ISSN(online): 2321-0613

Application of Cad/Simulation Towards Industrial Valve

T.Pavan Kumar[1] M.Praveen Kumar[2] S.Shravan Kumar[3]
[1]HOD [2,3]Student
[1,2,3]Department of Mechanical Engineering
[1,2,3]NNRG,Hyderabad

Abstract— Pumping applications with high head, surge tanks, or multiple pumps, have long proved a challenge to system operators trying to minimize line surges resulting from slamming non-return valves. Traditional non-return valves use outside levers with weights or springs in an attempt to reduce slam. By doing so, they pull the disc down into the flow and reduce the length of the disc stroke so that it closes faster. Unfortunately, while outside levers and weights help solve one problem, they create two others. Increased pressure loss and maintenance are inevitable with traditional non-return valves. Pulling the disc down into the flow creates a blockage in the line and causes tremendous pressure loss and turbulence. The valve comprises bonnet, internal disc, drain pipe and inlet/exit duct for connection to upstream and downstream component and works at above or atmospheric pressure conditions depending upon process equipment configuration. In order to evaluate flow characteristics and pressure drop valves through measurement, investment for development test rig and associated instrumentation is required. Added to this, supply of working fluid at desired pressure to the valve is also needed to evaluate its performance. This is a slow process and requires the support of several agencies like skilled man power, space, procurement action of required instruments etc.

Key words: Application of Cad, Towards Industrial Valve

I. INTRODUCTION

In steam systems [1], the principal functions of valves are to isolate equipment or system branches, to regulate steam flow, and to prevent over pressurization. The principal types of valves used in steam systems include gate, globe, swing check, pressure reducing, and pressure relief valves. Gate, globe, and swing check valves typically isolate steam from a system branch or a component.

Pressure reducing valves (PRV) typically maintain certain downstream steam pressure conditions by controlling the amount of steam that is passed. These reducing valves are often controlled by transmitters that monitor downstream conditions. Pressure relief valves release steam to prevent over-pressurization of a system header or equipment. In coal fired power stations, these valves in different sizes are employed in extraction lines between turbine cylinders to heaters and condensers

A. Background Of Study

Traditional swing check valves use outside levers with weights or springs in an attempt to reduce slam. By doing so, they pull the disc down into the flow and reduce the length of the disc stroke so that it closes faster. Unfortunately, while outside levers and weights help solve one problem, they create two others. Increased head loss and maintenance are inevitable with traditional swing check valves. Pulling the disc down into the flow creates a blockage in the line and causes tremendous head loss and turbulence. With the disc oscillating in the flow, the shaft, bearings, and shaft seal are all subjected to wear and reduced service life

II. LITERATURE REVIEW

The function of a check valve is to protect mechanical equipment and to maintain the inventory of a system against postulated piping failure by preventing reversal flow and resultantly it helps systems to perform their intended functions. If transient incidents such as pump trips and postulated pipe breaks occur check valves complete the process of closing after some degree of reverse flow is established due to its inherent mechanical dynamics and dependence on the flow. The reverse flow causes the check valve to close rapidly through the remaining portion of its travel. The reverse flow is then stopped instantaneously by the closed valve causing a loud water hammer in the pipe.

Non-Return Valves are a substantial part of numerous accessories applied in water supply and other hydraulic systems [3]. The application of these valves is aimed to prevent reverse flow under different conditions. This is necessary, for example, to prevent emptying of pipelines after pump shutdown and work of a pump as a turbine at intolerable high rotation speed due to backflow [4]. The application of check valves may also be connected with the water hammer phenomena, e.g., the valves at the pump discharge providing slow closure after pump normal switching-off, or after an energy supply interruption; the valves in by-pass lines of pumps, etc.

Energy losses in pipelines related to hydraulic resistance of check-valves depend on their type, design and adjustment. The simple and very often used design of the check valves is based on the use of flow action on the valve without application of external energy source. The plate closing/opening of the water passage is under action of pressure and friction forces from the liquid flow and the moment of these forces is under steady flow conditions in equilibrium with the moment of counterweight and friction forces in the bearing. Swing type check valves and valves with tilting disc, if the counter weights are applied, both are the direct control action valves. The check valve design in which a spring is used for the valve closure as an alternative to the weight belongs also to this type of the valves.

III. GOVERNING EQUATIONS FOR FL0W TRANSPORT

The set of equations which describe the processes of mass, momentum, energy based on conservation principles [18] general form of the mass conservation equation and is valid for incompressible as well as compressible flows. The source Sm is the mass added to the continuous phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources. The general form of momentum equation is written as

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla\cdot(\rho\vec{v}\vec{v}) = -\nabla p + \nabla\cdot(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F}$$

where *p* is the static pressure, $\bar{\bar{\tau}}$ the stress tensor (described below), and $\rho\ \bar{g}$ and $\bar{F}$ the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively. also contains other model-dependent source terms such as porous-media and user-defined sources. The stress tensor $\bar{\bar{\tau}}$ is given by

$$\bar{\bar{\tau}} = \mu\left[\left(\nabla\vec{v} + \nabla\vec{v}^{\mathrm{T}}\right) - \frac{2}{3}\nabla\cdot\vec{v}I\right]$$

Where μ is the molecular viscosity, I is the unit tensor, and the second term on the right hand side is the effect of volume dilation Turbulent flows are characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms of known quantities.

The simplest "complete models" of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard k- ϵ model in FLUENT falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time robustness, economy, and reasonable accuracy for a wide range of turbulent flows, explain its popularity in industrial flow and heat transfer simulations. The standard k- ϵ model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ϵ). The model transport equation for k is derived from the exact equation, while the model transport equation for ϵ was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. In the derivation of the k- ϵ model, it was assumed that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k- ϵ model is valid only for fully turbulent flows.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\epsilon - Y_M + S_k$$

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial\epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(G_k + C_{3\epsilon}G_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon$$

In these equations, Gk represents the generation of turbulence kinetic energy due to the mean velocity gradients, Gb is the generation of turbulence kinetic energy due to buoyancy Ym represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. C1□ , C2□ , C3□ are constants. □k , □□ and are the turbulent Prandtl numbers for and , respectively. and Sk , S□ are user-defined source terms. The turbulent (or eddy) viscosity, □t is computed by combining and as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon}$$

The values of model constants are

$$C_{1\epsilon} = 1.44,\ C_{2\epsilon} = 1.92,\ C_\mu = 0.09,\ \sigma_k = 1.0,\ \sigma_\epsilon = 1.3$$

Heat transfer can occur by three main methods: conduction, convection, and radiation. Physical models involving only conduction and/or convection are the simplest, while buoyancy-driven flow, or natural convection, and radiation models are more complex. Depending on your problem, FLUENT will solve a variation of the energy equation that takes into account the heat transfer methods. The general form of energy equation is written as

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = \nabla \cdot \left(k_{\text{eff}} \nabla T - \sum_j h_j \vec{J}_j + (\overline{\overline{\tau}}_{\text{eff}} \cdot \vec{v}) \right) + S_h$$

Where, keff is the effective conductivity (k+kt), where kt is the turbulent thermal conductivity, defined according to the turbulence model being used), and is the □j diffusion flux of species j.

The first three terms on the right-hand side of equation represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively. Sh includes the heat of chemical reaction, and any other volumetric heat sources.

$$E = h - \frac{p}{\rho} + \frac{v^2}{2}$$

Where h is the enthalpy defined as

$$h = \sum_j Y_j h_j$$

For incompressible gas flow enthalpy is defined as

$$h = \sum_j Y_j h_j + \frac{p}{\rho}$$

Y j is the mass fraction of species j and reference Temperature is 298.15K

$$h_j = \int_{T_{\text{ref}}}^{T} c_{p,j} \, dT$$

Detailed mathematical descriptions for the above formulation can be seen in several published literature and software user manual.

A. Finite Volume Techniques

This control volume technique consists of integrating the governing equations about each control volume, yielding discrete equations that conserve each quantity on a control-volume basis. Discretization of the governing equations can be illustrated most easily by considering the steady-state conservation equation for transport of a scalar quantity ϕ. This is demonstrated by the following equation written in integral form for an arbitrary control volume as follows:

$$\oint \rho \phi \vec{v} \cdot d\vec{A} = \oint \Gamma_\phi \nabla \phi \cdot d\vec{A} + \int_V S_\phi \, dV$$

IV. MESHING AND MODELLING

A. Application Of Non-Return Valve And Geometrical Model

Applications of different sizes of Quick Closing Non-Returning Valves (QCNRV) are in extensive use at coal fired/Nuclear power station as shown in Figure:4.1 wherein ESV/CV is Emergency stop valve/control valve, HP, IP, LP indicates high/intermediate/low pressure, TG- Turbo-generator, CRH/NRV are Cold Reheat/Non return valves, IV – Isolation valve and LPBV – Low pressure bypass valve.

The valve geometry of three dimensional model is generated from the set of manufacturing drawings using bottom-up approach in different parts and the final assembly of the component

B. Altair Hyper Works – CAD Repair For Flow Volume Extraction

Hyper Mesh includes direct readers to popular native CAD file formats. Moreover, Hyper Mesh has robust tools to clean-up (mend) imported CAD geometry that contain surfaces with gaps, overlaps and misalignments which

Hinder high-quality mesh generation. By eliminating misalignments and holes, and suppressing the boundaries between adjacent surfaces users can mesh across larger, more logical regions of the model significantly increasing meshing speed and quality. Boundary conditions can also be applied to these surfaces for future mapping to underlying element data.

The Hyper Works products have gotten significant updates in usability. Across Hyper Mesh, Hyper View and Hyper Graph tools like the Entity Editor, Plot browser and the Advanced Build Plots panel ensure faster access to data and modification of multiple entities simultaneously. This release provides a variety of CFD related meshing feature updates and mesh quality review functionality. The already strong offerings for composites modeling have been refined with various updates and additions to existing features and procedures. Advancements in the tools for model assembly improve the handling of more and more complex models. Altair Hyper Works module hyper Mesh provides powerful geometry preparation engine with model import generated in other CAD Software packages of different file formats.

The geometric files upon its imports displays Solid model as a volume. Examination of this model through topological connections. The flow volume of the valve geometry extracted with dimensions.

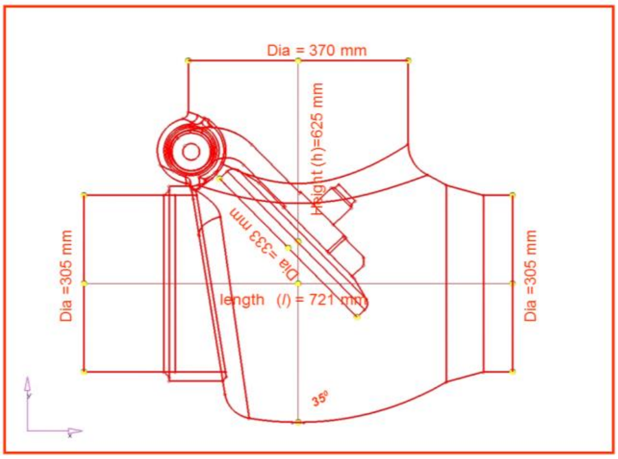

Fig. 4.1: Cad Model

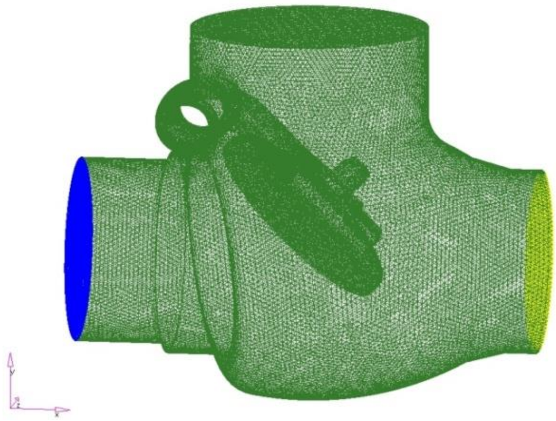

Fig. 4.2: Ansys Model

V. CONCLUSION

In steam systems, the principal functions of valves are to isolate equipment or system branches, to regulate steam flow, and to prevent over-pressurization. Valves which open fully but are not stop limited, liable to flutter. Flutter can also be caused by flow fluctuations due to presence of turbulent eddies. Effects of flutter can leads undesirable effects in terms of aerodynamics efficiency and structural integrity. In order to understand the flow pattern inside the valve and its influence towards pressure drop are studied through numerical simulation using Computer Aided Engineering and Simulation techniques on high speed digital devices.

Creation of solid models from the set of manufacturing component assembly drawings is the first step and successfully developed using the State of Art Modelling Software UG-NX. In order to study flow distribution, flow volume is extracted and necessary surface corrections are carried out to generate volume of valve body. Since the flow variables are functions of space coordinates, calculations of velocity, pressure and turbulence quantities needs the point data through different types of elements called Mesh using Altair Hyper works software. After checking the it's quality along with boundary conditions, mesh data has been exported to flow solver Ansys Fluent. In the pre-process setup, type of working fluid and inlet and exit conditions are used to simulate the flow in the valve body. After successful convergence, the results in the valve plane in terms of flow pattern, flow path and pressure pattern through contour plots are analyzed.

The flow distribution around valve disc takes several turns resulting to presence of turbulence vortices. The variation of velocity from the inlet to exit locations shows highly non-uniform with maximum velocity at bottom corner of exit location. The vector plot highlights the smooth flow at entry location to some distance and then decreases and forms turbulent vortices bottom of the valve body and loss of momentum in the flow around the disc resulting to unequal flow at exit location. A larger pressure is being built up by the forward flow compared to the backward flow, which will affect the hydrodynamic force and corresponding torque on the flapper. Using the average quantities of flow variables, pressure resistance coefficient is determined and the automation procedure has been developed for several disc positions in the valve chamber from opening to closed position.

The experimental measurements through laboratory test setup using probe for traversing mechanism with actuator fixed at the air duct fixture. Flow resistance coefficient is evaluated for full opening using the average values of pressure, inlet density and velocity for several opening positions. The performance characteristics of valve in terms of pressure resistance coefficient obtained both simulation and measurement compared and agreement is satisfactory. It is also noticed that at some valve positions, some deviations are observed which can be improved by considering more advanced numerical techniques related to mesh generation/turbulence models.

REFERENCES

[1] Dvir Y. (1997) Flow Control Devices, Israel, 101-132
[2] K. G. Saravanan and N. Mohanasundara Raju Structural Analysis of Non Return Control Valve using Finite Element Analysis International Engineering Research & Technology (IJERT) Vol. 4, April-2015
[3] Marcus N. (1966) Dynamics of controlled check valves for control of flap shock and water hammer. Int Jrnl Water and Irrigation Review, 16, (3), 31-34.
[4] Thorley A.R.D. and Enever K.J. (1979) Control and suppression of pressure surges in pipelines and tunnels, CIRIA Report 84, 19, 25-27.
[5] Kalizun V.I. (1973) Osnovi vodosnabjenia i kanalisazii Principles of Water Supply and Sewage, Moscow, 1-20.
[6] Anon (1988) Water Supply Accessories., Israel, 2-26
[7] Check Valve Analysis and Prioritization, Kalsi Engineering, USA, 2008
[8] Pordal H.S et al, Computational fluid dynamics a key analytical tool, Hydrocarbon processing, pp 85-91, 2001
[9] Ming C Leu Albin Thomas Krishna Kolan NX 9.0 for Engineering Design, Manual, Wisconsin University, USA
[10] Thomson, J.F et.al Numerical Grid Generation – Foundations and Applications, North Holland, Amsterdam, 1985
[11] Shekar Majumdar Pressure based finite volume algorithm for viscous flow computation, CFD Advances/Applications, NAL, Bangalore, 1994
[12] Bhasker C Post-Doctoral research report on high performance computing, ERC, Mississippi State University, USA, 2000.