NUMERICAL MODELING OF FLOW CONTROL VALVE

INTRODUCTION

Control valves are designed to regulate flow, level, temperature or pressure within a vessel, pipeline or process. They are used in the refinery, petrochemical, polymer, mineral processing and power industries. The property of flow field in a control valve plays an essential role in energy dissipation and system efficiency. Conventionally the pressure loss of hydraulic valves is calculated theoretically or by using empirical formula. Recent development of the CFD technique makes it possible to achieve a better solution of the pressure loss within the complex channel of various hydraulic valves without experimental expenses.

COMPUTAIONAL METHODOLOGY

- ✓ Cad Model Development
- ✓ Fluid domain extraction
- ✓ Mesh generation
- ✓ Simulation (setting the boundary conditions and solver controls)
- ✓ Interpreting the results

The geometry was developed in CAD software and imported to CFD modelling software where the fluid domain has been extracted and surface meshes were created. To perform a numerical simulation the geometry is divided into sufficient number of computational cells in which the governing equations of fluid flow will be solved.

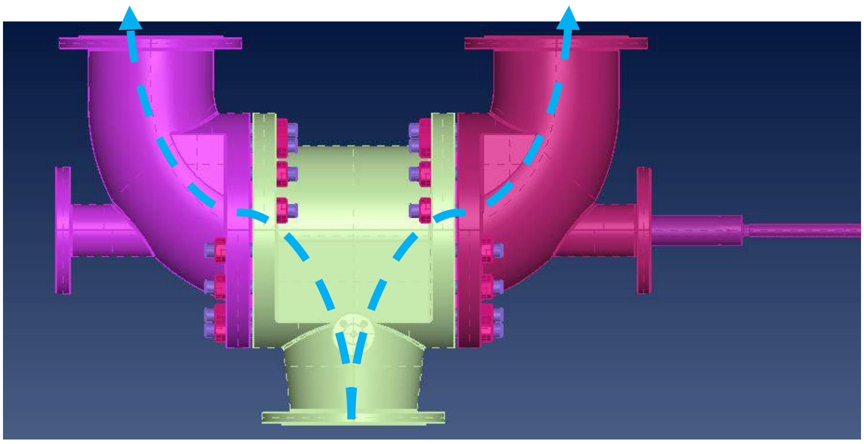

Fig 1 CAD geometry

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

The given is referred to as "selector" valves and is well suited to applications where mandatory pressure relief devices are installed as dual devices. It has two flow paths through which it can be operated to switch the flow between two paths using a round wheel connected through a spindle. The design of the valve should ensure that at least one of the two pressure relief device is always in service and full relief flow is guaranteed during changeover.

Considering the accuracy of computational results the mesh should have certain quality level and controlled by the parameters such as *equi-angle skewness, aspect ratio, equi-volume skewness*. The inlet and outlet portions are extended to a ratio of 3D and 6D length to attain the convergence accuracy in CFD analysis. (D-valve dia)

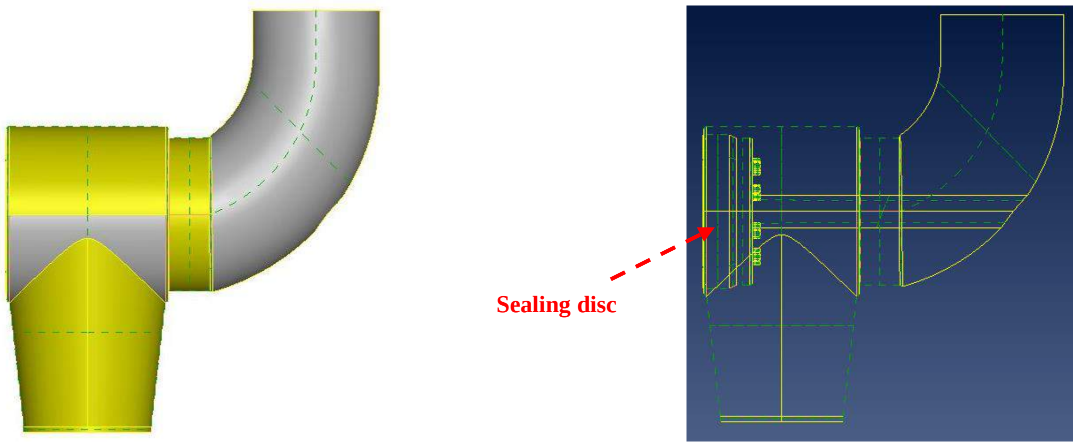

Fig 2 Extracted fluid domain of Right side flow path

BOUNDARY CONDITIONS

The mesh file is then imported into solver interface where the material properties and the valve inlet, valve outlet and wall conditions were specified. Water is considered to be the working fluid. As the flow inside the valve path going to be turbulence in nature, a suitable turbulence model has to be selected while doing the simulation.

- Inlet - pressure inlet with atmosphere temperature
- Wall - no slip condition & adiabatic
- Outlet - pressure outlet
- Viscous model - K-omega-SST

FLOW CHARACTERISTIC PARAMETERS

- Pressure loss co-efficient (ζ)
- Flow co-efficient (K_v or C_v)

i

Fig 3 Meshed geometry

CONTOUR PLOTS AND VECTOR PLOTS

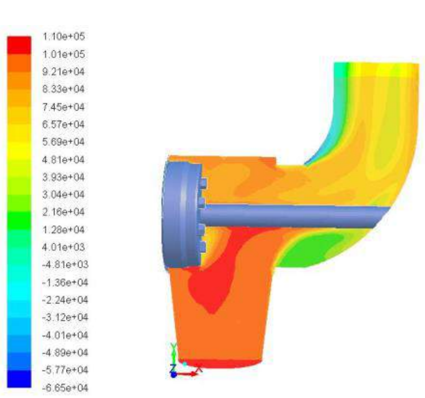

Fig 3 Total pressure contour (Pa)

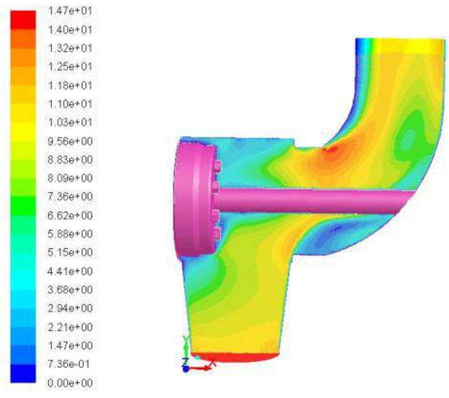

Fig 4 velocity contour (m/s)

Model/properties	Discharge (m3/s)	Pressure drop (Pa)	Pressure loss co-efficient Zeta (ζ)	Flow co-efficient (K) Kv
Right flow path	0.5298212268	38645.551	0.6626	3037.35

Table 1 flow parameters

POWER OF CFD

CFD provides a clear insight view at anywhere inside the valve geometry which is not possible to get from experiments. And with the aid of various contour plots and vector plots it is quite comfort to study the flow nature and to identify the pressure drop regions. So , CFD can comprehensively replace the time consuming and expensive experimental procedures.

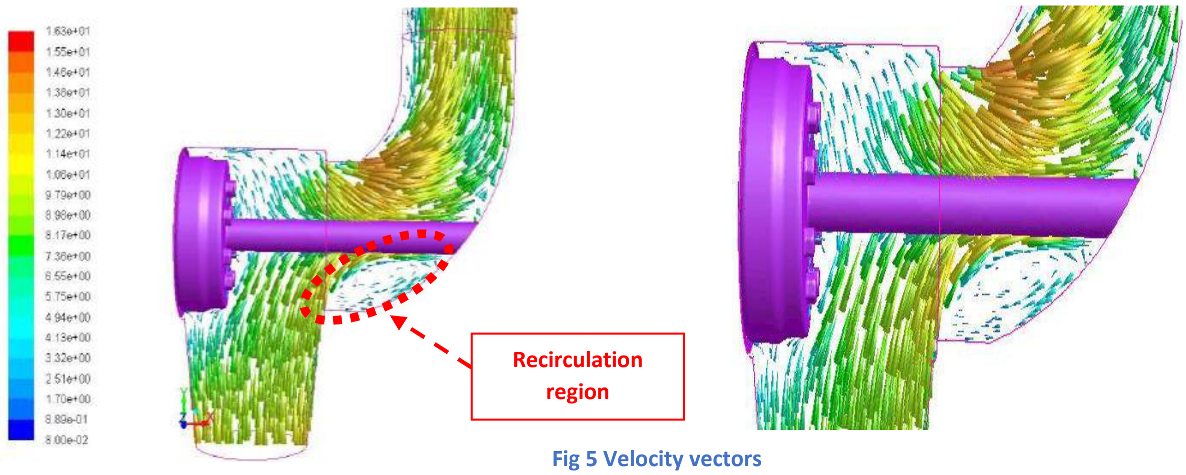

Fig 5 Velocity vectors

FINDINGS

Thus, CFD is comprehensively utlized to understand the behavior of fluid flow as well as to examine the critical regions where posible design improvements can be done to provide a better valve design,Which helps the customer to increase their product quality with less effort and also in a quicker time.

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com