ES440 Computational Fluid Dynamics

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

Introduction to Star-CCM+ software – Session 2

External Flow Simulation

Previously in CFD Star-CCM+ lab …

As you remember, so far we have learned:

- How to start Star-CCM+ (by clicking on the bright yellow start icon).
- How to simulate a simple internal flow.
- How to set relevant simulation parameters.
- How to set boundary conditions.
- How to run the simulation.
- How to stop and continue (rerun) the simulation.
- How to analyse the results (flow visualisation).

Introduction

The purpose of this session is to give you an idea of the procedures that need to take place in order to set-up any CFD simulation. You will be given a generic geometry file and be expected to follow the general steps given in order to set-up the simulation for running.

From this session onwards, we will be moving away from prescriptive tutorials on the exact steps you need to take in order to set-up a simulation and we will be aiming to get you to create a simulation from scratch.

If at any point you are unsure of how to complete a specific step, first look into the Tutorial help files for an explanation. If you are still confused, please ask for help from one of the lab demonstrators.

Setting up a simulation

The main steps involved in running a simulation are:

1. Define and import geometry.
2. Assess geometry and remesh if required.
3. Creating Continua and Setting Solver Options.

4. Define boundaries.
5. Apply boundary conditions.
6. Set convergence criteria.
7. Run simulation.
8. Post-process simulation results.

Each of these steps will be presented in more depth and discussed.

1. Define and Import Geometry

The CCM+ environment does not have an in-built geometry creation facility, instead relies on the import of any geometries required. For the purpose of this session, we will use the import facility within CCM+ to read in a standard CAD model and mesh.

TASK 1: Importing the Geometry

- Download the **Solidworks** file '**lillebumpforccm.sldprt**' from the es440 website.
- Once downloaded, open in **solidworks**, inspect its dimensions and save as a **parasolid**(*x_t) file so that it can be used in **Star-CCM+**.
- Open **Star-CCM+**, go **File>New Simulation** and select serial for the **run mode** and click ok.
- Then go **File>Import Surface** and select the saved parasolid, and then click open.

A note should be made that there are a variety of CAD formats that can be read into **Star-CCM+**. Students are free to use any format they wish for the purpose of their coursework but by experience it is found that using the **Parasolid** (*.x_t) format causes the least problems.

2. Assess geometry and remesh if required

From the previous step, you should have a geometry file input and displayed within **Star-CCM+**. The problem now remains, however, that the mesh may not be suitable for a CFD simulation. In general, we like the mesh to avoid very long and thin elements (as these can generate dodgy coefficients within the equations) and prefer generally square (or equilateral) elements.

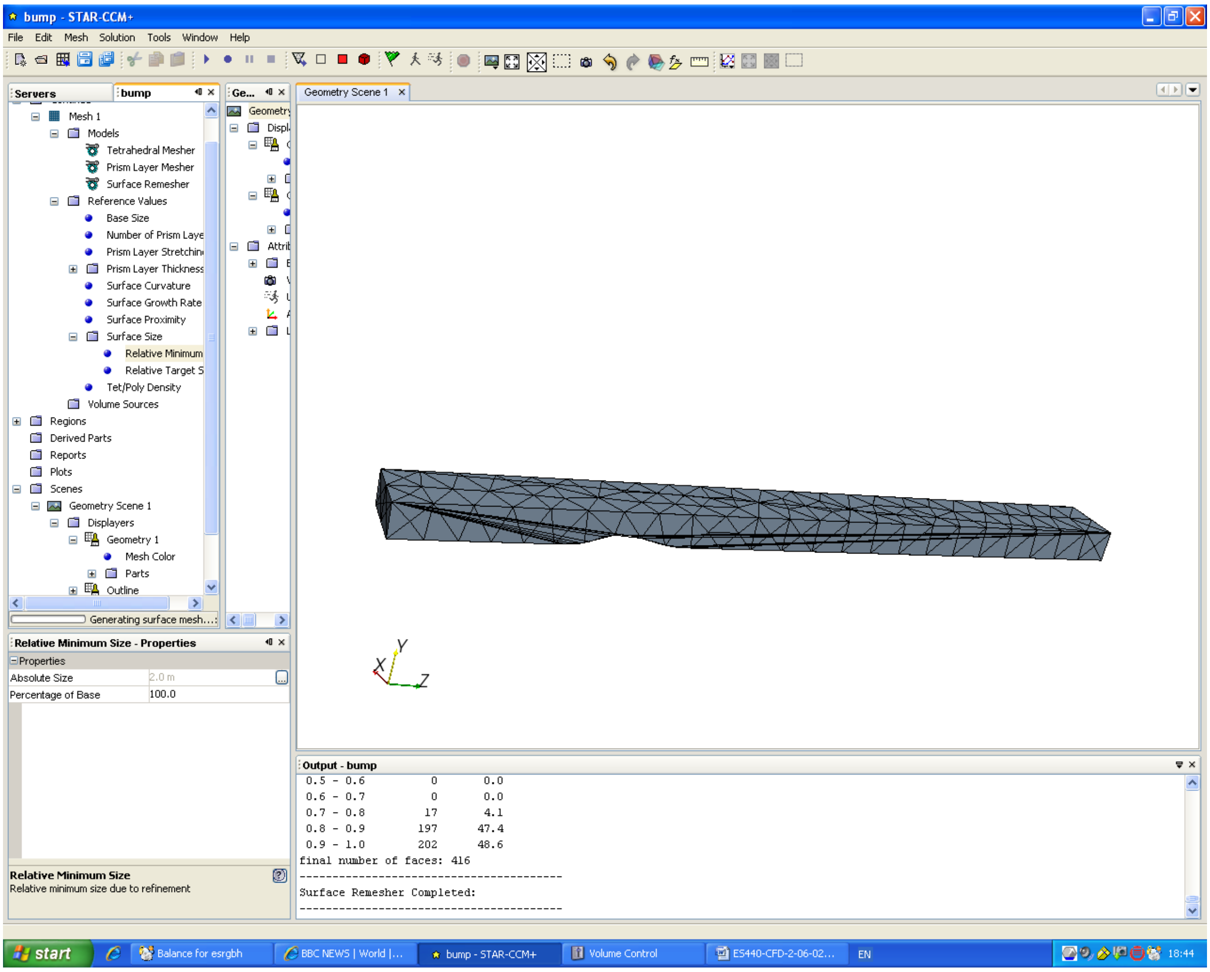
If this is not the case, it may be necessary to remesh the surface before we generate the volume grid (which will be the grid we actually use for the simulation.)

You should have noticed that from viewing the model on **solidworks** that the part is 23x1x2 metres in dimension

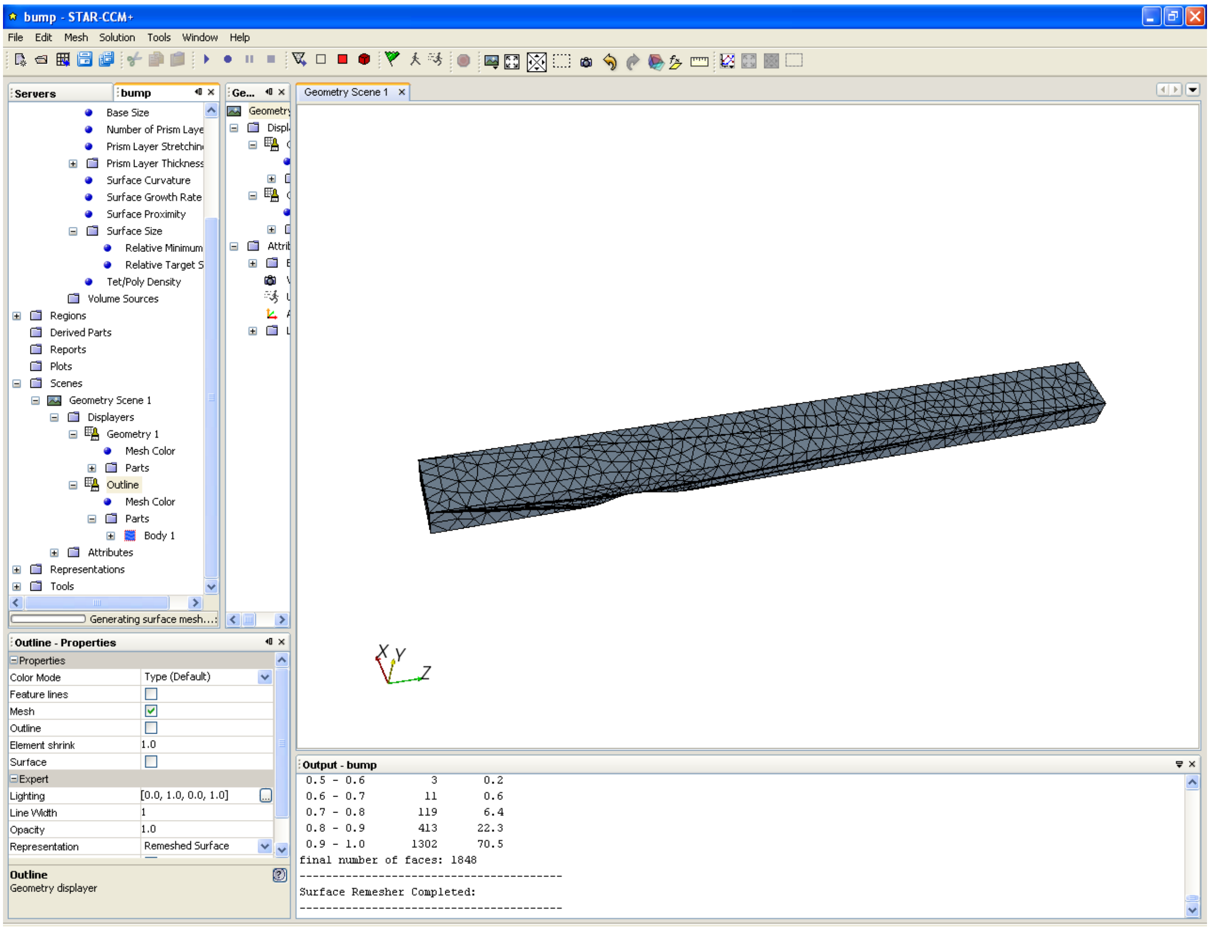
TASK 2: Surface Meshing

- Right click on **Mesh 1** under the **continua** folder in the simulation tree and select **meshing models**.
- For this simulation use Surface Remesher, Tetrahedral Mesher and Prism Layermesher.
- Under **Mesh 1>Reference values**> set base size at 2.0m, no. of prism layers at 3 and prism layer stretching at 1.5

- Under **Mesh 1>Surface size**> set relative minimum target size at 100.0 percentage of base.
- Then on the top tool bar press the red square button to generate a surface mesh.
- Now under **Scenes>Geometry Scene 1> Outline-Properties>** deselect outline and select mesh, and remeshed surface under the representation scroll bar, you should have a scene similar to this.

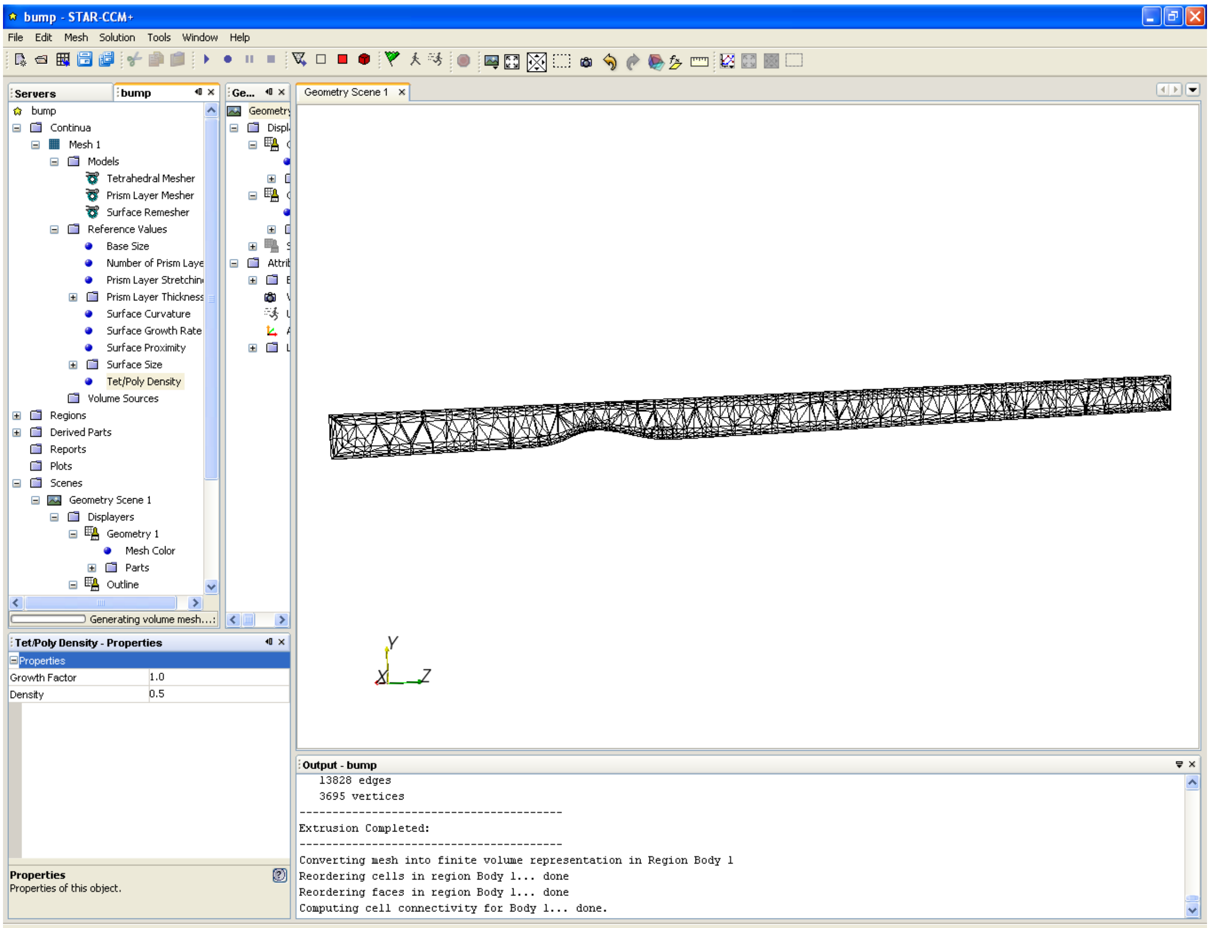


- What do you think? Well it's a bit coarse and could do with some refinement to improve the answer.
- Change the base size to 1.0m and minimum target size to 50%, this effectively reduces the minimum mesh size from 2.0m to 0.5m
- On the top toolbar go **Mesh**>clear generated mesh to delete the current mesh and then press the red square button to generate a new surface mesh.
- View the mesh and you should end up with something like the image below. It is a bit finer than the previous mesh and will do for the purpose of this exercise.



<u>TASK 3:</u> Generate a Volume Mesh using the Volume Mesh tool.

- Under **Mesh1>tet/poly density** change the Density to 0.5
- Press the red cuboid button in the top toolbar to generate a volume mesh.
- Then using the planar viewing techniques shown in the first Star-CCM+ session, generate a lengthwise cross sectional planar view of the volume mesh. It should look something like below.



- Now change the density to 1.0, clear the mesh solution, and generate a new volume mesh. Don't worry about the surface mesh as Star-CCM+ will automatically generate a surface mesh before creating a volume mesh if no surface mesh currently exists.
- The new volume mesh should be a bit more refined with more elements.

  N.B. Meshing will be covered in greater detail in Session 4.

3. Creating Continua and Setting Solver Options

At this point, you should have the geometry input and meshed up ready to begin the actual problem definition. Before defining the boundary conditions, we need to define a continua and solution parameters.

A continuum (in the context of Star-CCM+) is a region of continuous physical characteristic. For example, in our case, the air within the tunnel is a single continuum. If we were to have a block within the flow, that area would be a solid continuum.

We already have a mesh continuum defined by the surface and volume meshes. You will need to create a new Physics Continuum (I.e. an area in which some particular laws of physics apply).

TASK 4: Create a new physics continuum (call it 'Air Flow') and select the models to be applied within this continuum.

Right click continua and select new>physics continuum

- A physics continua called physics 1 should now appear under the continua folder in the simulation tree re-name this 'Air Flow'
- Right click on 'Air Flow' and select models
- Select the following:-
    - Three Dimensional
    - Gas material
    - Coupled flow
    - Stationary motion
    - Ideal gas
    - Steady flow
    - Turbulent regime
    - Spalart-Allmaras model

More will be explained regarding turbulence modelling in session 4)

4. Define Boundaries

At present, our domain is one homogenous block with no separately defined faces or regions. A region in Star-CCM+ is an area of the domain which has applied certain physical conditions in that either it is regarded a solid, fluid or porous. A region can have a multitude of faces each one able to have its own boundary conditions e.g. pressure outlet or velocity inlet.

As it is unlikely that anyone would want to model an isolated flow without any inlet or outlet, it is now necessary to define the faces of the domain and so separate the domain surface into faces ready for the application of boundary conditions.

To do this, we will use one of the '**Split**' tools.

TASK 5: Using the '**Split by Angle**' tool, divide the single boundary into its constituent faces.

- In the simulation **tree >Regions> Body1>Boundaries>Boundary 1>** right click and select '**split by angle**' tool.
- You should change the split angle from the default of 89 to 15 degrees.
- You should now have 6 faces/boundaries.

  (Hint: You may need to delete all Representations other than the Volume Mesh in order to use this tool.)

TASK 6: Rename all of the boundary faces as appropriate.

- Rename the faces in a similar fashion to how they were named in simulation in the first Star-CCM+ session.
- The flow is running length ways with the inlet the side closet to the bump.

Useful tips: You can split the model into multiple faces when importing the model at the start, select one boundary per face rather than one boundary for all faces.

5. Apply Boundary Conditions

Now that the boundaries have been created, it is necessary to define the boundary conditions.

TASK 7: Apply the boundary conditions.

- As shown in Star-CCM+ session 1, apply the boundary conditions.
- As previously mentioned all faces (apart from two) will be solid walls so will need to be set as '**Wall**' boundary conditions.
- One face will be an inlet and here we will set the velocity. Therefore this face will need to be defined as a '**Velocity Inlet**' condition.
- Finally, we will set the outlet as being at atmospheric pressure so this face will be defined as a '**Pressure Outlet**' condition.

  (Hint: Keep an eye on the properties window below the main tree.)

TASK 8: Set a velocity at the inlet.

- This is done exactly the same way as in Star-CCM+ session 1. Set the velocity a 1.0m/s.

  (Hint: You will need to look under the Physics Values option for the relevant surface)

Useful tips: With complicated models you could have easily a hundred faces or more, for management ease you can combine faces that have the same boundary conditions by holding **ctrl** for one or **ctrl+shift** to select a group of faces from the simulation tree (regions>body>boundaries)and then right click>combine)

6. Set convergence criteria

One measure of convergence of a simulation is the magnitude of the residuals. The residual defines the difference between the answer for the previous iteration and the answer for the current iteration. If the residual reduces in size, this shows that the answer between each iteration is closer and closer. As the simulation converges, each iteration will be more or less the same so the residual will be small.

However, the actual value for the residual is meaningless as we do not know the actual answer (that's why we're running the simulation!). Therefore, as a general rule, we aim to get the residuals down below 10^{-4} from the start of the simulation. This indicates that the residuals at the end of the simulation are around 0.0001 the level of the residuals at the beginning of the simulation.

It should be noted though that convergence does not equal a correct answer! There will always be inaccuracy within a simulation and the final answer may be wrong even though it is well converged.

TASK 9: Set the simulation stop once the X-momentum residual is less that 10^{-3}.

- Go to simulation tree>Stopping criteria>Right click>create from monitor>x-momentum
- X-momentum criterion>minimum limit 10^{-3}.

N.B. In your assignment you will probably want a lower residual stopping criterion than this, this stopping criteria should take ten minutes for the simulation to solve and is hence being used due to the shortness of the lab session.

7. Run simulation

The simulation is now ready to be run. This can be done by hitting the 'running man' icon on the tool bar. A plot of the residuals will appear and the simulation will commence.

It is nice, however, to create a scalar plot that can be viewed during the simulation. This one of the novel features of Star-CCM+ which allows a simulation to be viewed in real-time as the solver runs.

TASK 10: Create a new Scalar displayer by taking a plane section down the centre of the channel Use this plane to display velocity magnitude contours.

- As in Star-CCM+ session 1, create a new plane section by producing a new derived part.
- Add this plane section to the parts within a new scalar displayer scene and remove any other parts.
- Select a smooth filled plot.

TASK 11: Run the simulation and keep an eye on the contour plot until you think that the answer has settled down.

8. Post-process simulation results

Once the simulation is finished, the results can be plotted to display a variety of fluid properties. You have already produced a velocity contour plot on the centre plane. See if you can complete these final tasks.

TASK 12: Produce a scalar displayer, presenting the Wall Shear Stress contours on the bottom wall (the one with the bump on)

- Create a new scalar scene.
- Remove all parts from the parts list apart from the bottom wall (the wall with the bump).
- Select wall shear stress magnitude and a line plot.
- You may have to run the simulation for one more iteration by pressing ‘step’ looks like the ‘walking man’ to make the plot appear.

  (Hint: You will need to create a new Scalar Displayer, and select the bump surface as the part to display.)

TASK 13: Produce a line plot of the pressure on the bump surface

- Plots>Right click>Create a new xy plot
- Selecting the bottom wall as the part to be used
- Select scalar under x-type

- Changing the scalar value to position z-component
- For scalar value under y-type select the pressure
- Notice the large pressure drop at z=-6

On the next session...

- Grid refinement

Don’t miss out!