ANALYSING THE HYDRO DYNAMIC AND STRUCTURAL INTEGRITY OF BALL VALVE

INTRODUCTION

Ball valves are mostly used in shutoff applications. They are not recommended to be used in a partially open position for a long time under conditions of a high pressure drop across the valve, thus the soft seat could tend to flow through the orifice and block the valve movement.

The seat material resistance of the ball valve limits the working temperature and pressure of the valve. The seat is plastic or metal made.

Ball valves have many good points and are often considered superior to many other kinds of valves. They are very easy to use and can both maintain and regulate three things-

1. High pressure,
2. High volume and
3. High flow of temperature.

TYPE OF BALL VALVES

Float ball valve: The ball is held on two seat rings.

Trunnion ball valve: The ball is supported on its vertical rotation axis by a Trunnion. The Trunnion absorbs the pressure from the flow, therefore the contact between the ball and the seat is not excessively stressed and the operating torque can remain low. This design is recommend for big diameters and high pressure. It has a mechanical means of anchoring the ball at the top and the bottom, this design is usually applied on larger and higher pressure valves (say 4 inch and above 600 psi and above)

SCOPE:

Analysing the pressure load on the ball material. Since a ball valve may operate at a wide range of pressure it is very much essential to test the structural strength of ball , sealing portion while designing the valve.

COMPUTAIONAL METHODOLOGY

Cad Model Development

- ✓ Fluid domain extraction
- ✓ Mesh generation
- ✓ Simulation from CFD and coupled with FEA.
- ✓ Interpreting the results

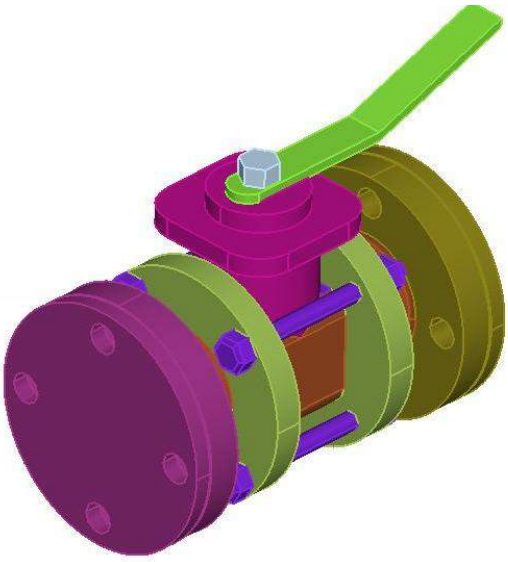

Figure 1 CAD model

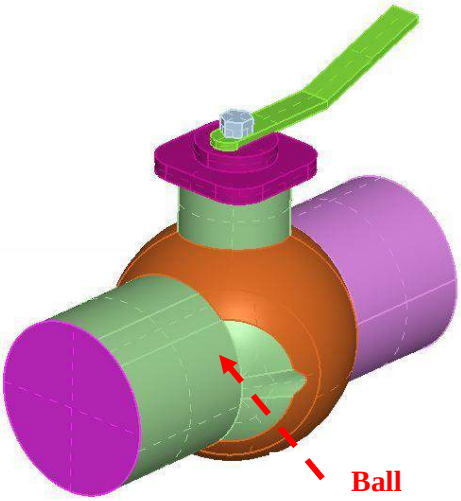

Figure 2 Extracted fluid domain

A three dimensional valve has been created using solid works tool. In order to prepare a computational domain the geometry is imported into CFD pre-processor where the fluid domain has been extracted as detailed below.

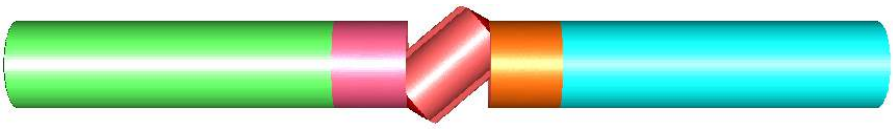

Fig 3 Fluid domain extension

Grids are then generated to the required fluid region using Ansys T-grid. 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 attain the convergence accuracy in CFD analysis as shown above.

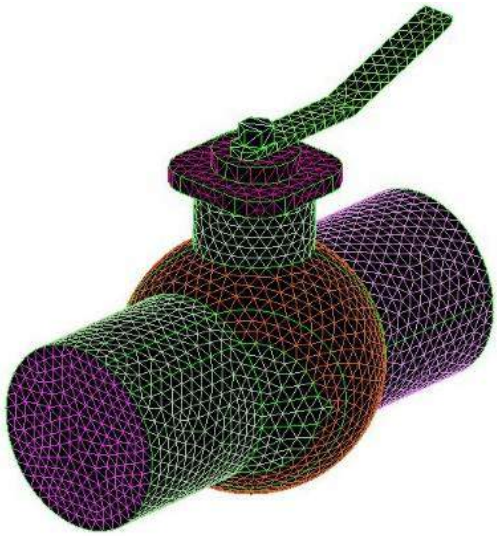

Figure 4 surface mesh

Surface and volume mesh

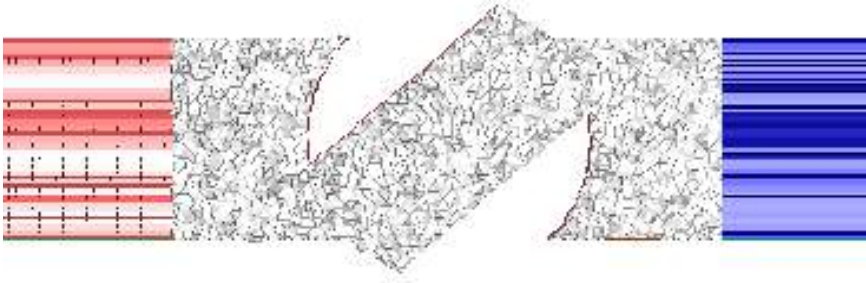

Figure 5 volume mesh cut plane

BOUNDARY CONDITIONS

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

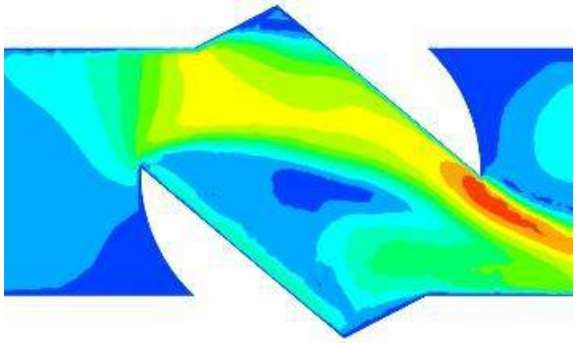

Figure 6 velocity contour

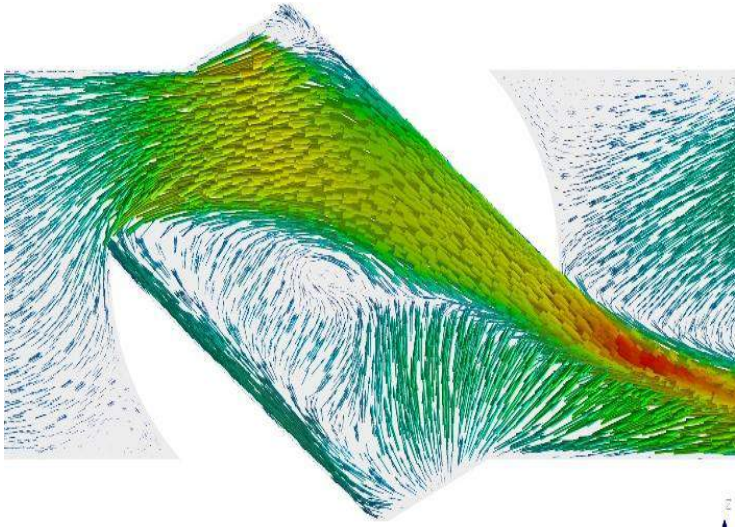

Figure 7 velocity vector

COUPLING CFD AND FEA

CFD provides a clear insight view at anywhere inside the valve geometry which is not possible to get from experiments. And by coupling the hydro static pressure into FEA the structural failure canbe predicted as shown below.

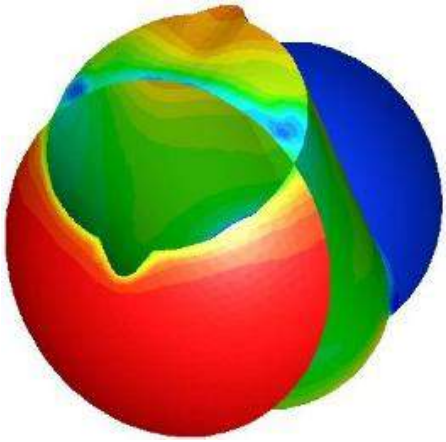

Figure 8 hydro static pressure acting on ball surface

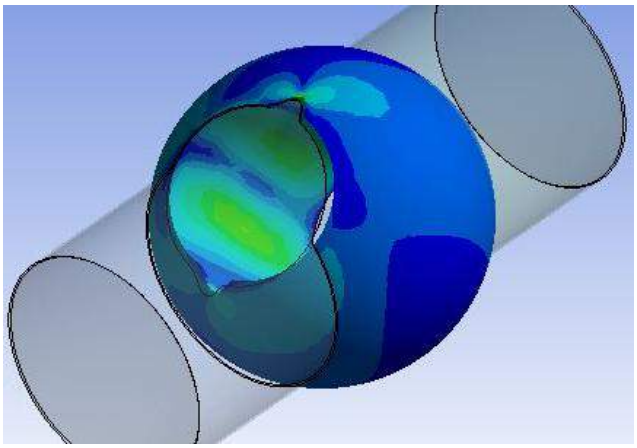

Figure 9 Stress distribution