"CAD MODELING & OPTIMISATION WITH FEA OF FLOW REGULATING VALVE FOR REDUCING COST"

Anant A. Deshpande[1] , Dhiraj V. Astonkar[2] , Dushyant B. Pawar[3]

[1] M.E. (CAD/CAM) Student, Department of Mechanical Engineering, Dr. Sau. Kamaltai Gawai Institute of Engineering & Technology, Darapur, Tq. Daryapur, Dist. Amravati, Maharashtra, India
[2] Assistant Professor, Department of Mechanical Engineering, Dr. Sau. Kamaltai Gawai Institute of Engineering & Technology, Darapur, Tq. Daryapur, Dist. Amravati, Maharashtra, India
[3] Assistant Professor, Department of Mechanical Engineering, Dr. Sau. Kamaltai Gawai Institute of Engineering & Technology, Darapur, Tq. Daryapur, Dist. Amravati, Maharashtra, India

Abstract

In a water distribution system, Flow Control Valve is used to control the flow rate in the pipeline connections. In this thesis, a fixed flow control valve is investigated to reduce the flow rate and set to deliver the pre-set flow of 5-6 LPM (litre per minute). Which helps to distribute the water for a maximum period and maintains the usage only for the drinking purpose? Geometry of FCV with a ball check valve is implemented, where the ball check helps to stop the back flow of the fluid from the valve. Detailed inspection of dynamic changes in pressure and flow velocity in the valve is conducted through simulation. The study of fluid properties describes the expected design and specifies the flow structure in the valve. The results of this project demonstrate a good performance of the design-build and influence the requirements. The obtained values in the simulation, analytical and experimental results are compatible, which concludes the survey of FCV is equipped to custom.

Keyword: *Cavitation, Flow Control Valve, Orifice*

Introduction:

Flow control valves include some simple orifices, which are to sophisticate the closed-loop electrohydraulic valves, where it automatically adjusts to variations in temperature and pressure.

The purpose of flow control in a hydraulic system is to regulate the speed. Here are the some of the devices, which control the speed of an actuator in the component of a machine to help in moving and controlling the hydraulic system by regulating the flow rate. The flow rate can determine the rate of energy transfer at any specified pressure. Where the actuator force multiplied by the distance, through which it moves is equal to the work done on the load, while the energy transformed must always be equal to the work done. Actuator speed determines the rate of energy transfer, and thus the speed is a function of flow rate.

Now coming to the directional control, it doesn‘t deal with primary energy control but rather through directing the transfer system to the appropriate place in the system of a specified time.

Direction control valves are a thought of a fluid adjustment, which makes desired contacts. It is that they direct the high-energy input stream to the actuator inlet and deliver a little energy return path.

The control of energy transfer system through flow controls and pressure a slight concern if the flow steam can‘t reach at the perfect place and time. Then the secondary directional control devices are set to solve the time cycle. Due to the fluid flow, often can be regulated by directional control valves in which approximate flow rate or pressure control are achieved

The main objectives of this paper are:

- To build geometry model of flow control valve.

- Static analysis on FCV to study the pressure on valve.
- Comparisons of static analysis with different pressures

Theoretical Model:

Fluid dynamics is a field of study in fluid mechanics, which deals with fluid flow in motion. It has a wide range of applications in calculating the forces and moments on valves, determining the mass flow rate of liquids through pipelines. The solutions of the fluid dynamics problem characteristically involve by solving various properties of the fluid, such as pressure, flow velocity, density and temperature as functions of time and space.

Conservations laws

The three conservations laws used for fluid dynamics are:

Mass continuity: The rate of change of fluid flow inside the constant volume must be equal to the net rate of fluid flow into the volume.

$$\frac{\partial}{\partial t}\iiint_v \rho dv = - \oiint_s \rho.u.dS$$

In the above equation, ρ is the fluid density, u is the flow velocity vector, *t* is time. By divergence theorem the differential form of continuity equation is,

$$\frac{\partial \rho}{\partial t} + \Delta.(\rho u) = 0$$

Conversation of momentum: The application of Newton's second law of motion to the control volume due to the net flow of air in volume and the action of external forces on the air within the volume.

Conversation of energy: The energy can be converted from one from to another from, conservation of energy gives the total energy in a closed system.

Reynolds Number

Reynolds number is a ratio of inertial forces to viscous forces within a fluid which is subjected to relative internal movement due to different velocities, on the boundary surface of a pipe. Reynolds number (Re) is a dimensionless quantity in fluid mechanics.

$$\text{Re} = \frac{\rho v D}{u}$$

Where,

ρ = density of the fluid

v = characteristic velocity of fluid with respect to object

D = diameter of the hydraulic pipe

u = dynamic viscosity of the fluid

Laminar Flow

Laminar flow is a type of fluid flow, in which the fluid travels smoothly or in regular paths. In laminar flow, occasionally the streamline velocity, pressure and flow properties at each point in the fluid remain constant. It is in straight pipe considering the relative motion of a set of concentric cylinders of fluid, fixed on the outside of the wall. Laminar flow is common only in some cases where the flow is relatively small.

Turbulent Flow

Turbulent flow is a type of fluid flow, in which the fluid enters and undergoes irregular fluctuations, or mixing. In turbulent flow, the speed of the fluid at a point is continuously undergoing changes in both direction and magnitude.

The most of the flow considered as a turbulent flow, usually, the examples of turbulent flow water transport in pipelines, flow through turbines and pump.

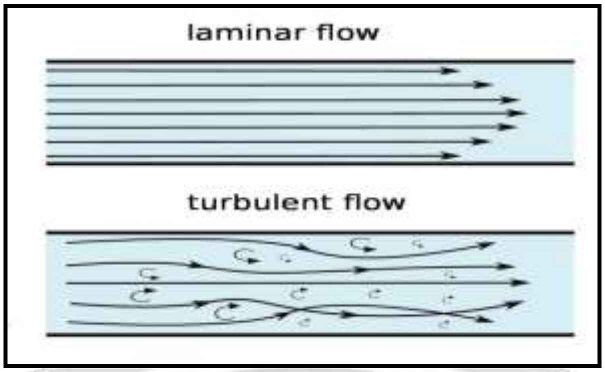

Fig.-1: Fluid flow view

In this paper, through calculations and the flow pattern expected is turbulent flow in the present flow control valve. The computational analysis is done in turbulent flow module to calculate the volumetric flow rate.

Fluid Flow equations

The fluid flow equation is derived to calculate the flow rate of the valve and describes the motion of viscous fluids. The equations of fluid mechanics used in this thesis are Bernoulli equation, continuity equation, pressure, orifice fluid flow and Navier-stokes equation.

Fluid flow in Orifice plate

Considering the steady flow of a constant density fluid in the valve, Bernoulli equation is used to consider being a statement of conservation of energy principle to suitable for flowing fluids.

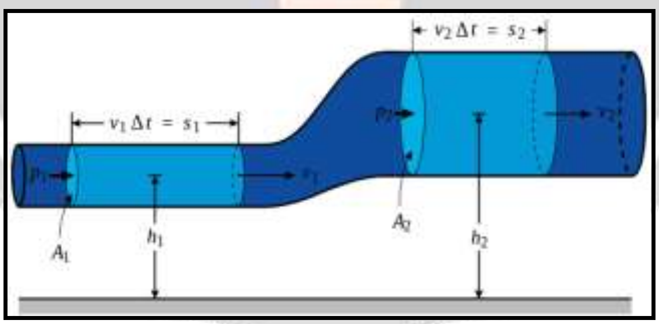

Fig.-2: Bernoulli's principle

$$\frac{P_1}{\rho \cdot g} + \frac{u_1^2}{2g} + z_1 = \frac{P_2}{\rho \cdot g} + \frac{u_2^2}{2g} + z_2$$

Comparing with continuity equation

$$A_1 u_1 = A_2 u_2 = Q_v$$

Where,

P_1= inlet pressure

P_2= outlet pressure
u_1 = inlet velocity
u_2 = outlet velocity
A_1 = area of the inlet pipe
A_2 = area of the outlet pipe
Z_1, Z_2 = potential energy
ρ = density of the fluid

$$\frac{u_2^2}{2}\left[1-\left(\frac{A_2}{A_1}\right)^2\right]=\frac{P_1-P_2}{\rho}$$

$$Q_v = A_2u_2 = C_dA_2\left[\frac{2(P_1-P_2)/\rho}{1-\left(\frac{A_2}{A_1}\right)^2}\right]^{1/2} \quad where \quad \Delta P = h\rho g$$

$$Q_v = C_dA_2\sqrt{2h\rho g}$$

Now, in our case A_2 is the area of the orifice plate, h is pressure head in meters and C_d is the coefficient of discharge. C_d = 0.70 for this condition.

Table - 1: Analytical Calculations of Orifice plate

Inlet pressure in bar's	h pressure head in meters	Area of Orifice plate A_2 in m^2	Orifice diameter in mm	Orifice velocity in m/sec	Flow rate Q_v in m^3/sec
0.4	4	1.257e-5	4	6.1	7.79e-5
0.5	**5**	**1.257e-5**	**4**	**6.92**	**8.71e-5**
0.6	6	1.257e-5	4	7.59	9.54e-5
0.7	7	1.257e-5	4	8.1	1.03e-4
0.8	8	1.257e-5	4	8.7	1.10e-4
0.9	9	1.257e-5	4	9.2	1.16e-4
1	10	1.257e-5	4	9.7	1.23e-4

The obtained flow rates for the different initial pressure values are calculated. The initial pressure can be given to the pressure gauge which is connected to the valve for the require pre-set flow. In this thesis, the investigation is done with the initial boundary conditions of pressure 0.5 bar and validating the result with other two methods.

CAD Model:

CAD is the Computer Aided Design.CAD is utilized in the conceptualization, design and documentation of product. The PRO/E software offers several different approaches to develop a solid model of regulating valve like part designs, surface designs etc. A solid model was created using PRO/E wildfire 4.0. With the help of PRO/E different concept of CAD modeling has created.

Introduction to CAD Software [Pro/E Wildfire 4.0]

Pro/ENGINEER is a parametric, feature based, solid modeling System. It is the only menu driven higher end software. Pro/ENGINEER provides mechanical engineers with an approach to mechanical design automation based on solid modeling technology and the following features.

3-D Modeling

The essential difference between Pro/ENGINEER and traditional CAD systems is that models created in Pro/ENGINEER exist as three-dimensional solids. Other 3-D modelers represent only the surface boundaries of the model. Pro/ENGINEER models the complete solid. This not only facilitates the creation of realistic geometry, but also allows for accurate model calculations, such as those for mass properties.

Parametric Design

Dimensions such as angle, distance, and diameter control Pro/ENGINEER model geometry. It can create relationships that allow parameters to be automatically calculated based on the value of other parameters. Whenever modify the dimensions, the entire model geometry can update according to the relations created.

CAD Geometry:

Feature-Based Modeling

It can create models in Pro/ENGINEER by building features. These features have intelligence, in that they contain knowledge of their environment and adapt predictably to change. Each feature asks the user for specific information based on the feature type. For example, a hole has a diameter, depth, and placement, while a round has a radius and edges to round.

Associativity

Pro/ENGINEER is a fully associative system. This means that a change in the design model anytime in the development process is propagated throughout the design, automatically updating all engineering deliverables, including assemblies, drawings, and manufacturing data. Associatively makes concurrent engineering possible by encouraging change, without penalty, at any point in the development cycle. This enables downstream functions to contribute their knowledge and expertise early in the development cycle.

Capturing Design Intent

The strength of parametric modeling is in its ability to satisfy critical design parameters throughout the evolution of a solid model. The concept of capturing design intent is based on incorporating engineering knowledge into a model. This intent is achieved by establishing feature and part relationships and by the feature-dimensioning scheme. An example of design intent is the proportional relationship between the wall thickness of a pressure vessel and its surface area, which should remain valid even as the size of the vessel changes.

Combining Features into Parts

The various types of Pro/ENGINEER features serve as building blocks in the progressive creation of solid parts. Certain features, by necessity, precede others in the design process. The features that follow rely on the previously defined features for dimensional and geometric references. The progressive design of features can create relationships between features already in the design and subsequent features in the design that reference them.

Part Modeling:

- *Starting Out in Part Mode*--Describes how to start creating a part with Pro/ENGINEER.
- *Sketcher*--Describes how to create sketches in a stand-alone Sketcher mode.
- *Datum*--Describes how to create datum features: datum planes, datum points, datum curves, datum axes, coordinates features, graphs, evaluate features.
- *Sketching on a Model*--Describes how to create 3-D sections in the process of feature creation.
- *Feature Creation Basics*--Describes how to create extruded and revolved protrusions.
- *Sweeps, Blends, and Advanced Features*--Describes how to create sweeps, blends, and advanced features.
- *Construction Features*--Describes how to create construction features, such as holes, slots, and cuts.

- *Rounds*--Describes how to add rounds to part geometry.
- *Tweak Features*--Describes how to create tweak features, such as draft, local push, and section dome.
- *Creating Surface Features*--Describes how to create surface features.
- *Creating Advanced Surface Features*--Describes how to create advanced surface features.
- *Working with Quilts*--Describes operations that one can perform on quilts.
- *Freeform Manipulation*--Describes how to dynamically manipulate a surface of a part or quilt.
- *Patterning Features*--Describes how to pattern features.
- *Copying Features*--Describes how to create and place groups of features, and how to copy features.
- *Modifying the Part*--Describes how to modify and redefine the part.
- *Regenerating the Part*--Describes how to regenerate the part and resolve regeneration problems.

Assembly

One can combine features not only into parts but also into assemblies. Assembly mode in Pro/ENGINEER enables to place component parts and subassemblies together to form assemblies, as well as to design parts based on how they should fit together. It can be then modify, analyze, or reorient the resulting assemblies.

3D model of FCV

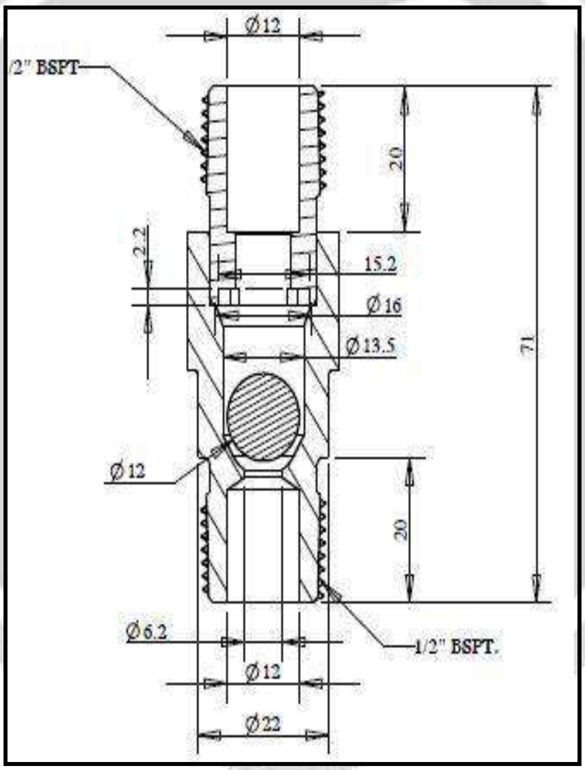

Fig.-3: Dimensions of Flow Control Valves

Flow Control Valves

FCV are taken and created in three parts:

1. FCV outer body

2. Ball check

3. Outlet insert

FCV Outer body:

It is the initial part of the body which has the inlet pipe connection to the valve. The ball check is rested in this body when there is no flow.

Fig.-4: Flow Control Valves (FCV) Part1

Ball check:

It is rested in FCV outer body and plays the key role in FCV by making pressure drop and controlling flow. The return flow is stopped by this valve.

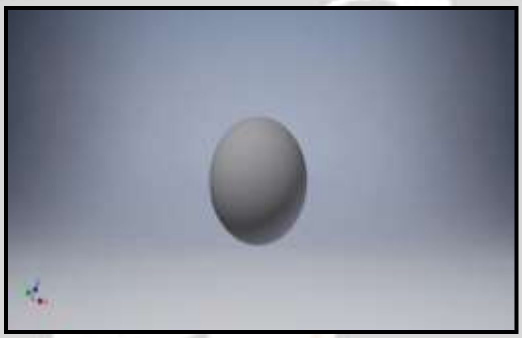

Fig.-5: Flow Control Valves (FCV) Part2

Outlet Insert:

This outlet insert contains the outlet connection of the valve to the pipe and fixed to the FCV outer body. It helps to stop the ball while the flow runs.

Fig.-6: Flow Control Valves (FCV) Part3

CAD Model Assembly

In assembly modelling, combining the parts and subassemblies to form an assembly that functions as a single unit. The assembly constraints make relation to one another for parts and subassemblies. It defines a set of features in an assembly that interact with multiple parts [3].

Inserting existing parts into an assembly or sketch of modelling commands to generate parts in the background of the assembly. It can also group designed parts that function together as one unit. Merging various subassemblies to shape an assembly makes design changes easier to accomplish.

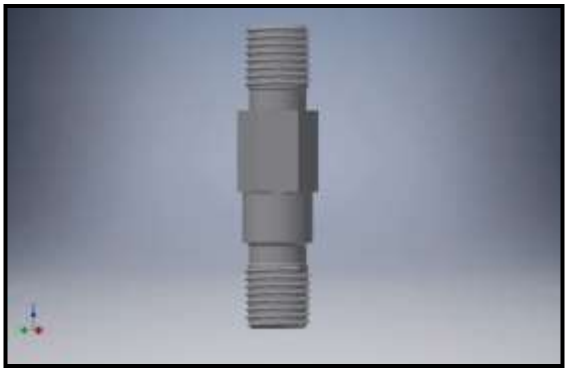

Fig.-7: Flow Control Valves (FCV) Assembly

CAD Model Re-assembly

The purpose of doing this re-assembly is to create a revised design which is imported for Simulation Software. The main reason to adjust the design is that in existing CAD model the ball check positioned in the FCV outer body. When the fluid enters in the valve the position of ball reach to the top section of FCV outer body and allows the fluid to pass through the nozzles and blocks the excess amount of flow

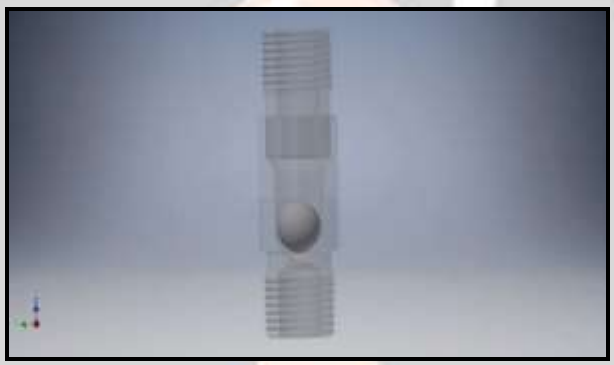

Fig.-8: Flow Control Valves (FCV) Re-assembly

The Flow Control Valves (FCV) Re-assembly model shows the position of the ball, and it also comes back to the same positon while the flow comes to the end and helps to stop the back flow from the valve. Considering the fluid flow, the position of the ball is assembled to the maximum point, where this happens while the fluid hits the ball. The outlet insert stops the ball by closing the main central orifice near the nozzles and allows to pass the fluid in the smaller area.

Fig.-9: Transparent Flow Control Valves (FCV) Re-assembly

The Transparent Flow Control Valves (FCV) Re-assembly model shows the final position of check ball reaches. This design helps to extract the fluid domain and then to import into FEA software.

Optimisation of FEA

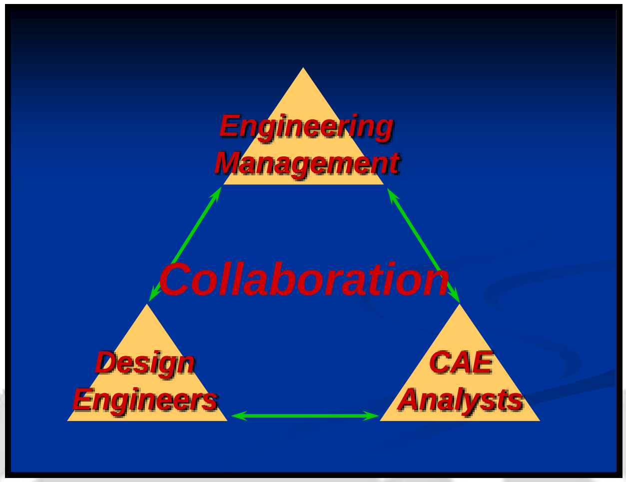

Fig.-10: Analysis collaboration

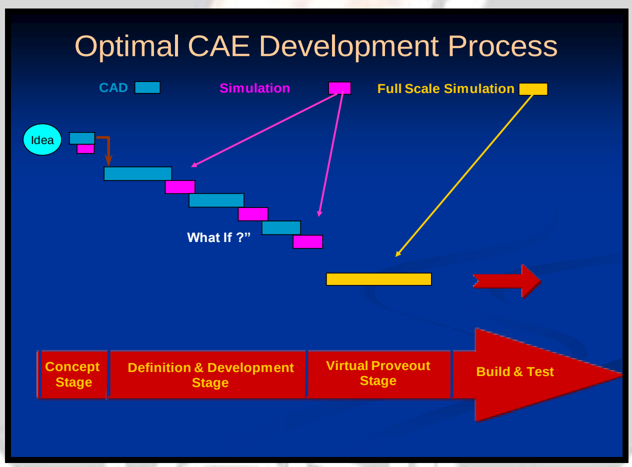

Fig.-11: CAE Development process

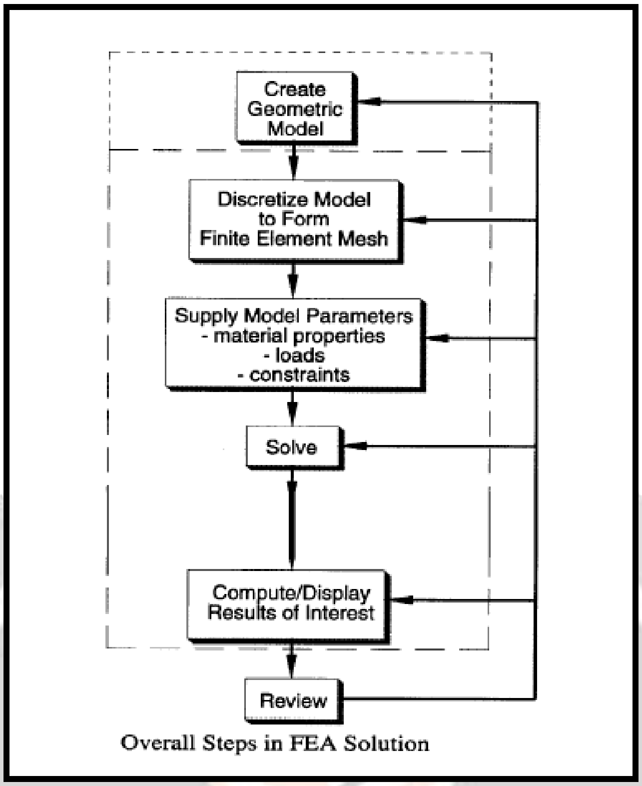

Fig.-12: Flow chart of analysis process

A General procedure for Finite Element Analysis:

Preprocessing

The preprocessing step is, quite generally, described as defining the model and includes

- Define the geometric domain of the problem.
- Define the element type(s) to be used.
- Define the material properties of the elements.
- Define the geometric properties of the elements (length, area, and the like).
- Define the element connectivity (mesh the model).
- Define the physical constraints (boundary conditions).
- Define the loadings.

The preprocessing (model definition) step is critical. In no case is there a better example of the computer-related axiom "garbage in, garbage out." A perfectly computed finite element solution is of absolutely no value if it corresponds to the wrong problem.

Solution

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s). The computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow. As it is not uncommon for a finite element model to be represented by tens of thousands of equations, special solution techniques are used to reduce data storage requirements and computation time. For static, linear problems, a wave front solver, based on Gauss elimination, is commonly used. While a complete discussion of the various algorithms is beyond the scope of this text, the interested reader will find a thorough discussion in the Bathe book.

Post processing

Analysis and evaluation of the solution results is referred to as post processing. Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution. Examples of operations that can be accomplished include:

- Sort element stresses in order of magnitude.

- Check equilibrium.
- Calculate factors of safety.
- Plot deformed structural shape.
- Animate dynamic model behavior.
- Produce color-coded temperature plots.

While solution data can be manipulated many ways in post processing, the most important objective is to apply sound engineering judgment in determining whether the solution results are physically reasonable.

ANSYS Design Modeler

ANSYS DM is an application from ANSYS Software which provides modelling functions exclusive for simulation that includes CAD geometry modification, clean-up and repair and several custom tools developed for fluid flow and structural analysis. It is a powerful geometry modelling for engineering applications in simulations. While the creation of design models is an essential part of product development process. It represents the first step of the simulation process. Besides existing a vital component of engineering simulation, design and manufacturing geometry links with the engineering simulation, and plays a serious role in the simulation of product development.

Extracting the fluid volume

The first step in any CFD analysis is to identify the extents of the fluid domain in which the governing equations are solved. This requires the isolating in a section of a larger physical system. Deciding where the computational domain begins and ends has an impact on the accuracy of the model.

Flow Domains can be divided into two basic classes:

1. Internal flow domain
- Flow through pumps, pipes etc.
- Flow is bounded in all sides by walls
2. External flow domain
    - Flow around the objects like aircrafts etc.

The fill operation is exclusively suitable for extracting fluid domains.

1. Creating a frozen body from the interior voids of CAD geometry
2. There are two way to define voids:
    - By Cavity- Requires picking the faces in the void.
    - By Caps- Requires the surfaces body 'capping' the openings.

Now in this thesis, it requires the Internal Flow Domain has the flow is passing through the valve. Creating the fluid volume by considering the method 'By caps.

The re-assembled design is imported in ANSYS DM. Extracting the fluid volume by closing all the holes in FCV, by creating the surfaces to the holes.

Fig.-13: Creating surface to the holes.

The holes are closed by creating surfaces and now making 'fill' operation to the design choosing the extraction type as 'By Caps'. Selecting the targeted surfaces and applying the whole body of FCV the fluid volume is generated.

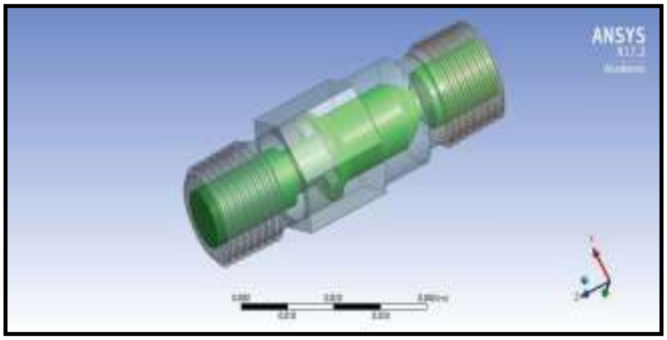

Fig.-14: Generating fluid domain

Suppressing the exterior parts in the excluding the ball check of the FCV body, the required combination of fluid domain path and ball solid path is obtained.

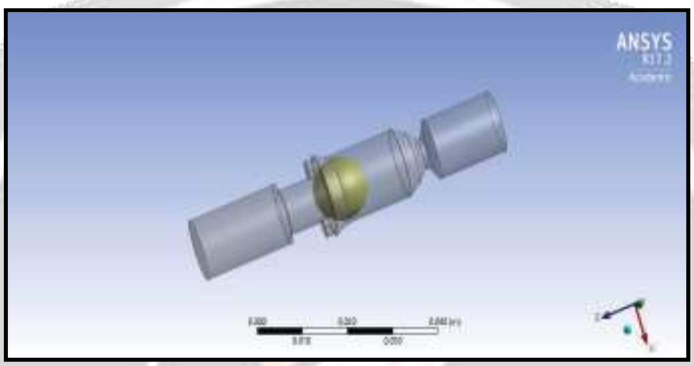

Fig.-15: Extracted fluid domain with ball check

Comparison of Results

The comparison of fluid flow rate between numerical model and theoretical model results are done in this thesis. The measurement scale of LPM (litre per minute) is used to measure the flow rate of flow control valve.
The comparison is done for the inlet pressure of 0.5 bar analysis in three cases.

Table - 2: Results for flow rate and flow velocity at 0.5 pressure bar

Flow parameters	Numerical model	Theoretical model	Experimental results
Flow rate in LPM	5.38 LPM ($8.9668\times 10^{-5} m^3/sec$)	5.22 LPM ($8.71\times10^{-5} m^3/sec$)	5.4 LPM ($9.001\times 10^{-5} m^3/sec$)
Flow velocity at notch	7.11 m/sec	6.9 m/sec	-

Conclusion

In this paper, the study mainly includes the comparison of the flow rate in numerical, theoretical and experimental models with initial pressure of 0.5 bar. The theoretical derivations, cavitation process and analysis are evaluated. The conclusions made from this work are as follows:

- The study investigated the optimization of the valve core structure by the pressure drop compensation coefficient revising method, and the effect of different key parameters on the flow rate control accuracy was discussed
- Through the CFD simulations the perfect required design can be investigated and helps to manufacture the correct model which saves a lot of time and money.
- Cavitation could possible to take place due to the empty space in the solid body and high pressure recovery than the vapor pressure of the valve.
- From the investigation performed in this thesis it can conclude that the flow control valve can obtain a pre-set flow of 5-6 LPM for the initial pressure of 0.5 bar.
- The study of flow pattern shows that the fluid flows around the sphere and has the higher velocity near nozzles.
- From now, it can confirm that use of Flow Control Valve is very suitable for household water distribution instead of using Ferrule valve.

References

[1]. Akbarian, E., Najafi, B., Jafari, M., Ardabili, S. F., Shamshirband, S., & Chau, K. W. (2018). Experimental and computational fluid dynamics-based numerical simulation of using natural gas in a dual-fueled diesel engine.*Engineering Applications of Computational Fluid Mechanics*, *12*(1), 517–534. https://doi.org/10.1080/19942060.2018.1472670

[2]. Aung, N. Z., Yang, Q., Chen, M., & Li, S. J. (2014). CFD analysis of flow forces and energy loss characteristics in a flapper-nozzle pilot valve with different null clearances. *Energy Conversion and Management*, *83*, 284–295. https://doi.org/10.1016/j.enconman.2014.03.076

[3]. Cui, B., Lin, Z., Zhu, Z., Wang, H. J., & Ma, G. F. (2017). Influence of opening and closing process of ball valve on external performance and internal flow characteristics. *Experimental Thermal & Fluid Science*, *80*, 193–202.
https://doi.org/10.1016/j.expthermflusci.2016.08.022

[4].Ghalandari, M., Bornassi, S., Shamshirband, S., Mosavi, A., & Chau, K. W. (2019). Investigation of submerged structures' flexibility on sloshing frequency using a boundary element method and finite element analysis. *Engineering Applications of Computational FluidMechanics*, *13*, 519–528.
https://doi.org/10.1080/19942060.2019.1619197

[5]. Hou, C. W., Qian, J. Y., Chen, F. Q., Jiang, W. K., & Jin, Z. J. (2017). Parametric analysis on throttling components of high multi-stage pressure reducing valve. *Applied Thermal Engineering*, *128*, 1238–1248.
https://doi.org/10.1016/j.app lthermaleng.2017.09.081.

[6]. Huovinen, M., Kolehmainen, J., Koponen, P., Nissilä, T., & Saarenrinne, P. (2015). Experimental and numerical study of a choke valve in a turbulent flow. *Flow Meas & Instrum*, *45*, 151–156.
http://doi.org/10.1016/j.flowmeasinst.2015.06.005

[7]. Iannetti, A., Matthew, T., Stickland, X., & William, M. D. (2016). A CFD and experimental study on cavitation in positive displacement pumps: Benefits and drawbacks
of the 'full' cavitation model. *Engineering Applications of Computational Fluid Mechanics*, *10*, 57–71.
https://doi.org/10.1080/19942060.2015.1110535

[8]. Li, B., Guo, S., Mao, X. Y., Wu, C., Zhang, D. M., Shang, J., & Liu, Y. S. (2017). Characteristic analysis and experiment of a dynamic flow balance valve. *Earth and Environmental Science*, *100*, 012107.
https://doi.org/10.1088/1755-1315/100/1/012107

[9]. Li, S. Y.,Wu, P., Cao, L. L., Wu,D. Z., & She, Y. T. (2017). CFD simulation of dynamic characteristics of a solenoid valve for exhaust gas turbocharger system. *Applied Thermal Engineering*, *110*, 213–222.
https://doi.org/10.1016/j.applthermaleng.2016.08.155

[10]. Liu, J. B., Xie, H. B., Hu, L., Yang, H. Y., & Fu, X. (2017). Realization of direct flow control with load pressure compensation on a load control valve applied in overrunning load hydraulic systems. *Flow Measurement and Instrumentation*, *53*, 261–268.
http://doi.org/10.1016/j.flowmeasinst.2016.07.004

[11]. Pietrzak, M., & Witczak, S. (2015). Experimental study of airoil-water flow in a balancing valve. *Journal of Petroleum Science & Engineering*, *133*, 12–17.
https://doi.org/10.1016/j.petrol.2015.05.019

[12]. Qian, J. Y.,Wei, L., Jin, Z. J.,Wang, J. K., Zhang, H., & Lu, A. L. (2014). CFD analysis on the dynamic flow characteristics of the pilot-control globe valve. *Energy Conversion and Management*, *87*, 220–226.
https://doi.org/10.1016/j.enconman.2014.07.018

[13]. Qian, J. Y.,Wu, J. Y., Gao, Z. X., Wu, A. J., & Jin, Z. J. (2019). Hydrogen decompression analysis by multi-stage Tesla valves for hydrogen fuel cell. *International Journal of Hydrogen Energy*, *44*(26), 13666–13674.
https://doi.org/10.1016/j.ijhydene.2019.03.235

[14]. Ramanath, H. S., & Chua, C. K. (2006). Application of rapid prototyping and computational fluid dynamics in the development of water flow regulating valves. *International Journal of Advanced Manufacturing Technology*, *30*(9–10), 828–835.
https://doi.org/10.1007/s00170-005-0119-5

[15]. Ryu, S. R., Rhee, K. N., Yeo, M. S., & Kim, K. W. (2008).Strategies for flow rate balancing in radiant floor heating systems.*Building Research & Information*, *36*(6), 625–637.
https://doi.org/10.1080/09613210802450697

[16]. Salih, S. Q., Aldlemy, M. S., Rasani, M. R., Ariffin, A. K., Ya, T. M. Y. S. T., Al-Ansari, N., Yaseen, Z. M., & Chau, K. W. (2019). Thin and sharp edges bodies-fluid interaction simulation using cut-cell immersed boundary method. *Engineering Applications of Computational Fluid Mechanics*, *13*, 860–877.
https://doi.org/10.1080/19942060.2019.1652209

[17]. Shen, X. R., Lu, B., Li, J. L., & Li, Z. Z. (2006). The numerical analysis and experimental investigation of automatic flux compensation valve. *Chinese Pneumatics & Seals*, *1*, 56–58.

[18]. Taylor, S. T., & Stein, J. (2002). Balancing variable flow hydronic systems. *Ashrae Meeting*, *4*, 17–25.

[19]. Wu,D., Li, S.,&Wu, P. (2015). CFD simulation of flow-pressure characteristics of a pressure control valve for automotive fuel supply system. *Energy Conversion & Management*, *101*, 658–665.
https://doi.org/10.1016/j.enconman.2015.06.025

[20]. Xu, X. G., Liu, T.Y., Li, C., Zhu, L.,&Li, S. X. (2019). A numerical analysis of pressure pulsation characteristics induced by unsteady blood flow in a bileaflet mechanical heart valve. *Processes*, *7*(4), 232.
https://doi.org/10.3390/pr7040232

[21]. Zhou, L., Bai, L., Li, W., Shi, W. D., & C, W. (2018). PIV validation of different turbulence models used for numerical simulation of a centrifugal pump diffuser. *Engineering Computations*, *35*(1), 2–17.
https://doi.org/10.1108/EC-07-2016-0251