ES440 Computational Fluid Dynamics

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

Introduction to Star-CCM+ software – Session 1

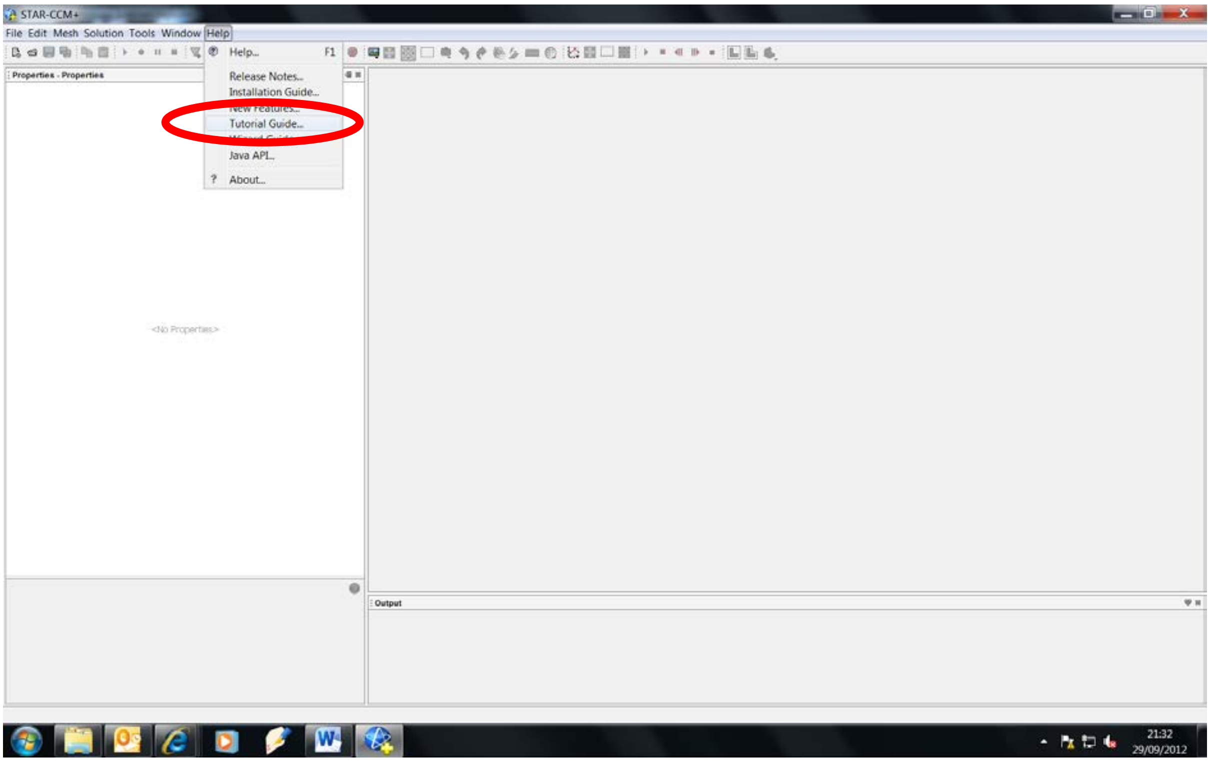
Internal Flow Simulation

How to start

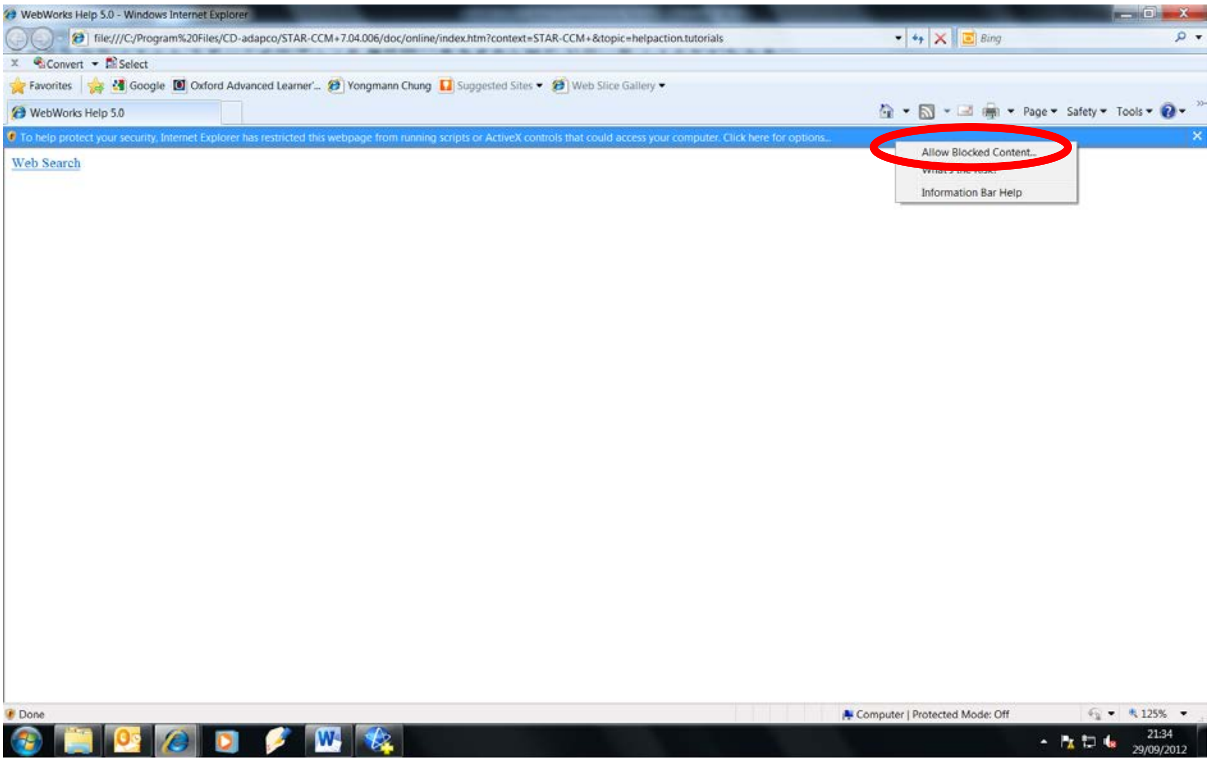
Double click on the ‘**STAR-CCM+(v.7.04.006)**’ icon on the desktop.

You should get the start-up screen as shown below.

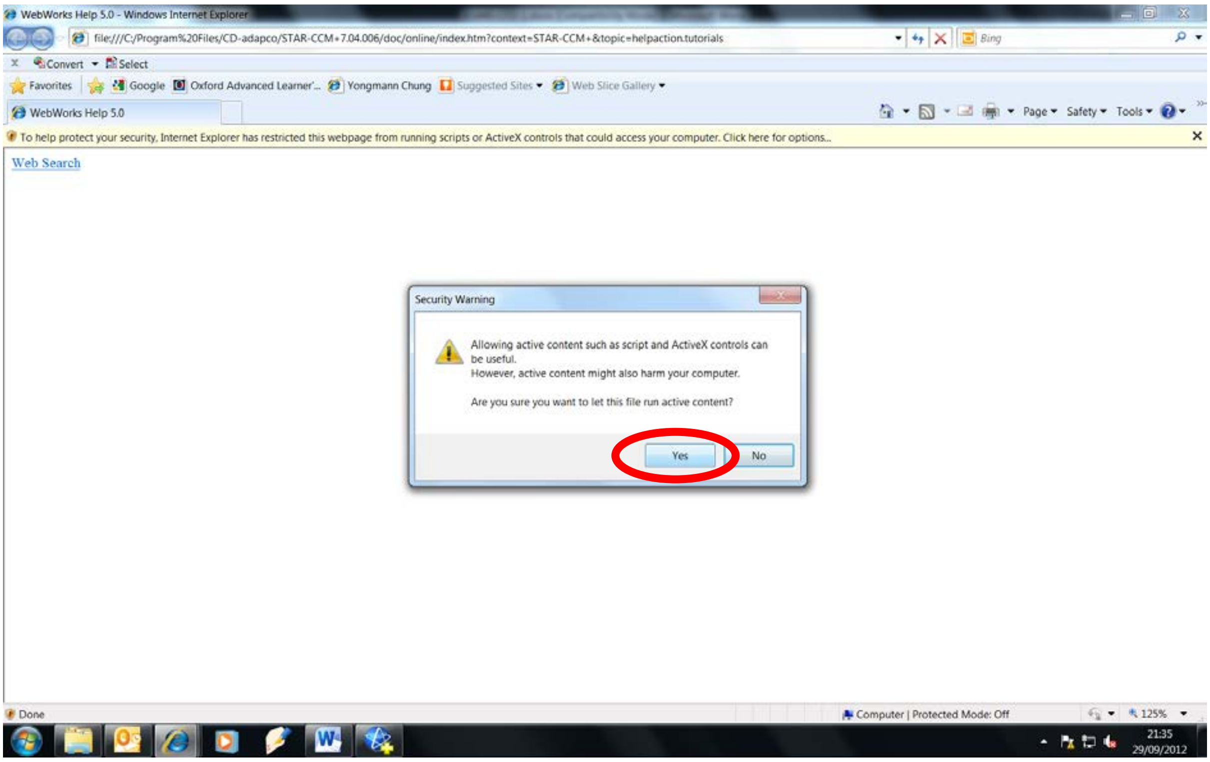
Once it has started, click on the '**Help**' button on the toolbar at the top as shown below. Then, click on '**Tutorial Guide**' to access the tutorials.



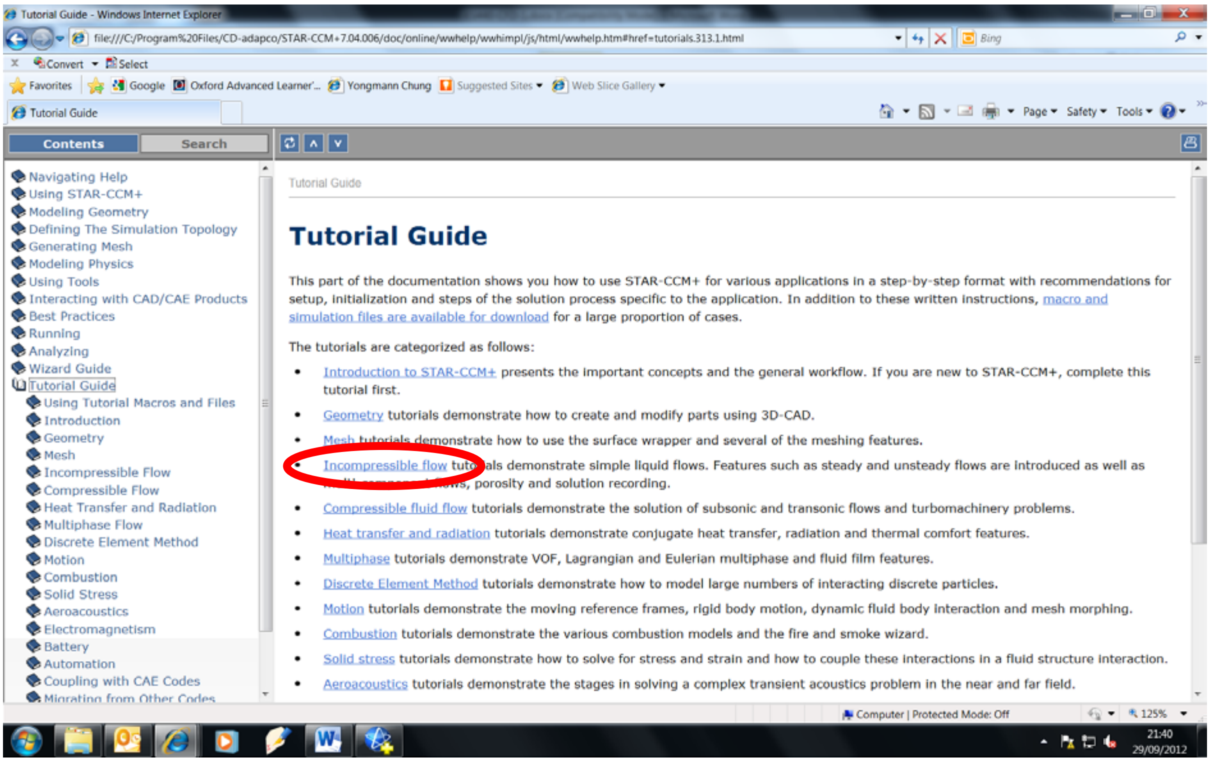
If the help file does not display (the default CADLAB settings may be to block all pop-ups), right-click on the yellow banner at the top of the page and choose '**Allow Blocked Content**'.



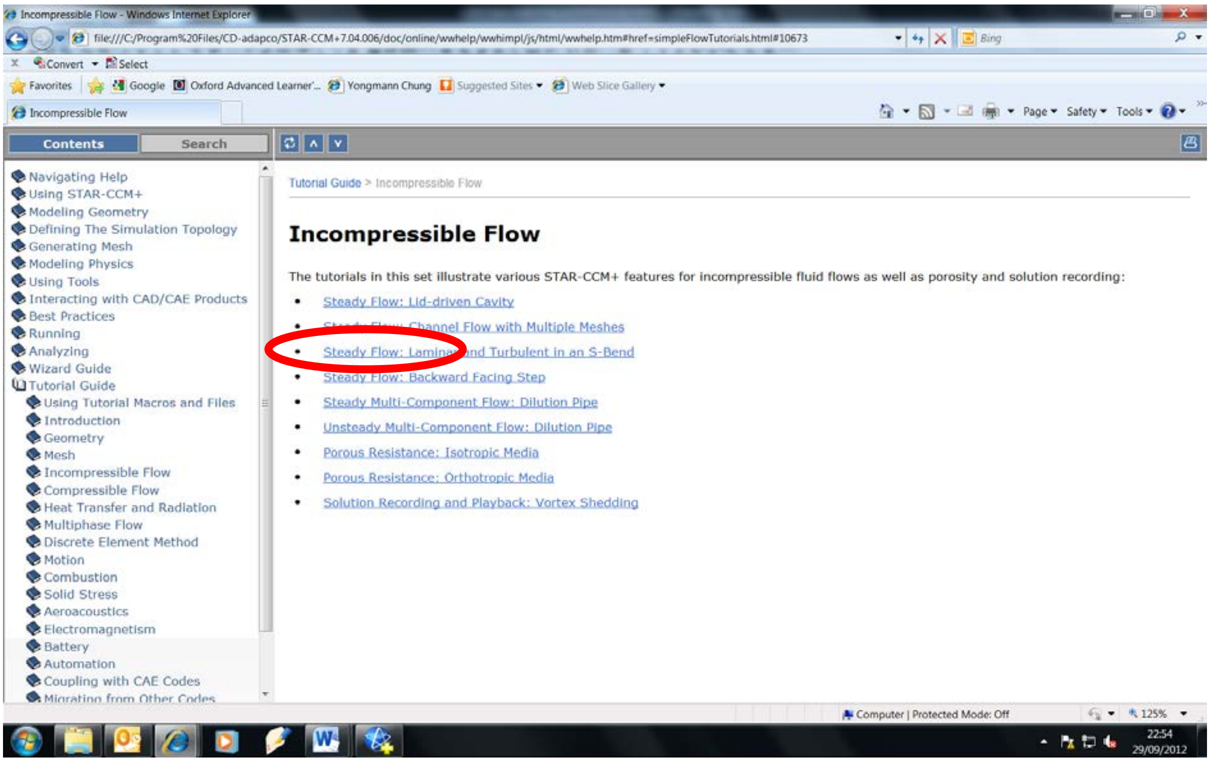
Then, click **Yes**.



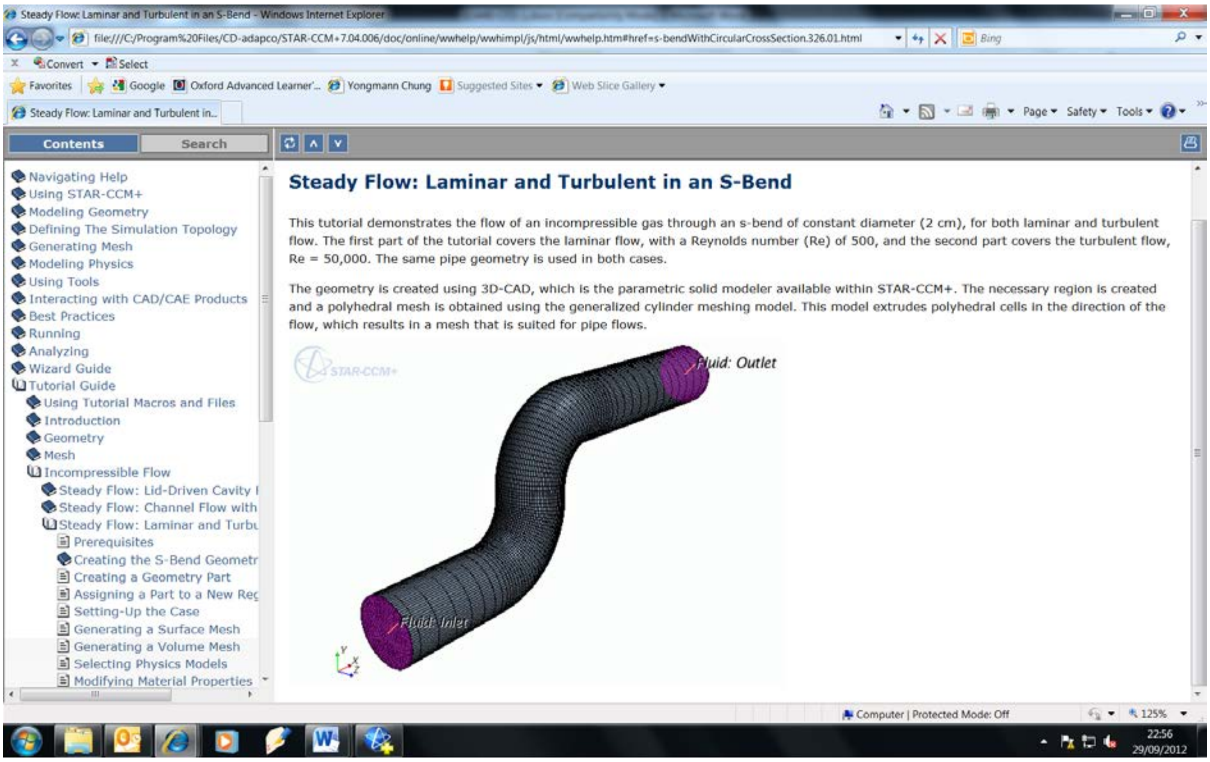
The Tutorial Guide screen should look like the screen below. Choose '**Incompressible flow**' for now, but later try more tutorials relevant to the project.



Then s ‘**Laminar and Turbulent in an S-Bend**’ tutorial.



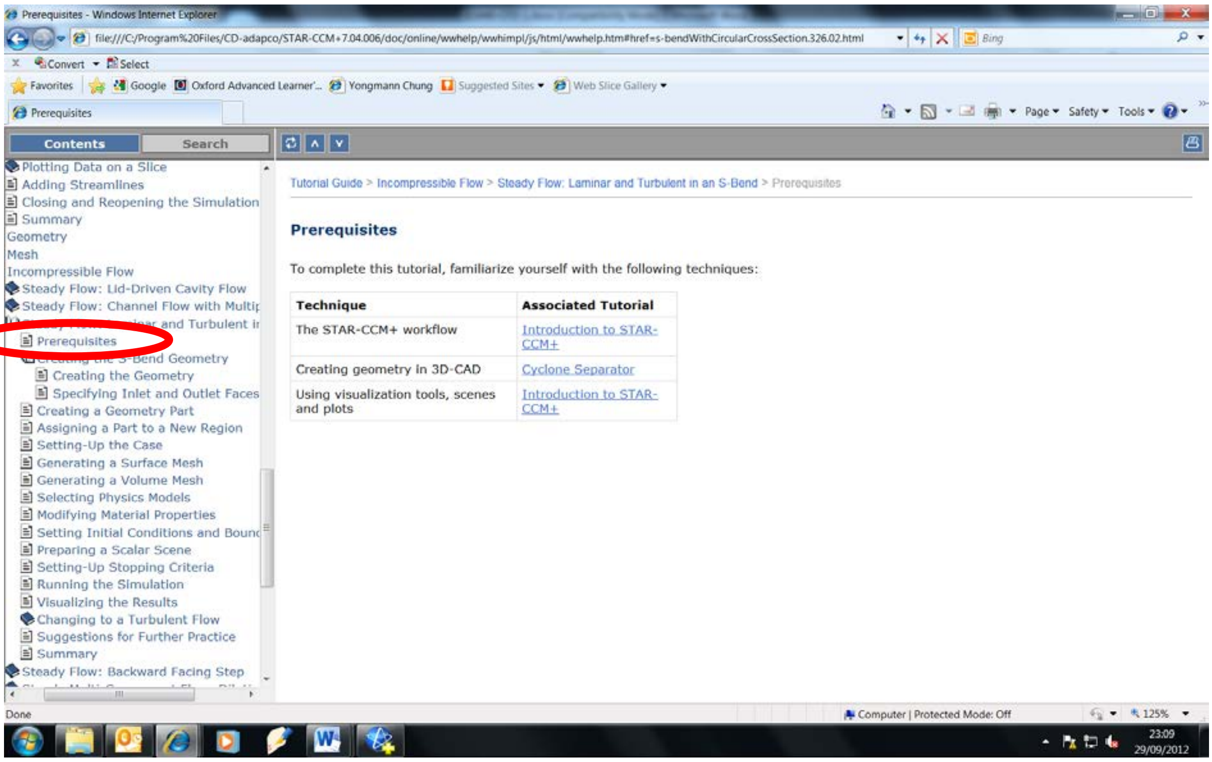
This is for a curved pipe, but for now we will use a simple straight pipe instead.



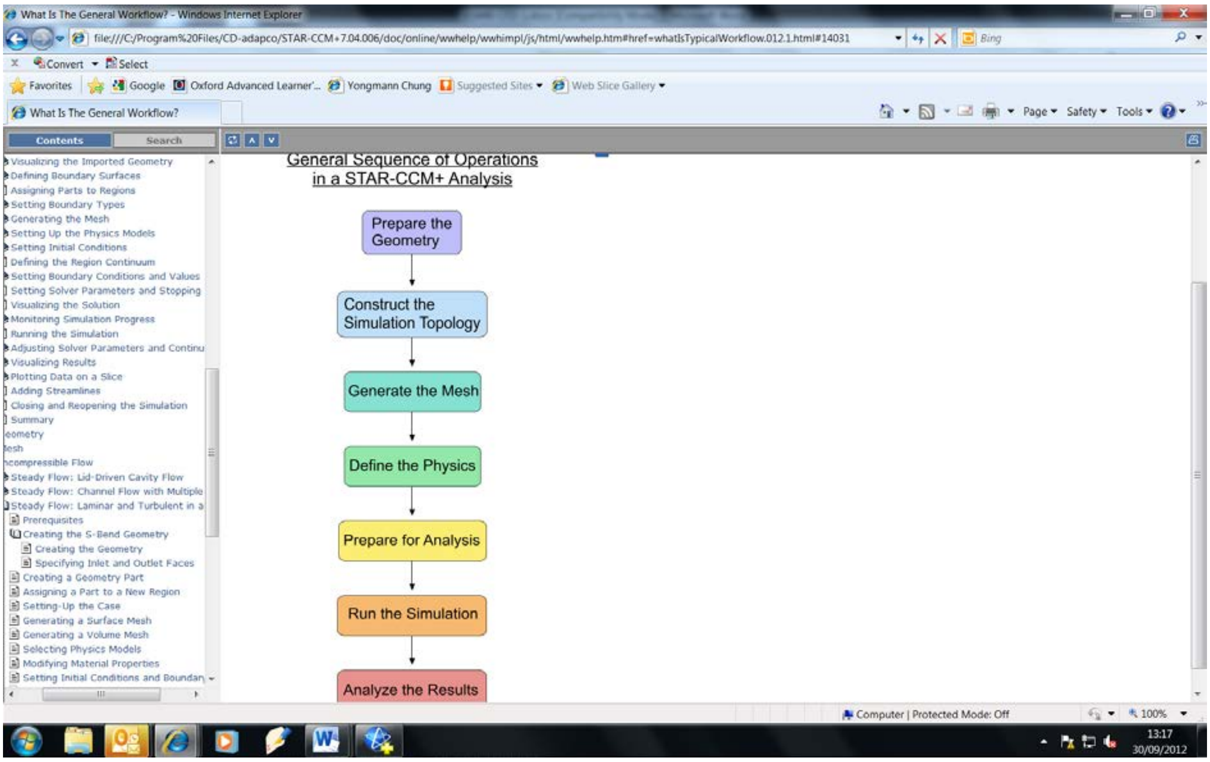
There are many steps on the left menu, and we will proceed step-by-step.

Step 0: Prerequisite

- Try '**Prerequisite**' on the left menu. Do not worry about the '*Cyclone Separator*' part.

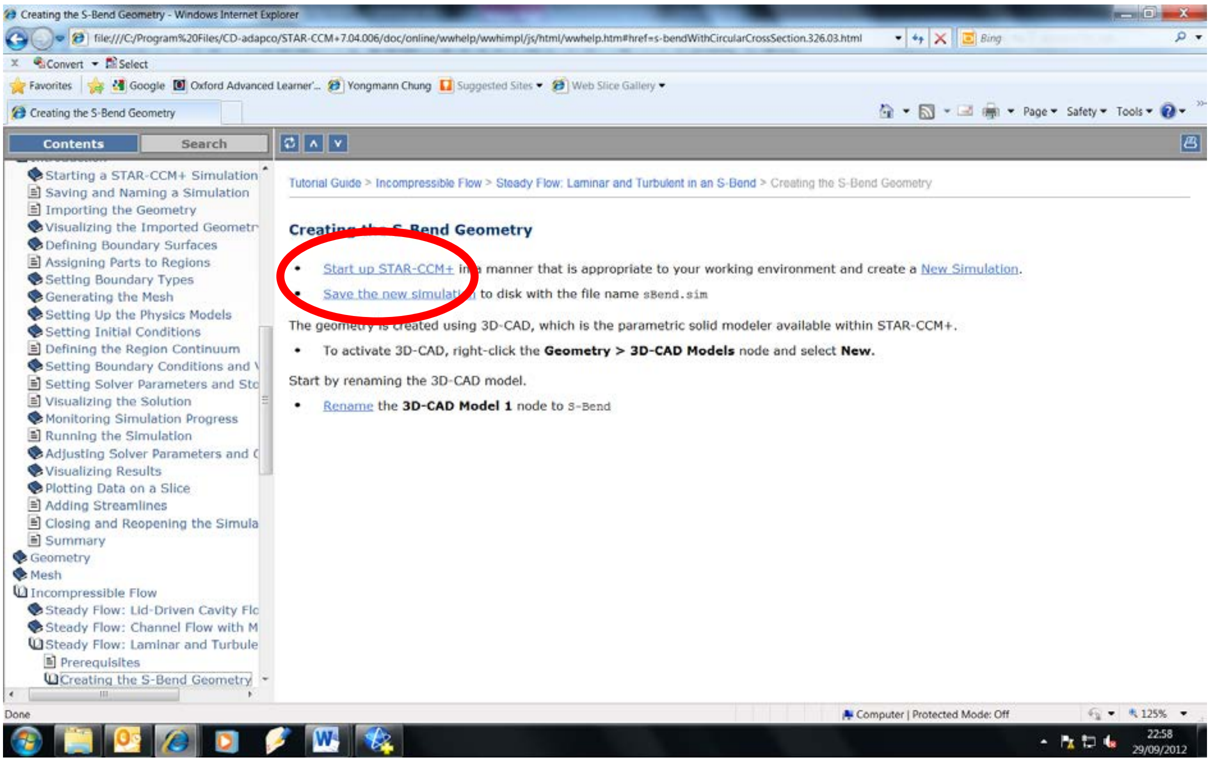


- Be familiar yourself with the **General Sequence of Operations in a Star-CCM+ Analysis**.

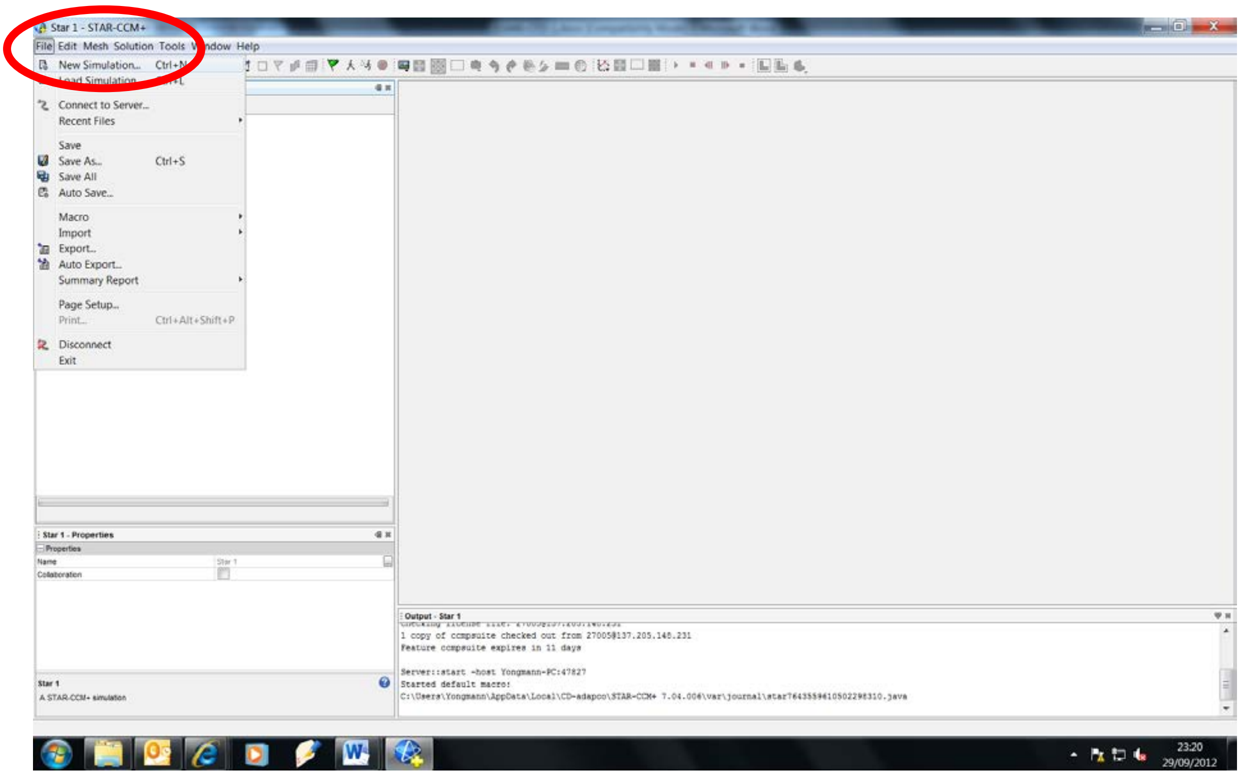


<u>Step 1</u>: Prepare the Geometry

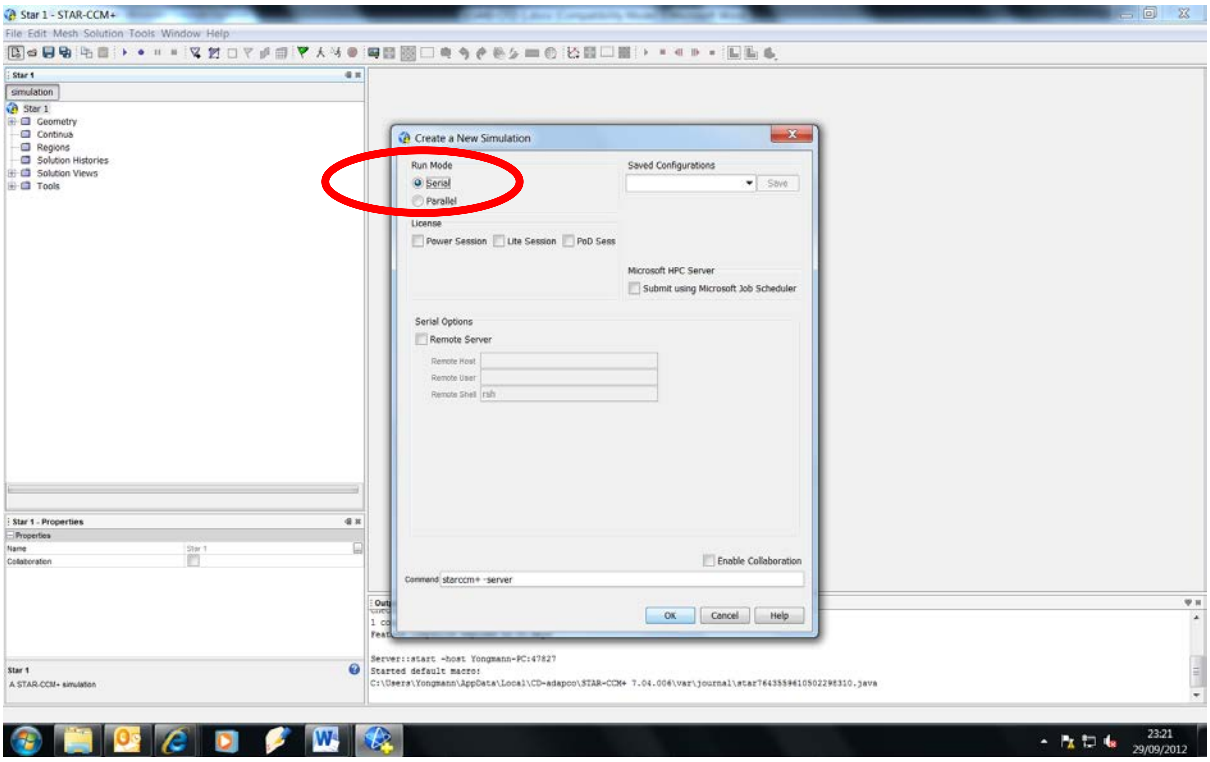
- Now try '**Creating the S-Bend Geometry**' on the left menu.
- We will try ***only*** the first part because the geometry will be created in **SolidWorks**.



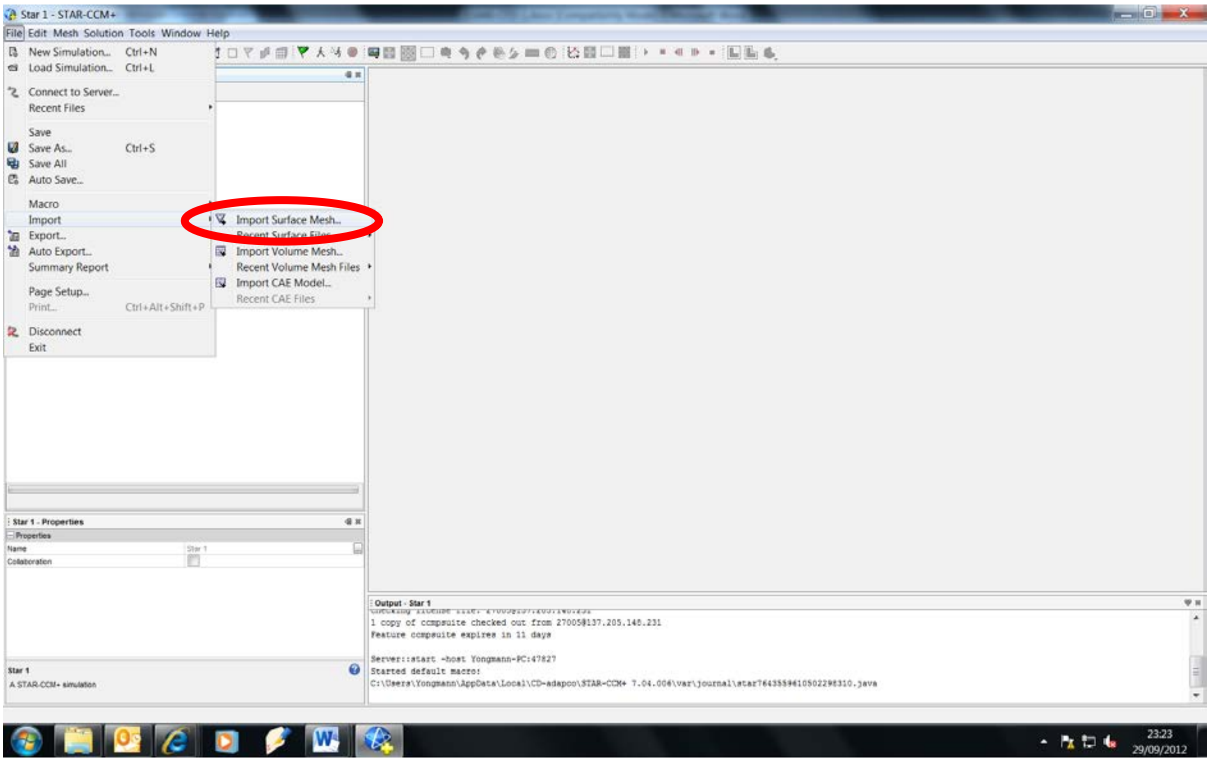
- Start a new simulation.



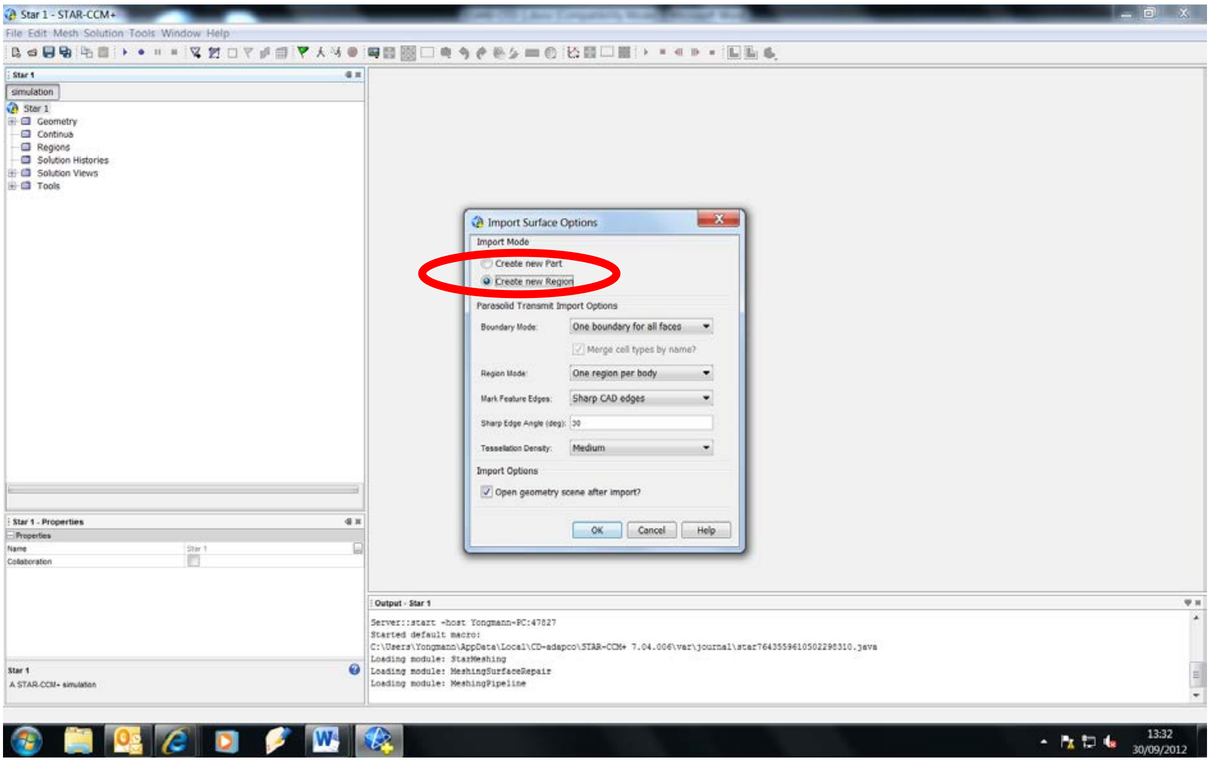
- Choose the **Serial** Run Mode, and click **OK**.



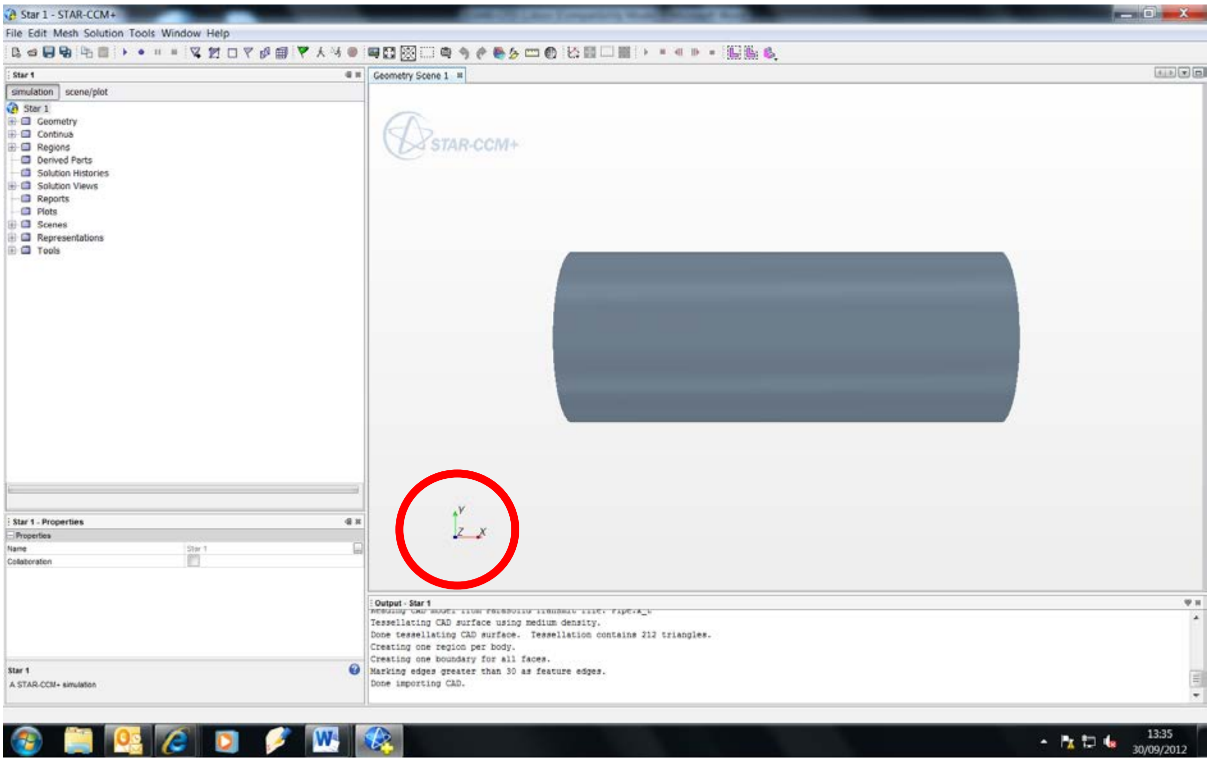
- Now, download the geometry file '**pipe.x_t**' from the ES440 website.
- Then, choose '**Import Surface Mesh**' to import the geometry file.



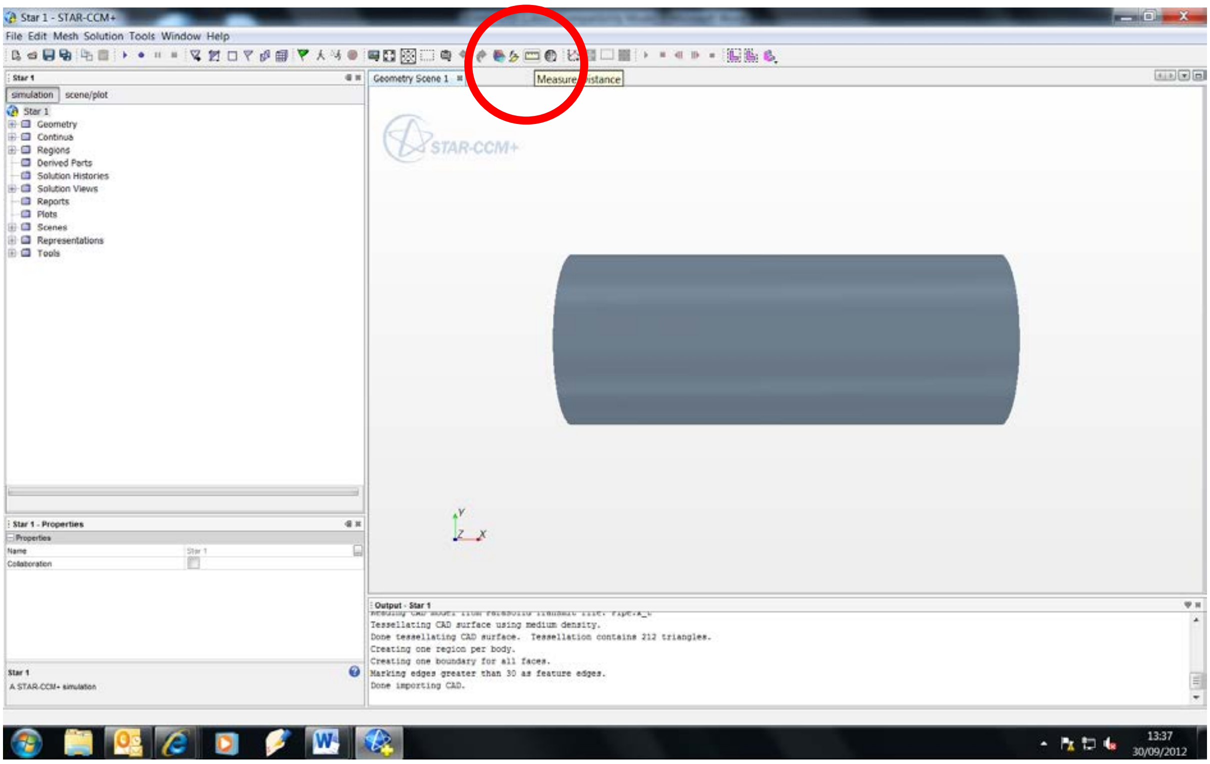
- Select '**pipe.x_t**' file. Please note this is in **parasolid**(*.x_t) file format of **SolidWorks**.
- Choose '**Create new Region**' Import Mode, and click **OK**.



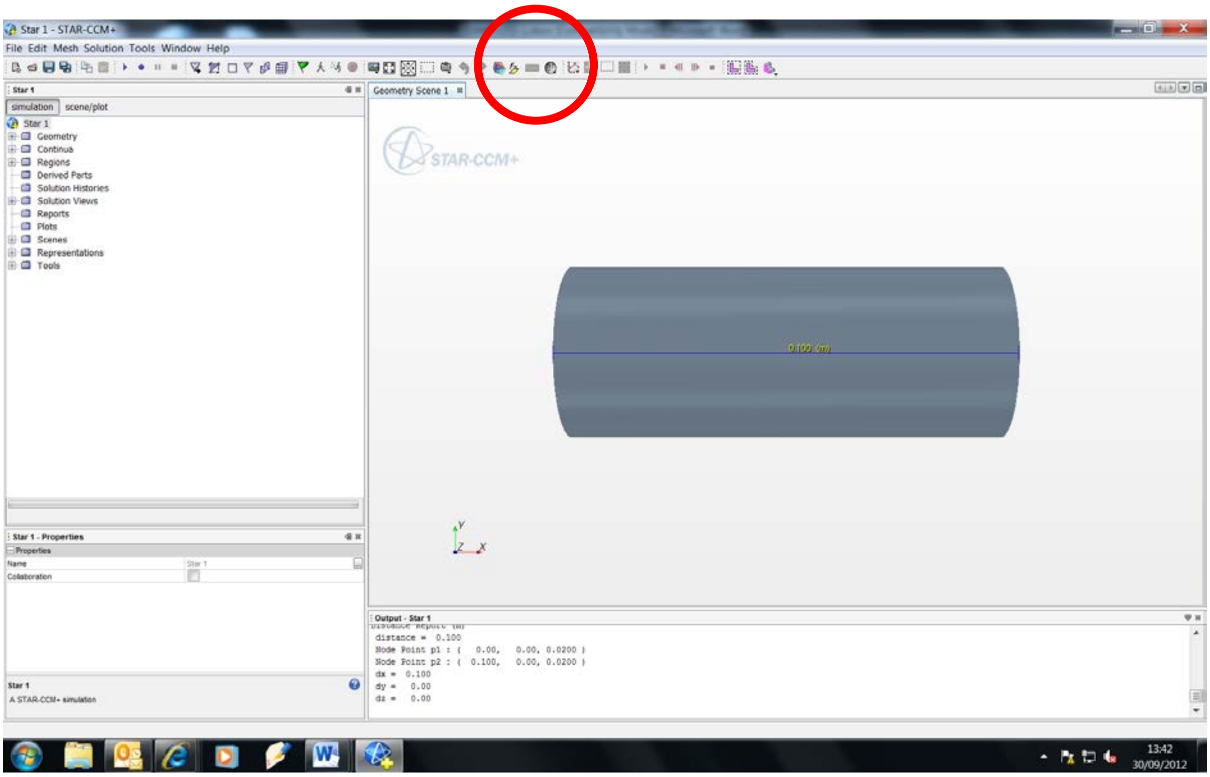
- First, we need to check if the geometry file has been imported correctly.
- Check the coordinate system, to confirm that the pipe flow is in the x direction.



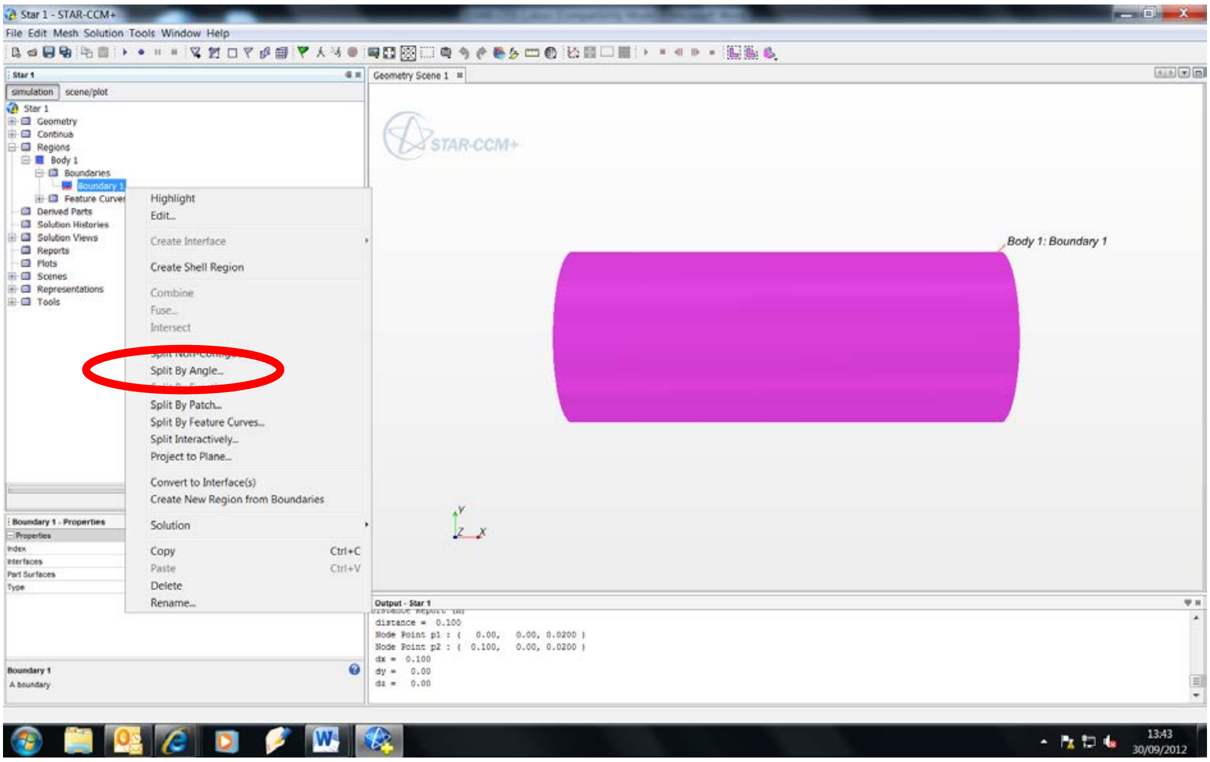
- Click the ‘**Measure Distance**’ button to check the dimension of the model.



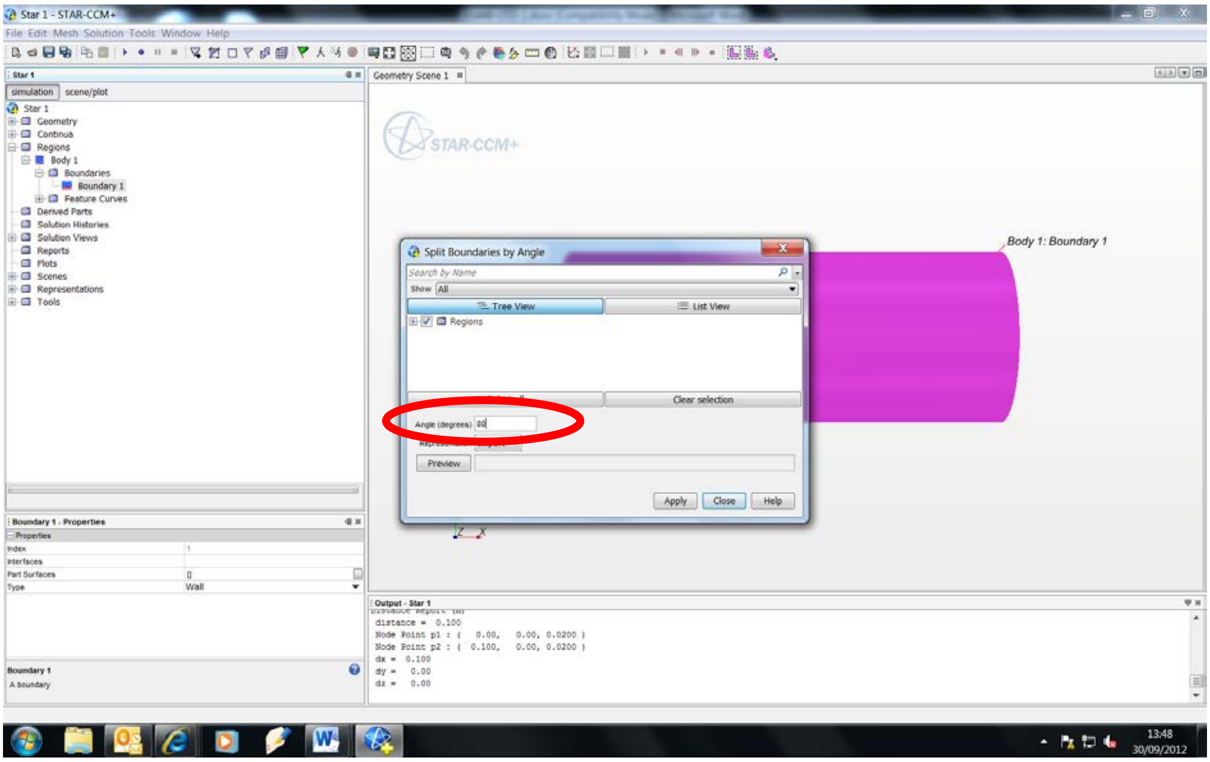
- Click on one end, and drag the cursor to the other end.
- The length of the pipe should be 10 cm or 0.1 m.



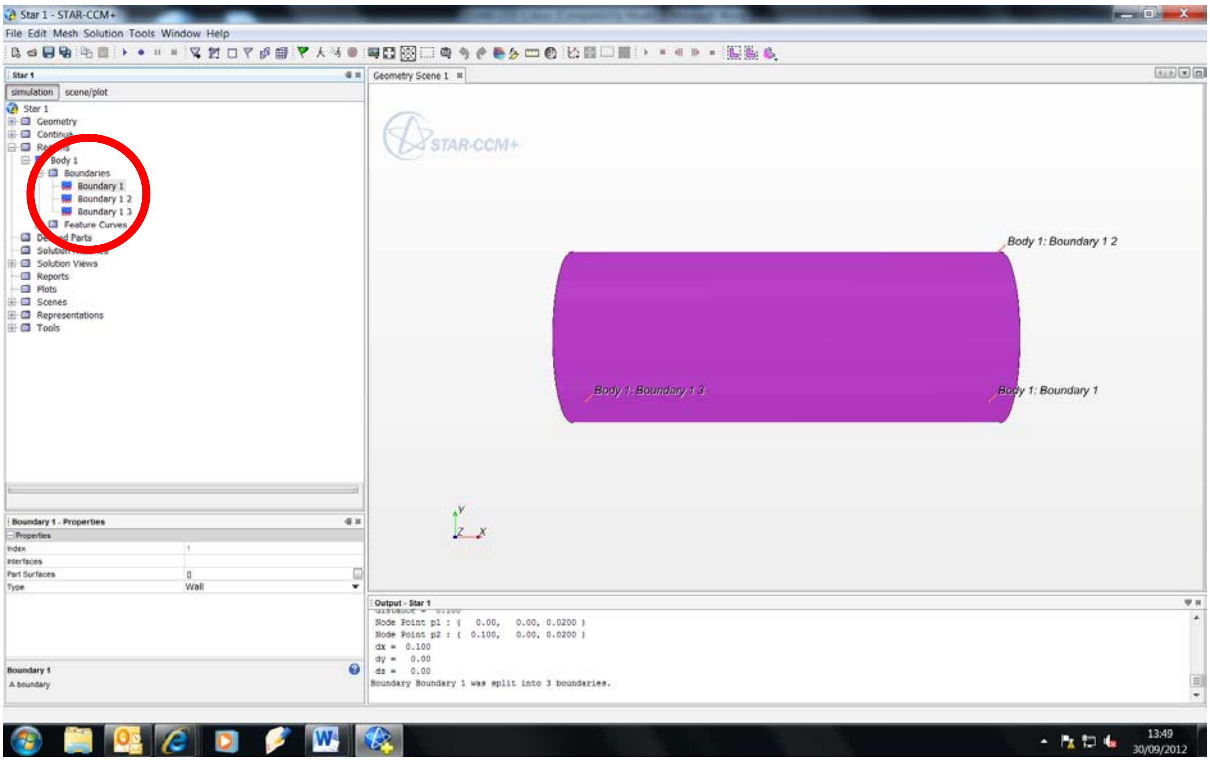
- Now, we need to define/split the boundaries for inlet, outlet, and pipe wall.
- Go **Regions>Body 1>Boundaries>Boundary 1**.
- Then, right click on **Boundary 1**, and choose '**Split By Angle**'.



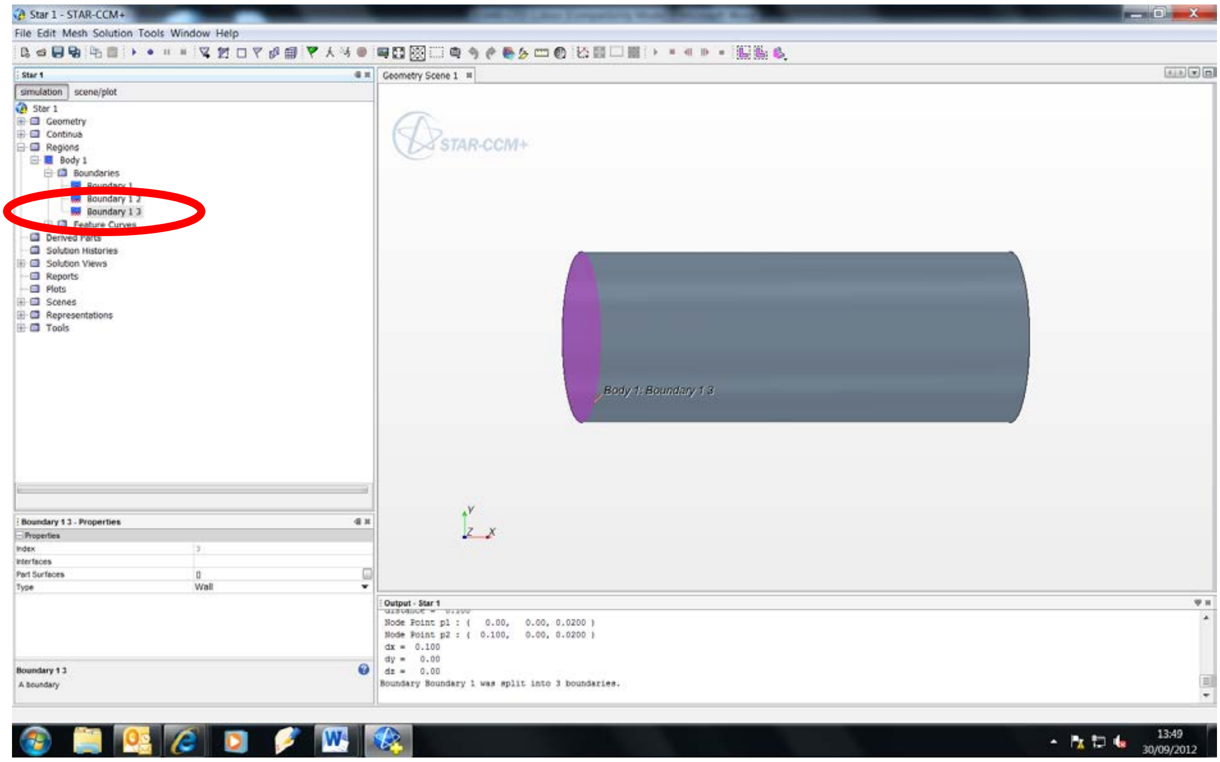
- Select the angle at **80** degrees, and click **Apply** and **Close**.
- You may need to try other angles in the project.



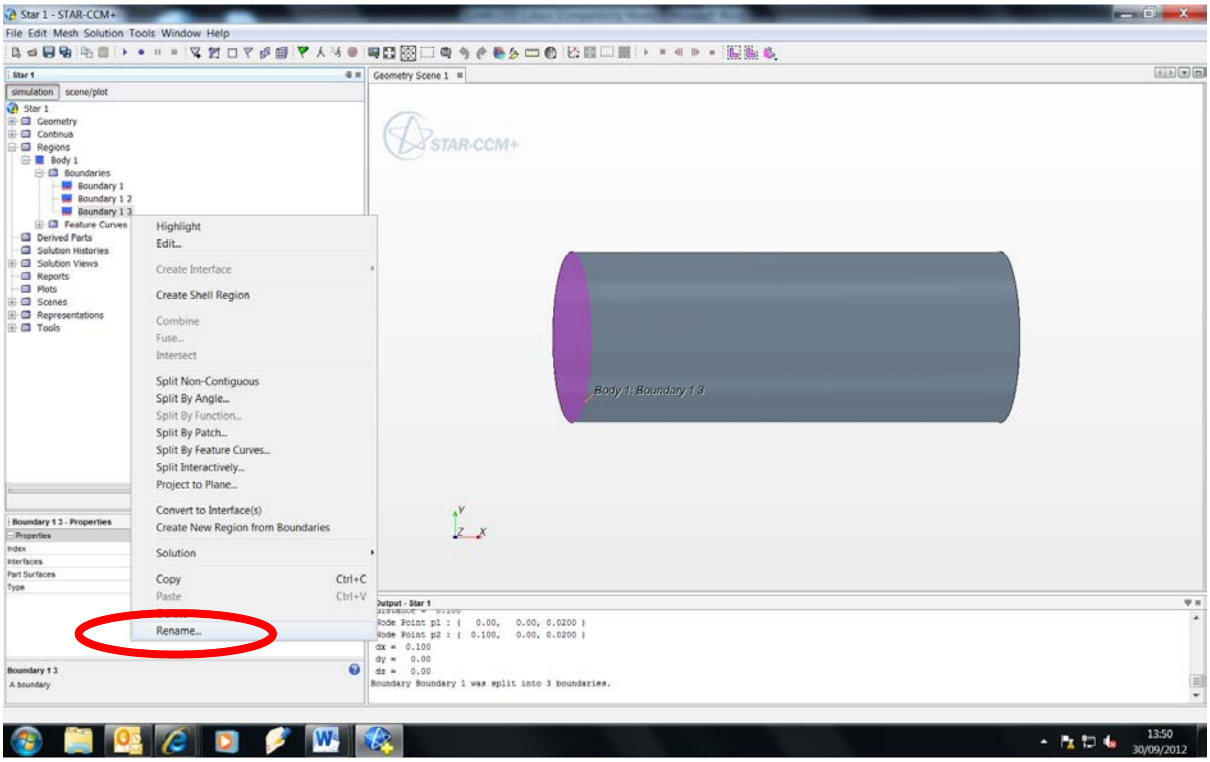
- Now, there are three boundaries created.



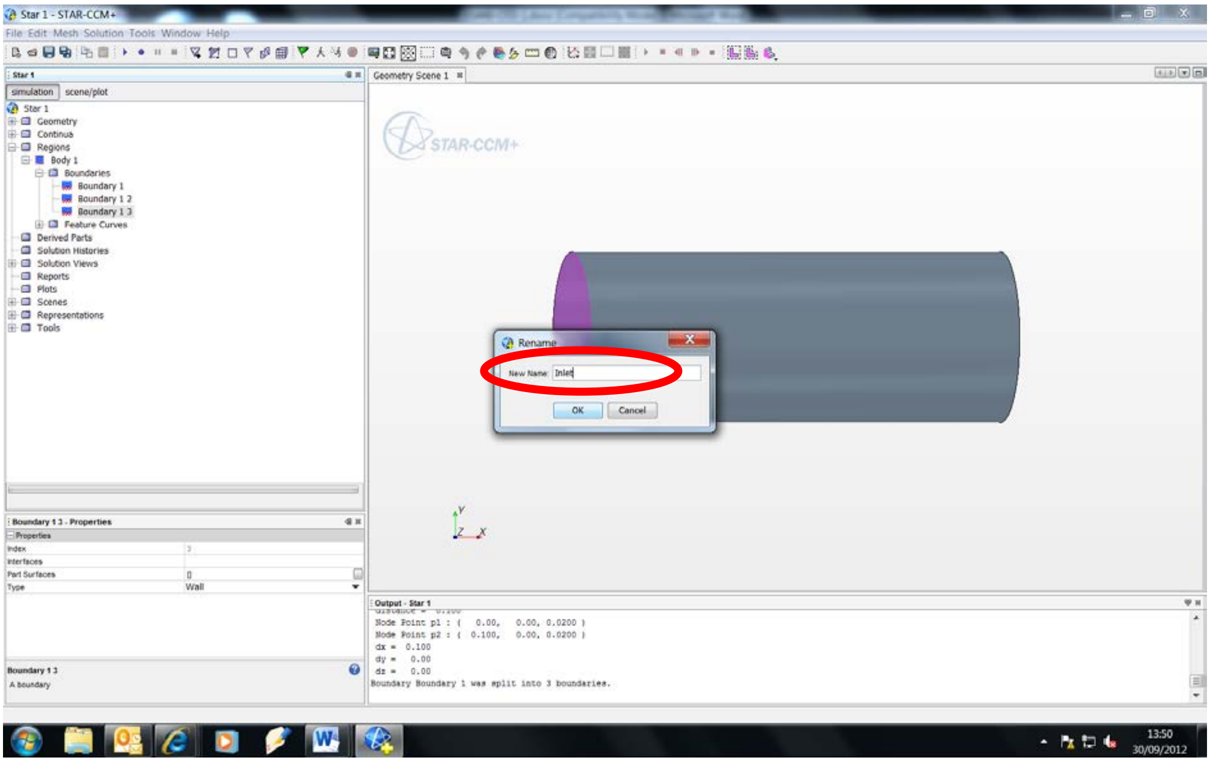
- Click ‘**Boundary 1 3**’. The left circular cross-section is highlight.
- This will be renamed as **Inlet**.
- Do not worry if a different section is highlighted in your computer.



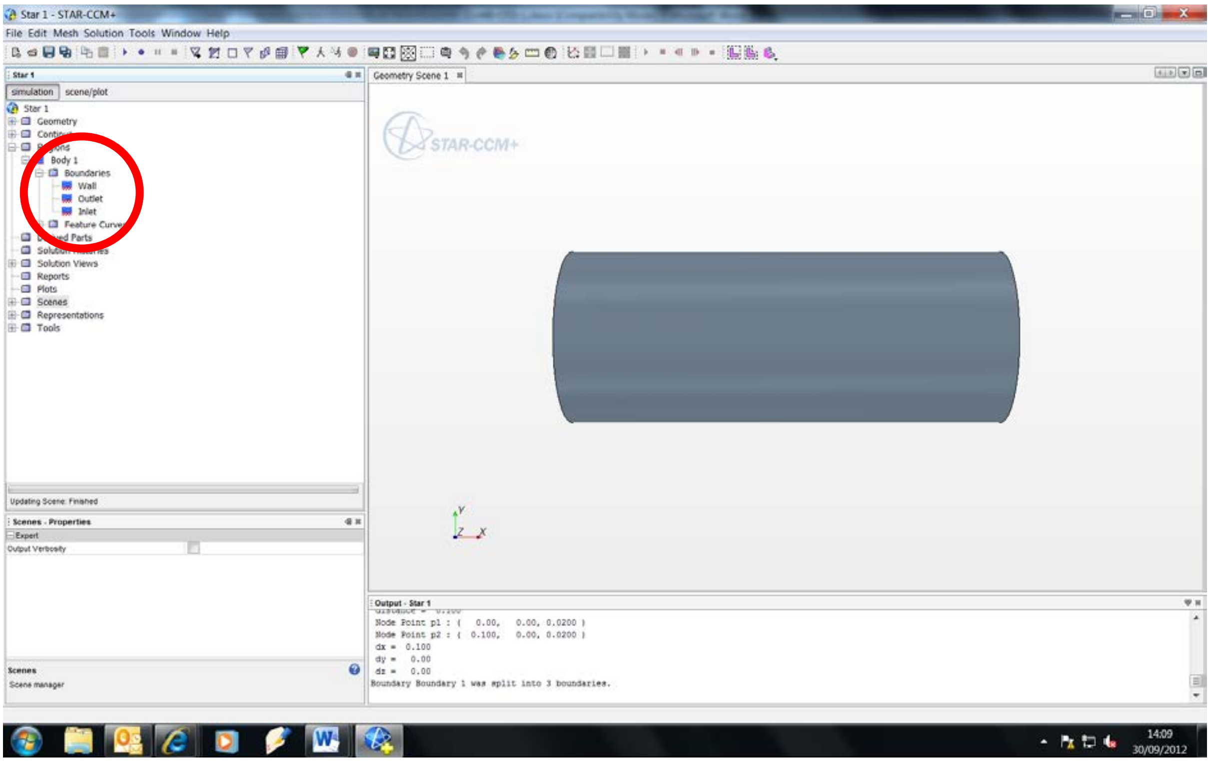
- Right click on ‘**Boundary 1 3**’, and choose ‘**Rename**’.



- Rename ‘**Boundary 1 3**’ as ‘**Inlet**’, and click **OK**.

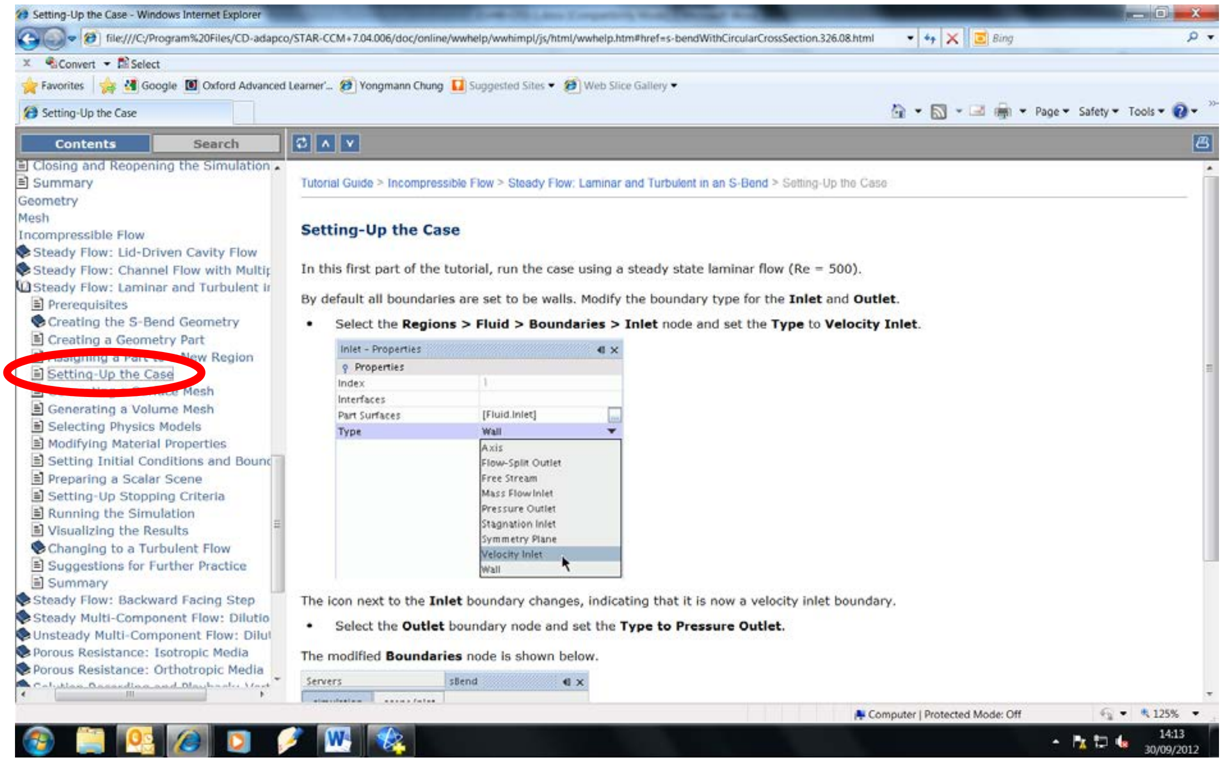


- Rename ‘**Boundary 1 2**’ as ‘**Outlet**’, and ‘**Boundary 1**’ as ‘**Pipe Wall**’.



<u>**Step 2**</u>: Construct the Simulation Topology

- Choose ‘**Setting-Up the Case**’, and follow the instuctions.

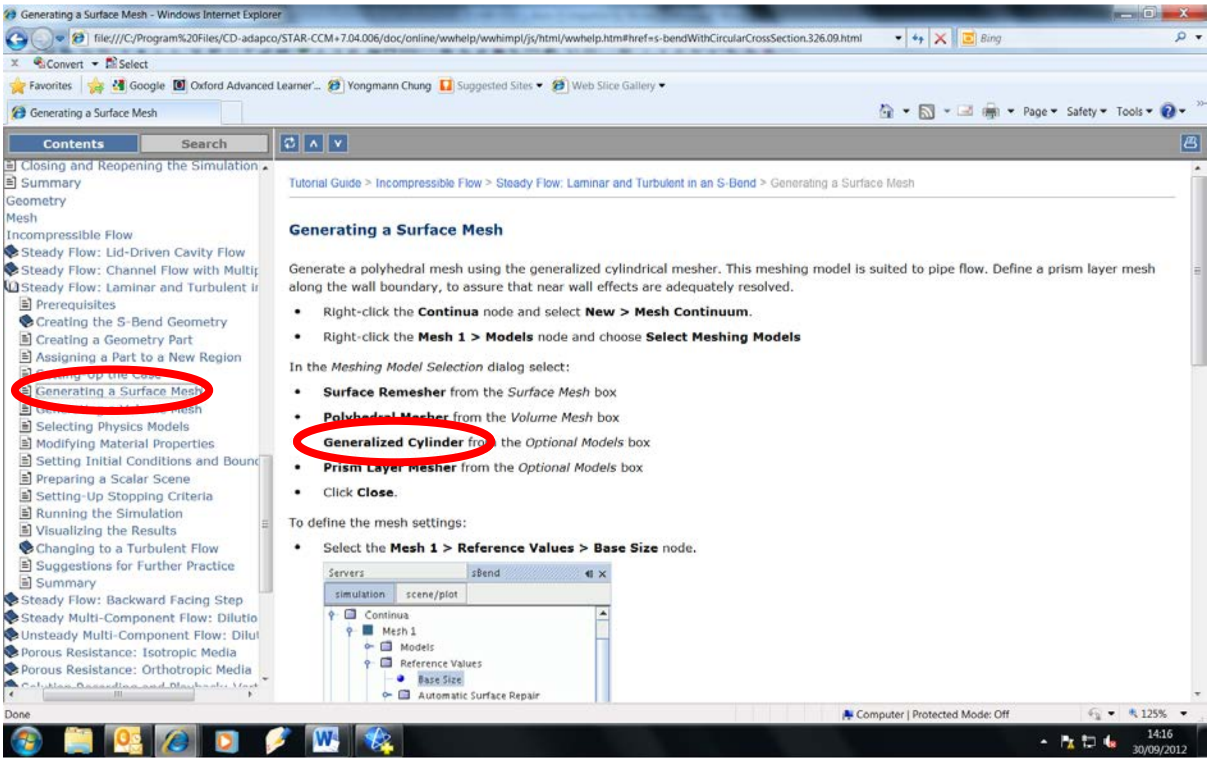


Step 3: Generate the Mesh

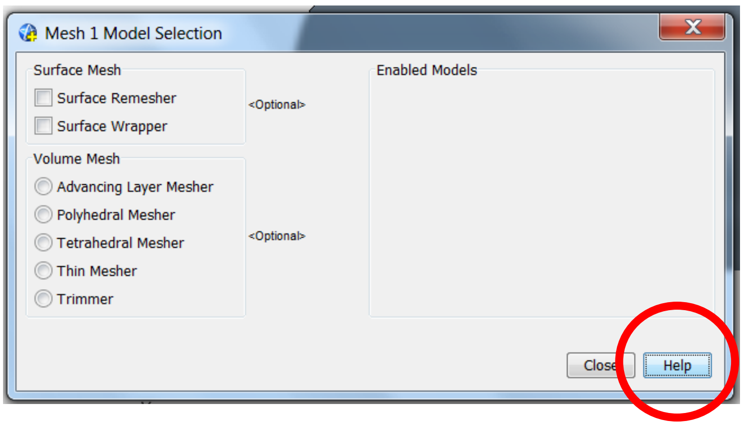
This step is very important, and a significant amount of project time will be typically spent to generate a high quality mesh (or computational grid as we call it).

This step is split into two parts: Surface mesh, and volume mesh.

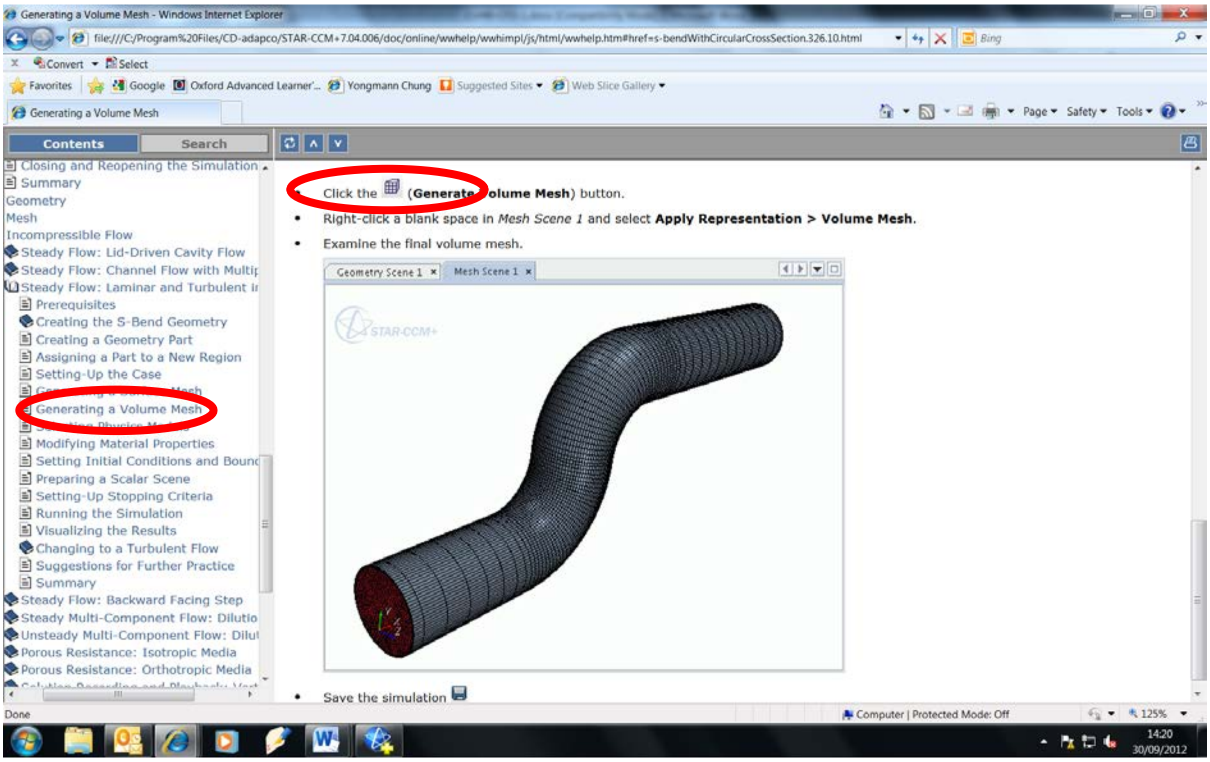
- Choose a '**Generating a Surface Mesh**' on the left menu.
- Follow the instructions except the *Generalised Cylinder* option.
- First, try even without the '**Prism Layer Mesher**' option.
- Select only '**Surface Remesher**' and '**Polyhedral Mesher**'.



- The available Mesh Model options are:
- Click the **Help** button for further information.



- Now, choose a '**Generating a Volume Mesh**' on the left menu.
- Ignore the *Generalised Cylinder* part, and continue from click the '**Generate Volume Mesh**' button part.



- Continue the rest of this tutorial, which includes:

Step 4: Define the Physics

Step 5: Prepare for Analysis

Step 6: Run the simulation

Step 7: Analyse the Results

Note: Please let me know if there are any factual errors or typos in this brief.

Tutorial Tips

- During the tutorial you will have to '**run**' (solve) the simulation. Press (Ctrl+R) or press the '**run**' (it has a running man as a symbol) on the toolbar at the top of the screen.

- You do not have to wait for the simulation to fulfil the stop criteria, as you can stop the simulation at any iteration number by pressing the '**stop current task**' button on the top toolbar (it has a red circular button right of the running man)

- After stopping the simulation by pressing 'Stop current task' button, the simulation can be recommenced from the last fulfilled iteration by pressing the '**run**' button again.

- If you need to change a simulation parameter or initial condition, stop the run, and go to top menu '**solution>clear solution**', and change the parameter, and then run the simulation again.

- You can save a simulation when the run is paused. It will save the results to the current iteration number. Star-CCM+ can then be closed, and re-opened. The simulation can be reloaded and you run the simulation again from the previous saved iteration number.

Exercise

- Stop your simulation in the middle of the run, and then save the intermediate results.
- Then, reload the simulation, and continue to run until the stop conditions are met. This tip will be particularly useful when you run large-scale simulations which take a long time to complete.
- Try different flow visualisation options available.
  – Plot streamlines, and iso-pressure contour lines.
  – Plot velocity and pressure profiles.
  – Try more.

On the next session ...

- CFD simulation of an external flow

Don't miss out!