flotek.g 2017- “Innovative Solutions in Flow Measurement and Control - Oil, Water and Gas”
August 28-30, 2017, FCRI, Palakkad, Kerala, India

Computational Fluid Dynamics Simulation

Kamala Kannan K R
McWane India Private Limited (MIPL)
Coimbatore, India.
kr.kamalakannan@mcwaneindia.com

VelMurugan C
McWane India Private Limited (MIPL)
Coimbatore, India.
velmurugan.chandran@mcwaneindia.com

ABSTRACT

This paper is prepared to promote the basic understanding of Computational Fluid Dynamics (CFD) simulation, its merits, demerits and its applications. We will compare and correlate the experimental results observed in an actual flow test with CFD-simulation to validate the accuracy and feasibility of CFD simulations.

A DN 1200 Butterfly valve is chosen as the reference model for our work. Butterfly valves are commonly used in industrial applications to control the flow of both compressible and incompressible fluids. A butterfly valve typically consists of a metal disc formed around a central shaft, which acts as the discs axis of rotation. The valve opening angle is increased from 0º (fully closed) to 90º (fully open). After comparing the results it is observed that the experimental results correlate with the CFD simulation results within an acceptably small variation.

It may be conducted that realistic fluid flow problems can be solved using CFD provided that matching boundary and loading conditions are modeled.

KEY WORDS

CFD, butterfly Valve, numerical Simulation, experimental validation

INTRODUCTION

CFD is a method of obtaining a discrete solution of real world fluid flow problems. A discrete solution is obtained at a finite collection of space points and at discrete time levels. The numerical methods applied today were developed during the 1970's to 1980's. Development is ongoing particularly in the physical parameterization of the flow models.

The establishment of the computational domain also known as pre-processing, defines the geometry or domain of the case. The domain is then divided into a large number of cells, which form a network called a computational grid. The grid is used to approximate the derivatives of the Navier-Stokes equations. This results in a large number of algebraic nonlinear equations.

In addition auxiliary equations are formed to describe the flow physics. A common phenomenon to be modeled in the case of all practical flows is turbulence. Depending on the case, there may be many other differential equations and constitutive models that describe the flow. The task of the flow simulation is to solve these equations with the user-specified boundary conditions.

The solution contains a huge amount of data. In order to gain an insight to the calculated result, visualization methods are employed to simplify the data. This data analysis is generally termed post-processing.

The role of CFD has become so strong that it can now be considered a third area of fluid dynamics. The other two classical fields are experimental and theoretical fluid dynamics.

The fundamental governing equations are- the continuity, momentum and energy equations.

1. Mass is conserved. (Continuity equation)

2. Newton’s second law. F=ma (momentum equation)

3. Conservation of energy (Energy Equation)

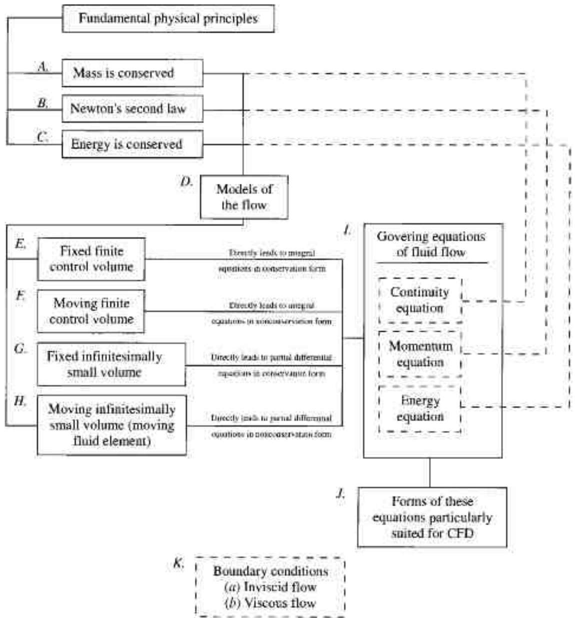


Fig 1- Fundamental physics & principals

EXPERIMENTAL SETUP

A butterfly valve is chosen to illustrate and verify that CFD-simulation correlates with experimental results. The experimental testing was conducted at a certified laboratory.

The 1200mm butterfly valve is fixed at the test bench and experimental results were recorded for various disc opening conditions. The 90^{o} (fully) open condition is used as the reference condition. As a result the Flow coefficients (Cv) were calculated by retrieving Input parameters from upstream and downstream locations on the test line. Fluid pressure is calculated at 6D distance at downstream (Outlet) and 2D distance at Upstream (Inlet).

Fig 2- Butterfly valve at test line

NUMERICAL SIMULATION

A CFD analysis is conducted using a numerical approach. Key physics settings like fluid medium and flow equations are defined before the analysis. Water is assumed as the fluid medium and the Spalart Allmaras turbulent equation was employed for the analysis. The flow was governed by continuity, momentum and energy equations. Inlet and outlet loading data are used from the experimental setup.

The key steps in applying the boundary and loading conditions using CFD are as follows:

- Pre-modelled CAD file is imported to the software.
- Boundary walls are meshed using fine elements for better result accuracy.
- Boundary conditions like inlet, outlet and wall are defined.
- Loading condition at Inlet of pressure 0.45bar is defined and flow rate is defined at the outlet.

- Wall function is defined at the wall regions.
- Solution setup and flow equations were defined.
- Solution is solved and results are plotted at the inlet and outlet conditions, similar to experimental setup.

From CFD-Simulation theoretical values are retrieved to calculate the Cv(Flow coefficient)

RESULTS

<u>Results from CFD simulation:</u>

Fig 3-Pressure at Inlet

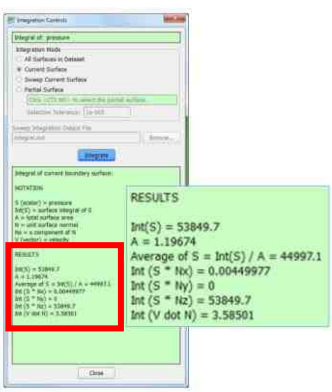


Fig 4-Integrated pressure result at inlet

Fig 5-Pressure at Outlet

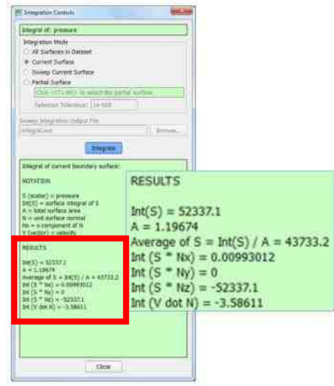


Fig 6-Integrated pressure result at outlet

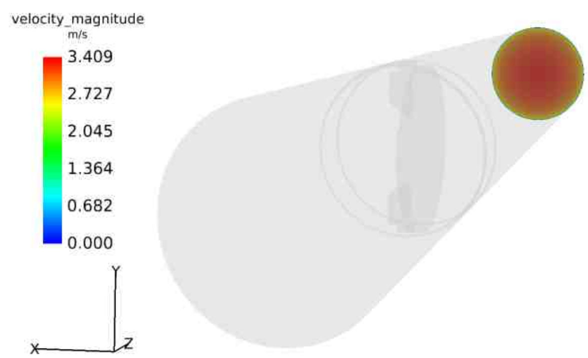


Fig 7 -Velocity at outlet

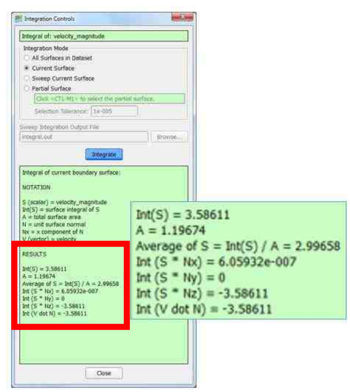


Fig 8-Integrated velocity result at outlet

Butterfly Valve 90^0 open condition
Cv = Qa / N1 * √ (Gf/dP)

Rho = 996.927 kg/m^3

Gf = Density of water during test / density of water at 15.6^o c

Gf = 996.927 / 999.007

= 0.997918

dP = Up stream Pr – downstream Pr

= 44997.1 – 43733.2 N/m^2

=**<u>1263.9 N/m^2 (0.012639 bar)</u>**

Qa = Area X velocity

=1.1304 X 10787.688

=**<u>12194.4025152 m^3/hr</u>**

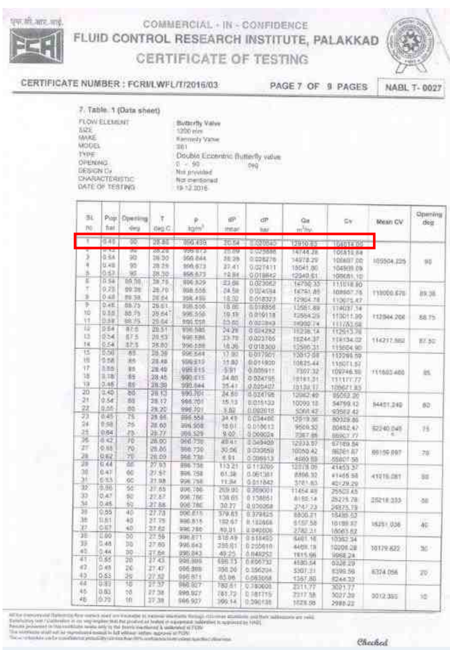
N1 (constant) = 0.865

After applying the above values we get

Cv = Qa / N1 * √(Gf/dP)

Cv = 100317.81 @ 90^O Open condition

<u>Results from Experimental test calculations:</u>


COMMERCIAL - IN - CONFIDENCE
FLUID CONTROL RESEARCH INSTITUTE, PALAKKAD
CERTIFICATE OF TESTING

CERTIFICATE NUMBER : FCRI/LWFL/T/2016/03 PAGE 7 OF 9 PAGES NABL T- 0027

7. Table. 1 (Data sheet)

FLOW ELEMENT	Butterfly Valve
SIZE	1200 mm
MAKE	Kennedy Valve
MODEL	[illegible]
TYPE	Double Eccentric Butterfly valve
OPENING	0 - 90 deg
DESIGN Cv	Not provided
CHARACTERISTIC	Not mentioned
DATE OF TESTING	19.12.2016

Sl. no	Pup bar	Opening deg	T deg C	ρ kg/m³	dP mbar	dP bar	Qa m³/hr	Cv	Mean CV	Opening deg

Fig 9-Experienmental test report

Cv = Qa / N1 * √ (Gf/dP)

Density of Water = 996.927 kg/m^3

Gf = Density of water during test / density of water at 15.6^o c

Gf = 996.927 / 999.007

= 0.997918

dP = 0.020540 bar

Qa = 12910.83 m^3/hr

N1 (constant) = 0.865

After applying the above data's (values) we get

Cv = Qa / N1 * √ (Gf/dP)

Cv = 104014.09 @ 90^O Open Condition

After comparing Cv values from the experimental tests and the CFD simulation, we can observe that (Cv) values from CFD deviate about **3.55%** from experimental results

COMMON APPLICATIONS WHERE CFD are USED.

The origin of CFD is in the aerospace industry. Today it has become an essential tool in almost every field of engineering science. Some of the Prime industries where CFD is frequently used are

- Aerospace / Aeronautics
- Automotive or Automobile
- Building HVAC (heating, ventilation, and air conditioning)
- Chemical / petrochemicals
- Energy / Power Generation
- Manufacturing / Process engineering
- Oil & gas industry
- Product design and optimization
- Bio medical
- Civil engineering applications
- Naval architecture applications

CONCLUSION

Traditional methods of prototype testing have ever increasingly been replaced by simulations using computers, Even so the fundamentals of the analysis have not changed as both the formulas and equations are the same. In many cases it is difficult or impossible to do prototype testing (e.g. wind flow over buildings, chemical flow etc.) CFD may help determine performance factors and design for proper functioning in situations that traditionally have not been conveniently subject to analysis. In many other cases prototype testing is too complicated and expensive to be practical. CFD simulations may help reduce the cost and time with associated with these analyses improving product performance, reliability and lowering overall cost.

ACKNOWLEDGMENTS

We are gratitude to Mr. Muthu and our colleagues for their support and assistance during preparation of paper. We would also like to thank McWane India for providing cool workspace and facilities. We also thank our parents and God for their blessing and suitable environment to give our potential.

NOMENCLATURE

CFD - Computational Fluid Dynamics
DN – Nominal diameter
dp – pressure drop (change in pressure)
Qa – Flow rate
C_v – Flow coefficient
D – Valve Diameter
Pr - Pressure
N1 – constant
Rho – Density
Gf – Specific gravity of Fluid

REFERENCES

[1] David C. Wilcox. "Turbulence Modelling for CFD"

[2] John D. Anderson, JR, 1995. "Computational Fluid Dynamics: The Basics with Applications"

[3] https://en.wikipedia.org/wiki/Computational_fluid_dynamics.

.