Optimization of the Design of a T-Joint in Pipes based on Simulation on ANSYS

Sahil Sihol
UG Student
Department of Mechanical Engineering
Chandigarh University, Gharuan 140413, Mohali, Punjab, India

Abstract

The basic approach to all piping systems is to write the Bernoulli equation between two points, connected by a streamline, where the conditions are known. For example, between the surface of a reservoir and a pipe outlet. The total head at point 0 must match with the total head at point 1, adjusted for any increase in head due to pumps, losses due to pipe friction and so-called "minor losses" due to entries, exits, fittings, etc.[1] Friction losses are a complex function of the system geometry, the fluid properties and the flow rate in the system which can be significantly reduced with the help of optimized geometry and thereby reducing minor losses and losses due to system geometry. It is apparent that the optimized design is better than the conventional design of T-joint.

Keywords: T-Joint, Streamline Flow, Turbulent Flow, Pipe Bends, Eddy-Formation

I. INTRODUCTION

In the present study, the conventional design of a T-Joint has been optimized by a minor geometrical change in the upper section of the 'T'. The modified geometry is shown in figure-1. The modified section streamlines the flow of fluid in the lower section of the T-joint and prevents the formation of eddies compared to the conventional design of the T-joint in which the fluid gets direction after collision of the streamlines due to which there are losses because of the formation of eddies and random movement of the flowing fluid, moreover, when the fluid finally reaches the lower section of the 'T' then at the point of intersection of the joint, that is, at the bend there is a fairly good amount of gap from the boundary surface due to separation which again leads to formation of vortices as was observed later from the details revealed after the simulation results. This again can be a question for later improvisation of the joint. However, in the present study only the upper section is considered for optimization since it was the prime hypothesis while beginning the study and also this can be regarded as the advantage of virtual-experimentation, that is, simulation as we get to find huge possibilities of optimization and thereby increasing the efficiency of a product at a very low cost compared to the experimental investigations.

Although many similar investigations on various platforms have been done earlier for mixing of fluids in pipes but in this investigation, which is also based on simulation performed in *Fluid Flow(fluent)* module of ANSYS 15.0 however, focuses not only upon the mixing of the fluid but also how the design can be improved to enhance the mixing and reduce losses which occur due to obvious reasons such as collision of stream as they enter in opposite direction which makes the flow turbulent and causes formation of eddies in the conventional design.

II. EXPERIMENTAL SET UP

A. Design of Experiment

In the current study there are two designs of the T-joint on which individually the simulation is performed keeping all the boundary conditions within and out same for both the analysis. The CAD models was created in SolidWorks 2014. The model geometry of optimized model and conventional model are shown with their dimensions in millimeter in figure-1 and figure-2 respectively as drawings; these CAD models are the computational domains on which the simulation has been performed. The modification is just the mirror image of the intersection of lower part of the 'T' with a curvature of 20 mm this was taken so as to maintain a good symmetry of model. However, with a large sample of simulations this dent can be standardized based on diameter of pipe and fluid velocity; this can be a further aspect of the current study.

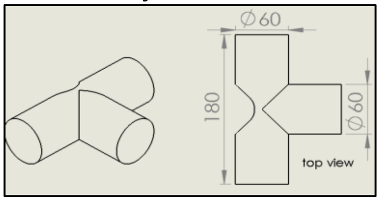

Fig. 1: Optimized CAD Model

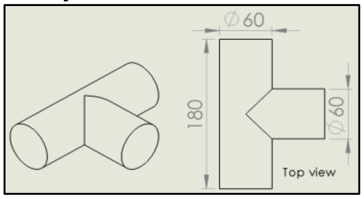

Fig. 2: Conventional CAD Model

III. SIMULATION METHODOLOGY

The simulation was performed using the working fluid as Water with properties described in Table-1.

Table - 1
Properties of Fluid used in Simulation

Property	*Value*	*Units*
Density	*998.2*	*kg/m^3*
Specific Heat	*4182*	*J/kg-K*
Thermal Conductivity	*0.6*	*w/m-K*
Viscosity	*0.001003*	*kg/m-s*

A. Meshing

The meshing details are as follows in Table-2 and the meshed geometry is shown in Figure-3:

Table - 2
Meshing Parameters and Details

Meshing type	*Unstructured; Tetrahedron*
Average Aspect Ratio	*15.41*
Minimum Orthogonality	*0.2939*
Sizing	
Use Advanced Size Function	*On: Curvature*
Relevance Center	*Fine*
Smoothing	*Medium*
Transition	*Slow*
Span Angle Center	*Fine*
Minimum Edge Length	*0.114610 m*
Inflation Option	*Smooth Transition*
Transition Ratio	*0.272*
Maximum Layers	*5*
Growth Rate	*1.2*
Statistics	
Nodes	*20023*
Elements	*102558*

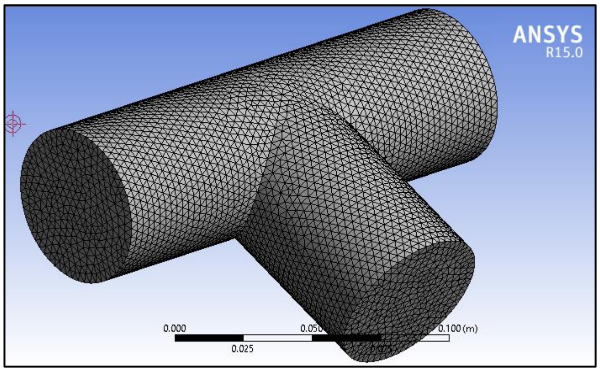

Fig. 3: CAD model after Mesh Generation

B. Boundary Conditions

The Boundary Conditions of inlets and wall are described in Table-3:

Table - 3
Boundary Conditions

Inlet Velocity of fluid from each inlet	*3 m/s*
Turbulent Intensity	*5%*
Turbulence Viscosity Ratio	*10*
Temperature	*300 K*
Shear Condition	*No Slip*
Model Used	*Realizable k-epsilon with enhanced wall treatment*
Spatial Discretization Gradient	*Green-Gauss Node Based*
Number of Iterations performed	*1000*

IV. RESULTS AND DISCUSSION

Simulation Results of Conventional T-Joint i.e., velocity streamline, velocity vector and pressure contours are shown in figures 4, 5 and 6 respectively.

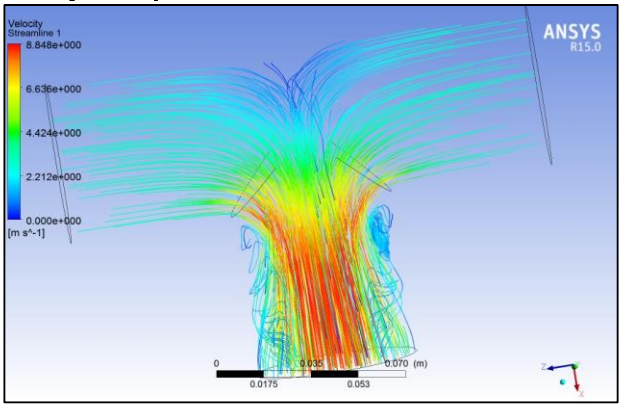

Fig. 4: Velocity Streamline of Conventional Model

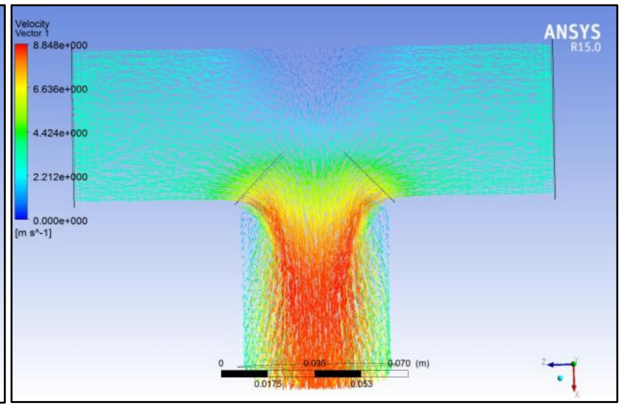

Fig. 5: Velocity Vector Distribution of Conventional Model

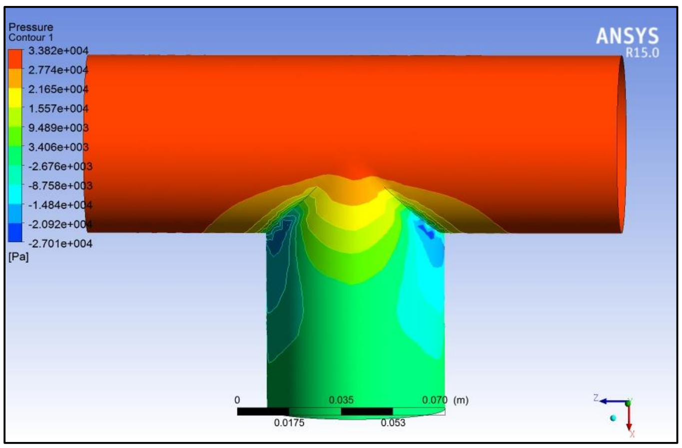

Fig. 6: Pressure Contour Distribution of Conventional Model

It can be observed with a little consideration in Figure-4 how the streamlines of flowing fluid are formed, the collision and thereby the formation eddies in that region are formed and in the figure-5 we can see a region of no-velocity exists after the collision which also wastes energy by making vortices in that region it can also be noted from the figure-6 that there exists a pressure variation at the junction point this is due to the impact force of each fluid stream and separation of flow from the contact surface and as the fluid advances the pressure also stabilizes at the time of exit.

Simulation Results of Optimized T-Joint i.e., velocity streamline, velocity vector and pressure contours are shown in figures 7, 8 and 9 respectively.

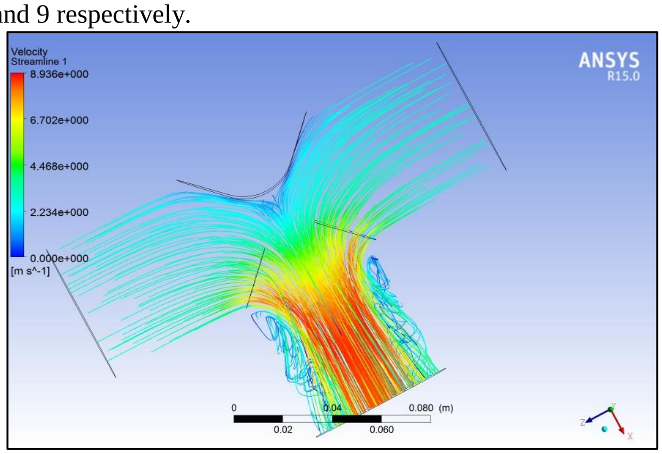

Fig. 7: Velocity Streamline Distribution of Optimized Model

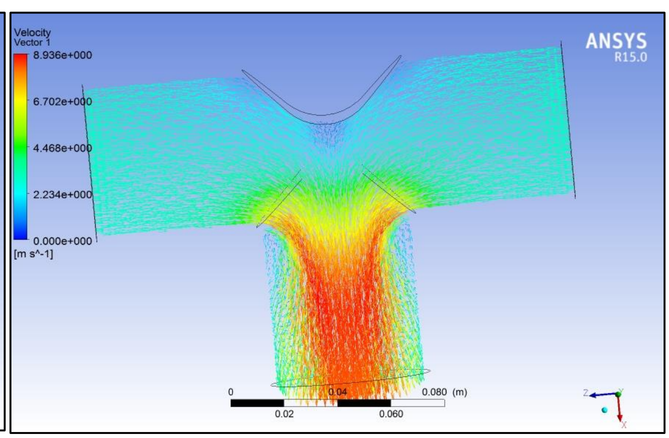

Fig. 8: Velocity Vector Distribution of Optimized Model

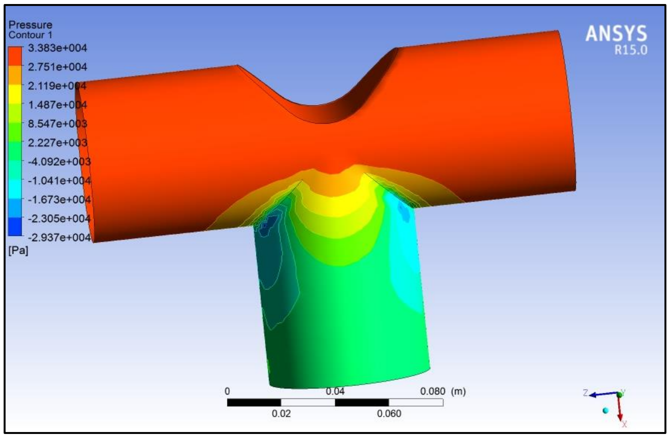

Fig. 9: Pressure Contour Distribution of Optimized Model

We can clearly observe by comparison of the results of conventional and optimized models that the motion of fluid is more streamlined, the unnecessary part in the conventional model which was causing loss of energy is not only eliminated but also it is helping in streamlining the flow. The pressure variation has also significantly changed at the junction which implies that there is less separation of flow in that region, moreover, the variation gets stable earlier and it can also be noted that at the region which is optimized there is no pressure variation which means that it has not affected the fluid flow adversely in any manner.

V. CONCLUSION

In this paper, it was attempted to optimize the design of conventional T-joint used in piping systems and the results of the simulation clearly indicated the success of the optimization which can be achieved by making a dent over that region with an appropriate die while manufacturing. However, it must be noted that the design can still be optimized further for the formation eddies in the lower section of the 'T' which was realized as a result of the simulation also a formula can be derived with enough simulations to standardize the amount of dent required these can be the further aspects of the current study.

REFERENCES

[1] Queens University, Losses in Pipes Retrieved From http://my.me.queensu.ca/People/Sellens/LossesinPipes.html 20 December 2016.
[2] Modi PN, Seth SM Hydraulics and Fluid Mechanics including Hydraulic Machines 19th ed.: Rajsons Publications; 2013.

