

KEB-45250 Numerical Techniques for Process  
Modeling  
Exercise 6 - Fluid flow  
22.02.2018

Antti Mikkonen

## Introduction

Today we will practice basic CFD software usage. We will be using Fluent but most CFD programs work the same way.

The first problem is about as simple as a CFD problem gets and provided with minimal guidance. Ask for help.

The second problem is a classical tutorial case provided by EDR&Medeso. It comes with a complete step-by-step guide.

## Problem 1

Consider a straight pipe with length  $L = 0.5m$  and diameter  $d = 0.02m$ . The inlet velocity is  $V = 5m/s$ . Fluid density  $\rho = 1kg/m^3$  and kinematic viscosity  $\nu = 0.01m^2/s$ .

Calculate pressure drop using Fluent and compare the result with an correlation based solution (maybe with Python)

$$\Delta p = \frac{1}{2} f \rho V^2 \frac{L}{d}$$
$$f = \frac{64}{Re} \tag{1}$$

Check Reynolds number. Do you need a turbulence model?

In Fluent use a area averaged surface monitor to calculate the pressure drop.

The mesh is provided with name pipe.unv in POP. To import it: File->Import->I-deas Universal. Remember to use a axisymmetric simulation.

Visualize your results with contour plots.

Try changing the discretization methods. Does the solution change? Why/why not?

After validating your model for pressure drop, try adding a heat equation to your solution. Set wall temperature  $T_w = 0K$  and inlet temperature  $T_i = 100K$  for nice visualization. Use heat conductivity  $k = 0.01W/mK$  and heat capacity  $c_p = 1000J/kgK$ . Note that the temperature acts as a passive scalar in our incompressible simulation.

Add a mass flow rate averaged temperature for outlet.

Visualize your results with contour plots.

## Problem 2

Complete the WS09\_Vortex\_Shedding from the Fluent\_QUICKSTART\_2\_days\_17.0\_v1-trainee/Day 2/workshop\_input\_files/Fluent provided by EDR&Medeso. The files are available in 0\_siirto directory in O: drive.

Cylinder in cross flow is one of the classics example cases. The tutorial contains many unnecessarily advanced features for us, but is still suitable.

## Extra 1

Try the other tutorials. Try adapting them to something that interest you. It is very difficult to teach how to do CFD. One must learn by experience.