

KEB-45250 Numerical Techniques for Process Modeling Exercise 7 - Turbulence 8.03.2018

Antti Mikkonen

Introduction

Today we will practice basic turbulence modeling with Fluent. We will start with a classic test case with a step-by-step guide provided by EDR&Medeso.

The second problem is a more difficult case with impinging jet heat transfer. The purpose is to show differences between different models. Experimental data is provided but we will probably not get a good match.

The third problem is similar to the project assignment on this course. The home assignment will be incompressible and more simple but with similar mesh and geometry.

Problem 1

Complete the workshop `Fluent-Intro_17.0_WS06_Turbulent_Flow_Past_a_Backward_Facing_Step.pdf` from the `Fluent_QUICKSTART_2_days_17.0_v1-trainee.zip` provided by EDR&Medeso. You can find the package from `0_siirto/turbulence`.

Problem 2

Impinging jets are a simple example of a flow that is difficult to model with CFD. The turbulence models are generally designed for wall aligning flows and fail in the impingement region.

Below, in Fig. 1, we have experimental data from Ercoftac. Our models are likely to be inaccurate. With care, it is completely possible to model an impinging jet with reasonable accuracy, but today we will not have time.

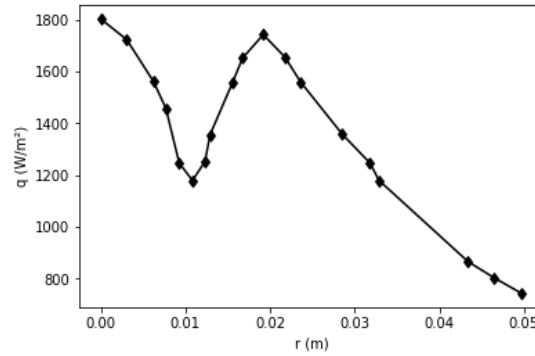


Figure 1: Measurements from Ercoftac

Load a ready made case from 0_siirto/turbulence/forStudents. Study the existing case and try different turbulence models and wall functions. Plot y^+ and Total Surface Heat Flux at wall_glass.

Compare results of different turbulence models with each other and with the experimental data. Why there are large differences between some models and small ones between others.

Extra 1

Try the Fluent-Intro_17.0_WS04_Fluid_Flow_Around_the_Airfoil.pdf workshop provided by EDR&Medeso.