

F TME226, Assignment 2: Turbulent flow using STAR-CCM+

IN this task, a commercial CFD software (STAR-CCM+ 12.06.011) will be used. The task is to do simulation of a two-dimensional hill flow. Several turbulence models will be used and the results will be compared with experimental data. Before doing the task, it is recommended to first do one of the tutorials in the User Guide (Version 12.06) of STAR-CCM+. The tutorial which is similar to this task is the "Steady Flow: Backward Facing Step".

You can do the assignment on your own or in a group of two. It is recommended (but not required) that you use \LaTeX (an example of how to write in \LaTeX is available on the course www page). You find \LaTeX [here](#). You can also use \LaTeX [on-line](#).

F.1 Backward Facing Step Tutorial

This tutorial is a good bridge before doing different cases. Here are some steps to access the tutorial:

1. Open a terminal window. In the terminal window, type starccm+
2. To start a new simulation, click **File** → **New**
3. Tick the Power-On-Demand box and fill the license box with the POD Key.
4. Download the tutorial instruction and data from the course homepage

F.2 2D Hill Flow

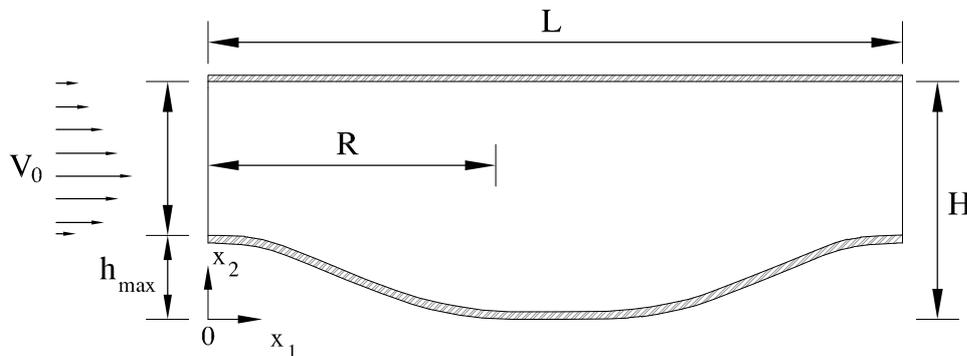


Figure F.1: flow over two consecutive hills

In this case, a two-dimensional, steady and incompressible flow over two consecutive hills mounted on the bottom of the channel will be studied. The height of the channel is $H = 151.75 \text{ mm}$. The maximum height and length of each hill are $h_{max} = 50 \text{ mm}$ and $R = 192.8 \text{ mm}$, respectively. The space between each of the consecutive hills is $9.0h_{max}$. The fluid is water of 20° Celsius and the Reynolds number ($Re = 37000$) is based on the mean centerline velocity at inlet and the inlet channel height. The inlet boundary condition (left boundary) is imposed as the velocity profile of the