To predict transitional and fully developed turbulent flow in a backward facing step geometry

This was original a workshop at the COBEE 2015 conference in China

Peter V. Nielsen and Yuguo Li

Background

It is known from experiments and simulations that low Reynolds' number effects may take place in room air flow. We know that this type of flow is difficult, to predict in details by CFD due to limitations of available turbulence models in connections with RANS equations, limitations of computer capacity and limitations in other types of methods as e.g. the use of LES. Errors due to choice of numerical methods, choice of the right physical model of the engineering problem you solve, choice of relevant boundary conditions and errors due to user's experience are also a part of relevant problems in doing CFD predictions.

Significant development and improvement has taken place in the CFD community in the last 30 years and, at least, some of the above mentioned problems have been addressed. It is therefore interesting to see how far we are today collectively with the relevant software and hardware.

Invitation

We invite you to participate in a workshop which will be finalized at the COBEE conference in July 2015.

The purpose of this unique workshop is to focus on errors existing in CFD simulations, especially in connection with low turbulent flow, and discuss what we can do, using a simple method.

We present a simple CFD problem (with no benchmark or experimental data for comparison) for all participants of the workshop to simulate using well-known and widely used commercial CFD software. We ask you to send your CFD results to us, and we shall consolidate all the results, and present in the workshop.

Consider this problem was a request from your client – your client is interested to have an accurate prediction of the penetration length for the whole Reynolds number range from zero to 10,000.

All participants who have submitted the simulations are invited to join, and we will discuss together on the following topics

1. How large are the differences in CFD results? How significant are the differences? Why are there differences? What can we do to minimize potential CFD errors?

2. How are the different schemes able to handle the low turbulent flow in the selected case? How can we find/define the limits for use of different turbulence models?

3. What are the challenges in applying CFD to a problem? How to build confidence in CFD simulations?

The CFD problem

We propose a simple CFD problem that is easy for all to simulate even on your PC. It is incompressible and two-dimensional in the laminar regime and perhaps in the fully turbulent regime, but not necessarily in the transition regime. It may be considered as a simple building ventilation problem in the turbulent regime, and it is described as The Backward-Facing Step or Sudden Expansion Flow in the fluid dynamics community. The flow is isothermal.
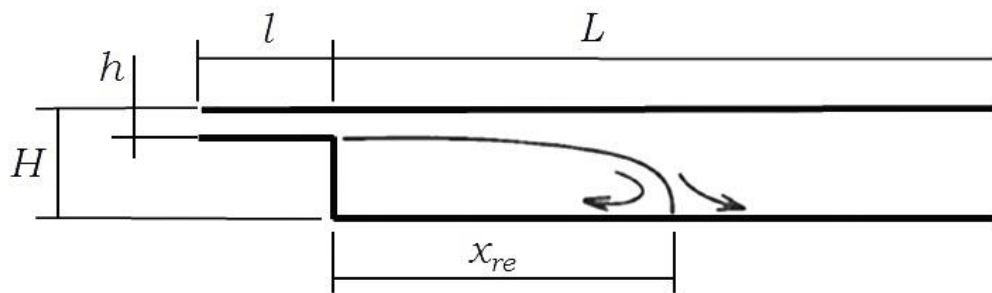

Figure 1. The geometry to use in the test case.

Figure 1 show the geometry of our proposed case. The flow is typical for isothermal room air flow in deep rooms. *H, h* and *l* are room height, supply slot height and length of supply opening.

L is the length of the model/room (note that this length should be sufficient otherwise, it may affect your prediction), and x_{re} is the length from the end wall to the location where the reattaches flow is separated in a flow back to entrainment into the wall jet and a forward flow towards the exit (i.e. reattachment point). The length x_{re} is referred to as the penetration length of the supply jet.

The following dimensions should be used:

> *h/H* = 1/5

> *l/h* = 4

> *L* should have a sufficient length without influencing the obtained penetration length x_{re}/H.

x_{re} is the distance to the first reattachment in the flow (there can be more in the case of laminar flow).

The flow might be transient and three-dimensional in a certain range of Reynolds Numbers (we can at least not exclude the possibility). We will therefore also define the test case as a 3D geometry with:

> *W = 2H*

We consider the velocity and the penetration depth in the 3D case in the vertical median plane at $y = 0.5W$.

The inlet flow is a Top Hat profile with constant velocity u_o everywhere in the profile.

The Reynolds number is:

$Re = (h \cdot u_o)/\nu$

Where ν is kinematic viscosity.

Requirements at the COBEE 2015 conference

You are invited to participate in a workshop where you predict the flow in the geometry from very small Reynolds numbers with laminar flow to high Reynolds numbers with fully developed turbulent flow. The range should be from Re = 1 to Re = 10,000. It is especially interesting to see if it is possible to handle the low Reynolds number regime or at least a part of it.

We ask you to select a CFD code (commercial code or your own development) and we ask you to select an appropriate turbulence model. Alternative you could also use LES. You are free to select the boundary conditions.

You are reminded to take a special care for the Low Reynolds number region. If your code or software tells you that your chosen turbulence model may not work for a particular region, please report it in your results. Do also tell us if your code gives any other warning.

You may record the effective dynamic viscosity at the recirculation centre, and plot the number of iterations as a function of the Reynolds number. These will be useful for interpretation of our results.
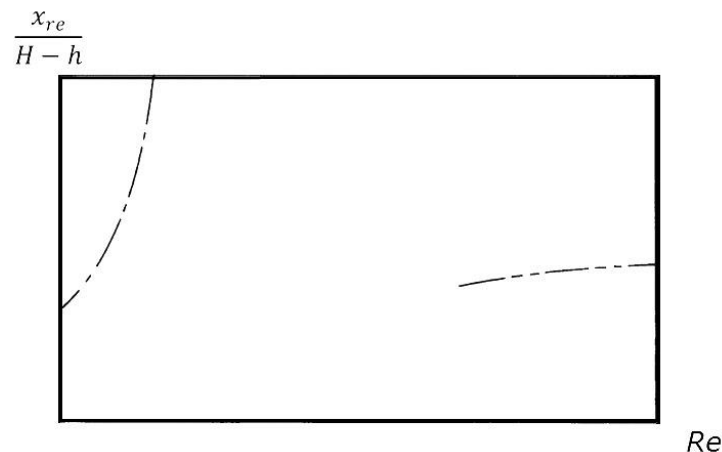

Figure 2. An illustrative graph for penetration length from Re = 1 to 10,000, as predicted by CFD. Note that this is just an example, hence no scales are given.

It is shown in Figure 2 that we may not obtain reliable prediction for low (not very low) Reynolds numbers (not fully developed turbulence region). You may find out what you can do for this low Reynolds number region. Do not connect results predicted by two different turbulence models in the final Excel scheme, or connect laminar predictions with predictions made by a turbulence model (which is also avoided in figure 2 in this instruction).