Design validation with CFD simulation and geometrical optimization

Research paper

www.bellinosrl.com

Author: S. Debernardis

Bellino s.r.l.
SP 1 Bari-Modugno km1,5
70026 Modugno - BA

Abstract

Control valves have the task to regulate fluxes through pipes in any fluid plants. This paper addresses the efforts in term of research to design more and more efficient control valves manufactured by Bellino srl. Some study cases explain the evaluation of their flow rates and show some comparisons with CFD simulations.

Introduction

Bellino srl designs and manufactures different types of control valves according to different customer requirements. The design of control valves starts with the need to optimize the knowledge acquired with the manufacturing of control valves over a long time, mainly based on two types of bodies, and different types of trims. A complete design of a control valve takes in consideration many requirements, which widely range from process specifications and customer needs, to applicable international standards.

Among all these design variables, and different available working conditions, this paper addresses only the issues about the flow rate through the valve in the fully-open position, which mainly characterizes the valve. The long collected experience is in the form of good choosing a complete series of dimensional, geometrical and performance parameters, commonly known to all designers of control valves. Only the expertise of a designer is able to choose the best configuration, not only to respect the control of the flow rate, but also all other specifications already reported before. The flow rate of a control valve is defined like the 'CV' value. More precisely, the valve 'Cv' defines how many gallons per minute of water flow through the valve when the difference of pressure between the inlet and outlet (as the standard [1] defines) is 1 psi, with the plug in fully open position [1].

A validation of manufactured valves, using computational fluid dynamic (CFD) simulations, in order to validate what is not .

possible to test on the hydraulic bench, is the main purpose of this paper After the CFD simulations, a comprehensive comparison among numerical results follows in the reading, with some highlights on some critical values, which are considered important, or if some discrepancies rise against the classical design. Finally, the paper describes some geometrical optimizations already used for the valve 300 Series (S300), and some evidences of the optimized geometry are shown in the end.

State of Art

The design of a control valve has to follow different requirements: physical processes to deal with, required reliability, recommendations by authorities or regulating agencies, costs and uniformities of some specifications used in plant [2]. Considering all that, the first parameters to take into account to address the selection of basic control valves, in sizing and types, are based on:

- Characteristics of the fluid and flow rate
- Inlet and outlet pressure, and temperature
- Valve service
- Allowable leakage
- Allowable noise level
- Piping specification

The sizing starts with the choice of some parameters like:

- Pressure recovery factor
- Incipient cavitation index
- Critical pressure factor
- Rangeability
- Shut-off value
- Calculated noise produced
- Type of trim and body
- Materials suitable for the fluid and the service
- Connection type

Simplifying the dimensioning procedure, we can state, without errors, that:

$$Q = k_v * \sqrt{\Delta p} \tag{1}$$

Where:

- Q $[\frac{m^3}{h}]$ is the flow rate
- Δp [bar] is delta pressure between inlet and outlet
- k_v $[\frac{m^3}{h} \cdot \frac{1}{\sqrt{bar}}]$ flow rate coefficient (K_v=0.865*Cv $[\frac{gal}{min}]$)

Now another parameter is useful to put in relation the flow rate and the area of the opened section of trim, it is usually called resistance:

$$A = r \cdot Cv \quad (2)$$

Where:

- A [mm^2] is the area of opened section in the trim
- r [$\frac{mm^2}{\frac{gal}{min}}$] resistance parameter

The experience of all previous designs gives Bellino the value to assign to the resistance parameter *r,* and consequently the flow rate coefficient *Cv*, according to all setting conditions discussed above. All the collection of these parameters is the expertise of Bellino in designing control valves.

The goal of this research is the validation of models, condensed in the values of *Cv* and *r* assigned to different types of valves, working in different conditions, through computational fluid dynamic analyses.

Methods and solutions

In this paper, the comparison of two valves is the main deal of the presented work. They are the valve 10'' ANSI 150 200 Series (S200) with GVCH trim and the valve 10'' ANSI 150 S300 with GVCH trim. Even if results are not specific for all control valves, it addresses the difference between two bodies used in almost all Bellino production, which are S200 and S300 shape.

Each valve is characterized by a Cv value that, as stated above, is the number of US gallons per minute of water at the temperature of 60°F, that flows through that valve when there is 1-psi-pressure drop in steady conditions [1].

The simulation and the evaluation of the flow rate through this specific valve using a computational fluid dynamic tool, setting a difference of 1 psi pressure between inlet and outlet sections, means exactly measures the Cv of that valve. To get an effective estimation of the Cv, the simulation should reproduce a hydraulic bench test according to the standards [3].

Standard references

The international standard CEI IEC 60534-2-3 [3] describes the practical test procedure to evaluate the flow capacity of a control valve. Apart from the definition of all the involved parameters, and apart from all the devices required to set a basic flow test system, the standard fixes the following important physical characteristics of the pipeline of the testbed:

- the test specimen (valve, in our case) should be installed between two pressure gauges
- the distance between the inlet section of the valve and the inlet pressure gauge should be twice the nominal pipe diameter of the nominal pipe connected to the valve
- the distance between the outlet section of the valve and the outlet pressure gauge should be 6 times the nominal pipe diameter of the nominal pipe connected to the valve

About the fluid to use, water within a temperature range of 5°C to 40°C shall be the basic fluid used in the test procedure for incompressible flow, which is what we are interested in.

At first, the procedure explains how to evaluate the maximum flow rate for the specific valve (chocked flow). The control system fixes the values for the inlet pressure and the flow rate. In a steady-state condition, a pressure differential is recorded. A second measurement is performed with conditions that have the 90% of the initial pressure differential. In this second condition, if the flow rate has a change below a range of 2% of the previous setting, the flow rate is considered like the maximum flow rate. If not, a higher inlet pressure should be considered for further replications. In order to prevent vaporization of the liquid, and so problems of cavitation during the measurement, some limitations are dictated on the inlet pressure set. The standard suggests to calculate the minimum inlet absolute test pressure, taking also into account the liquid pressure recovery factor of that valve, with the following equation:

$$p_{in_{min}} = \frac{2 \cdot \Delta p}{{F_L}^2} \tag{3}$$

where $p_{in_{min}}$ should have a minimum value of 0.1 bar. In our specific case, $\Delta p = 1psi$ and $F_L \approx 0.9$, so it returns $p_{in_{min}} = 17000\ Pa$. According to this specification of the standard, it is usually used this pressure $p_{in} = 1psi + 1bar \gg 17000\ Pa$.

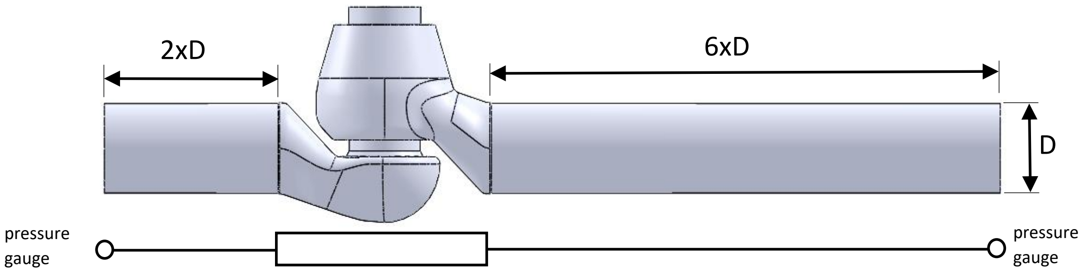
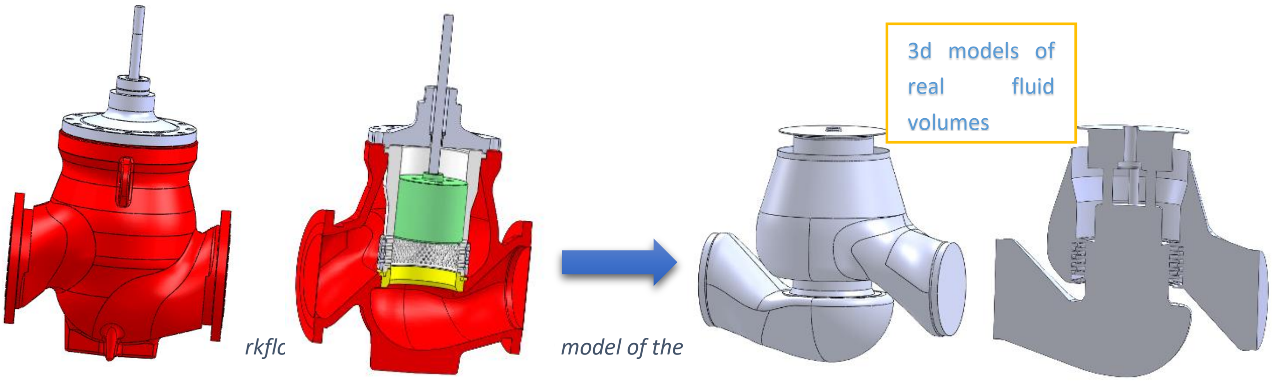


Figure 1 Settings of the dimensions of CAE model according to standards

The Figure 1 explains the configuration prescribed by the standards and its application on 3D modelling and CAE simulation. In this figure it is possible to see the fluid volume inside the valve (upper part), and the schematic representation of the test (lower part). Thus, the inlet section and outlet section have a respective distance of 2xD and 6xD, where D is the inside diameter of inlet and outlet section, equal to the inside diameter of two pipes connected to the valve.

CFD simulations

The computational fluid dynamics (CFD) simulations require a 3D model to be discretized in a 3D grid, composed by 3D cells, on which to apply a series of numerical equations, iterated up to the reach of the numerical convergence. To process the analysis, the starting point is the exact geometry of the valve as built, and then, from it, the derived real fluid volume. Bellino engineering uses SolidWorks as CAD software to design all parts to manufacture.



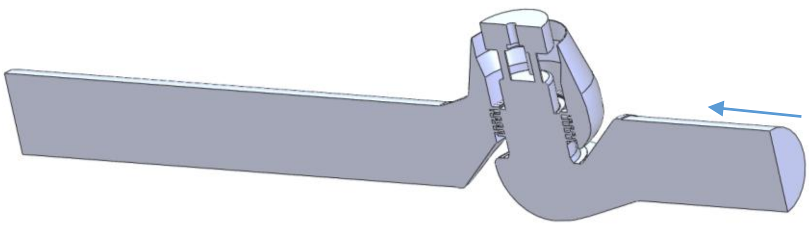


Figure 3 3D model of the fluid volume used for the simulation

Then, the 3D modelling workflow to obtain the fluid volumes, has to extend respectively the inlet and outlet sections, in order to respect the above-described standards.

This study considers two valves:

- valve 10’’ ANSI 150 S200 with GVCH trim
- valve 10’’ ANSI 150 S300 with GVCH trim

manufarctured by Bellino. These valves have the same trim type, the same nominal size but two different body shapes, named S200 and S300. In term of geometrical shape, it means that the flow has two different lenght and shape of ducts, between inlet section and trim, and between trim and outlet section.

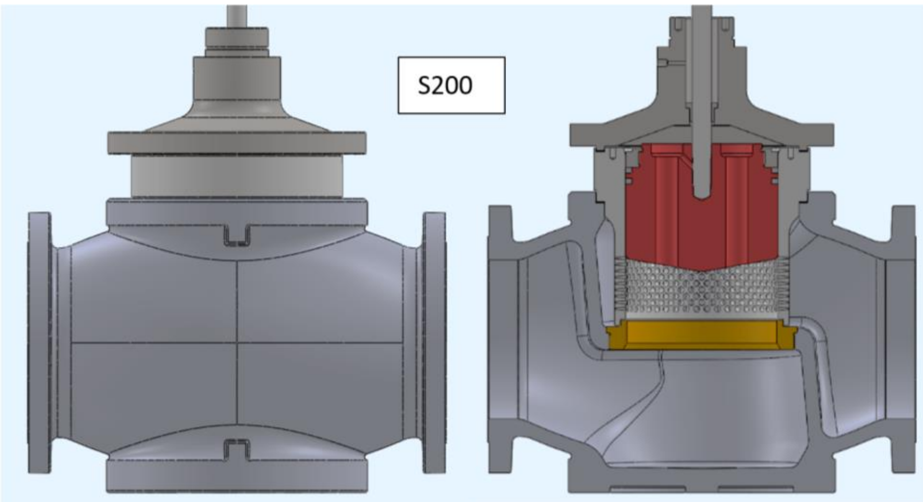


Figure 4 Description of the S200 valve body

The difference between the bodies is well described by the Figure 4 and Figure 5

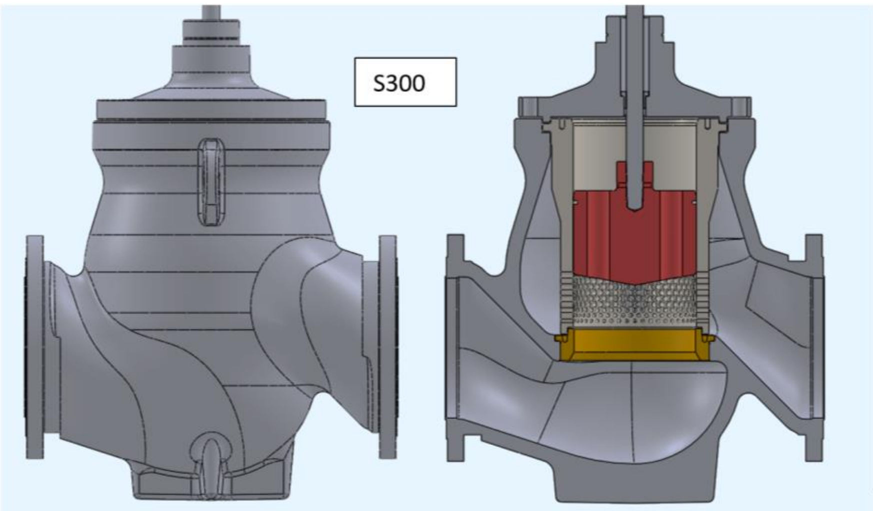


Figure 5 Description of the S300 valve body

Different Esi-Group software was used to perform all the analyses here reported: VisCart was used to create the unstructured mesh, and ACE+ to perform the CFD simulations, using the packages Flow and Turbulence. Both the valves have a plane of symmetry that corresponds to the cutting planes that show the interior of the valves in Figure 4 and Figure 5 Thanks to this possibility, it was enough to consider a half of the complete 3D fluid volume model for the analysis, so it was processed a fluid volume like that shown in Figure 3 . Therefore, after the meshing of these models, the computational fluid dynamics simulations run. Because of narrow holes in the region around the trim, a finer mesh was there considered.

parameter	S200	S300
No. of nodes	16166540	11223343
No. of cells	12547107	8765261
Type of cells	mixed	mixed
Max skewness	5	5
Max aspect ratio	99.2	74.6

The solver was set for incompressible turbulent flow simulation, and used the SST-K omega model for turbulence. The fluid assigned to the simulated volumes was water, with all properties already in the database of the software for that specific fluid. To perform the analyses, the following boundary conditions were applied for both the valves:

PATCH	Pressure [Pa]	Turbulence intensity	Dissipation rate	Temperature [K]
inlet	6895	5%	Hydraulic diam. set	300
outlet	0	5%	Hydraulic diam. set	300

The results are in the following table:

valve	Nominal CV [gallon/min]	Flow rate calculated by CFD simulation [gallon/min]	difference
S200	1100	924	16%
S300	1100	1023	7%

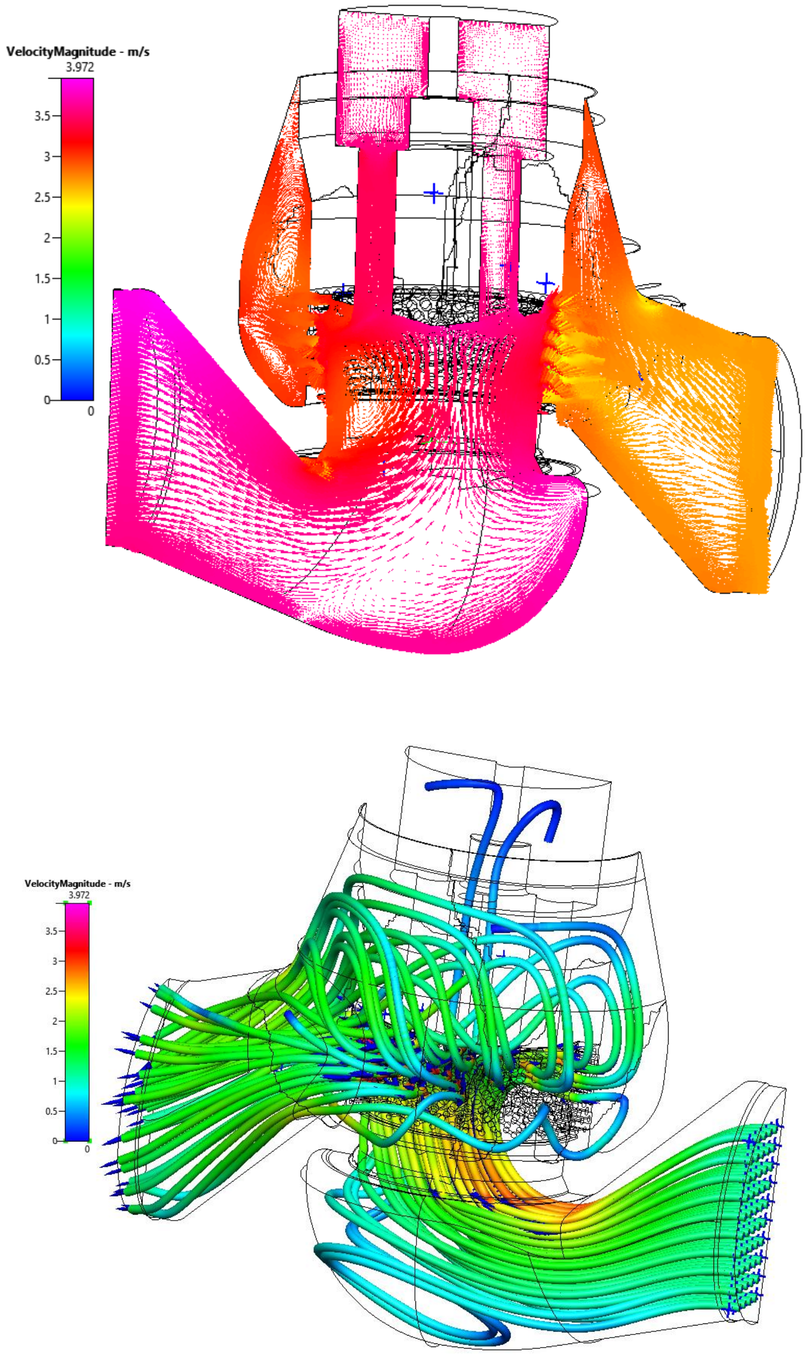


Figure 6 Some of the simulation results of the valve with S300 body

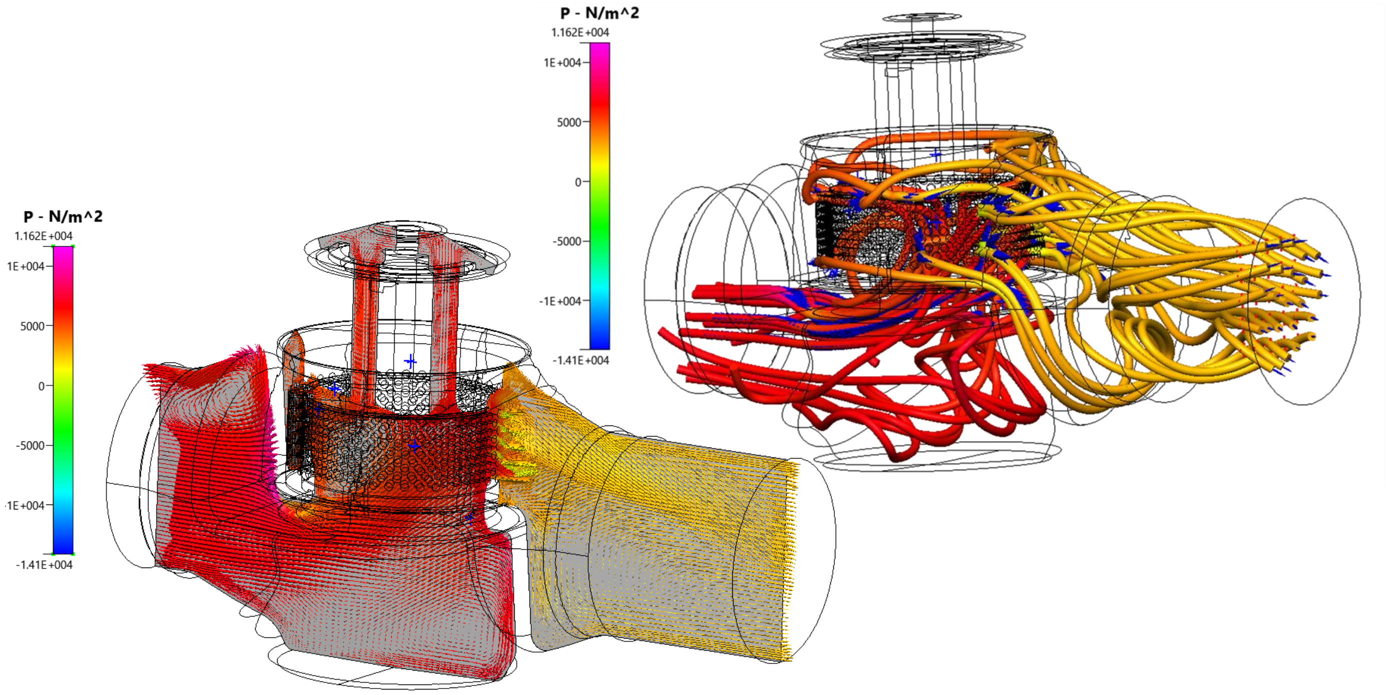


Figure 7 Some of the simulation results of the valve with S200 body

Comparison of design values and CFD results

In the previous paragraph, it is reported the difference in Cv found by the CFD simulations respect to design value. This Cv difference is about 7% for the S300-body valve, and 16% for the S200-body one. In this paragraph, the authors address the motivations of the discrepancy and the lesson learned about them.

Firstly, numerous CFD simulations about different-size valves gave a sample of results, on which statistical analyses return a bias error among CFD simulations and hydraulic testbed tests estimated of 4 ± 0.5% (95% of confidence). These tests were conducted on small valves, which were simpler to test on the hydraulic bench, due to low needs of flow rate. The measured error indicates that a CFD simulation calculates usually a lower flow rate respect to the testbed value of 4%. Experience explains that it can depends on these qualitative following factors:

- Simplification on geometry: even if the simulated fluid volume is almost the same as in the real case, some simplifications are necessary to obtain a good quality mesh. Thus, it means that small interstices are deleted and some ripples on the surface are rounded with the criteria that these modifications don't have to interfere directly with the global flux or with turbulent swirls. The Figure 8 shows some of these examples.
- Settings about simulations consider usually pure water with a volumetric mass of 1000 kg/m^3 and viscosity according to software database values. In a real hydraulic testbed, used water is not the distilled one, but what is considered pure water. It always contains some percent of mineral content.

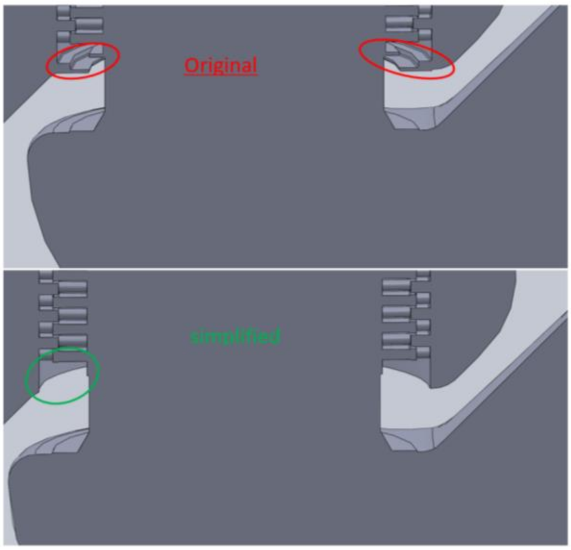


Figure 8 Examples of simplification done on fluid volumes of valves

- Evaluation of viscous effects in computational fluid dynamics needs a lot of attention. It depends on the quality of mesh, the boundary layers on the surfaces, the settings about roughness of surfaces and real roughness of surfaces of the test valve.
- Other many computational factors can influence the accuracy of the flow rate evaluation like used numerical solver, turbulence model, global mesh quality, etc.… The combination of them can influence a lot the reach of convergence and so the accuracy of solution.

Bellino, due to the high number of valve configurations, created a function to calculate the design value of Cv (hereafter *deriving function*). This function takes into account a big series of parameters like trim, body, fluid, work conditions, etc.…

The *deriving function* estimates a value of Cv equal to 1100 for the valve 10'' ANSI 150 S300 with GVCH trim. Since the total discrepancy between designed value and CFD-calculated one for this valve is 7%, it means there is another error part due to the *deriving function*. Summing up the errors about the calculation of the flow rate of this valve with S300 body, on one side due to CFD simulations, on the other side due to the *deriving function* used in Bellino, it returns:

$$\emptyset_{CFD} \cdot (1 + \epsilon_{CFD}) = \emptyset_R = \emptyset_D \cdot (1 - \epsilon_D) \quad (4)$$

where:

- $\emptyset_R$ is the flow rate measurable at the test bench;
- $\emptyset_{CFD}$is the flow rate calculated by a CFD simulation;
- ϵ_{CFD} is the bias error of performed CFD simulations evaluated with a statistical analyses, compared to the hydraulic testbed tests;
- $\emptyset_D$ is the flow rate calculated by the *deriving function* developed by Bellino;
- ϵ_D is the error found for the *deriving function*.

Therefore, according to this reasoning the correct estimation of the flow rate ($\emptyset_R$) for the valve 10'' ANSI 150 S300 with GVCH trim is 1067 [gal/min]. Applying the same correction to the valve 10'' ANSI 150 S200 with GVCH trim, the results are the followings:

valve	Designed value [gal/min]	Real flow rate value [gal/min]	CFD flow rate value [gal/min]
S200	1100	963	924
S300	1100	1067	1023

Consequently, the notable difference between the real Cv of two valves, is ascribable to the geometry of their bodies.

Actually, the geometry of the valve 10'' ANSI 150 S200 with GVCH trim has some flaws in terms of fluid dynamic fitting set because it does not follow the regular flux of the fluid (see Figure 4), creating some stagnation points. The images below explain better these conditions (see Figure 9).

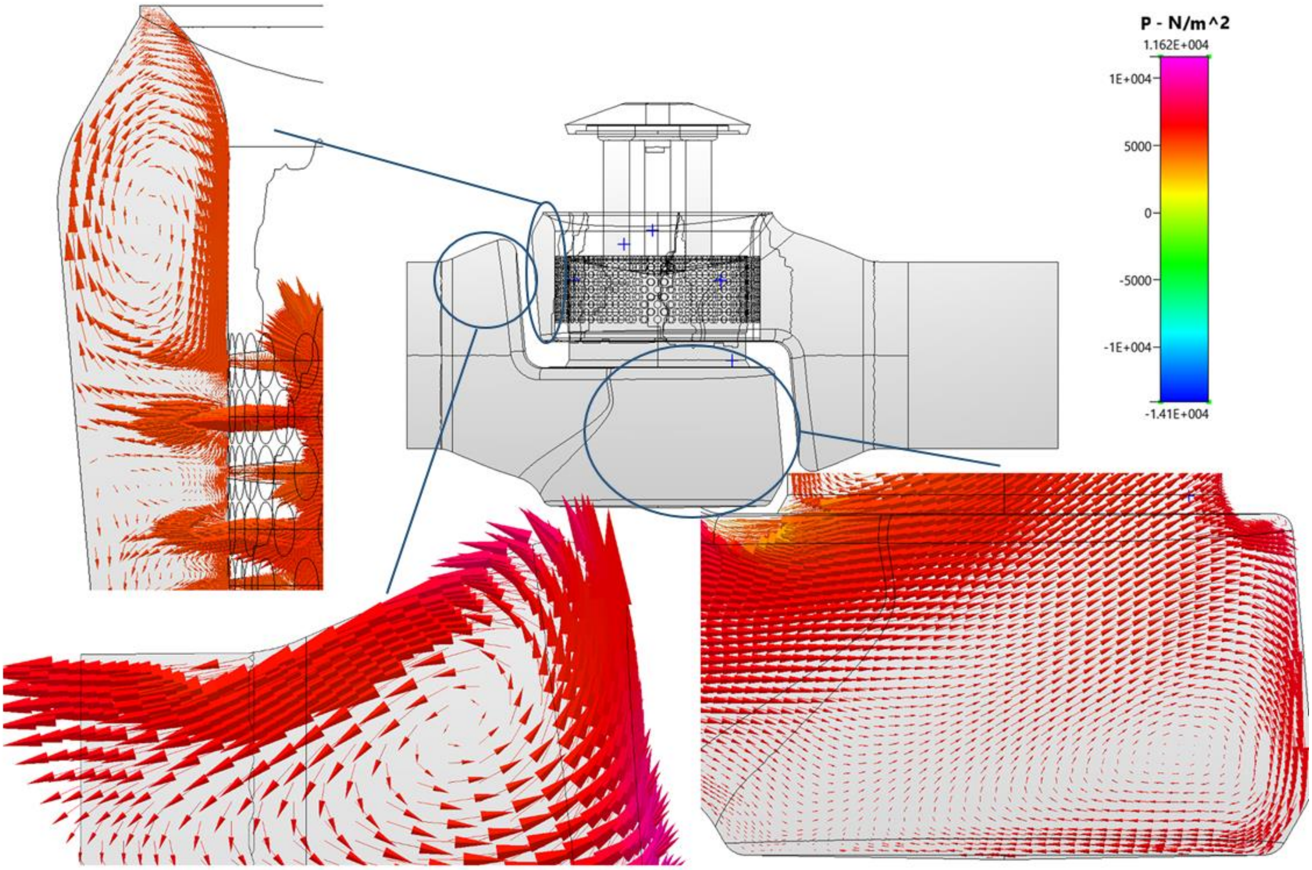


Figure 9 Worse geometrical features

Vectors represent the velocity magnitude and colors represent the pressure according to the color map defined by the legend. In that image, it is possible to see some stagnation points where some swirls go around them and waste fluid energy. Actually, these geometrical flaws are the main part of the inefficiency of valves with this type of body. In spite of the above considerations, there are various reasons why Bellino produce valves with these bodies:

- The valves with the S200 body weight 5-10% less than similar valves with S300 body
- The S200 body, compared to the S300 one, has a lower height, so usually it is easier to use them in replacing valves that are out of service, in an existing and working plant. Practically the S200 body is less bulky
- Because that specific shape which optimizes the use of materials, usually valves with S200 bodies have a lower price
- The shape of the S200 body is easily used for 3-way valves because of its symmetry

Optimization

The innovation proposed by Bellino to face to the previous described characteristics is the valve S300. It has a better performance in terms of fluid dynamic behavior, that is a bigger flow rate with the same boundary conditions. In fact, as the Figure 6 describes, the streamlines colored by the velocity map, have a simpler path, compared to the streamlines in Figure 7 It can be synonymous of a more efficient valve.

The Figure 6 shows how the main fluid dynamic flaws of valve with the S200 body are not in the valve with the S300 body, apart from some impossible removable features due to the manufacturing of them.

After these analyses with the CFD, globally Bellino can state that usually the valve with S300 body has a better performance than the S200 one, and in the specific case of the 10’’ ANSI 150 valve with GVCH trim, the difference is about 10%.

Conclusions

The paper show some results about computational fluid dynamics studies conducted on some valves designed and manufactured by Bellino. The studies compared the measure of the flow rate with different methods, which are the hydraulic bench tests, computational fluid dynamic analyses, and designed methods used in Bellino. The purpose of this research is to improve the design methods used in Bellino, because it takes into account many parameters, apart from fluid dynamic issues, therefore, it is very important in the global design of reliable and efficient control valves provided to customers of Bellino. Thus, the comparison was performed by the use of CFD simulations, when the direct evaluation of the flow rate on a hydraulic bench was not possible. Apart from improving the design procedure, this research also shows the value of its products in terms of performance, and why some products are similar but not equal.

At the conclusion of this paper, some results can be summed up in the following list: i) a better understanding of all phenomena involved in the fluxes inside the control valves; ii) confirmation that all efforts made in the past years to optimize the main part of all manufactured control valve, have been following a path in the right way; iii) recognition of other new critical parameters to be analyzed in the future.

References

[1] CEI/IEC, *Industrial-process control valves -,* International Electrotechnical Commission, 1997.

[2] G. J. Borden e P. G. Friedmann, Control Valves, New York: Instrument Society of America, 1998.

[3] Nuovo Pignone, Manuale di progettazione valvole.