. Generally operating pressures are one bar i.e., 101325 Pascal's and its

value in the pressure out boundary condition should be mentioned as zero. .

The calculate command solves transport equations using iteratively to reduce the errors and its convergence history for the case one of y-duct geometry is shown in Figure: 3.23
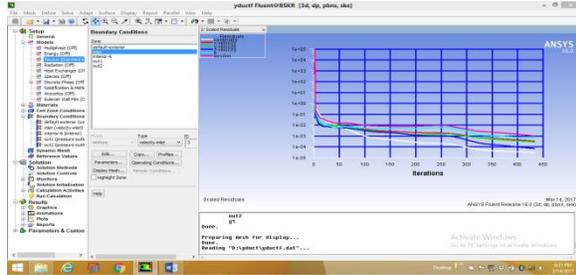


FIGURE 3.23 Command solves transport equation using iterations

In order to see the results are computed in right direction, using report mass flow flux values of at inlet and outlets are computed and shown below.

In order to see the results are computed in right direction, using report mass flow flux values of at inlet and outlets are computed and shown below.

| MassFlowRate | (kg/s) |
|---|---|
| inlet | 0.00017288377 |
| out1 | 0.00014094961 |
| out2 | 3.1955055e-05 |

The balance of mass flow across inlet and exit locations ensures the mass conservation principle which implies that quantity of mass entered in the volume of the component should leave with the same amount.

## 3.5 Simulation of Discrete Phase Model using Ansys Fluent:

The flowing fluid in the y-duct is subjected to micron level dust/carbon particles, component performance via particle motion can be simulated using discrete phase model available in Fluent Solver. However the computational grid used for flow simulation along with the turbulence model selected is remain same. When discrete phase model is made on, a pop-up

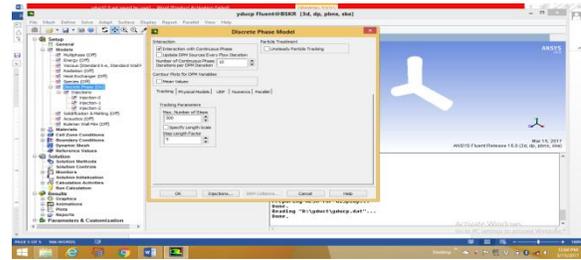menu shown in Figure 3.24 displays on the flow solver.



FIGURE 3.24 Discrete phase model

Under Discrete phase model, the solver expects to choose injection of particles which a pop-up-menu shown in Figure: 3.25

In the analysis three injection locations are chosen with the particle sizes 1 micron, 10 microns and 100 microns. The particles are considered to spherical whose drag law available in the solver is utilized.
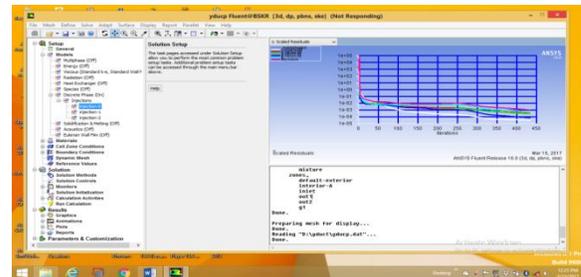


FIGURE 3.25 injections of particles

The flows with particles are simulated whose convergence history of equation residuals is shown in Figure 3.26.
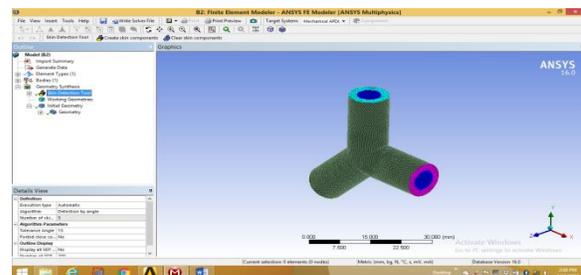


FIGURE 3.26 convergence history of equation residual

## 3.6 Structural Analysis of Y duct using Ansys Work Bench:

The geometry of Y-duct with thickness created and mesh generated in Altair Hypermesh has been saved with the user preferences as NASTRAN. The mesh

file has been saved as long format of NASTRAN Solver deck as .bdf File. When structural analysis for Y-duct envisaged, one of the essential condition is pressure load which comes from CFD solution
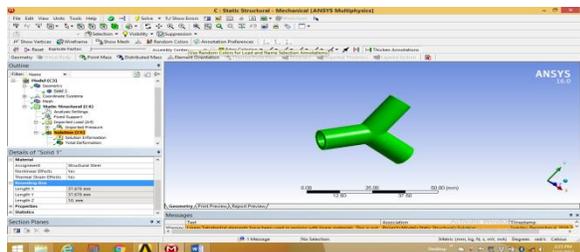


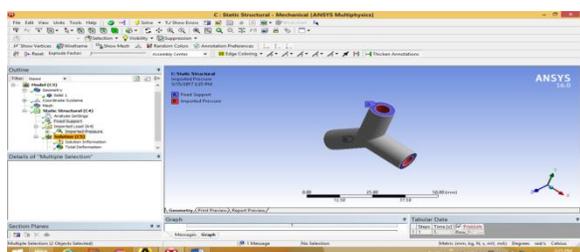Figure 3.27 Structural analysis graphic user displays



Figure 3.28 Structural Analysis

Under analysis settings, solver expects user inputs as constraints as fixed support has been used and imported pressure load inner surface of y-duct has been assigned. With these boundary conditions, the solver executed to simulate the structural analysis for total deformation and equivalent stresses.
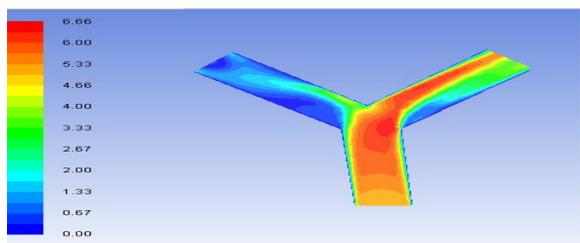
## 4. Discussion of Results



FIGURE 4.1 velocity magnitudes in the plane of computational along z direction

It is observed that velocity of fluid up to junction of y-duct is uniform and at the location of turning, the flow left side taking place on upper portion creating turbulence at bottom portion of right side cylinder. However there exists small quantity of flow left side

cylinder upper portion and at rest of the place flow is creating strong turbulence.
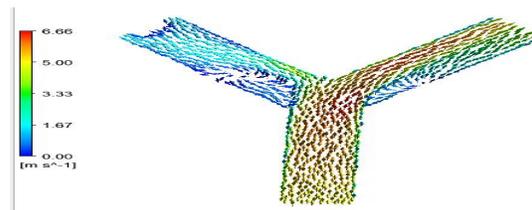


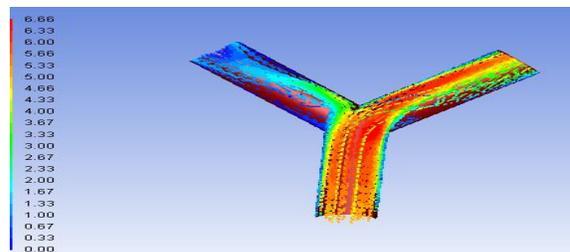FIGURE 4.2 Backflow occurring in y-duct vector plot
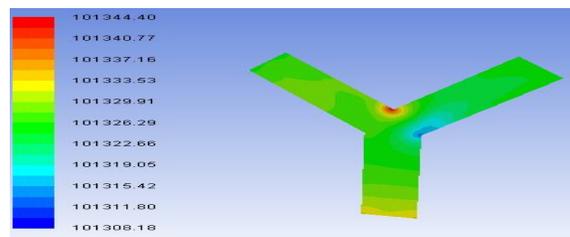


FIGURE 4.3 Flow Recirculation



FIGURE 4.4 Absolute pressure variations in the plane

The pressure on Pascal's indicates lowest at junction location of Y-duct and maximum exhibits at the location where three pipes in x y z direction meeting. For considered inlet velocities, the pressure drop across inlet and exit can be estimated using reports/surface integrals/mass averaged quantity inlet to outlet1 is as follows.

| Absolute Pressure | (Pascal) |
|---|---|
| Inlet | 101333.07 |
| out1 | 101325.00 |

Similarly the pressure drop across inlet to out2 of y-duct pipe estimated with the help of reports/surface integrals/mass weight average/absolute pressure as

| Absolute Pressure | (Pascal) |
|---|---|
| Inlet | 101333.07 |
| out2 | 101325.00 |

The pressure drop across inlet and exit location for the considered flow velocity and pressure comes to be 8pascals.The absolute pressure on wall surface in terms of Pascal's are shown in Figure 4.5

Where in at inlet pressure is high and decelerates up to junction point forming right side cylinder low pressure and generating high pressure on left side cylinder upper portion. The pressure distribution is non-uniform requires geometrical or flow conditions correction
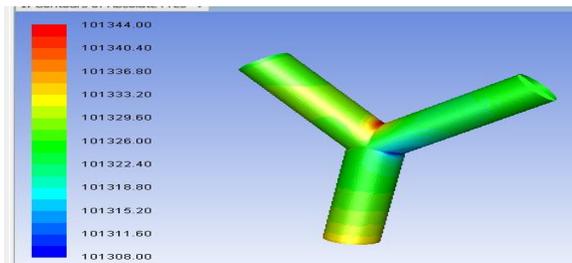


FIGURE 4.5 Absolute pressures on wall surface in terms of Pascal's

When the particles are added to flowing with the same velocity, the trajectories of particles in case of 1 micron follows the flow motion. However 10 micron size particles injected from the inlet location is shown in Figure 4.6
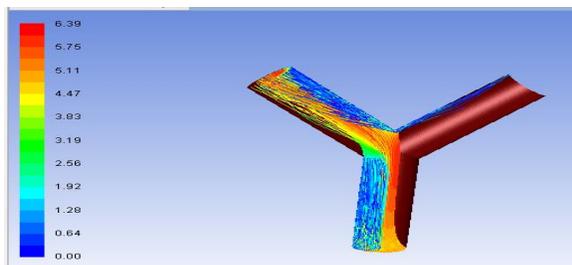


FIGURE 4.6 Inlet locations

The trajectory of particles from inlet with higher velocity travels up to junction location leaves through left side cylinder of y-duct. Due to exchange of mass/momentum the particles have lost inertia and exhibits lower velocities and diverts on the both side of the cylindrical tube. Concentration of higher velocity may lead to erosion problem of Y-duct pipe. The trajectories of 100 micron size particles on

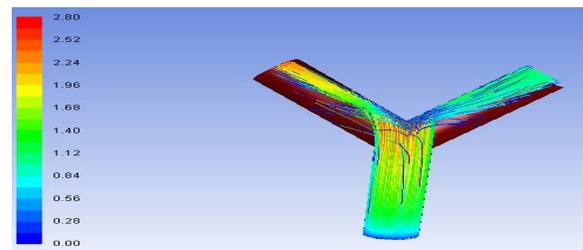particle velocity released from inlet location of the y-duct is shown in the Figure 4.7



FIGURE 4.7 velocity released from inlet location of the y-duct

Higher size particle follows with fluid velocity at inlet location and increases its velocity upto junction point and then diverts towards both the cylindrical tubes in x and y directions. However, the particle concentration is higher on left side cylindrical tube than right side cylindrical tube. In this plot, the particle higher velocity concentrates at junction location of y-duct tube.

The effect of particles in flowing fluids through its trajectories determines the possible location of erosion damage which can be obtained through tracks in Figure 4.8
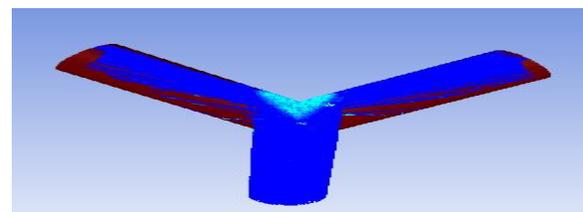


FIGURE 4.8 Erosion damage which can obtain through tracks

For the fixed support of y duct ends, pressure load mapping from CFD result file analysis has been carried out to predict the structural deformations and equivalent stresses on the component in the Figures 4.9& 4.10.

The deformation is though small and variation in terms of maximum taking place wall surfaces of the y-duct and minimum is occurring at end locations of cylindrical tubes.
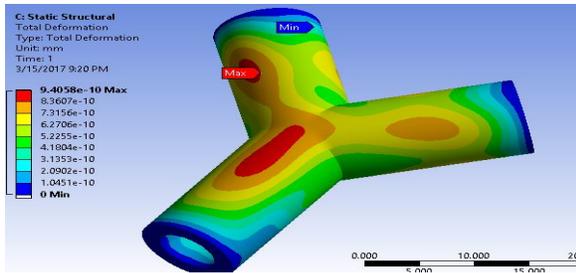
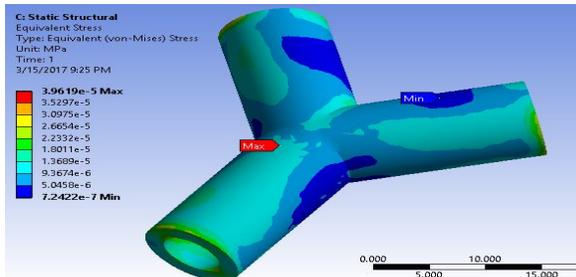FIGURE 4.9 structural deformations and stresses


FIGURE 4.10 structural deformations and stresses

Equivalent stress shown on mega Pascal's shows on the deformed body taking place at junction location and minimum occurs on left cylinder top portion and right side cylindrical surfaces whereas the same has been taken place bottom side of flow direction cylinder. Equivalent stress variation on component surface indicates that stresses are below yield and safe.

**5. Conclusions:**

The problem of y branch duct for transport of fluids in pipeline applications are geometrically constructed using bottom-up approach. In order to understand the pressure drop and flow fluid especially when it is diverting both the sides of cylinders, modern computational tool CFD is employed. The pre-requisites for geometrical input, computational grid required are generated using commercial software's Altair Hypermesh in three dimensional environments. For the considered flow velocities at inlet, working pressure the computational grid has been simulated for turbulent flow and several results in terms of velocity magnitude, vectors, flow path and absolute press in the plane of computational domain has been

obtained and explained graphically. The variations and highlights of each variable are discussed at length in the results and discussions.

It is of interest when the flowing fluid suspended particles especially the problem is viewed from the point of two phase flow applications; discrete phase model available in fluent software is employed. After successful simulation, different types of particle trajectories released from the inlet of computational domain tracked and explained graphically. The particle trajectories on velocity scale provides higher velocity regions which may also erosive nature is also computed and shown graphically.

The pressure fluctuations, unequal velocity distributions, particle trajectories, the component may become weaken from structural aspects. To examine the component deformation is small and equivalent stresses are below yield, static structural analysis has been carried out and interpreted the output variables physically through graphical illustrations.

**5.2 Future scope of work**

The problem of y branch duct for flow distribution, particle trajectories and structural aspects provide valuable basis for refining the work. One of the problem noticed when the flow is along flow direction when diverting through other cylinders it is taking only one side. This needs improvement if the y duct is to be properly designed one. This aspect can be refined through flow simulation with different mesh type elements and size of computational grid.

The study also reports the particle trajectories and impact parameters indicate the erosion location at junction location which needs to be improved. In order to improve problem requires number of simulations for different mass flow and velocity and pressures to predict realistic pressure drop with realistic measurement data. It is also essential reduce

presence of flow recirculation so that low pressure zones and high velocity regions in the duct chamber can be minimized.

The problem also finds air-conditioning and ventilation ducts which are subjected to thermal gradient. With the suitable flow conditions future scope of work for numerical simulations to estimate wall heat transfer, temperature profiles are essential using the CFD solver. To complete the analysis, thermal stress with the mapping of fluid pressure load acting on the y duct wall surface provide design solutions to understand the deformations and equivalent stresses prevailing on the y duct external surfaces. Considerations of more complex configurations of y duct with down ward components like a valve in piping network also forms the future scope of work. Analysis procedure for different physical aspects along with model construction and computational mesh generation will form valid reference for future research.

# 6. REFERENCES

1. Romero-Gomez, et al .*Mixing at Cross Junctions in Water Distribution Systems - Part I. A Numerical Study*, ASCE Journal of Water Resources Planning and Management, pp. 284-294. 2008

2. Austin, R. G., et.al *Mixing at Cross Junctions in Water Distribution Systems – Part II. An Experimental Study*, ASCE Journal of Water Resources Planning and Management pp. 295-302, 2008

3. Aslam A. Hirani et al. *CFD Simulation and Analysis of Fluid Flow Parameters within a Y-Shaped Branched Pipe* Journal of Mechanical and Civil Engineering Volume 10, Issue 1 PP 31-34, 2013

4. Chen C et. al *A comparison between alternative methods for gas flow and performance prediction of internal combustion engines* SAE 921734 Warren dale, 1992

5.Mitianiec,W, *Determination of thermodynamic gas parameters in branched pipes in internal combustion engines*, Journal of Physics: Conference Series 530, 2014

6. Shirish L. K et.al *Literature Review –on CFD Study of Producer Gas Carburetor IJSRD* - International Journal for Scientific Research & Development Vol. 2, Issue 05, 2014

7. T. R. Anil, et. al .*CFD Analysis of Mixture Flow in a Producer Gas Carburetor* International Conference on Computational Fluid Dynamics, Acoustics, Heat Transfer and Electromagnetic CFEMATCON- 06, 2006.

8. Beach R et.al *Measure Guideline: Optimizing the Configuration of Flexible Duct Junction Boxes* NREL Technical Monitor: Michael Gatwick, internal report 2014

9. Rutkowski, H. Manual D—Residential Duct Systems, 3rd edition, Version 1.00. Arlington, VA: Air Conditioning Contractors of America. 2009

10. Hein, A G, *Electro-static Precipitation Dust Movement and Gas Flow optimization*, 8th International Conference on ESP, Ala-bema, USA, 2001.

11. Gallimberti, *Recent Advancements in the Physical modeling of Electro-static Precipitators,* Journal of Electrostatics, Vol 43, pp 219-247, 1998

12. Beach, R et.Al*CFD Analysis of Flexible Duct Junction Box Design*. National Renewable Energy Laboratory. DOE/GO-102013, 2013.