# ES440 Computational Fluid Dynamics

Dr Yongmann M. Chung
School of Engineering, University of Warwick, UK
September 2012

## Introduction to Star-CCM+ software – Session 3

### Mesh Independence

### Previously in CFD lab 2…

As you remember, so far we have learned:

- How to import SolidWorks geometry
- How to create a mesh and change mesh parameters
- How to set-up a physics continuum
- How to set-up convergence criteria
- How to define boundaries and their conditions

The purpose of this session is to teach you about how to choose an adequate mesh. As computing power is expensive (time-wise) it is important for us that the solution is as accurate as possible within the quickest possible time. Once this is achieved, the solution is said to be grid independent, i.e. if we made the mesh finer, the solution would still be more or less the same so being independent of the mesh.

The aim for today is to generate several different meshes, and to evaluate the mesh variables

### Checking Grid Independency

There are a number of ways to check that a solution is grid independent. Personally, I prefer to monitor some flow property of interest and note when the solution for that flow property remains the same for finer grids.

For your work, I would like you to monitor two variables. Firstly, the pressure distribution on the bump surface is of importance as it shows how the flow is accelerating/decelerating and can signal the onset of flow separation which is usually of importance to us. Secondly, I would like you to monitor the wall shear stress magnitude on the bump surface as this provides a measure of the skin friction drag being generated on the bump as well as providing another signal of flow separation (recall for your fluids courses that a boundary layer will separate once the wall shear stress reaches zero).

## Reloading Last Week's Work

As with the previous session, there are a number of tasks to be completed although, as at the end of session 2, these will be deliberately quite vague to get you thinking for yourself.

- **TASK 1** - Open up your work from last week and make sure you completed all the tasks (especially Task 13 about displaying the pressure distribution on the bottom).
- **TASK 2** - In a similar way to the pressure distribution, display the wall shear stress distribution on the surface.
- **TASK 3** – Change the x-momentum stop criteria to a 1e-4 and continue running, you will have to switch off the maximum number of steps stopping criteria
- **TASK 4** - Output pictures of these results for later comparison (Hint- You can use a screenshot if necessary but you could always try right clicking on the plot and looking under Hardcopy)

## Generating the new grids

In Star-CCM+ it is relatively easy to generate new grids although it may be necessary for you to start from scratch if you previously had to delete certain representations of the mesh. Do not worry, as you have already set-up one simulation, it shouldn't take you too long to set up another one and setting up another grid simply requires you to entire a different mesh density when you get to that part.

- **TASK 5 -** Clear the previous solution (the option is under solution) and clear the previous mesh (The option is under mesh), Run the simulation with same mesh parameters as before but with the grid base size changed to 0.5 m
- **TASK 6** - Run the simulation until the results have settled down (either the residuals or the flow properties, you decide!).
- **TASK 7** - Again, output pictures of these results for later comparison.
- **TASK 8** - Repeat steps 4-6 but for a finer grid with a base of 0.25 m. I have run this simulation for a 1500 steps already and you can download it from the website to run its called **bump25.sim**, run for another a few hundred steps. (to open the residual plot and scenes find them on the simulation tree and right click>open to view them, **do this before you run them!)**

- **Task 9** – Download **bump15.sim** from the website this has a base size of yes you guessed it 0.15m. Run this simulation for 20 further steps you should note the computational expense of the simulation

- **Task 10** – Download **bumpmix.sim** from the website this has a base size of 0.3m but the relative minimum target size is now 25% run for another 20 steps

**Note:** *For **bump15.sim** and **bumpmix.sim** we have let the solutions fully converge, have a look at the residual graph to see the outcome of running your simulation till convergence is met*

## Conclusion

What you should notice is that the grids get finer the shear stress results at the top of the bump get slightly higher while they get slightly lower at the back of the bump. You will notice with the pressure distributions that the pressure is slightly lower at the top of the bump and slightly higher at the front.

So what does this tell you, well with the finer grids the results do get more accurate but is it worth the extra computational time? As always, an optimum has to be made. You will have also noticed from your observations that apart from the area around the bump the results for all the cases are much closer together. The simulation with a grid size of 0.15m is too slow and 0.25 m grid size could be considered a bit slow as well. The **bumpmix.sim** simulation is a good compromise though as it allows a slightly finer mesh around the bump, while giving a coarser mesh downstream, saving computational time. In your assignment you should use relative minimum target size intelligently to get a finer mesh around areas of interest while keeping the computational time acceptable.

The other option under the surface size parameters 'relative target size' merely changes the percentage of the base size that the mesher actually makes the mesh to. So if the base size is 0.1 m and the relative target size is 25.0, then the mesher will make elements to a base size of 0.025 m.

Once again, if you have any problems at all, please ask for help. However, for your own benefit, please have a good go yourself first!

## Domain size

As equally important as mesh independence is domain independence. The domain should be large enough that it does not alter the results significantly yet not so large that it becomes computationally expensive. As in the case of the mesh size an optimum must be found using the same trial and error technique as used for mesh size.

For example, for external flow simulations make sure that there is a large enough gap between the inlet and the object that the inlet is not affected by the consequent high pressure field in front of the object, the same goes for the pressure changes that could affect the outlets, and if the gap between the object and side walls is not great enough then a ‘channelling of the flow’ could occur affecting the results.