Determination of the Valve Sizing Coefficient of Globe Valve using CFD

K Yogesh, Kaivalya Bakshi

Abstract - *In this paper the flow past a 2" sized globe valve is numerically simulated and the valve sizing coefficient (C_v) is calculated using CFD. A theoretical study using elementary techniques is exercised to understand the significance of the flow coefficient (C_v) in predicting the mass flow rate through the valve. This exercise gives way to a clear understanding of the inherent simplicity of the function of a valve. A commercial grid generation tool ICEM-FLUENT is used for mesh generation. Commercial fluid dynamics code, ANSYS FLUENT is used for the flow modeling. For the static analysis, fully opened valve position is used and 1 atm through 7 atm of pressure difference (ΔP) is applied across the valve numerically. The numerical data obtained from the simulations is then used to calculate the flow coefficient (C_v) computationally. From the results obtained it is found that the flow coefficient was indeed a constant for the same set of pressure differential and mass flow rate. The flow coefficient value is thus proved to be a function of the valve structure. By comparing between numerical and experimental results it is concluded that flow coefficient (Cv) can be predicted with agreeable accuracy using computational techniques.*

Keyword – CFD, globe valve, valve sizing coefficient, ANSYS FLUENT, ANSYS ICEM-FLUENT

I. INTRODUCTION

Computational Fluid Dynamics (CFD) emerged over the past few decades as a bankable design tool with the advent and establishment of high speed and storage capacity computers. Over the past few decades it has consolidated its widespread use as an investigative tool too. Many commercial CFD tools now have the requisite proficiency to aid and deal effectively with a variety of industrial, manufacturing and research problems. Dedicated commercial CFD software or codes that are as much competent are readily available, have endowed the contemporary engineer to reiterate his analysis or design and supplement necessary documentation with quantitative data. The same data can also be used to affirm and conceive interpretations. CFD is an equal partner with pure theory and experiment in the analysis and data generation required for industrial devices, systems and products. It synergistically works with the other two approaches of pure theoretical analysis and experiment that are employed to solve real life industrial fluid dynamics problems. This third approach is now an integral part of analysis executed for any purpose.

Design procedures for fluid interacting mechanical components, products and systems and instrumentations involve numerous iterations to achieve optimum results. Any particular iteration requires analysis using all or part of the three afore mentioned approaches considering standard practices, requirements and suitability. Each iteration is followed by purposeful improvements by altering the existing design. Expediting and decreasing the cost per iteration is essential in today's fast paced and consumer driven market. CFD empowers the design engineer to understand fluid material interactions and form set notions regarding improvements in design by providing fluid visualizing techniques. It also conveniently provides data that confirm the validity of, or, suggest improvements in experimental setups and results thus accelerating iteration periods cost effectively.

Valves control the fluid flow and pressure in a system or process. The selection of their types, design and material plays a vital role in the performance and reliability of any system. Now, with the emergence of robust computational fluid dynamics (CFD) tools and powerful computers, the analysis of valve performance, and thus the job of designing valves to suit a particular application can be done much faster. Apart from this, CFD analysis can reveal the complex flow structure inside the valve, which the experiments hardly provide. Hence as discussed above CFD approach betters the iteration process when conducted in lieu with experimentation. Even otherwise, experimentation needs to be supplemented with CFD analysis as a validation technique. Indeed the complexity of the valve geometry or shape warrants the use of sophisticated commercial CFD softwares for analysis.

II. DEVICE DESCRIPTION

There exist different kinds of valves for different purposes. Of these, globe valves are used in systems where good throttling characteristics and low seat leakage are desired with relatively high head loss allowable in an open valve. Globe valve is commonly used as the control valve for high pressure system due to lesser leakage and higher erosion resistance compared to other valves. Globe Valves are named for their spherical body shape with the two halves of the body being separated by an internal baffle. Figure 1 shows a general type globe valve with all its relevant components.

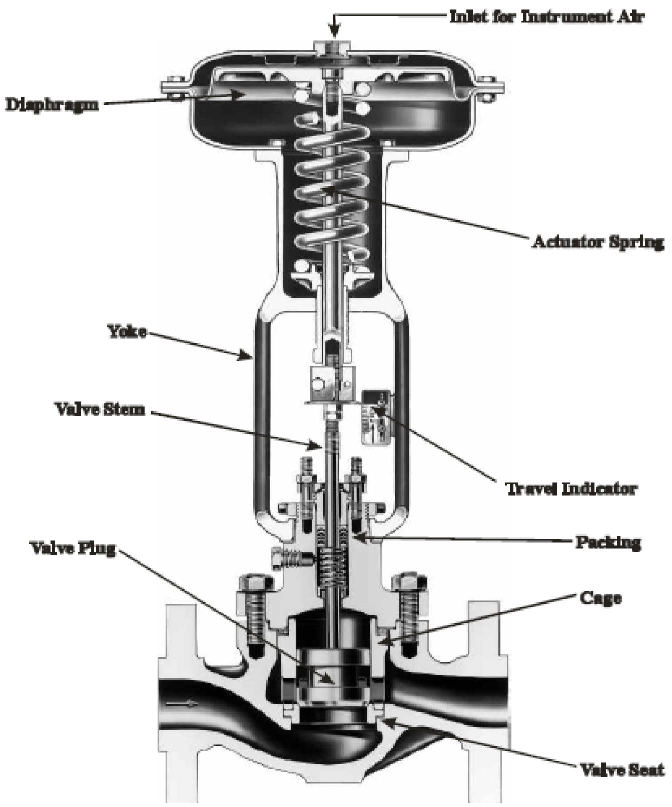


Figure 1: Schematic diagram of globe valve

Moreover, globe valves are usually operated in fully opened condition owing to their large head losses even in that condition. Thus the analysis of fluid flow through globe valves in fully opened condition garners special attention. Consequently many studies concerning this phenomenon of fluid flow through fully opened globe valve have been conducted.

In this paper, the turbulent modeling of a 2" size globe valve, for a range of pressure difference spanning from 0.5 bar to 7 bar has been performed in order to understand its performance and to estimate its Sizing Coefficient (C_v). The conventional definition for C_v has been used. Its use is vindicated by the simple mathematical manipulations which affirm its invariance.

III. Physical Description of the Flow Through the Valve

The fluid in a typical piping system passes through various fittings, valves being one of them. These fittings such as tees, elbows, inlets, enlargements, contractions alter the smooth fluid flow and cause additional energy losses besides the typical contact viscous friction energy losses. In a typical piping system they are minor compared to the pipe friction losses and are hence called *minor losses* as opposed to *major losses* caused by the viscous friction at the contact surface of the pipe. Usually the major losses exceed the minor ones, but it may not always be the case.

The minor losses are expressed in terms of *the head loss coefficient*, or *resistance coefficient* (k):

$$k = \frac{h_L}{\frac{v^2}{2g}}$$

Where, h_L is the *additional irreversible head loss* in the piping system caused by insertion of the component and is defined as:

$$h_L = \frac{\Delta P}{\rho g}$$

Where, ΔP is the pressure difference across the valve.

The irreversible head loss is caused by the skin viscous friction and the induced swirling turbulent eddies that are formed in the valve and continue downstream. These losses are irreversible since the energy dissipated by them is converted to heat.

The understanding of the *vena contracta* is instrumental in revealing how control orifices function. Whenever a flow is contracted that is forced to flow through a constricted inlet, with sharp corners, the flow separates around the corners and gives rise to eddies. This eddy formation results in viscous dissipation caused by intense mixing and part of kinetic energy is converted to heat energy as evidenced by the slight rise in fluid temperature. The vena contracta is that cross section beyond the contraction where the fluid velocities are parallel to each other and normal to the constriction cross section.

Thus by varying the *resistance coefficient* (k) the head loss can be varied and by means of which the *vena contracta* can be altered. This gives us a control of the fluid velocity at the outlet or the *vena contacta*.

Valves are commonly used in piping systems to control the flow rates by simply altering the head loss until the desired flow rate is achieved. For valves it is desirable to minimize the head loss in fully open condition. In order that we might be able to better predict the flow rate obtained the flow coefficient (C_v) has been defined.

IV. Valve Sizing Coefficient

Masoneilan suggests the use of 'valve sizing coefficient' as a geometrical constant or characteristic of a globe valve. Following derivation for the expression regarding the coefficient has been adopted from Zappe and is concluded with a formula for it as a function of the valve properties.

Derivation for the formula of the valve sizing coefficient:

By changing the plug position, the minor losses (h_L) are changed. Bernoulli's Equation in modified form:

$$\frac{1}{2}\frac{v_1^2}{g} + \frac{P_1}{\rho g} = \frac{1}{2}\frac{v_2^2}{g} + \frac{P_2}{\rho g} + h_L$$

….assuming one dimensional flow

$$h_L = \frac{\Delta p}{\rho g}$$

….since $v_{1=}$ v_2

Notation:

v_1 =velocity at valve inlet,
v_2 =velocity at valve outlet
P_1 =pressure at valve inlet,
P_2 =pressure at valve

Now to relate ΔP with flow rate (Q) we employ C_v .
Expression for C_v is obtained from equations 1,2 :

1. modified Bernoulli's equation

$$h_L = \frac{\Delta p}{\rho g}$$

Assuming one dimensional flow

2. Borda-Carnot Equation

$$h_L = k\frac{v_c^2 - v_2^2}{2g}$$

Where, v_c =velocity at vena contracta
k=const. & k<1

(I)Equating right hand sides of equations 1& 2,

$$\frac{\Delta P}{\rho g} = k\frac{{v_c}^2 - {v_1}^2}{2g}$$

Since v_1 =v_2

Let A_c =area of vena contracta
A_1 =area of c/s at valve inlet

(II) Continuity Equation for single dimensional flow,

$$v_c A_c = v_1 A_1$$

Since A_c <$A_{1,}$ we can safely assume that

$${v_1}^2 < {v_c}^2$$

(III) $$\frac{\Delta P}{\rho g} = \frac{k{v_c}^2}{2g}$$ …..From (II)

(IV) $$\frac{\Delta P}{\rho g} = \frac{k{v_1}^2{A_1}^2}{2g{A_c}^2}$$ …..From (Continuity Eqn)

(V) $$Q = v_1 A_1$$ …..Equation for Discharge

(VI) $$\frac{\Delta P}{\rho g} = \frac{kQ^2}{2g{A_c}^2}$$ …..From (IV) and (V)

(VII) $$Q = \left[A_c\sqrt{\frac{2}{k}}\right]\sqrt{\frac{\Delta P}{\rho}}$$ …..From (VI)

Since A_c and k are constants

So we get,

$$Q = C_v * \sqrt{\frac{\Delta P}{\rho}}$$

Where, C_v =constant

A standardized definition of C_v as advocated by the ISA and IEC in US customary units:

Flow Coefficient (C_v):

The flow of water through a valve at 60 ºF in US gallon/minute at a pressure drop of 1 lb/in² (U.S.) as defined by the equation-

$$C_v = \frac{Q}{\sqrt{\frac{\Delta P \times 62.4}{D}}}$$

Where, D=density of working fluid in pounds/cubic feet.

To reiterate the physical understanding that *Cv is a function of Valve geometry we may use the following formula:*

$$C_v = \sqrt{\frac{2}{k}} \times A_c$$

Where, k=constant from Borda Carnot equation
A_c=Area of vena contract=Area of c/s at valve seat

Valve sizing and the coefficient C_v:

The valve size for a globe valve refers to the outer diameter of the port at the inlet or outlet. Valve sizing thus refers to the selection of an appropriate port outer diameter. The selection of a control valve is based on the required flowing quantity of the process, in other words the volume flow rate (Q). The selection of a correct valve size, as determined by formula, is always premised on the assumption of full knowledge of the actual flowing conditions, Q and the operating pressure differential (ΔP).

To relate these various operating conditions viz. Q and ΔP, the flow sizing coefficient (C_v) is employed. By definition, *the valve flow coefficient, Cv, is the number of U. S. gallons per minute of water that will pass through a given flow restriction with a pressure drop of one psi.* Basically, it is a capacity index upon which the engineer can rapidly and accurately estimate the required size of a restriction in any fluid system. Hence the *flow coefficient* (C_v) is called the *valve sizing coefficient.*

It is defined by the formula:

$$Q = C_v * \sqrt{\frac{\Delta P}{\rho}}$$

Where ρ=density of the working fluid.
It can be observed from the formula that C_v is independent of the flow conditions and is therefore a function of solely the valve structure.

V. LIMITATIONS OF ANALYTICAL AND EXPERIMENTAL METHODS IN CALCULATING C_v

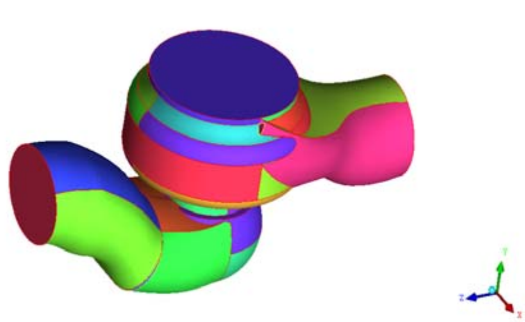
Analytical methods to determine C_v suffer a major setback due to the inherently turbulent nature of the flow. There simply does not exist, a complete solution for such complex flows that could determine the values of the necessary variables from the provided boundary conditions. In other words the flow is too complex so that theoretical analysis is generally not possible.

Hence we turn to experimentation, which very successfully provides the data necessary to calculate C_v for various valves. However it is time consuming and relatively expensive process. Errors that are instrumental, human and general in nature play a role in the measure of accuracy while measuring flow parameters during experimentation. The presence of fittings causes a further loss of pressure other than that due to the valve. This affects the C_v calculation and must be considered by the use of other factors such as F_p, to make the process less prone to error.

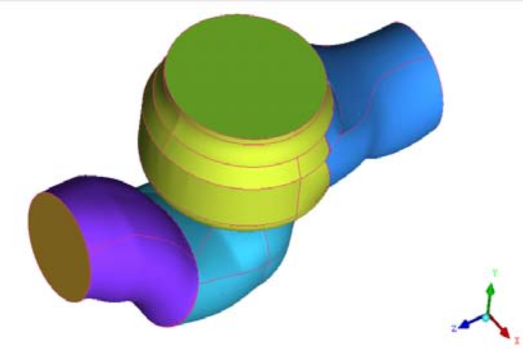
VI. CFD ANALYSIS

Geometry Repair and Meshing

The meshing of the geometry was carried out using ANSYS ICEM-CFD*v10* codes. ICEM-CFD is a highly versatile and vivid tool. It is capable of handling both structured and unstructured grids with ease.
The geometry of a globe valve is very complex. The Valve has many features that are of no importance in a numerical simulation. Such surfaces were removed and were replaced with much simpler ones. Such a modification helps to reduce the computation time and resources hugely, and these modifications will not affect the final simulation results much.

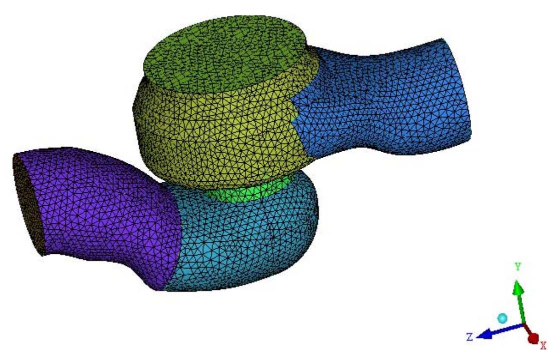


Imported Geometry



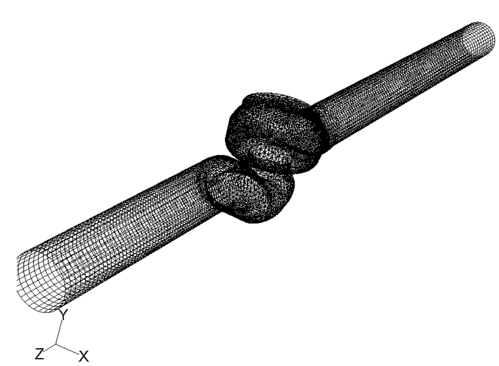
Repaired Geometry

Even after performing the cleanup, the repaired geometry is too complex to mesh. Using a structured grid to mesh this would have been a tedious affair, though the results obtained using them would have been more accurate in general cases. Considering the fact that the problem involves a fairly simple flow, it was decided that an unstructured mesh would suffice. Hence, the fluid domain was divided into tetragonal elements using the OCTREE approach.



Final Valve mesh

To capture the flow development before and after the globe valve fluid domain, pipes of length 5D and 15D were added ahead and behind the valve respectively. The pipe geometries, being simple were meshed structurally using hexagonal elements. The three fluid domains thus obtained were merged together to obtain a hybrid mesh. Again, considering the simplicity of the flow, the hybrid mesh interface was made non-conformal.



The whole fluid domain

Turbulence Model
ANSYS Fluent V6.2.16 was used as the solver. The mesh files (.MSH) were imported in Fluent. Simulation was done at various pressure differences between Inlet and Outlet. The fluid material is water, hence the flow is incompressible. Segregated type of solver was chosen as the fluid flow is incompressible. The flow is steady state case. The fluid flow is inherently turbulent, hence Standard k-ε turbulence model was chosen. The flow is single species, single phase and no heat transfer phenomenon is involved, hence necessary solver model was chosen and applied.
Boundary Conditions:

- The interface surface between the pipes and the valve mesh was assigned “interface” boundary condition.
- All other surfaces except the inlet and outlet were assigned “no-slip” boundary condition.
- The inlet and outlet were given “pressure inlet” and “pressure outlet” boundary condition.
- The values of the inlet outlet pressures were varied in successive simulations.

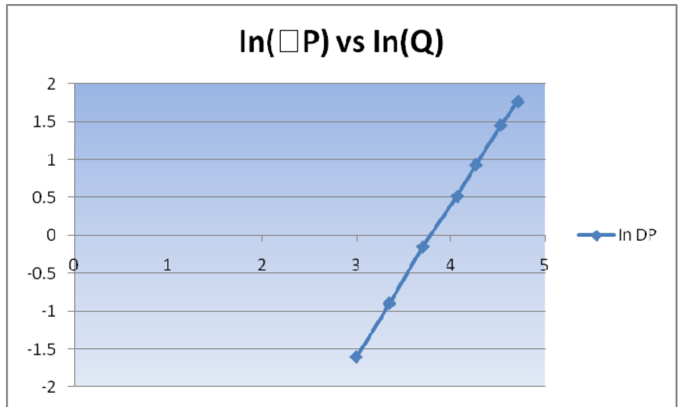
VII. RESULTS

The valve sizing coefficient C_v was calculated using the relationship prescribed by the International Standards Association (ISA) and the International Electro technical Commission (IEC). The results have been tabulated,

S.N	ΔP Sys	Mass Flow Rate Q	P at Valve Inlet	P at Valve Outlet	ΔP Valve	Cv
	atm	*m^3/hr*	*atm*	*atm*	*bar*	*Usgpm*
1	0.25	19.9443	1.2101	1.0123	0.2004	51.4597
2	0.5	28.3385	1.4214	1.0918	0.4069	51.312
3	1	40.5009	1.8522	1.0034	0.8601	50.4418
4	2	58.5154	2.672	1.0161	1.6778	52.1802
5	3	71.2282	3.5355	1.0238	2.5404	51.5714
6	5	92.5478	5.2281	1.0015	4.2826	51.6553
7	7	111.8016	6.5877	1.074	5.8603	53.34

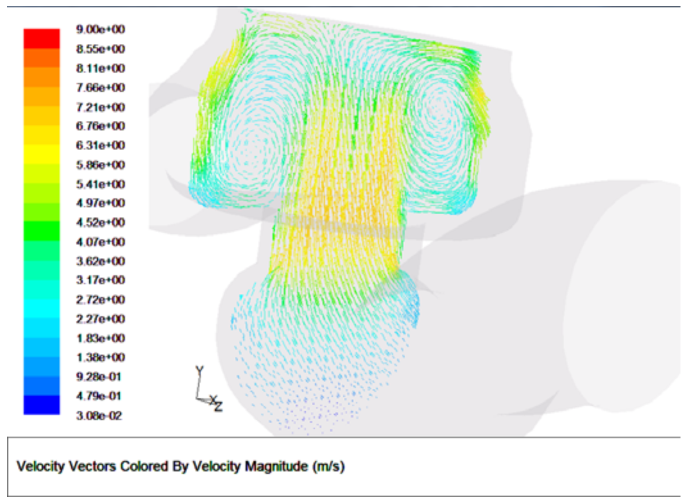
It is clear that since the value of C_v is largely invariant for the values of the pressure differential for which simulations have been conducted, the simulation results are not deviant from experimental evidence.
In order to enhance visualization of the relationship between the operating pressure differential (ΔP in bar) and volume flow rate (Q in m^3/hr) an appropriate graph depicting this relationship has been provided below. The corresponding x and y values have also been tabulated here.

ln Q	2.99	3.34	3.70	4.07	4.27	4.53	4.72
ln ΔP	-1.61	-0.89	-0.15	0.52	0.93	1.45	1.77

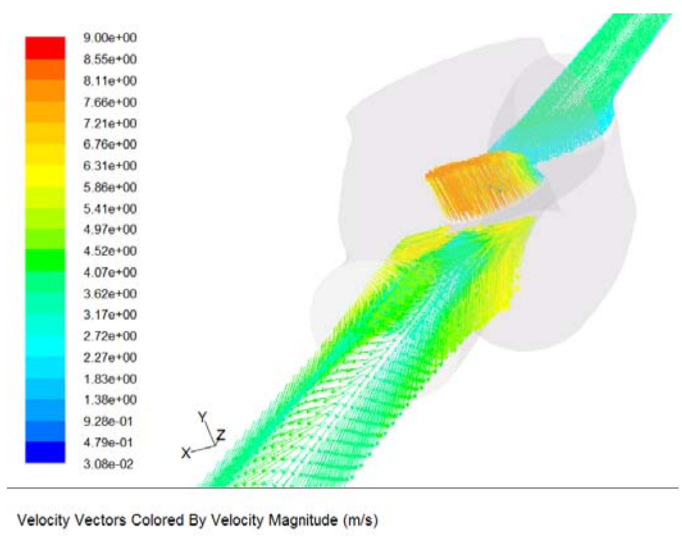


VIII. OBSERVATIONS

The flow characteristics can be studied with the help of two iso-surfaces “z=0” and “y=0”. *Vector plots* on these surfaces show the velocity direction at each point.

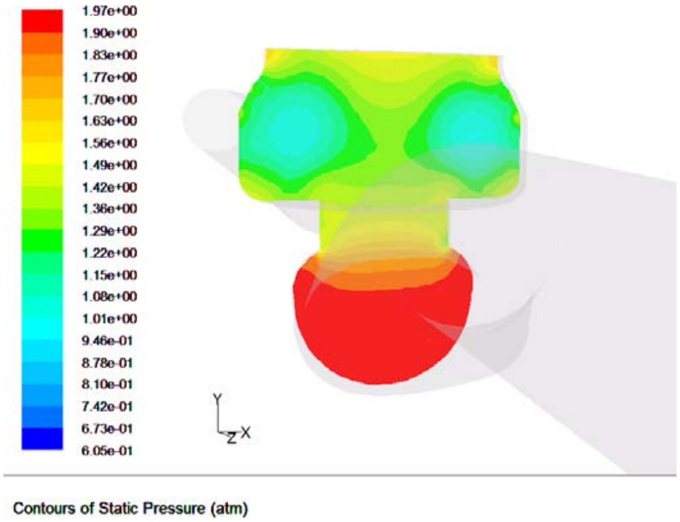


Velocity vector plot at ΔP=0.25 atm on iso-surface “z=0”

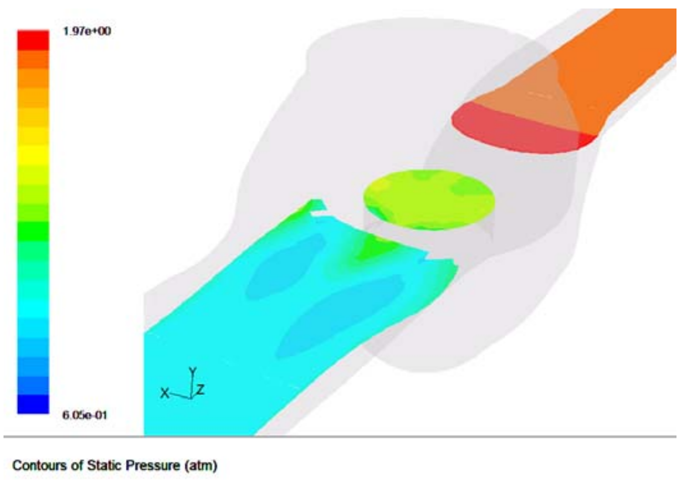


Velocity vector plot at ΔP=0.25 atm on iso-surface “y=0”

Contour plots showing the pressure distribution in the fluid region is also plotted at surfaces “z=0” and “y=0”

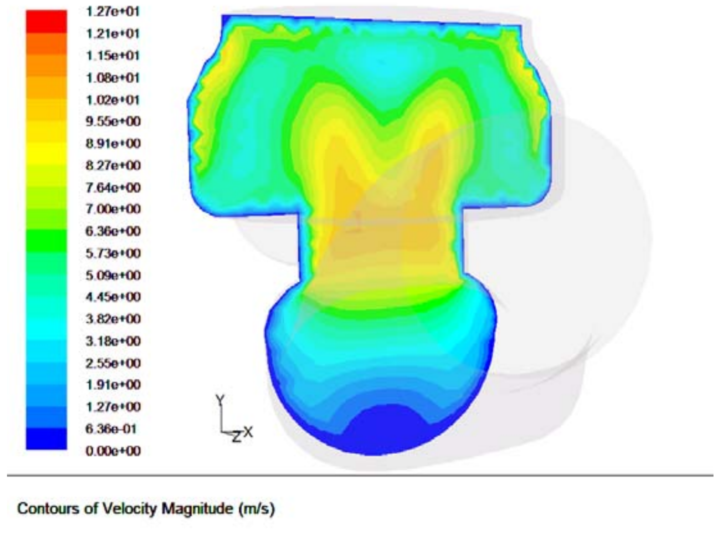


Static pressure contour at ΔP=1 atm on iso-surface “z=0”

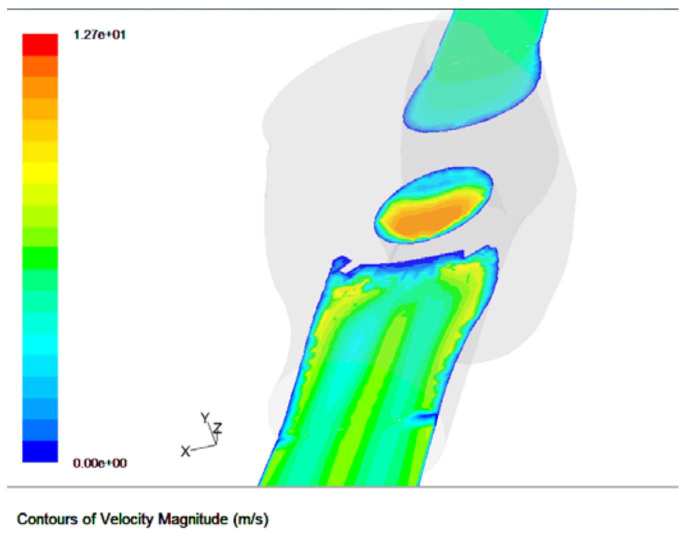


Static pressure contour at ΔP=1 atm on iso-surface “y=0”

Similarly, the contour plots of velocity at the iso-surfaces is as follows,



Velocity magnitude contour at ΔP=1 atm on iso-surface “z=0”



Velocity magnitude contour at ΔP=1 atm on iso-surface “y=0”

IX. CONCLUSIONS

The valve sizing coefficient (C_v) for the Globe Valve (2 in) geometry was calculated in fully opened condition. The numerical simulation has revealed a few important conclusions. To begin with, the values of C_v remain invariant of the change in operating pressure differential. We can thus infer that C_v is a function of only the geometry of the valve at low and medium pressure differentials.

Secondly, the values of C_v obtained through Numerical Simulation closely approximate the experimental data for such 2 in valves. CFD simulations may therefore be further used to find out to predict with good accuracy the flow variables as also the value of Cv for other similar valve geometries with variable openings. CFD will be a viable tool in such cases to reduce costs, resources and precious man hours spent in experimentation.

X. REFERENCES

[1] Zappe, R. W., and Smith, P., "Valve Selection Handbook", Gulf Professional Publishing.
[2] Masoneilan, Masoneilan Control Valve Handbook, Masoneilan Valve Division
[3] British Control Valve Handbook
[4] Cengel, Y. A., and Cimbala, J. M., "Fluid Mechanics-Fundamentals and Applications"
[5] "Control Valve Handbook", Emerson Process Management.
[6] FLUENT, 2005, ''User's Guide,'' Fluent Incorporated, Centerra Resource Park, 10 Cavendish Court, Lebanon, NH 03766.
[7] Anderson, J. D., "Introduction to Computational Fluid Dynamics".
[8] http://www.cfd-online.com/Wiki/
[9] http://www.answers.com