CFD Analysis of A Pressure Relief Valve

Kishan Patel
Student, M.E. (CAD/CAM),
A. D. Patel Institute of Technology, New V.V.Nagar, Gujarat

Abstract— Diaphragm valve is a shut off valve, which is used for open/close application. One of the variant of diaphragm valve is a weir type diaphragm valve, in which a dam like structure called weir is placed in the middle of the valve. When diaphragm is stretched up to weir, the flow stops. However, even in open condition, the weir obstructs the flow. That is why if design is not proper, drastic pressure drop can occur. This make designers to use CFD analysis to find values of various flow parameters and simulate flow in weir type diaphragm valve which helps to improvise flow characteristics of the valve. Generalised procedure to perform CFD analysis of a weir type diaphragm valve is discussed in this paper.

Keywords: - Pressure *relief valve, CFX analysis*

I. INTRODUCTION

Pressure relief valve is a safety valve, which is used for safety applications. Pressure relief valve consists of a valve body (with two or more ports), a diaphragm, and a plunger which upon exertion of force closes or opens the valve. Pressure relief valves are flex-body valves in which the valve body consists of a rigid and flexible section. The flexible body section is provided by a diaphragm which is the closure member. The seat is provided by the rigid body section and may be either flat or spherical depending upon the application. Two types of pressure relief valve – spring operated and pilot operated .Figure 1 shows spring operated PRV and figure 2 shows pilot operated PRV.
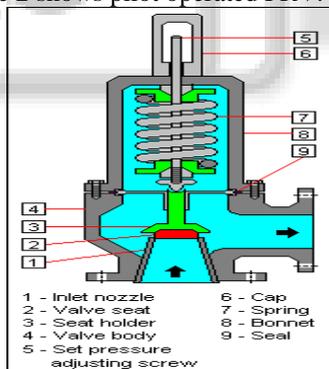

Fig. 1: spring operated type pressure relief valve

The shape of the plunger seat in the flow passage is designed to reduce the flow variations to a minimum, while still providing a smooth and streamlined flow passage. The life of diaphragm improves as a result of reduction in flexing stress. Due to the short stroke of the valve, plastic materials such as PTFE.

It can be used for diaphragm, which is too inflexible for longer strokes. The back of the diaphragms is lined with an elastomer to provide uniform seating stress upon valve closing. Weir-type diaphragm valves may also be used in general and high vacuum service. However, specially reinforced diaphragm might be required in high vacuum service. Because the diaphragm area is large compared with the flow passage, the fluid pressure imposes a correspondingly high force on the raised diaphragm. The resulting closure torque limits the size to which diaphragm valves can be made.
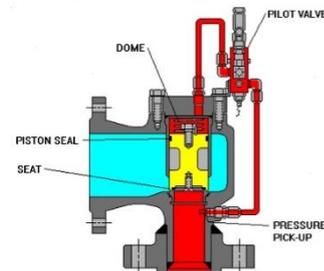

Fig. 2: Pilot operated type pressure relief valve[1]

It can be used for diaphragm, which is too inflexible for longer strokes. The back of the diaphragms is lined with an elastomer to provide uniform seating stress upon valve closing. Weir-type diaphragm valves may also be used in general and high vacuum service. However, specially reinforced diaphragm might be required in high vacuum service. Because the diaphragm area is large compared with the flow passage, the fluid pressure imposes a correspondingly high force on the raised diaphragm. The resulting closure torque limits the size to which diaphragm valves can be made.

A. Hongjun Zhu et al. [2] presents CFD analysis of a needle valve to predict the flow erosion rate and flow-induced deformation of needle valve. Simulations are conducted in ANSYS workbench 14.0, in which FLUENT is used for calculating flow parameters. Meshing with different refinements at different locations according to geometry and required accuracy are employed. CFD analysis is done to observe effect of valve opening and inlet valve channel size.

B. A. Beune et al. [3]has used CFD analysis to analyse the opening characteristic of high-pressure safety valves. Several predefined meshes are employed to cover mesh deformation without deteriorating the quality, for complex geometries of high-pressure safety valve.

II. CFD ANALYSIS

ANSYS Fluent is an extensively used tool for CFX analysis all over the world. All the simulations and analysis shown in this paper are also performed in ANSYS Fluent software package.

Likewise most other CFX tools, ANSYS Fluent also divides its full analysis in to three elements. Pre Processor, Solver and Post-Processor.

A. Pre Processor

First we need to define geometry and topology of working fluid and surrounding solid parts. For that a 3D model is required to generate. ANSYS DesignModelerprovides modelling abilities, however for complex parts commonly used modelling softwares like Creo or Solid Works are recommended.
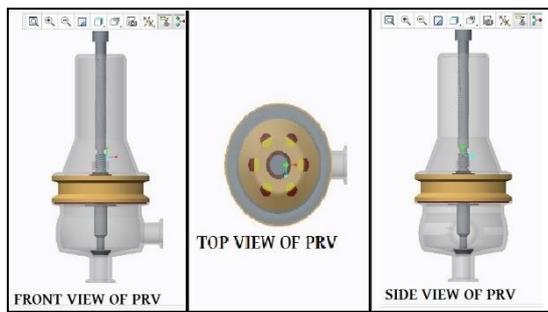
Fig. 3: 3D model of pressure relief valve

A 3D model of diaphragm valve generated in Creo Parametric is shown in figure 3. As we can see different components are assembled using Creo Parametric software. The model is assembled with a diaphragm, plunger alongwith base casing and converted to STEP format to make it compatible with ANSYS.

STEP model is imported into ANSYS Workbench's FLUENT module. Internal hollow portion of the valve is converted to fluid state from solid to produce geometry of fluid flow. For that purpose tool named 'Fill' is used in ANSYS DesignModeler as shown in figure 4.
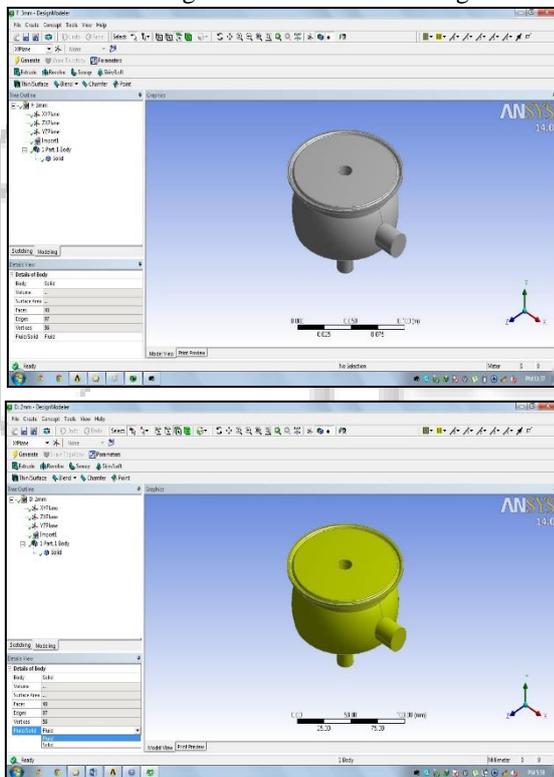

Fig. 3: Filling cavity in model

Now surfaces from which flow would come in and go out, need to be specified. Surface of orifice from one side is selected and named 'inlet' and surface from oppposite side is named 'outlet' by same way. Tool named 'named selections' in ANSYS DesignModeler is used for that purpose.

Next step is meshing. ANSYS uses ANSYS ICEM CFX for meshing which can be opened directly from workbench using 'Mesh' tab. 'Generate Mesh' tab in meshing software is used to generate mesh. Different mesh controlling parameters can be inserted to generate mesh with desired criteria. However, in current case no control parameters are used, that means automatic mesh is generated by default. Meshed component is shown in figure 5.The mesh is then updated to link output data to Fluent solver.

Now, next step is to set boundary conditions and values of other flow parameters. Conditions are assigned for this case are shown in table 1.
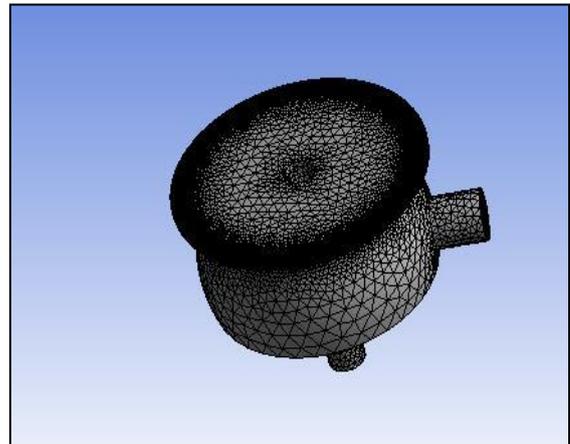

Fig. 4: Meshed diaphragm valve

Table 1. Conditions assigned for the case

Model	Viscous-Laminar
Fluid material	Water-liquid
Solid material	Steel
Inlet boundary condition	Pressure inlet(4bar)
Outlet boundary condition	Pressure outlet(1.bar)

Solution control parameters are to be assigned next. Simple solution method is chosen for current case. In solution monitors, convergence criteria can be assigned.
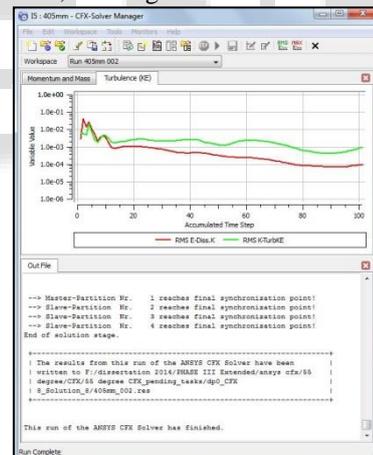

Fig. 5: Graph of residuals vs. iterations

B. Solver

After assigning all required data, solving process starts. First solution is to be initialized for calculating initial data. No. of iterations to be solved are allocated and setup is started.

ANSYS shows graph of residuals vs. iterations while solving process as shown in figure 6. When values of residuals declines from assigned criteria, solution gets converged. If solution doesn't converge, but values of residuals is steady, one can assume that solution is converged.

C. Post-processing

Results of solutions can be seen from post-processing module of ANSYS Fluent. Graphs, contours, streamlines can be generated and plotted in this module. Simulation of

flow can be animated to find discontinuity in flow. Results of current case are shown in figure 7.
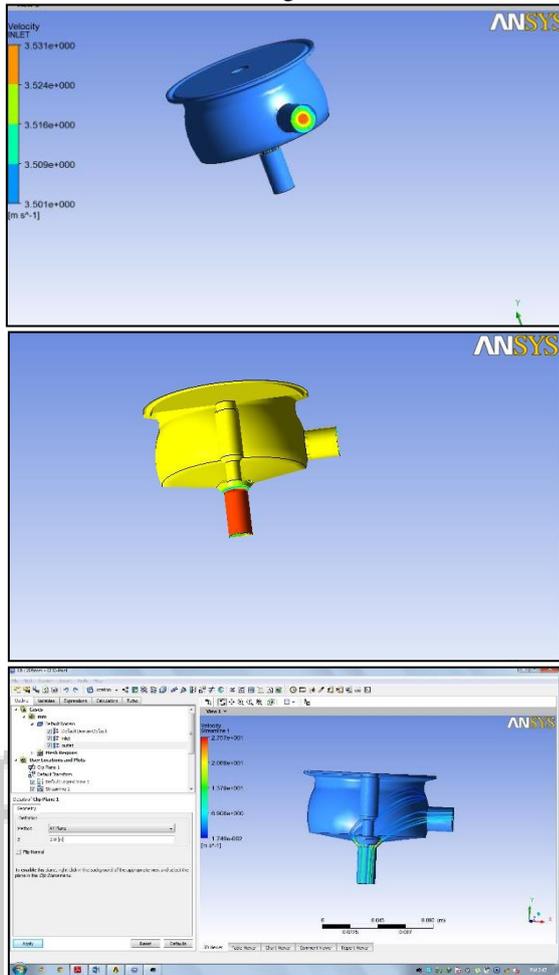

Fig. 6: Post-processing

D. Summary

As discussed in this paper, CFX analysis is found to be an excellent tool for designing a weir type diaphragm for improved flow characteristics. ANSYS Fluent can be used for CFX analysis to find out values of various flow parameters at different locations of valve. Conditionof flow in the valve can be understood easily with the help of graphical results generated in the software.

REFERENCES

[1] Peter Smith, R. W. Zappe, "Valve Selection Handbook: Engineering fundamentals for selecting the right valve design for every industrial flow application",5th Edition, Gulf Professional Publishing, 2004

[2] Hongjun Zhu, Qian Pana, Wenli Zhanga, Guang Fenga, Xue Li,"CFD simulations of flow erosion and flow-induced deformation ofneedle valve: Effects of operation, structure and fluid parameters", Nuclear Engineering and Design273 (2014),pp. 396–411.

[3] A. Beune, J.G.M. Kuerten, M.P.C. van Heumen, "CFD analysis with fluid–structure interaction of opening high-pressure safety valves", Computers & Fluids 64 (2012),pp. 108–116.

[4] ANSYS. ANSYS FLUENT tutorial guide, Release 14.0. Canonsburg; 2011.