Change in the Flow Rate Through a Deformed Valve

D. Kekejian[*1], Y. Martinez S.[**2]
[1]Yerevan State University, 1 Alex Manoogian, Yerevan, Armenia.
[2]Instituto Tecnologico y de Estudios Superiores de Occidente, ITESO
Tlaquepaque, Jalisco, Mexico
*davidgenius1@hotmail.com
** yurims@iteso.mx

Abstract: In this work, a new design is suggested which functions both as a pipe for fluid transfer and as a valve which decreases the flow of the fluid in the opposite direction in case there are pressure fluctuations in the pipe. Therefore, we call it a "Deformed Valve". To design the structure, we had to use different geometric shapes and calculate the velocity of the fluid for each shape to determine the best geometric conditions in which our Valve is optimized.

1. Introduction

Optimization is one of the greatest challenges in Engineering science. To make a design optimized to the best is very useful in order to manufacture perfected devices. In this work, an optimization problem is solved for a pipe. By choosing the right geometry we were able to make the ratio of the flow in one direction to the opposite direction as maximum as possible. This type of Deformed Valves can replace the ordinary valves that are used in pipes which makes the manufacturing process difficult and expensive. By using the suggested design it is possible to build more efficient pipes that are cheaper too.

Keywords: Deformed Valve, Optimized Pipe, Turbulent Flow, Maximum Flow Ratio

2. Theoretical Analysis and Estimations

To solve the problem, we did some Theoretical Analysis of some important facts in fluid dynamics which are the following

2.1 The Venturi Effect

The Venturi effect [1] is the reduction in fluid pressure that results when a fluid flows through a constricted section of pipe. The Venturi effect is a jet effect; as within a funnel the velocity of the fluid increases as the cross sectional area decreases, with the static pressure correspondingly decreasing. According to the laws governing fluid dynamics, a fluid's velocity must increase as it passes through a constriction to satisfy the principle of continuity, while its pressure must decrease to satisfy the principle of conservation of mechanical energy. Thus any gain in kinetic energy a fluid may accrue due to its increased velocity through a constriction is negated by a drop in pressure.

2.2 Local Pressure Losses

When local obstacles, flow area and/or flow direction changes occur, additional pressure losses appear. These losses are termed as local (or minor) losses [2].

2.2.1 Sudden Expansions

One of the common flow obstacles is a sudden cross-sectional area change. When flow has a direction from the smaller to the larger cross-sectional area, the obstacle is called a sudden enlargement. By applying the conservation of mass, momentum and kinetic energy we get

Mass balance:
$$U_1 A_1 = U_2 A_2 \quad (1)$$
Momentum balance:
$$F = \rho(U_1 A_1 U_1 - U_2 A_2 U_2) + p_1 A_1 - p_1 A_1 \quad (2)$$
Here F is the force exerted by walls on the fluid .
Solving for the pressure difference gives,
$$p_2 - p_1 = \rho U_2 (U_1 - U_2) = \rho U_2^2 (\frac{A_2}{A_1} - 1)$$
Since $A_2 > A_1$, the momentum balance predicts a rise in the pressure.
Mechanical energy balance:
Applying the Bernoulli equation yields:
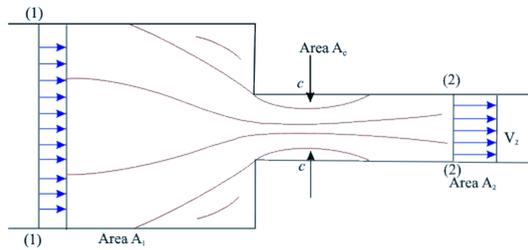$$0.5\rho U_1^2 + p_1 = 0.5\rho U_2^2 + p_2 + \Delta p_I \quad (3)$$
For sudden enlargement, the local pressure loss and the loss coefficient are as follows,

$-\Delta p_I = \xi_{enl} 0.5\rho U_1^2 \; ; \xi_{enl} = (1 - \frac{A_1}{A_2})^2 \; (4)$

Where ξ_{enl} is the local loss coefficient, ρ is the density of the liquid, U_1 and U_2 are the velocities respectively at the surfaces A_1 and A_2.

2.2.2 Sudden Contractions

When flow has a direction from larger to smaller cross-sectional area, the obstacle is called a sudden contraction. This type of obstacle is not a simple analogue to the sudden enlargement case due to the presence of a so-called vena contracta just downstream of the obstacle, as shown in Figure 1

Figure 1. Flow through a sudden contraction

In a similar manner as for the sudden enlargement, the local pressure loss and the loss coefficient for the sudden contraction can be shown to be as follows:

$-\Delta p_I = \xi_{enl} 0.5\rho U_2^2 \; ; \xi_{enl} = (\frac{A_2}{A_c} - 1)^2 \; (5)$

where A_c is the smallest flow cross-section area which occurs at a certain distance downstream of the area contraction, at the vena contracta location. This cross-section area is not known exactly and depends on the area ratio A2/A1. It is customary to give the contraction loss coefficient in a graphical or tabulated form as a function of the area ratio A2/A1, (see Table 1). Alternatively, the loss coefficient can be obtained from the following correlation,

$\frac{A_c}{A_2} = 0.62 + 0.38 \left(\frac{A_2}{A_1}\right)^3$

A_2/A_1	0	0.2	0.4	0.6	0.8	1
A_c/A_2	0.586	0.598	0.625	0.686	0.790	1
$(A_2/A_c -1)^2$	0.5	0.45	0.36	0.21	0.07	0

Table 1. Values of the local loss coefficient for a sudden contraction.

For an arbitrary obstacle, such as channel bending, orifice or grid spacer in a fuel assembly, the local pressure loss is calculated from Eqs. (4) and (5), where the local loss coefficient will depend on the geometry details of the obstacle and is typically establish in an experimental manner.

## 3. Use of COMSOL Multiphysics

3.1 Geometry and parameters

To not complicate the problem and to make the solving sequence easier we solved it in 2D.To design the pipe we used two rectangular figures that are connected to each other with a triangular like shape which we like to call it a "Deformed Valve". We defined the 1st rectangle, (see Figure 2), as the following
Width=0.5m, since it equals to the diameter of the pipes used to provide water in PWR reactors. Height=1m,position in corner with x= − 0.25 and y= −0.5+p,where p is a parameter defined as the following $p = (0.25 - r) \times \tan\left(a \times \frac{\pi}{180}\right) - 1$
Where a is the connection angle (marked in red, Figure 2) and r is the length of the opening (marked in blue, Figure 2).For this problem we chose r to be 0.025m for avoiding solution errors during computing, and changed a by using parametric sweep = range(15,5,85)
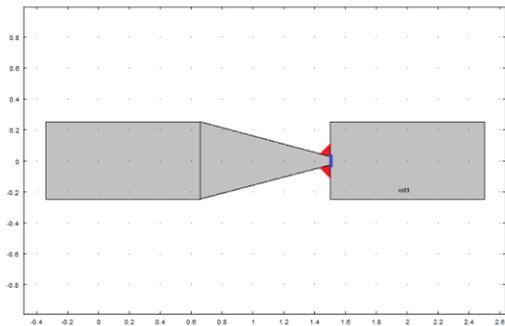
Figure 2. The Geometry of the pipe

For the triangular like shape, we used three linear segments. From (0.25, –0.5+p) to (r, –1.5) to (–r, –1.5) to (–0.25, –0.5+p).Concerning the 2nd rectangle, it has the same size as the first but the position is center (0, –2).The geometry is designed in a way that the length of the openings remain the same when the connection angle is changed. Also, we placed it horizontally to neglect the hydrostatic pressure of the fluid.

3.1 Materials

We used liquid water, since it is the most common liquid and is largely used in industrial applications. Its density equals to 1000kg/m^3 and has a dynamic viscosity equal to 1.002×10^{-3} Pa.s

3.2 Physics Interface - The Turbulent flow

In fluid mechanics or more generally continuum mechanics, incompressible flow refers to a flow in which the material density is constant within a fluid parcel – an infinitesimal volume that moves with the velocity of the fluid. An equivalent statement implying incompressibility is, that the divergence of the fluid velocity is zero.
The turbulent flow module solves The Reynolds-Averaged Navier-Stokes (RANS) equations [3]. The Reynolds-averaged representation of turbulent flows divides the flow quantities into an averaged value and a fluctuating part, $u_i = U_i + u_i'$, where U_i is the time averaged part and u_i' is the fluctuating part. Decomposition of

flow fields into an averaged part and a fluctuating part, followed by insertion into the Navier-Stokes equation, then averaging, gives the Reynolds-averaged Navier-Stokes (RANS) equations:

$$\rho \frac{\partial \mathbf{U}}{\partial t} + \rho \mathbf{U}.\nabla \mathbf{U} + \nabla.(\rho \mathbf{u}' \times \mathbf{u}'') =$$
$$= -\nabla.P + \nabla.\mu(\nabla \mathbf{U} + (\nabla \mathbf{U})^T) + \mathbf{F}$$

$$\nabla.\mathbf{U} = 0$$

$\nabla.(\rho \mathbf{u}' \times \mathbf{u}'')$ represents interaction between the fluctuating velocities and is called the Reynolds stress tensor.
In some cases, the accuracy provided by wall functions is not enough. In these cases, a so called low Reynolds number model can be used. The phrase "low Reynolds number" refers to the region close to the wall where the viscous effects dominate.

The fluid properties were taken from the material and the temperature was set to 293.15K.

The wall boundary condition was set to No Slip which prescribes $\mathbf{u}$=0 ,that is the fluid on the wall is not moving.

The initial values of pressure and velocity were set to zero.

The inlet boundary condition was set to "pressure, no viscous stress" with p=10atm. For fully developed pipe flow, the turbulent Intensity is about 0.05 and the turbulent length scale is about 3.8% of the hydraulic diameter. For our case, it is 0.019m.

The outlet boundary condition was also set to "pressure, no viscous stress" with p=1atm.

3.3 Meshing

The mesh generator divides the domains into triangular or quadrilateral mesh elements. If the boundary is curved, these elements represent only an approximation of the original geometry. We used triangular mesh, because it is hard to make a smooth edge flow of quadrilateral elements specially at the connection area where the fluid passes from the Deformed Valve into the 2nd rectangle. The triangular size was set to fine .However, to increase the smoothness and
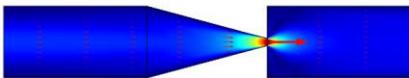
resolution we added a distribution of fixed number of elements equal to 50 at the connection area.
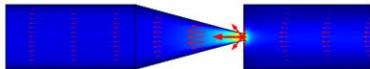
3.3 Results

We studied two cases.
1-When the water is flowing from left to right (see Figure 3).
2-When the water is flowing from right to left (see Figure 4).

**Figure 3.** The flow of water in the 1st case when the angle is 75d

Figure 4. The flow of water in the 2nd case when the angle is 75d

After the solution sequence done for each angle, we calculated the mean velocity for each angle at

the connection area by using line integration. Then plotted the results vs. the connection angles for the two cases (see Figure 5 and 6).
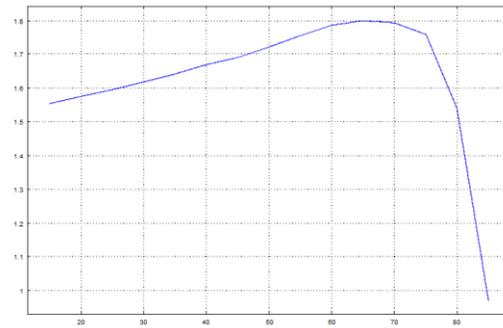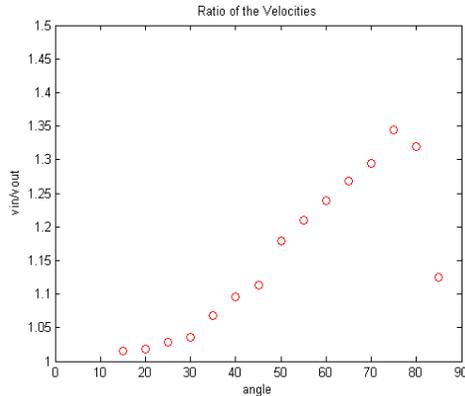

Figure 5. The velocities vs. the connection angles for the 1st case
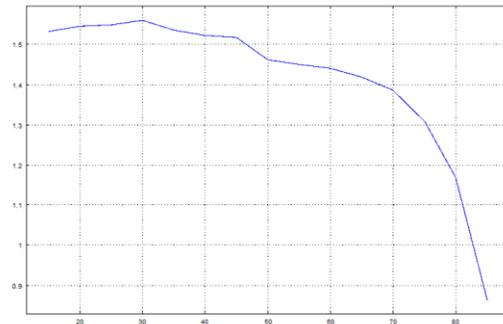
Figure 6. The velocities vs. the connection angles for the 2nd case

But when we calculated the ratio of the velocities in the 1st case to the 2nd we got the following results (see Figure 7)

Figure 7. The ratio of the velocities vs. the connection angles

As a result, we see that the ratio increases as the angle increases,it reaches a maximum value at 75 degrees and then decreases.
Therefore, by making the connection angle equal to 75 degrees we can make the ratio of the flow in one direction to the opposite direction as maximum as possible.

4. Conclusions

These results could be useful for diverse types of applications, for example, this type of pipes can be used in complex structures, such as Nuclear Reactors , to decrease the flow of the coolant (water) in the opposite direction in case of accidents, such as the blow of a pipe which will cause a pressure drop and may change the flow direction.
In future, we expect to include Heat Transfer module and design refrigerators working with shock waves which will be called Plasma-Stirling cooling machines.

9. References

1. The Venturi effect, Wolfram Demonstrations Project. Retrieved 2009-11-03.
2. Henryk Anglart, Thermal-Hydraulics in Nuclear Systems,69-71,2008
3. D.C. Wilcox, "Turbulence Modeling for CFD," 2nd ed., DCW Industries, 1998.