

Flow Analysis of Two Way Gravity Diverter Valve for Discharge of Granular Material from One Point to Another in Chute and Free Fall Application

Nirav Patel¹

¹M.E. Student ((CAD/CAM))

¹A. D. Patel Institute of Technology, New V. V. Nagar, Gujarat

Abstract— Diverter valve is used to divert the flow of material with multi-diversion ports. This two-way gravity diverter valve diverts the free fall material with the help of flapping system. The work has been carried out for the CFD analysis of Gravity Diverter Valve using the discharge of powder material, which is free fall from silo. How the flow is changed with the flap position, actuating with the pneumatic cylinder is examined.

Key words: Gravity Diverter Valve (GDV), CFD analysis

I. INTRODUCTION

Diverter valve is a direction control valve is used to divert the material flow. Some of the types according their configuration are shown in Figure 1 to Figure 5 [5].

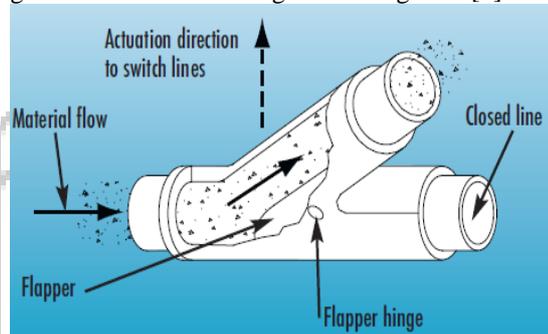


Fig. 1: Flapper Diverter Valve

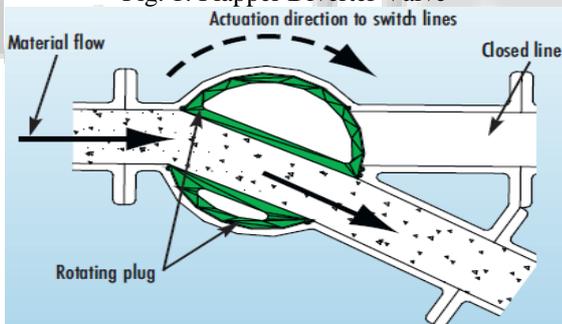


Fig. 2: Rotary plug diverter valve

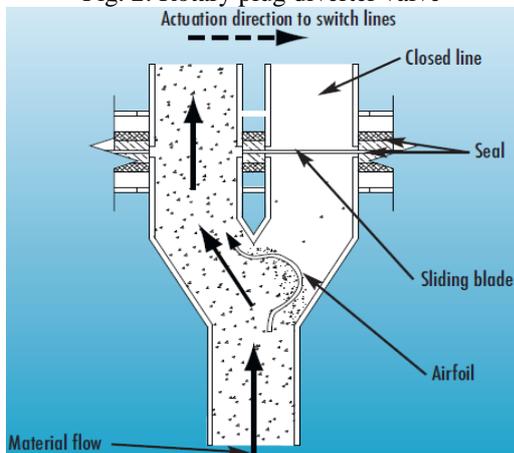


Fig. 3: Sliding blade diverter valve

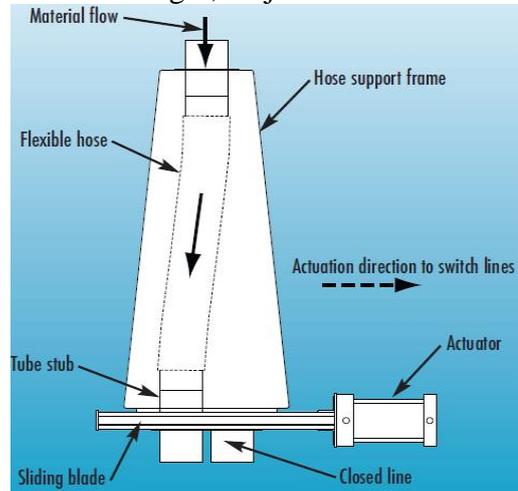


Fig. 4: Flexible tube diverter valve

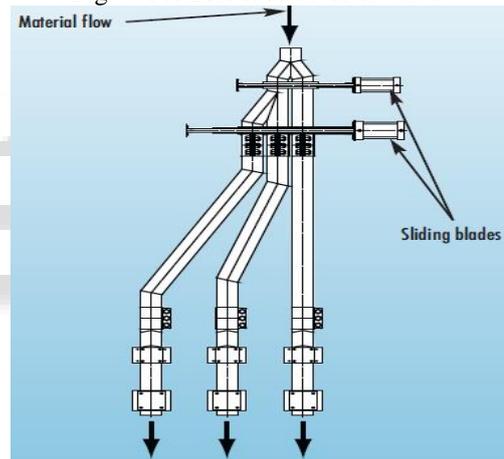


Fig. 5: Multiport diverter valve

From the above configurations of diverter valves, we considered the flapper diverter valve for gravity flow. Gravity diverter valve has main valve body with oscillating flap for direction control. Flap Position can be changed with the use of double acting pneumatic cylinder. Gravity flow of material is diverted by changing the flap position. Fig. 6 shows the sectional view of valve having straight way flow path from top to bottom and Fig. 7 shows the sectional view of inclined way flow path.

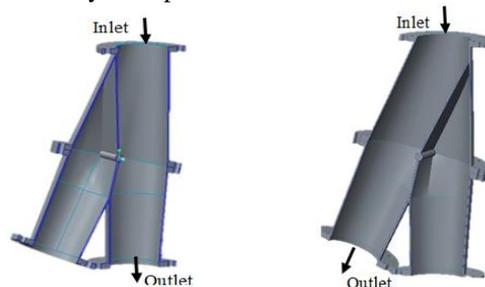


Fig. 6: Straight way flow path Fig. 7: Inclined way flow path

Gravity diverter valve is operated at the room temperature and CaCO_3 is used as material flow, which free fall from silo. CFD analysis is done for examine the disturbance in flow at flap and shaft position.

- 1) 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.
- 2) 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 CFX is a tool used for CFD analysis such as ANSYS FLUENT. Analysis and simulations of the GDV are performed in ANSYS CFX that are shown in paper.

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

A. Pre-Processor

Fluid volume and solid boundary are required for flow simulation. Fulfilling those requirements, we need geometry and topology of fluid volume. However, ANSYS has design modeler that provides modeling facility, for some complex parts commonly modelling software's like Creo and Solid Works are recommended.



Fig. 8: 3D Model of GDV

A 3D model is generated of Gravity diverter valve in Creo Parametric is show in Fig. 8. We can see in Fig. 3 there is one inlet of valve and two outlets. Flow will directed by flap position and either of the both outlets have open-close condition.

The model is 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 filled with cavity to produce geometry of fluid flow. For that purpose tool named 'Fill' is used in ANSYS Design Modeler as shown in Fig. 9.

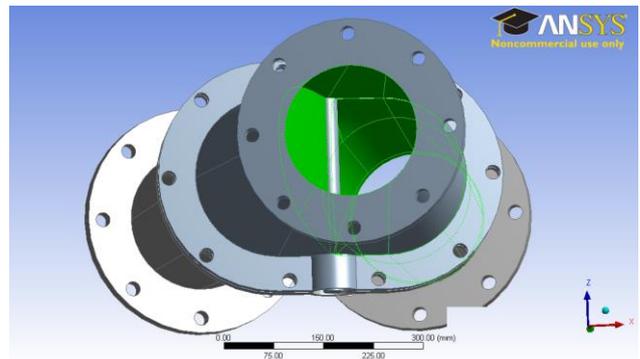


Fig. 9: Filling cavity in Model

In ANSYS Design Modeler the surfaces where the flow comes in and out is named as Inlet and Outlet respectively. Tool named 'Named selections' in ANSYS Design Modeler is used for that purpose.

Next step is meshing. ANSYS uses ANSYS ICEM CFD 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 Fig. 10. The mesh is then updated to link output data to CFX solver.

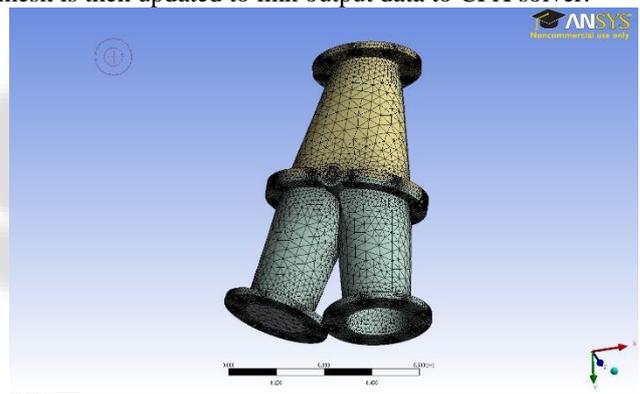


Fig. 10: Meshing of Model

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.

Model	K-epsilon
Fluid Material	CaCO_3
Solid Material	Steel
Inlet Boundary Condition	Mass Flow Rate (0.18 kg/s)
Outlet Boundary Condition	Pressure (2 bar)

Table 1. Conditions assigned for the case

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

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. When values of residuals decline from assigned criteria, solution is converged. If solution does not converge, but values of residuals are steady, one can assume that solution is converged.

C. Post Processor

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 Fig. 11.

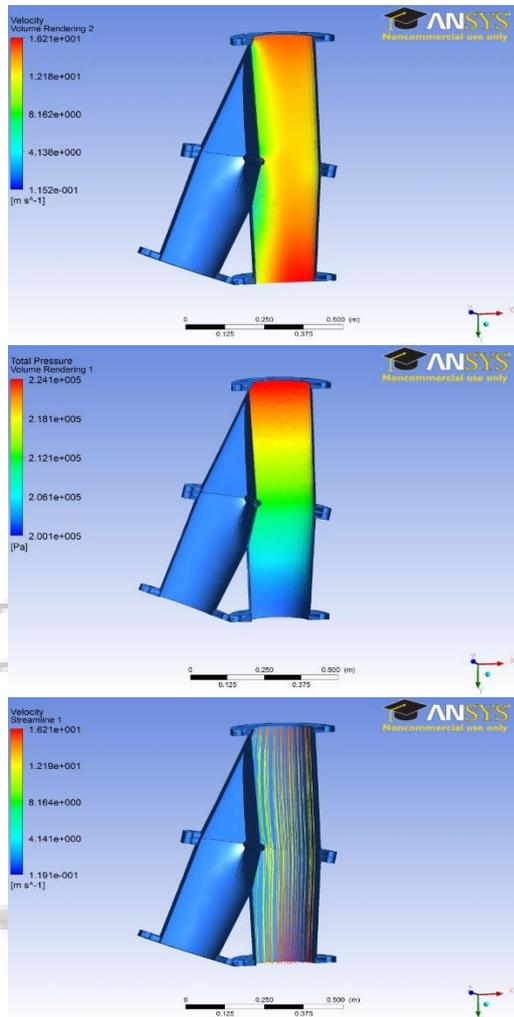


Fig. 11: Post processing

III. SUMMARY

As discussed in this paper, CFD analysis is found to be an excellent tool for designing a Gravity Diverter Valve for improved flow characteristics. ANSYS CFX can be used for CFD analysis to find out values of various flow parameters at different locations of valve. Condition of 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 of needle valve: Effects of operation, structure and fluid

parameters", Nuclear Engineering and Design 273 (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 CFX tutorial guide, Release 14.0. Canonsburg; 2011.
- [5] Robert Harkin, "Factors to consider when selecting a diverter valve"
- [6] www.podwerbulk.com

