OPTIMISED FLOW

www.guichon.com

BASICS OF INDUSTRIAL VALVES

GUICHON

OPTIMISED FLOW

In some processes the valves used must be designed in a manner to optimise flow. The first criteria to define is the diameter of the flow path in function of the operating conditions, particularly the flow rate, the Delta P (ΔP) and the type of fluid. Next, some specifications will require additional studies on the turbulence and pressure losses linked to the equipment, on the reduction of non circulation zones or even on the optimisation of the speed of the fluid flow for the process.

OPTIMAL DIAMETER CHOICE FOR THE FLOW PATH

WHAT IS THE NOTION OF CV/KV?

To choose a diameter for the flow path, the notion of Cv/Kv should be defined:

Kv : Flow coefficient indicating the number of m3 of water at 20°C passing through the valve with a pressure drop of 1 bar, over an hour period.

Cv : Equivalent to the coefficient Kv, but expressed in US gallons per minute of water at 60°F, passing through the valve with a pressure drop of 1psi.

The relation between Kv and Cv is expressed as follows:

$$Cv = 1.156 \times Kv$$

It is important to differentiate the valve flow coefficient "Cv/Kv of the valve" (equally called Cvs/Kvs when fully opened) that represents the valve capacity in terms of flow rate, from "calculated Cv/Kv" which is determined from the process needs (in function of the operating conditions and the fluid).

DEFINING THE PROCESS NEEDS IN TERMS OF CV/KV

The formulas to calculate Cv are listed in the standard NF EN 60534. It is set to define the type of fluid (compressible/ incompressible), the flow regime (turbulent/full flow) as well as the presence of elbows on the line to choose the adapted formula.

In the simplest case, corresponding to a valve installed without an adjacent connection and an incompressible fluid with a non-engorged turbulent flow regime, the expression of Kv is the following:

$$Kv = Qv \times (\frac{G}{\Delta P})^{\wedge}0.5$$

With :
Qv : volumetric flow rate in m^3/h
G : density in kg/dm^3
ΔP : Pressure drop (bar)

HOW TO SELECT THE VALVE DIAMETER

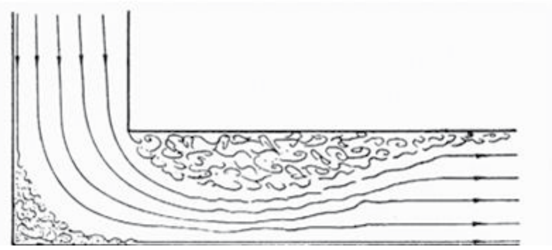
Having calculated the Cv/Kv required by the process, the selection of a superior Cvs/Kvs can be made from the manufacturers sizing chart. If a control valve is required, the sizing rules, unique to each manufacturer should be applied according to the control characteristic (see chapter entitled, "Functions").

PRESSURE DROP AND TURBULENCE

WHAT IS A PRESSURE DROP?

A pressure drop is energy loss of the fluid. These losses are due to friction of the fluid on the inner walls of the valve, and to other flow obstructions along the way (elbows, change of direction, narrowed flow path, etc.).

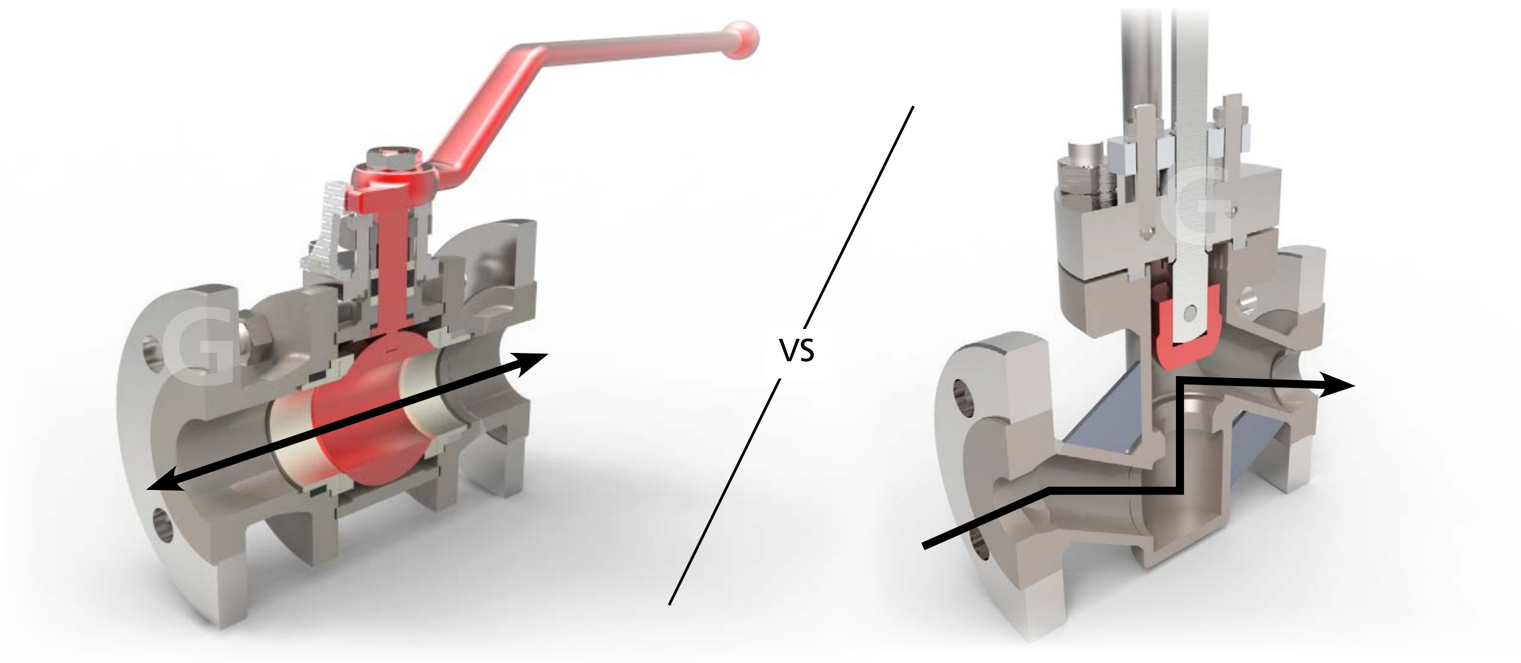
Physically, the pressure drop of a valve is the difference between the inlet pressure and the outlet pressure (ΔP).



■ Flow in a 90° elbow

TECHNOLOGIES WITH LOW PRESSURE DROPS

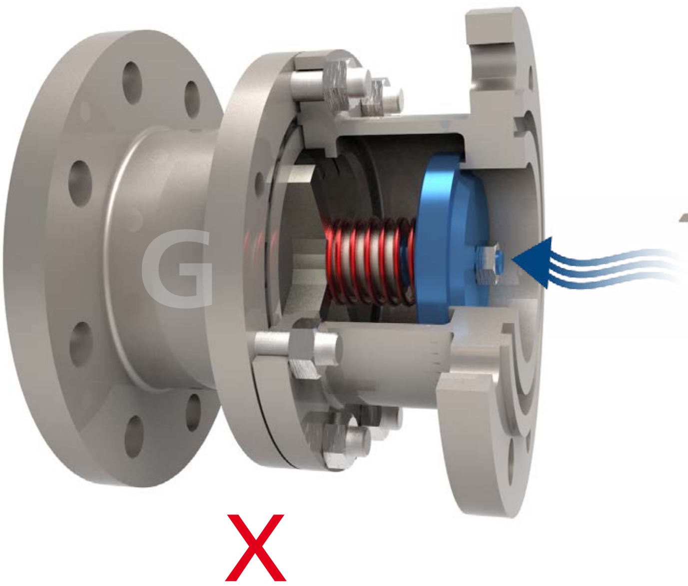
Certain valve technologies have a low pressure drop. For example, a fully opened ball valve has nothing hindering the fluid flow and the pressure losses are comparable to that of a straight pipe. On the other hand, for example, a straight pattern globe valve, even when fully opened, has a sinuous body design which creates an inevitable pressure drop.



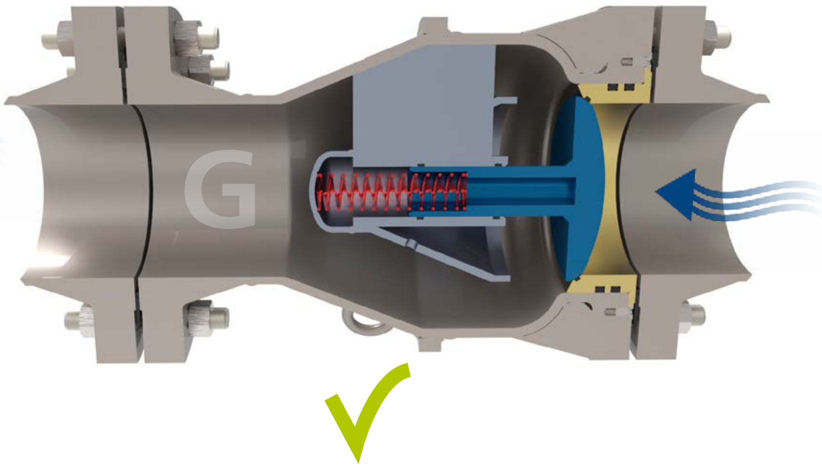
OPTIMISED FLOW

VALVE OPTIMISATION FOR REDUCED PRESSURE DROP

On certain valve equipment, it is possible to minimise the pressure drop. In this case, the objective will be to diminish any flow obstructions through optimising the body and obturator designs.

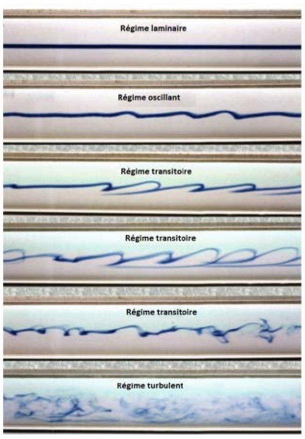


- **"Classic" non-return check valve**
  When the check valve is open, the fluid bypasses the obturator (blue disc), perturbing the flow path.



- **"Low Pressure Drop" non-return check valve**
  The disc design is optimised, "curved shape", the flow path is not as perturbed.

WHAT IS TURBULENCE?



The phenomena of wind in the atmosphere is an example of turbulence. Mechanically speaking, turbulence is the flow regime of a fluid, in which a transient motion is superimposed with an agitated, random motion.

Indeed there exists several flow regimes, from laminar to turbulent.

The flow regimes are characterised by the Reynolds number. The Reynolds number (***Re***) is dimensionless. It can be calculated accordingly:

$$Re = 1000 \times \frac{V.d}{\vartheta}$$

With :

V : The flow velocity in m/s
d : The hydraulic diameter of the tube in mm
v : The kinematic viscosity of water in mm^2/s!

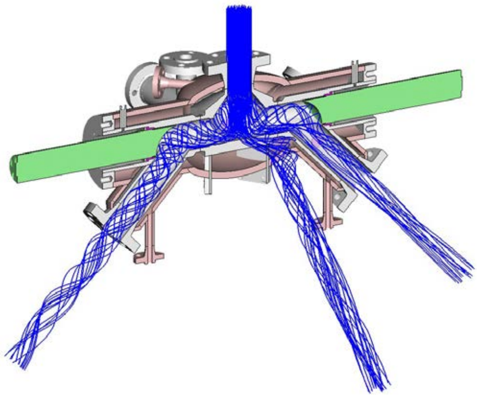
$$Re = \frac{\rho . V . d}{\mu}$$

With :

p : the volumetric mass in kg/m^3 (fluid density)
V : The flow velocity in m/s
d : The hydraulic diameter of the pipe in m
µ : The dynamic viscosity in Pa.s (or kg/m.s)

Calculating the Reynolds number allows the flow regime to be determined:

- If **Re** < 2000 : the flow regime is laminar, characterised by a calm and continuous flow.
- If **Re** > 4000 : the flow regime is turbulent, characterised by flow in the form of eddies and whirls.



MANAGING TURBULENCE IN A VALVE

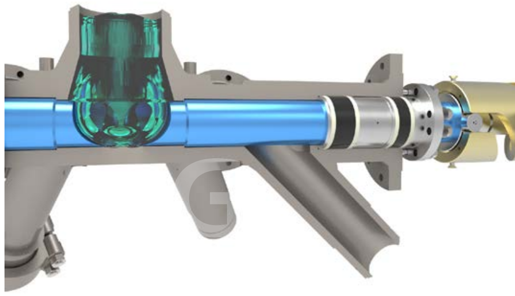
Modeling software and fluid calculations make it possible to simulate flows in such a way that turbulent zones inside valve equipment can be observed. Furthermore, there are technical solutions that allow for control of turbulence while fluid is flowing through the valve. One such solution is a rotating piston.

The schematic to the right shows a multi-way valve. The pistons are profiled to follow the internal shape of the body. In the closed position, the pistons are all in the same position.

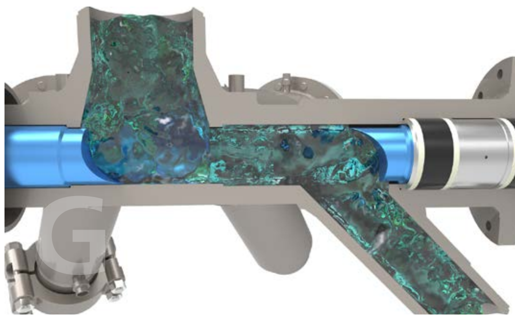
Upon opening, the piston moves back and rotates on its axis 180 degrees. When fully opened, the profiled piston tip allows the fluid to be directed towards the outlet leg, avoiding turbulent eddies when the fluid changes direction.

When the valve is installed on in-line piping, it is important to make sure that the design is appropriately adapted to avoid creating zones that perturb the good flow of fluid through the piping.

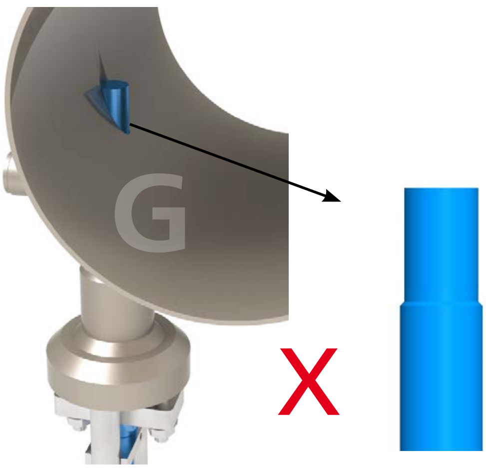
A standard valve will be equipped with a flat piston, contrary to a custom valve that can be equipped with a profiled piston to improve flow in the piping.



■ Profiled, rotating piston valve (Closed position)

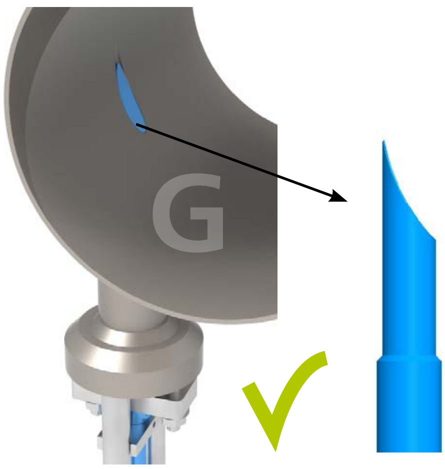


■ Profiled, rotating piston valve (Opened position)



■ Use of a flat piston on a tangential valve

VS



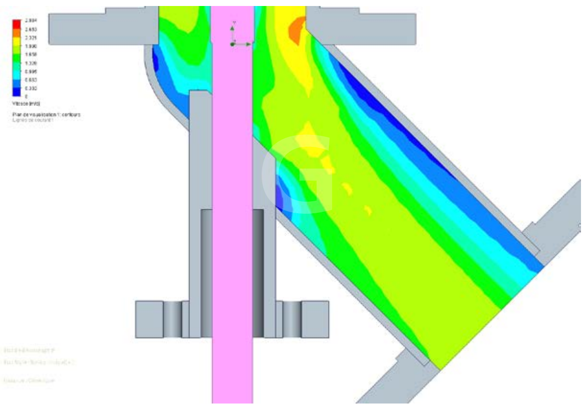
■ Use of a profiled piston on a tangential valve

OPTIMISED FLOW

RETENTION/NON CIRCULATION ZONES (DEAD SPACE)

A retention/non-circulation zone is a space where the fluid stagnates. A valve body design said to be "dead space free" can be essential for avoiding stagnation of viscous fluids, a localised solidification or contamination between two samples.
It is important to note that dead spaces can produce block formations that can disturb fluid flow and eventually damage the surfaces/elements assuring tightness and/or the actuation system.
In order to illustrate a non-circulation zone, we will look at an example of a tank bottom drain valve and a multiway valve used for diverting a fluid.

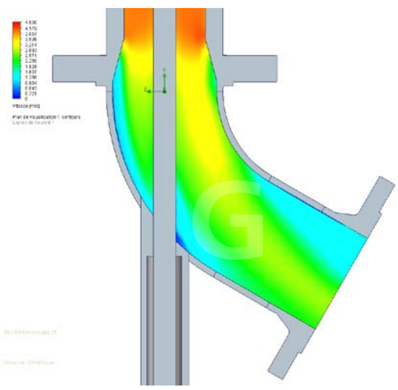
Example 1: To drain a tank, an operator has 2 solutions:



1 The standard tank bottom disc valve solution

The design of this valve is standardised as much as possible to reduce costs. The flow is therefore not fully optimised.

For example, the stuffing box protrudes into the valve body, creating a source for fluid retention. The dead spaces are represented in blue on the image to the left.



2 The customised tank bottom disc valve solution

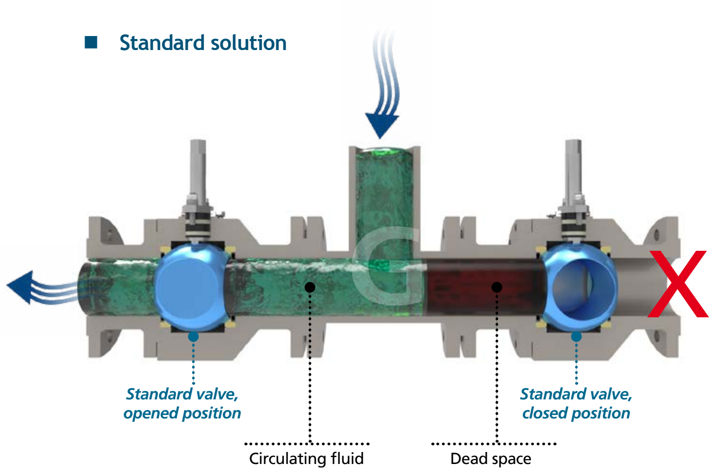
With a customised valve body, the dead spaces are significantly diminished (blue zones reduced). In this valve, the fluid will not stagnate.

Example 2: To divert a fluid, an operator has 2 solutions:

1 Standard Solution: 2 ball valves coupled by a tee fitting

When one of the two valves is closed, a retention zone is created inside of the closed valve. In this case, we observe that when the fluid is directed to the left, it stagnates on the right side of the tee connection, and vice versa. The fluid contained in the retention zone is susceptible to transform into another state (solidification, crystallisation…).

When the valve to the right will open, the fluid from the dead space will be transferred into the process and will contaminate the next production batch.
If the fluid is sticky or viscous, a part of fluid residue from the retention zone will equally adhere to the piping, causing a reduction in the flow path. In this case, the operator must proceed to disassemble the valves for maintenance.



2 Solution with a custom multi-way valve

For this custom solution, there is only one valve body and it is perfectly adapted to the process line. The connections are limited to the strict minimum, which eliminates potential leak points (no "tee" fitting). Thanks to two symmetric pistons, we observe that there are no longer any retention zones.