

INTRODUCTION

A valve is a mechanical device which controls the fluid flow through system by changing the flow area & thus increasing the resistance across valve. Therefore flow rate & pressure drop across valve are of utmost importance for optimum valve performance. Valves are an instrumental component of industries such as chemical, processing, mining, plastic manufacturing and many other fields due to multiple availability of valve types and designs. CFD offers a safe, reliable and cost + time saving solution to the testing of various working parameters of valves. CFD helps to estimate the flow performance of the valve beforehand and optimize the design accordingly.

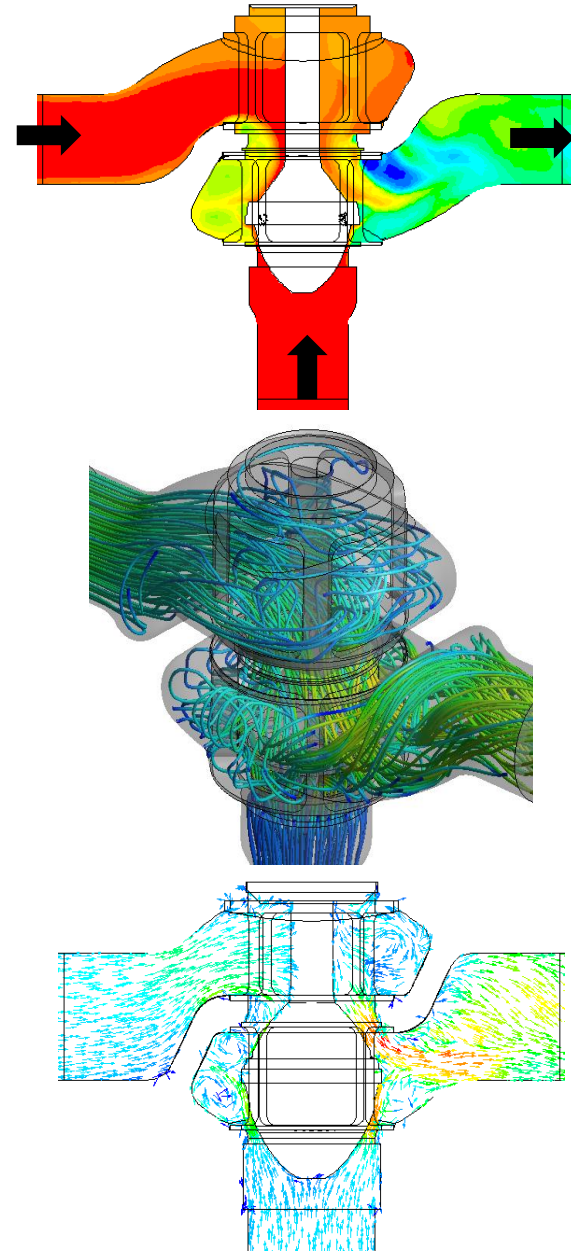
CHALLENGES

- Meshing of close proximity in flow domain.
- Determining Valve coefficient.
- Estimating percentage mixing for different type of fluids.

THE SOLUTION

Flow simulation of valve was carried out to determine the flow pattern, valve coefficient, pressure drop within the valve system & uniformity of both fluids at exit. Capturing the close proximity regions accurately was the biggest challenge faced while meshing the valve. The flow entering from the two pipe mixes in spindle region and flows through the outlet pipe. From simulation we are able to visualize the flow pattern within the valve. The pressure drop and valve coefficient for the valve was derived after the simulation and accordingly design modifications were suggested.

Due to increase in flow opening area of valve, the valve coefficient increased & thus reducing pressure drop across valve. Also, percentage of mixing of two fluids is checked by calculating the uniformity of at exit.



BENEFITS

- Prediction of valve coefficient & mixing of fluids.
- Reduced number of trials for optimum operation of valve.