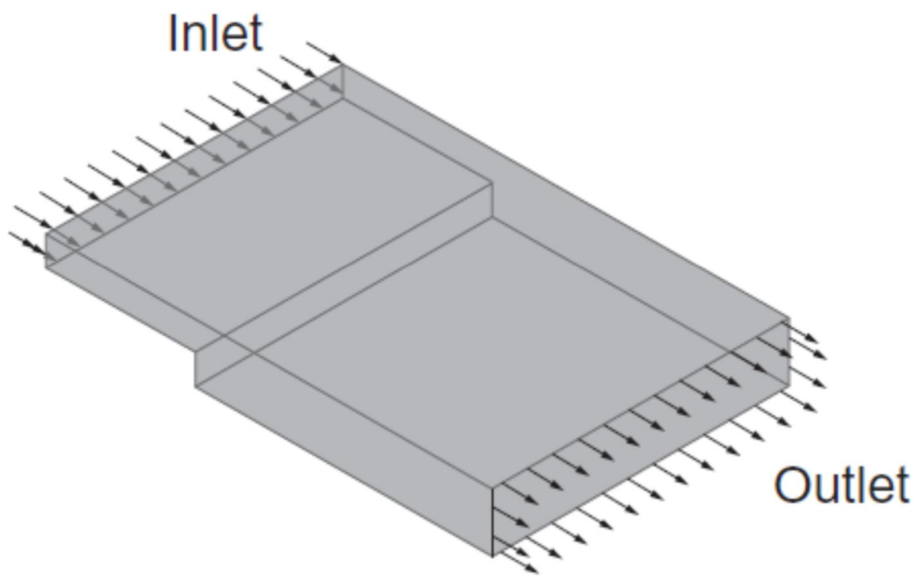
Introduction

The backward facing step is a benchmark problem for validation of CFD modeling techniques. Flow over the step undergoes separation, reattachment and recirculation. Turbulence models have been seen to give widely varying results for this problem, and the results are also sensitive to the numerical method and grid resolution.



Source: Computational Fluid Dynamics – A Practical Approach

In this homework assignment, you will conduct a CFD simulation of turbulent flow over a backward facing step using the Fire Dynamics Simulator (FDS) software (Version 6.6 or 6.7 – current in-use versions). The aims of this assignment are to:

1. Generate a 3D geometry and mesh in FDS (0.1 m to 0.2 m nominal cell size).
2. Set up monitors in the FDS model to record results and simulation diagnostics.
3. Simulate turbulent flow over the step and monitor results to demonstrate that a steady state solution has been reached.
4. Post process the results (via Smokeview and Excel) and compare with experiment data.
5. Check sensitivity of results to the geometry (domain width), grid resolution and turbulence model.
6. Document results in a short report.

Data and Model Set Up Instructions

Geometry for the step is provided in the figure below. The Reynolds number for the flow is defined as:

Re = ρ* U * H / μ

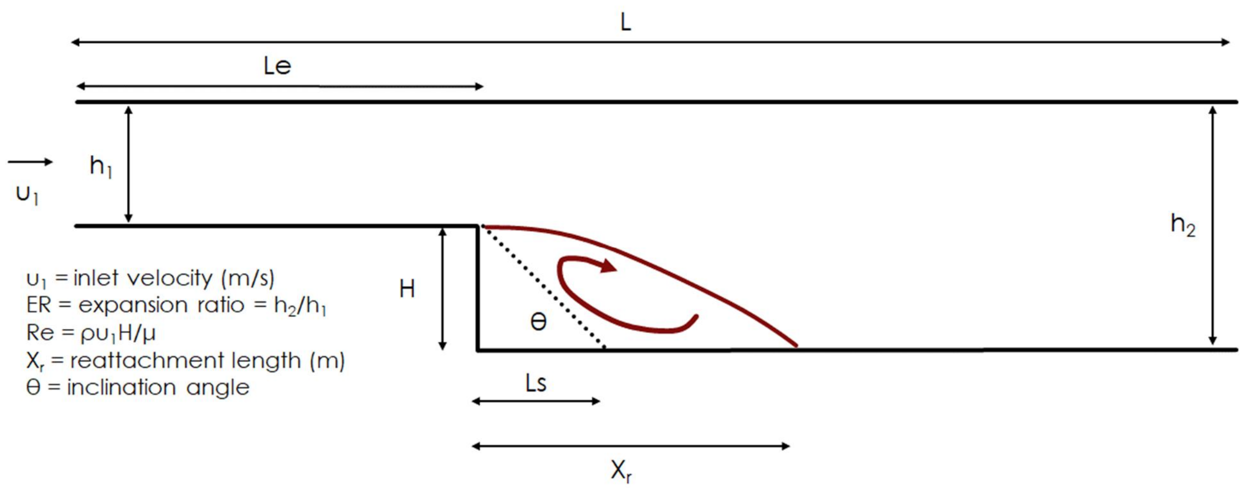
ρ = density (kg/m^3), nominally 1.2 kg/m^3

U = upstream bulk velocity (m/s)

H = step height (m)

μ = fluid viscosity ($N/s/m^2$), nominally 1.8e-5 Ns/m^2

The Reynolds number for this problem is 15,000. Expansion ratio (ER) for the geometry should be 2.0. Width of the geometry should be at least $2h_2$.



Download the following file from Courseworks and use it as a basis for building your model: CFD-03-10.fds

The CFD model inlet length prior to the step (Le), and outlet length downstream of the step, are up to the individual to determine (you should state why you used a certain inlet and outlet length). The length should be chosen such that boundary conditions are far enough away from the expansion to not influence the result of interest. You should choose a grid resolution that balances computation time versus resolving the flow adequately (hint: if your model takes more than about 60 minutes to run, the grid resolution is much too fine, and a mesh of around 100,000 cells should run reasonably quickly, but also note that you want to have at least five cells across the domain width and height).

Your simulation will be transient; you will need to choose a run time long enough to allow the flow to develop and settle to a pseudo-steady state. FDS uses large eddy simulation turbulence modeling, which resolves large scale turbulent motion, hence it is three-dimensional and transient (you should monitor the velocity at a point the domain and total volume flow to confirm this steady condition is achieved).

Experiment data are for this problem are taken from published data[1]. For this assignment, the key piece of experimental information to use is the reattachment length. In FDS this can be computed through visualization of the time-averaged component of x velocity (the location where the velocity is 0 is the reattachment length); visualization is accomplished with a contour slice of x velocity along the duct centerline and in Smokeview the slice files can be averaged in time. Tick marks can be placed to help visualize the exact length. Refer to file “Assignment 3 – FDS Basics.pptx” for a demo of how to do these two things. Reattachment length for this problem, based on test data is around 8.5H downstream of the step.

Submission Requirements and Instructions

1. Introduction and problem statement
    a. Geometry (base case parameters) – physical dimensions including step height, inlet duct height, domain length chosen downstream of the step, and domain width. Provide reasoning for the domain length chosen downstream of the step.
    b. Mesh resolution (show an image of the mesh / geometry, quote number of cells, note whether any special approaches were needed to generate the mesh – single grid or multiple grids).
    c. Velocity boundary conditions for the CFD model (show working for computation of inlet velocity).
    d. Submit your base-case model (as a text file, for information only in case of a result not making sense).
2. Run the model with FDS default parameters used (same as case CFD-03-10):
    a. Submit a graph of velocity versus time at a point in the domain and total volume flow rate to demonstrate that steady state is reached. Comment on what indicates steady state and beyond what time.
    b. Submit a mean (time-averaged) x velocity contour, label the recirculation zone and state why a certain averaging time was used (such as visual observations, integral flow time, etc.). Mark the recirculation point and approximate length (report the length in terms of step heights).
    c. Report the pressure loss through the geometry, compute K factor based on velocity before the step (ignore friction loss at the wall – make it part of the K factor you compute, show working).
3. Sensitivity analysis (summarize how each model change affects reattachment length, in step height, and pressure loss in a tabular form):
    a. Adjust grid resolution (coarser grid) – keep everything the same as above but make the grid coarser by a factor of 2 (not that each duct must have at least 5 cells across the height or width), report the reattachment length and pressure loss, and compare with the result from 3c.
    b. Adjust turbulence model – keep everything the same as 3a, but make the turbulence model “DYNAMIC SMAGORINSKY”, report the reattachment length and pressure loss, and compare with the result from 3a.

[1] Flow separation over the inclined step, Ruck and Makiola, Physics of Separated Flows—Numerical, Experimental, and Theoretical Aspects, Vieweg+Teubner Verlag (1993): 47-55.

   c. Adjust domain width – keep everything as per 3b and make the domain wider by a factor of 2, report the reattachment length and pressure loss, and compare with the result from 3b.
   d. Compare the results with respect to reattachment length sensitivity.
4. Document results in a short report/summary that responds to the questions 1 through 3 given above.

Assessment

Total value: 10% of final grade (10 points – see below for breakdown)

Assessment:

1.	Introduction and problem statement	3.0
2.	Base case runs	3.0
3.	Sensitivity analysis	3.5
4.	Presentation/report	0.5
	Total	10

Work is to be conducted individually. Be descriptive as possible in your answers. Highlight results. Organize text clearly and neatly. Clearly state all assumptions. Label and properly refer to figures, tables and equations presented (0.5 marks (out of 10) of the assignment grade is assigned for presentation).

Due date: October 21, 2019 at 6 pm (0.5 mark deduction for each day of delayed submission)