Determining the coefficient of hydrodynamic torque using CFD

In quarter-turn valves, torque is the turning effort required to rotate the valve's closure member (ball, disc, plug) or hold it in position. Knowing the valve torque is essential in actuator sizing. Major components of total shaft torque include seating, bearing, packing frictional torque and flow-induced hydrodynamic torque. This article discusses how to calculate hydrodynamic torque using CFD simulations.

Hydrodynamic torque (T_d) is a flow-induced torque that is purely due to the fluid forces acting on the rotating valve parts. Hydrodynamic torque is determined to be a function of valve design, valve opening position, pressure drop and flow direction (in case offset valves). It is a general industry practice to use hydrodynamic torque coefficient (C_{dt}) to obtain hydrodynamic torque for any operating pressure conditions.
The hydrodynamic torque coefficient is a non-dimensional representation of hydrodynamic torque normalized by static pressure drop across the valve and valve size. Dynamic torque coefficient is calculated using the equation:

$$C_{dt} = \frac{T_d}{\Delta P \times D^3}$$

T_d - Hydrodynamic torque
C_{dt} - Coefficient of hydrodynamic torque
ΔP - Pressure drop across the valve
D - Diameter of the valve

Traditionally, dynamic torque (and flow) coefficients are determined by valve flow loop test. The test is typically performed under fully turbulent, non-cavitating flow conditions using water at a uniform approach-velocity encountered in a long straight pipe.
The hydrodynamic torque is determined by averaging out total opening and closing torque because adding the torques together cancel out frictional torques. Pressure drops are measured two diameters upstream and six diameters downstream for several flow rates at each opening.
Dynamic torque for large-sized and high-pressure valves is evaluated from a scaled prototype of their product line due to limitations of experimental testing. But computer technology enables to predict flow coefficients using Computational Fluid Dynamic simulations.

Computational Fluid Dynamic simulations

Over the past decade with rapid development of computer technology Computational Fluid Dynamics (CFD) has become an important method of engineering design. CFD uses numerical methods to solve fluid flow equations and does not require a physical model of a valve. Computers are used to perform the calculations required to simulate the flow of the fluid. General, CFD simulation methodology is like this:

Pre-processing
- Obtaining fluid volume from the CAD geometry.
- Discrete the volume in finite number of cells to solve fluid flow equations numerically.
- The physical modeling boundary condition setting.

Solving
- Solving the numerical fluid flow equations iteratively using high-performance computers.

Post-processing
- Finally, a post-processing is used for extracting quantitative and qualitative data from the simulation.

CFD uses finite volume or finite difference approach to solve the governing equations iteratively for fluid velocities, mass flow, pressure, temperature, turbulence parameters and other fluid properties. These numerical techniques involve the discretization of the flow volume into a finite number of cells and solving the governing partial differential equations for each cell numerically. As a result, an approximate value of the variables at a specific point can be obtained.

Powerful method

Figure 1 shows viscous and pressure forces acting on a face of a finite volume cell and the radial distance from the axis of rotation of the valve disc. In this way, one can obtain total torque acting on moving components of the valve by integrating viscous and pressure torque around the axis of rotation for a particular opening condition.

$$F_{pressure} = p_i A_i \quad (1)$$

$$F_{viscous} = \left(\upsilon\rho \frac{\partial U_i}{\partial x_i} \right) A_i \quad (2)$$

$$Torque\ (T_d) = \sum_i (r_i \times F_{pressure}) + \sum_i (r_i \times F_{viscous}) \quad (3)$$

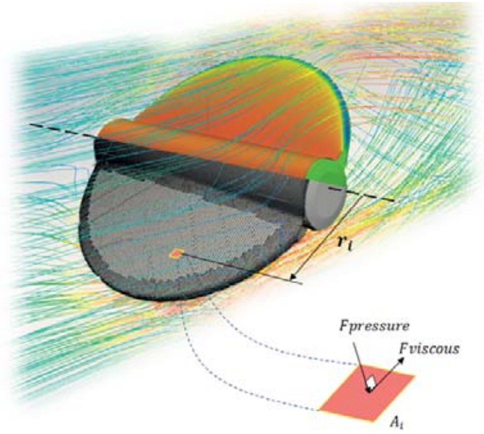

Figure 1: Fluid forces acting on discretized face element on a valve disc. Source: simulationHub Autonomous Valve CFD

Where, i is the cell face number attaches to the valve disc and stem, $F_{pressure}$ and $F_{viscous}$ are the pressure and viscous force acting on the face element respectively. r is radial distance of the face element from the axis of ration, ν is the kinematic viscosity of fluid, ρ the density of fluid, U is the fluid flow velocity, x is the linear dimension of the face in the direction of velocity, A is the surface area of a face element.

Obtaining hydrodynamic torque and performance coefficients using CFD seems a powerful method but it is not a piece of cake for everyone. Traditional CFD approach requires skills related to fluid volume extraction, meshing, boundary condition setup and even post-processing to obtain accurate and meaningful results. Another challenge is that CFD requires high computational machines to run simulations.

Autonomous CFD

These challenges can be solved using autonomous CFD approach by which all the CFD-processes (CAD-cleanup, fluid volume, meshing, solving, post-processing) can be automated using intelligent algorithms and solved over clusters of cloud computers. By doing so, designers can focus on the design part and all the CFD-processes are managed by intelligent algorithms. simulationHub Autonomous valve CFD is an industry-specific CFD-application developed for valve designers using the autonomous CFD approach. The software requires a CAD-geometry and inputs like opening condition and fluid flow direction in order to run a simulation. An app has been developed to provide the results in the form of C_v, K_v, and C_{dt} for each opening condition.

Conclusion

The above cases illustrate that the hydrodynamic torque coefficient for a valve depends on various factors such as system conditions, valve design, and disc position and rotation, etc. Predicting the operating torque using traditional flow loop testing can provide limited insights to the valve designer.

However, by using Computational Fluid Dynamics (CFD), the dynamic torque can be calculated and malfunctions or short-comings in their design would be identified before the valve has been manufactured. Advancement in upcoming CFD technology such as autonomous CFD using artificial intelligence and availability of high computational resources will revolutionize the valve design procedure.

Case studies

So, using this technology we have studied the behavior of hydrodynamic torque coefficient for butterfly valve for some cases.

Case study 1: based on valve eccentricity

In this case, the hydrodynamic coefficient of concentric and double offset butterfly valves of the same size is calculated for every 10-deg opening condition ranges from 20-deg to 90-deg. The results obtained from the simulation show (Figure 2) that dynamic torque coefficients are different for a concentric and offset type of valve, the maximum value occurs at approximately 60° to 80° opening condition.

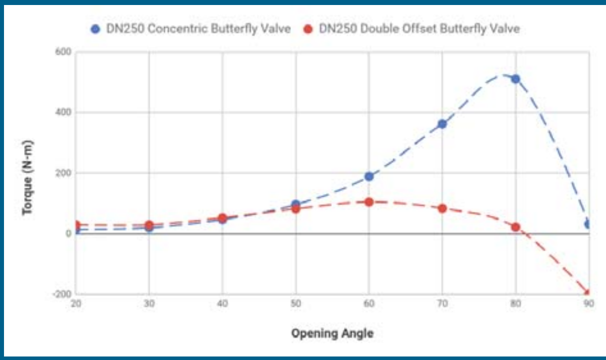

Figure 2: Dynamic torque coefficient for concentric and double offset valve. Source: Results obtained using Autonomous Valve CFD app

Case study II: based on flow direction

For a concentric butterfly valve, C_{dt} curve behavior is independent of flow direction and it is fixed for that valve. But in the case of a double-offset butterfly valve, due to geometry, drastic change in behavior is observed by reversing the flow direction from seat side flow to shaft side flow. The results show (Figure 3) that the behavior of C_{dt} depends on the direction of flow for offset valves. Also, C_{dt} changes its sign near 80° for a flow in shaft upstream direction.

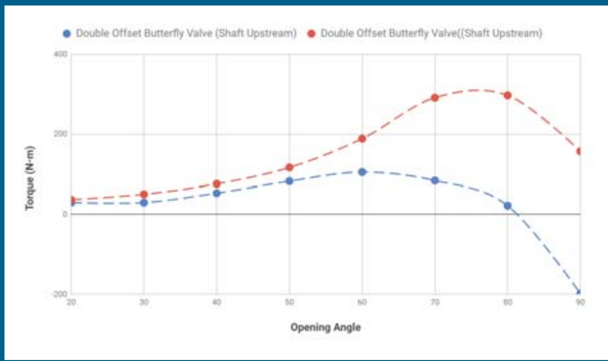

Figure 3: Dynamic torque coefficient behavior for different flow direction. Source: Results obtained using Autonomous Valve CFD app

CASE III: orientation & elbow bend

The hydrodynamic torque is purely resultant of fluid forces acting on valve disc, so any upstream flow disturbance can significantly affect hydrodynamic torque coefficient. The upstream flow disturbance most often caused by line elements (e.g., elbows, pump, tee etc.,). The following figure shows a butterfly valve installed in three different configurations relative to elbow fitting. From the results, it is evident that the C_{dt} characteristic curve depend on valve orientation with respect to the elbow and valve rotation.

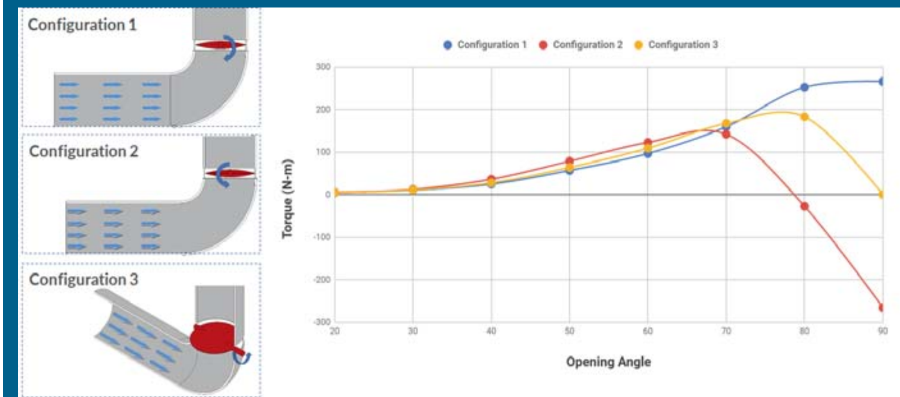

Figure 4: Dynamic torque coefficient characteristic for valves after elbow fitting in different configurations. Source: Results obtained using Autonomous Valve CFD app