Characteristic Analysis of Commercial Flap Non-Return Valve by using Ansys®

Jiby joy[1], Renjith R [2], Jose Cherian[3], Nikhil Mohanan [4]
[1]Student, [2]Assistant Professor, [3]Assistant Professor, [4]Student
Federal Institute of Science and Technology (FISAT)™,
Angamaly, Ernakulam, Kerala 683577

***Abstract*— A non-return valve is a valve that allows the direction of fluid flow only in one direction. Non-return valve plays a marginal role in pumping systems to avoid initial priming of pump set. The overall efficiency of pump system is much dependent on the foot valve. Since the efficiency of the whole system is reduced, when the starting stage (foot valve) is less effective. The traditional model non return valve is having more resistance to fluid flow due to its inefficient design and manufacturing [1].**

In this study, modelling and analysis of commercial flap non-return valve are done. The study was focused on finding the weak points of the foot valve. CREO Parametric 2.0 and ANSYS® CFD packages were used in this study. The various parameters analyzed during this analysis are flow pattern, velocity streamlines, pressure contours, turbulence etc.. The model and analysis of commercial flap non-return valve is performed on CFD software.

Keywords*— Analysis, ANSYS®, Foot valve, Fluent, Creo, Flow patterns, Velocity Contours, Pressure Contours, Effectiveness, Commercial Flap Non-Return valve

INTRODUCTION

The word ‘valve’ is derived from the Latin word *valva*, means the moving part of a door, in turn from *volvere*, to turn or roll. A valve is a device which regulates, directs or controls the flow of a fluid by opening, closing, or partially obstructing various passageways. In an open valve, fluid flows in a direction from higher pressure to lower pressure.[10] Valve is an essential component of a piping system that conveys liquids, gases, vapors, slurries etc..

Foot valve is a type of check valve and are placed at the pump’s wet well. Unlike other valves, a foot valve is created with a larger flow area than the actual pipe size to make sure that there is less head loss. Foot valves are either made of PVC plastic or stainless steel, and they are known for keeping the continuous presence of water within the pump.[11] Foot valves are used to maintain hydraulic pressure to keep the water flow in accordance with the given settings or configurations. There have been instances where the pressure can actually pop the valve out and cause major leakage; thus, it is important to design the valve in a better way and to use the right kind of material in the tubing to be able to support the force within the valve.[7]

The foot valve in a centrifugal pumping system keeps the suction pipe primed and blocks the entry of foreign material into the suction pipe. Foot valve consists of a casing, valve system and strainer. It offers resistance to the flow of water causing head loss and hence energy consumption, which is high in traditional foot valves. However, being a low cost component, people usually do not pay much attention to proper foot valve selection. As a result, they end up spending more for energy in the long run. In fact, a large number of foot valves are manufactured by local firms without meeting the standard design requirement. They are often energy inefficient but sold in the market at low cost. For most of the foot valves, the manufacturers do not provide information on the head loss or frictional factor features, making it difficult for consumers to choose better foot valves that are available in market. [1]

VARIOUS DESIGN PARTS AND SPECIFICATIONS FOR A STANDARD VALVE DESIGN

CASING

The casing size of foot valve was 180 mm dia. and 120 mm height with wall thickness of 5 mm. The casing was made of polypropylene. At the upper end, an eccentric mouth of 100 mm diameter and of length 30 mm was provided. The diameter of the casing was kept larger than opening size of the valve so that the flap valve could open to maximum angle of 90° and would not obstruct the water flow.

VALVE-SYSTEM

The valve system consisted of rubber washer with supporting synthetic rubber pieces at the top and at bottom of the washer. The shape of the top plate was made of both rectangular and semi-circular section with radius of 46 mm. The bottom plate was of circular cross section of 48mm size. The valve was made to rest on the flap by gravity force of fluid above valve, when the pump was not in operation. When water was sucked, the valve opened and was held to the casing. The weight of the flap valve system was 0.3 kg.

STRAINER

The strainer of the commercial foot valve was engraved with slots with trapezoidal shape on the surface. The numbers of slots made on the curved surface were 8. The width of slots was kept constant at 15 mm and the length of the slot increased at different angles on the curved surface. The central height of the 'Bell' curve at the bottom of the modified strainer was kept equal to the height of vertical slots. The edges of the slots were made round and smooth for the free flow of water.

There are 19 vertical slots were made along the periphery of the strainer with increase in slot dimensions. The combined open area of the vertical and bottom slots was increased in the modified strainer so as to keep the ratio of strainer opening area to the suction pipe area as 3 : 1. An exploded view of commercial Flap Non-return valve is in Fig. 1.

FLANGE

The inner surface of the flange opening was made smooth and curved to allow water flow with minimum frictional resistance.

Fig 1: Exploded view of Commercial Flap Non-Return Valve

The details and characteristics of the commercial flap non return valve are as follows:

- Ratio of strainer area to suction pipe cross sectional area is 3 : 1
- Ratio of flap open area to suction pipe cross sectional area ratio is 1.3: 1
- The angle of flap opening was kept at 90° for maximum efficiency.
- The CFV was made of virgin polypropylene to reduce friction.
- Strainer slots edges are made smooth to reduce frictional resistance to the streamlines

MATERIAL SELECTION:

Structure	Polypropylene
Flap Seal	Synthetic Rubber

DIMENSIONAL PARAMETERS:

Total height of valve	182mm
Height from bottom to flap of valve	91mm
Height from the flap to top of valve	91mm
Thickness of valve	7mm
Diameter of flap	48mm
Maximum width (diameter) of valve	110mm
Inlet Diameter of valve	44mm
Outlet Diameter of valve	90mm

METHODOLOGY AND STEPS

In this work, A generic model of the foot valve is prepared by using Creo Parametric software and this generic model is imported to ANSYS Fluent for the simulation and analysis of the flow pattern in the internal channel flow. After this meshing is done on the flow field volume. Hydrodynamic flow evaluation of water flow over an object can be performed using analytical method or CFD approach. On one hand, analytical method of solving water flow over an object can be done only for simple flows over simple geometries like laminar flow over a flat plate. If the flow gets complex as in flows inside a variable area housing, the flow becomes unpredictable and it is impossible to solve the Navier- Stokes and continuity equations analytically. On the other hand, obtaining direct numerical solution of Navier-stoke equation is not yet possible even with modern day computers. In order to come up with reasonable solution, a time averaged Navier-Stokes equation is being used (Reynolds Averaged Navier-Stokes Equations – RANS equations) together with turbulent models to resolve the issue involving Reynolds Stress resulting from the time averaging process. In present work the k-e turbulence model with non-equilibrium wall function is selected to analyze the flow over the generic valve design model. This k-e turbulence model is very robust, having reasonable computational turnaround time, and widely used by the flow based analysis industry.

STEPS OF ANALYSIS

- Select the model upon which analysis to be done.
- Designing the model in solid Creo 2.0 with proper dimensions & parameters.
- Import the Model into the analysis software.
- Select the Flow model and turbulence conditions
- Apply the boundary conditions.
- Analyzing the model in ANSYS 16.0 with fine mesh for higher accuracy
- Simulation & Testing of model for various properties such as flow pattern, pressure drop, velocity streamline etc.
- Study and Analyze the completed simulation model results

COMPUTATIONAL FLUID ANALYSIS

PREPROCESSING

Geometry Creation

CREO Parametric was used to create the model of the standard valve which is available in the regular market. The parts have been created on full scale with the valve full open condition. The net geometry consists of 4 parts. The strainer, flange, valve system and casing which were assembled to form the Geometry for analysis.

Defining the Geometry

The Surfaces of the designed parts were given specific names so as to identify them and to asses flow parameters based on those surfaces. The main selections for this one are the strainer surface, valve system surface, and top casing surfaces

Meshing

The meshing was done using ANSYS 16's default meshing client, Mechanical ICEM CFD (Integrated Computer Engineering and Manufacturing) Meshing Program.

PARAMETER SELECTION

Sizing

Advanced Sizing Functions	Based on Proximity and Curvature
Relevance Centre	Medium
Initial Seed Size	Active Assembly
Smoothing	Medium
Transition	Slow

Inflation

Inflation type Option	First Aspect Ratio
Method	Tetrahedrons

Solution Settings for Analysis Iterations

Pressure Velocity Coupling Scheme	Coupled
Solver Type	Pressure Based
Turbulence Model	Realizable K−ε Model with Non Equilibrium wall functions
Solution Method	Pressure-Velocity Coupling (Coupled) Based on Second order Upwind

Fluid Properties

Fluid	Water
Density	1000 kg/m^3

Inlet Conditions

Inlet Gauge Pressure	0 Pascal
Turbulence Intensity	1%
Turbulence Viscosity Ratio	10
Turbulence Specification Method	Intensity and Viscosity Ratio

Outlet Conditions

Outlet Gauge Pressure	0 Pascal
Turbulence Intensity	5%
Turbulence Viscosity Ratio	10
Turbulence Specification Method	Intensity and Viscosity Ratio

POSTPROCESSING

We visualize the results of the CFD simulation using ANSYS CFD-Post. This plotted vectors that are colored based on pressure, velocity, and turbulence, on a plane within the geometry. In addition, it can create output parameters within ANSYS CFD-Post for later reuse in ANSYS Static Structural for structural analysis based on Von Mises failure criterion

The following patterns were achieved from the above analysis

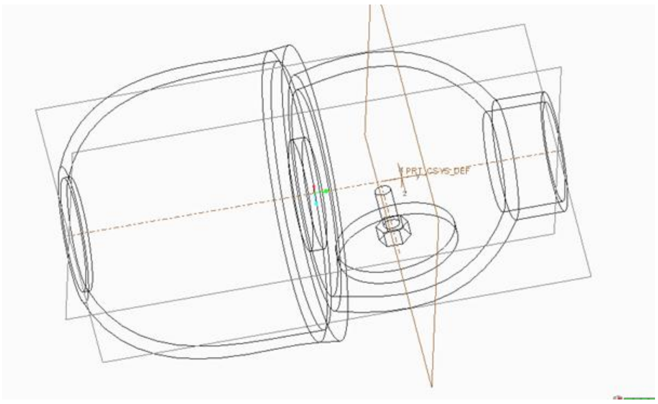

Fig 2

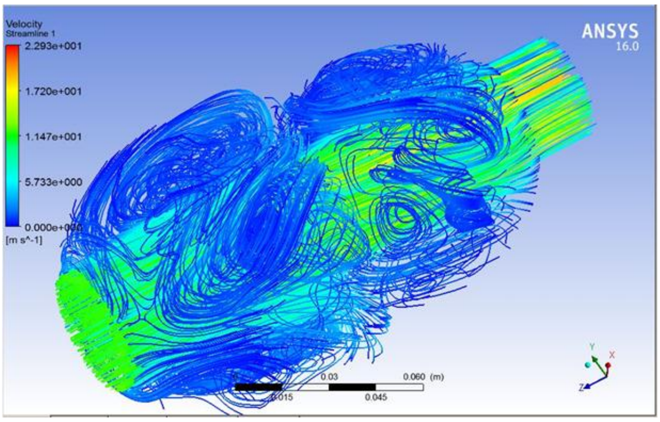

Fig 3

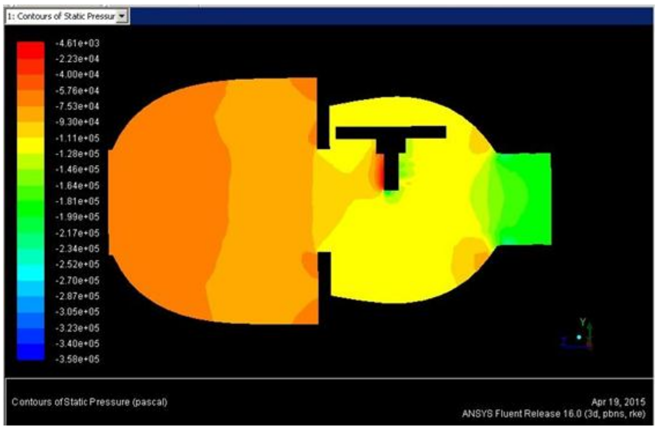

Fig 4

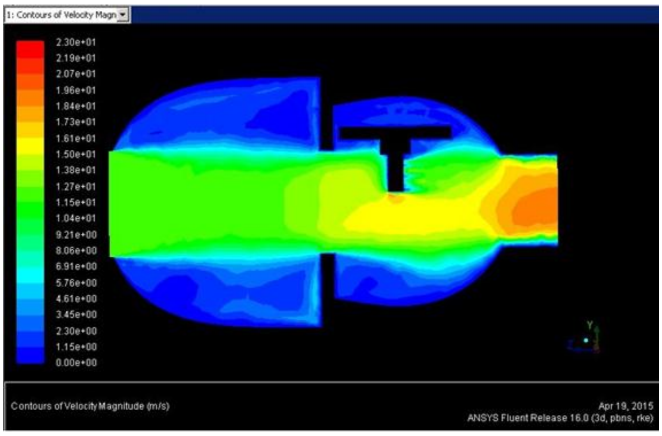

Fig 5

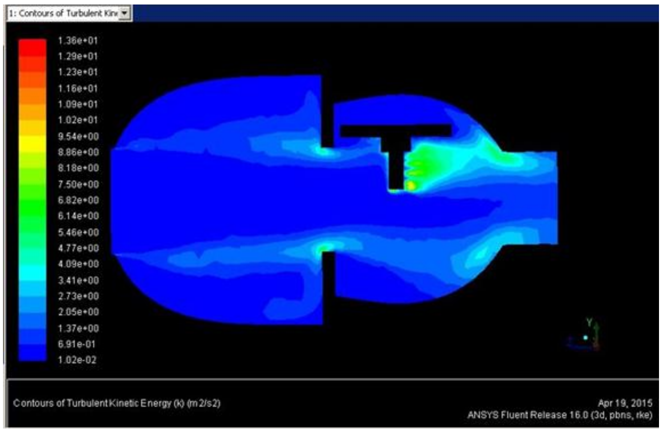

Fig 6

RESULTS AND DISCUSSION

From the Post-processing data, it can be see that there are regions of higher pressure and lower pressures in flow pattern, this leads to our understanding that the entire friction and head losses can be reduced if we can have a more uniform pressure distribution. Also from the Velocity plots it can be seen that the design increases recirculation and at the same time, adds to the turbulence at the wake. The Bob/Flap meanwhile produces a directional force at higher flow rates such that the entire setup can receive a net drop in power.

Fig2 shows the wire frame model of the commercial flap nonreturn valve.

Fig 3 shows velocity streamlines of water particles. The blue coloured lines shows the fluid is circulating throughout the valve's outer compartments at slower velocity rates, which in turn provide additional power to overcome the viscous flow losses whereas the green lines shows water with normal velocity. Light blue color indicates a transition between normal flow and slow speed flows.

Fig 4 shows pressure contours. In this, the pressure at the entry side cavity is very high since, the intake area and valve throat is which renders the entry flow cavity incapable of providing streamlined flow. This is compounded by the secondary cavity with moderate flow pressure. The drastic change in pressure values in the two compartments, suggests that there is a huge loss in the flow at this point. Hence in a more optimized model we may be able to see a more gradual drop in pressure as we move along the outer to the inner pipes.

Fig 5 shows velocity contours along the cross sectional plane. This show the maximum flow rate is achieved through a small area of the foot-valve cross section, such that the velocity beyond this segment seems to decrease at a quite fast rate, and hence a section conforming to the faster flowing area would increase the effectiveness of the foot valve, reducing material requirement up to an extent.

Fig. 6 shows Turbulence intensity of fluid flow along the cross sectional plane. For an effective flow within the valve with least drag resistance the turbulence should be minimal. Here from this figure, much turbulence variation is present which increases the drag component thereby reducing the overall effectiveness of the foot valve.

From the above discussion, it can be concluded that the present model foot valves are highly inefficient.

Hence, a far better design may be proposed based on the losses identified from this analysis, which can produce more effective foot-valve. Furthermore, an ideal foot-valve should have minimal velocity drop, minimal pressure variation and at the same time least possible turbulence intensity along the flow.

ACKNOWLEDGEMENT

I would like to express my sincere gratitude to all those who have helped me in the successful completion of the project entitled "*characteristic analysis of commercial flap non-return valve by using Ansys®*"

REFERENCES

[1] *K.S.ReddyJ, Harshna D.Baghde, A.C. VarshneyJ, D.M.BhandarkarJ and v.K.Pandey*-Development and Evaluation of low friction foot valve
[2] F.M. White, Fluid Mechanics, Fourth Edition, Mc-Graw-Hill, 1999.
[3] J.A. Fox, Hydraulic Analysis of Unsteady Flow in Pipe Networks, Wiley, New York, 1977.
[4] H.K. Versteeg, W. Malalasekera, An Introduction to Computational Fluid Dynamics, Prentice Hall, 1995.
[5] Patel S M. 1982 Sujala foot valve can save energy worth 10 crore rupees per year in agricultural pumps ofGujarat State. Agric. Enging Today, 6(1),27-30.
[6] Reddy K S; Varshaney A C; Gharde K D. 2002. Assessment of agricultural pumping systems. Technical report: CIAE *1*IDED *I* NATP - *TOEI* 2002 *1284*, CIAE Publication.
[7] Reddy K S; Gharde K D; Varshaney A C. 2003. "Performance evaluation of foot valves" Technical bulletin: CIAE/2003/1 00. Central Institute of Agril. Engineering, Bhopal.
[8] Sewa Ram; Chauhan H S; Trivedi S K; Sharma H C 1980. Energy loss in irrigation pumping sets -A survey. Energy Management, 4 (3), 293 -296.
[9] Sewa Ram; Sharma H S; Chauhan H S. 1982. Pantnagar foot valve. AE Bulletin: 82. Department of Agricultural Engineering, GBPUAT, Pantnagar.
[10] http://www.gpa.no/no/virksomhetsomraade/offshoremarine/valves/
[11] http://www.websterpumpsonline.com/websterpump_term_definition_a_m.php#.Ve-grbO1ktM