

## REVIEW OF FLOW DISTRIBUTION NETWORK ANALYSIS FOR DISCHARGE SIDE OF CENTRIFUGAL PUMP

**Satish M. Rajmane**

*Research Scholar,*

*WIT Research Center, Solapur University, Solapur, India*

**Dr. S. P. Kallurkar**

*Principal,*

*Atharva College of Engineering, Mumbai, India<sup>2</sup>*

### ABSTRACT

A computational fluid dynamics (CFD) analysis has been conducted to find the pressure losses for dividing and combining fluid flow through a junction of discharge system. Simulations are performed for a range of flow ratios and equations are developed for pressure loss coefficients at junctions. A mathematical model based on successive approximations then would be employed to estimate the pressure losses. The proposed CFD based strategy can be used for the analysis of all the three pipe branches of some diameter are selected along with equal length so that only the effect of bend angle can be studied. The effect of bend angle, pipe diameter, pipe length, reynolds number on the resistance coefficient is studied. The software used is CATIA for modeling and ANSYS fluent for analysis purpose.

**KEYWORDS:** Computational fluid dynamics, junction pressure loss coefficient, dividing manifold, is combining manifold, flow distribution.

### INTRODUCTION AND RELEVANCE

Centrifugal pumps are a sub-class of dynamic axis symmetric work absorbing turbo machinery. Centrifugal pumps are used to transport fluids by the conversion of rotational kinetic energy to the hydrodynamic energy of the fluid flow. The rotational energy typically comes from an engine or electric motor. The fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward into a diffuser or volute chamber (casing), from where it exits. Common uses include water, sewage, petroleum and petrochemical pumping. The reverse function of the centrifugal pump is a water turbine converting potential energy of water pressure into mechanical rotational energy.

A computational fluid dynamics (CFD) analysis has been conducted to find the pressure losses for dividing and combining fluid flow through a junction of discharge system. Simulations are performed for a range of flow ratios and equations are developed for pressure loss coefficients at junctions. A mathematical model based on successive approximations then would be employed to estimate the pressure losses. The proposed CFD based strategy can be used as a substitute to setting up and performing costly experiments for estimating junction losses.

## LITERATURE REVIEW

**Abdul Waheed Badar, Reiner Buchholz, Yongsheng Lou and Felix Ziegler**<sup>(1)</sup> have studied the A CFD analysis has been conducted to find the pressure losses for dividing and combining fluid flow through a tee junction of a solar collector manifold. Simulations are performed for a range of flow ratios and Reynolds numbers, and equations are developed for pressure loss coefficients at junctions. A theoretical model based on successive approximations then is employed to estimate the isothermal and non-isothermal flow distribution in laminar range through a collector consisting of 60 vacuum tubes connected in parallel in a reverse and parallel flow arrangement.

**Chris Smith**<sup>(2)</sup> have investigated using CFD in flow assurance Analysis of low material temperatures caused by rapid gas expansion (JT cooling) is possible using CFD. This includes cold spots and thermal gradients, which can go beyond the design limits of the material, and traditional pipeline tools don't give the distinction required. Situations can be evaluated including choke valves and downstream pipe work at start-up, and process equipment and pipe work during blow down.

**R. Vijayakumar, S.N. Singh, V. Seshadri**<sup>(3)</sup> has studied Unbalanced coal/air flow in the pipe system feeding the boilers will lead to non uniform combustion in the furnace thereby lowering the efficiency of the power plant.. The current practice of balancing the flow by introducing orifices in the pipe system is generally achieved using semi-empirical methods. This paper presents the CFD analysis for deciding the optimum geometry of the orifices to balance the flow in the existing power plant at NTPC.

**Balvinder Singh, Harpreet Singh, Satbir Singh Sehgal**<sup>(4)</sup> the objective of this study was to Pipe fittings are used in the plumbing systems to connect straight pipe or tubing sections for regulating or measuring fluid flow. The wye shape fitting will convert into T shape fitting. In the present work, effect of angle of turn/bend for a Y-shape pipe will be studied computationally. Water and air as a fluid is selected which flows through the plumbing system. The effect of bend angle, pipe diameter, pipe length, Reynolds number on the resistance coefficient is studied. It was observed that resistance coefficient vary with the change in flow parameters.

**S. Mokhtari, V.V. Kudriavtsev, M. Danna**<sup>(5)</sup> has studied uniformly of a dielectric film deposited on the silicon wafers greatly depends on the flow delivery systems & injector geometry. In this paper, we analyze flow distribution patterns through the simulated closed-end multiple outlet pipe/channel which delivers chemical precursors to the parallel slot linear injector of a chemical vapor deposition reactor. Instead of using semiempirical formulas for the flow resistance & flow network methodology, a finite difference multi-block CFD solution method is used to calculate local pressure patterns in the pipe & establish flow distribution trends & uniformly as a function of Re number & system aspect ratio. It was found that flow distribution changes qualitatively as we move from the low Re flow conditions to intermediate & high Re flow regimes.

## PROBLEM DEFINITION

The typical case of a centrifugal pump supplying fluid or water to multiple destinations is the subject matter of this work. The multiple channels at the delivery side poses challenge for

arriving at the right configuration of the pipe diameters, number of channels and the design of the manifold from where the sub divisions for the pipe emanate. The right value needs to be assigned for these design parameters so as to maximize the mass flow (discharge) while minimizing the pressure drop at the delivery side. The effort of this work is to find the best design for manifold at delivery side for a given pipe size at the suction end.

### Proposed Work

1. Study the existing system.
2. Calculate the pressure drop, velocity, discharge through numerical method
3. Analysis using Computational Method.
4. Study the results and identify the region for improvement such as minimizing the losses, pressure drop, and vacume creation in the system.
5. Revised the geometry and analysis for different variants.
6. Validation of through experimentation.
7. Identifying the best solution in the given alternatives.

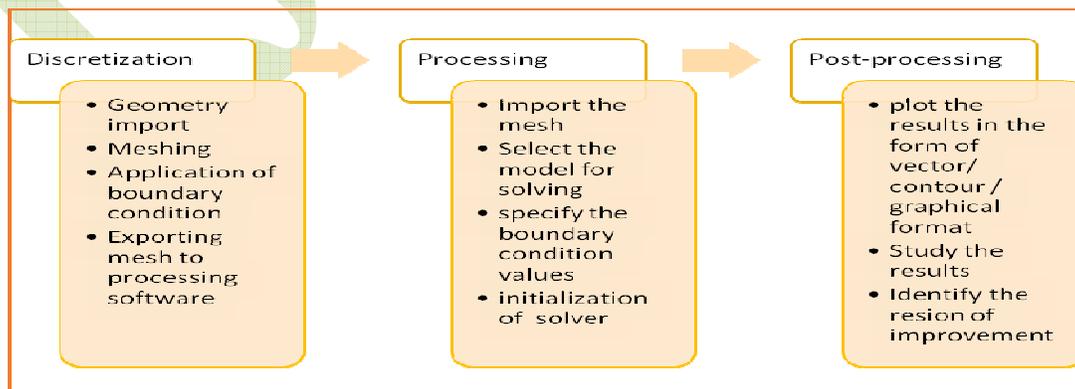
### Methodology

**Numerical Method:** For various configuration of a system, velocity, pressure drop, discharge would be calculated using mathematical model. With the help of this calculation we could compared the preliminary results obtained from the computational method.

**Analytical Method:** Computational method would be used for capturing the small changes in the flow pattern in the fluid and pressure variation in the system. The process is mainly carried out in three main steps as follows:

1. Discretization: Fluid domain created using CAD software is imported in the pre-processor for discretization process. Meshing is carried out to find out the small changes in the system. Typically Gambit or ICEM CFD software used for the meshing purpose
2. Processing: In this stage, solver setup preparation carried out in ANSYS workbench. Typically mass flow rate or velocity value would be given at inlet side and k-epsilon with 2 equations module would be used for solving purpose.
3. Post-processing: ANSYS workbench is used to study the results. Results are shown in the vector or contour or graphical format. We would also calculate the pressure drop, discharge and velocity in the numerical format.

### Flow chart:



## EXPERIMENTATION AND VALIDATION

Looking into the feasibility of making the prototype, the same might be built as a miniature version of the full scale model. This model would include the sump, piping system, centrifugal pump and a delivery pipe. During the experimentation over the prototype, discharge and pressure drop would be compared through numerical and computational methods. At a constant volume of discharge, time varies with geometry. During experimentation, time required for constant discharge and back pressure is determined for the optimum variant. In this process we shall use the 1 HP or ½ HP motor (Pump) and standard pipe size for making prototype.

For validation purpose, we could compare the output parameters using numerical, computational and experimental results for the given prototype.

## CONCLUSIONS

A CFD analysis is used to estimate the junction losses at the tee junctions of a collector manifold. A simplified model of the junction is built and simulated in FLUENT for a range of Reynolds numbers and riser-to-manifold flow ratios. The resulting junction loss coefficients have shown a strong dependency on the flow rate at low Reynolds numbers. The variable loss coefficients are implemented in a theoretical model to predict the flow distribution in a coaxial vacuum tube solar collector arranged in U- and/or Z-configurations. The model is validated with the experimental results for the same collector in U-configuration. The model agrees reasonably well (but not perfectly) to the experiments. The model can be used to predict flow distribution for any number of risers in the prescribed range of Reynolds numbers. Flow uniformity decreases with increasing flow rate and temperature. Parallel flow (Z-configuration) results in more but not perfectly uniform flow than the reverse flow (U-configuration). The proposed CFD-based method can replace the expensive and time-consuming procedure of setting up experiments for estimating junction losses.

## REFERENCES

- [1] Abdul Waheed Badar, Reiner Buchholz, Yongsheng Lou and Felix Ziegler; CFD based analysis of flow distribution in a coaxial vacuum tube solar collector with laminar flow conditions; International Journal of Energy and Environmental Engineering 2012.
- [2] Chris Smith (Production Assurance Manager at Xodus Group); A dynamic approach to flow assurance modelling; Technical Paper, Xodus Group.
- [3] R. Vijayakumar, S.N. Singh, V. Seshadri, (Deptt. of Applied Mechanics, IIT Delhi) CFD; Analysis for the Design of Optimum Geometry of the Orifices in the PC pipes for ensuring uniform feed rate of Coal through various Coal Burners in a Boiler.
- [4] Balvinder Singh, Harpreet Singh, Satbir Singh Sehgal; CFD Analysis of Fluid Flow Parameters within a Y-Shaped Branched Pipe; International Journal of Latest Trends in Engineering and Technology (IJLTET).
- [5] S. Mokhtari, V.V. Kudriavtsev, M. Danna; Flow informality and pressure variation in Multi-outlet flow distribution pipes, Advance in analytical, Experimental and computational Technologies in Fluids, Structures, Transient and Natural Hazards; PVP-Vol.355, AME 1997

- [6] H. Shaban, S. Tavoularis; Distribution of downward air–water flow in vertical tubes connected to a horizontal cylindrical header; Research Article Nuclear Engineering and Design, Volume 291, September 2015, Pages 90-100.
- [7] Sambhrant Srivastava, Apurba Kumar Roy, Kaushik Kumar; Design analysis of Mixed Flow Pump Impeller Blades Using ANSYS and Prediction of its Parameters using Artificial Neural Network; Research Article Procedia Engineering, Volume 97, 2014, Pages 2022-2031.
- [8] A.M.F. El-Shaboury, H.M. Soliman, G.E. Sims; Two-phase flow in a horizontal equal-sided impacting tee junction; Research Article International Journal of Multiphase Flow, Volume 33, Issue 4, April 2007, Pages 411-431.
- [9] Emmanuel Mignot, Cheng Zeng, Gaston Dominguez, Chi-Wai Li, Nicolas Rivière, Pierre-Henri Bazin; Impact of topographic obstacles on the discharge distribution in open-channel bifurcations; Research Article Journal of Hydrology, Volume 494, 28 June 2013, Pages 10-19.
- [10] Xiao-gang WANG, Zhong-min YAN, Wei-dong GUO; Three-dimensional simulation for effects of bed discordance on flow dynamics at Y-shaped open channel confluences; journal of Hydrodynamics, Ser. B, Volume 19, Issue 5, October 2007, Pages 587-593.
- [11] M. De Marchis, C.M. Fontanazza, G. Freni, A. Messineo, B. Milici, E. Napoli, V. Notaro, V. Puleo, A. Scopa; Energy Recovery in Water Distribution Networks, Implementation of Pumps as Turbine in a Dynamic Numerical Model; Research Article Procedia Engineering, Volume 70, 2014, Pages 439-448.

## AUTHORS BIOGRAPHY



### **Mr. S. M. Rajmane**

He is having ME from Solapur University. He is Research Scholar for WIT Research Center, Solapur University; Solapur He is having more than 10 years teaching experience. His area of interest is in Fluid machinery, FEM Design engineering.



### **Dr. S. P. Kallurkar**

He is having PhD from NITIE Mumbai. Presently working as Principal in Atharva College of Engineering Mumbai He is Research Guide at WIT Research Center, Solapur University, and Solapur He is having more than 25 years teaching experience