ES440 Computational Fluid Dynamics

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

Introduction to Star-CCM+ software – Session 4

External Flows and Unsteady Simulation

Previously in Star-CCM+ lab 3…

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,
- How to determine a suitable mesh size.

Introduction

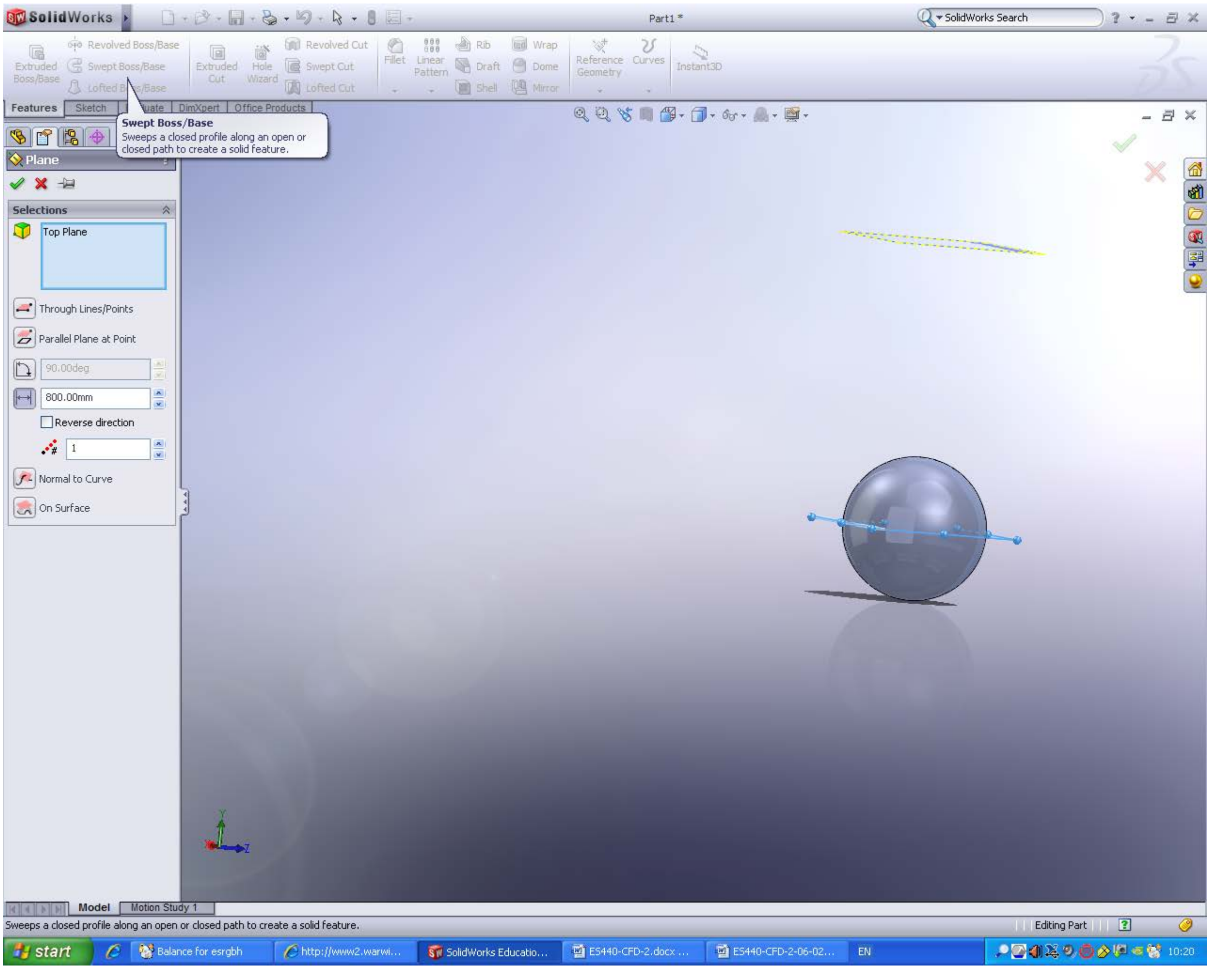
So far the geometry you have dealt with is simple pipe/duct flow, and in this session you will do an external flow, that is where you simulate the fluid flow around an object rather than through it such as the flow around an aerofoil. Also so far, you would have run steady simulations, this is where Star-CCM+ iteratively solves the problem but there is no advancement in time. For unsteady simulations, Star-CCM+ iteratively solves the Navier-Stokes equations, advances one time step, and then solves the N-S equations again. So, for a 10.0sec simulation with a time step of 1.0sec, Star-CCM+ would solve iteratively 10 times.

1. Creating a Solidworks Model

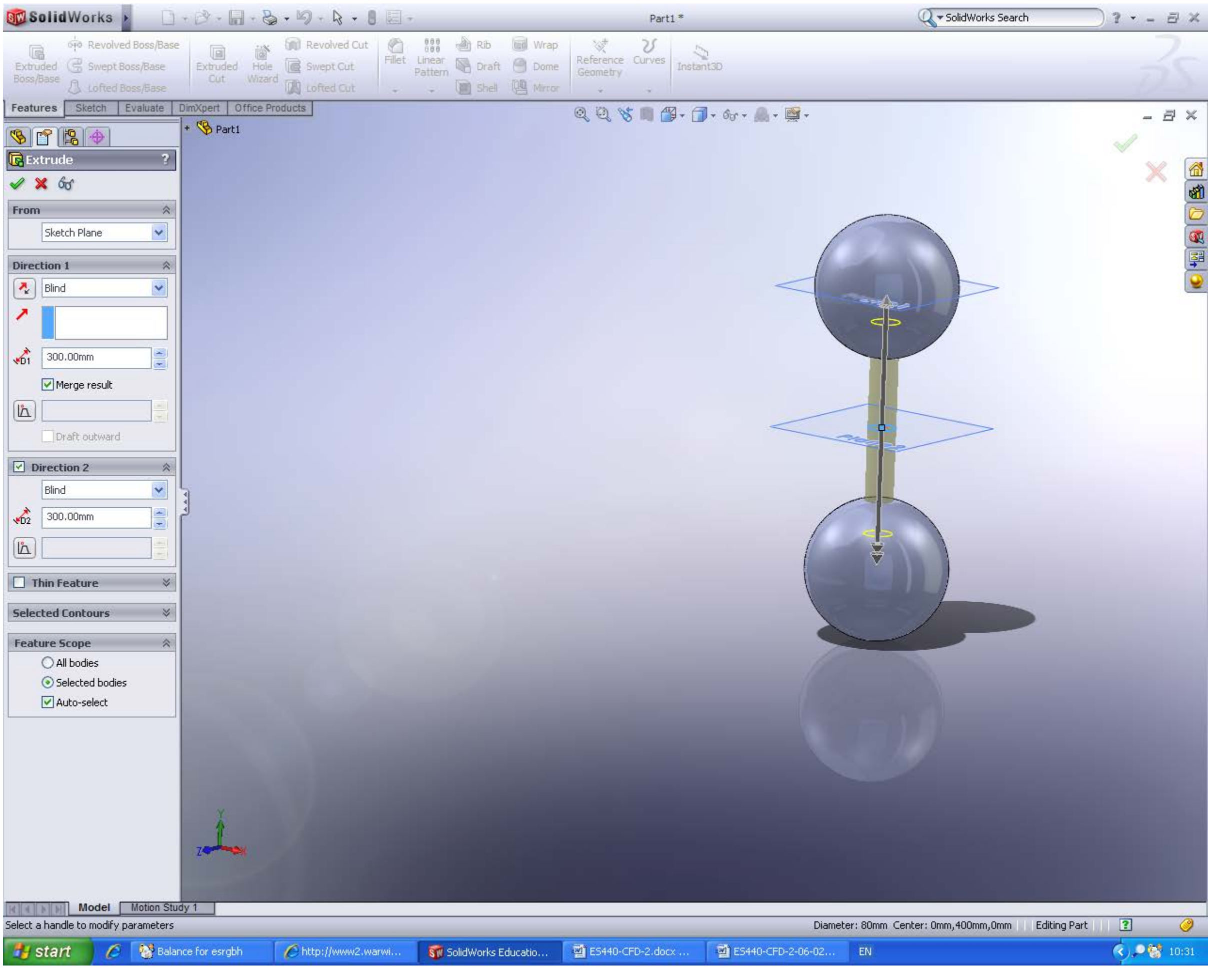
Now let's see how to create complicated geometries for Star-CCM+ with solids, fluids and porous components (Look for information on Regions in Star-CCM+ Help for more details, if you wish to use them for your coursework.). So far, in all the Star-CCM+ sessions for this module, we have used only a single fluid region. For example, in the previous pipe simulations (Star-CCM+ Lab1), the SolidWorks model was actually a model of the fluid region within the pipe and not pipe itself. Hence with external flow simulations, we need to supply Star-CCM+ with a SolidWorks model not of the solid region we are investigating, but a model of the **fluid region** (computational domain) around it!

Task 1: Creating a solid work model of the part

- Open a new part in **Solidworks,** create a **semi-circle** on the **top plane sketch** with a radius of **200mm,** with the **centre of the straight line at the origin**.
- Exit the sketch and do a **360^o revolved boss** to create a 400mm Diameter sphere**.**
- On the part tree, **click on the top plane to highlight** it and with it remaining highlighted, go **>insert>reference geometry**. Change to **offset distance to 800mm**. Your screen should look like below :

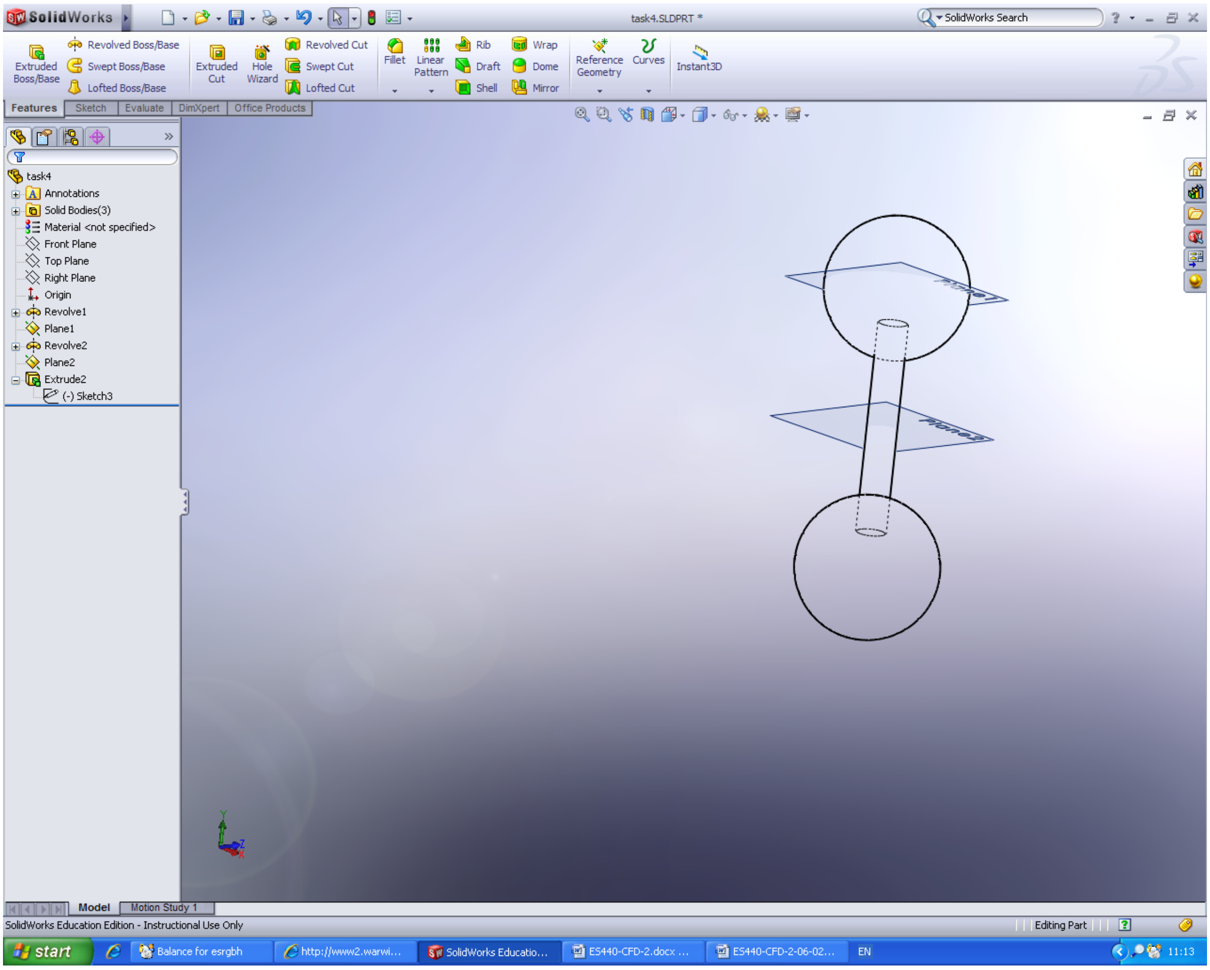


- **Press the green tick to create the new plane**, and create an **identical sphere to the last one** on the new plane with its **centre in line with the centre of the previous sphere.**
- As done previously, create a new plane from the top plane in the tree, this time with a **400mm offset** so that it is in the **middle of the two spheres.**
- On the new plane create a **circle with a radius of 40mm** and exit the sketch.
- **Extrude this new circle both ways 300mm** so that the generated rod **penetrates both spheres**. Your screen should look like below. **Untick the merge result**, and save the part at this point.



- **The reason for unticking the merge result is to leave you with three separate bodies. Otherwise the rod that penetrates both parts would have merges with the two spheres, resulting in one solid part. In this next section you will learn how to combine parts, and this tool removes conflicting geometry between two solids to form one solid body. You will more than likely have to use this tool in your coursework as solid bodies that clash will result in a mesh error on star-CCM+ and so imported parts should be one solid body!**

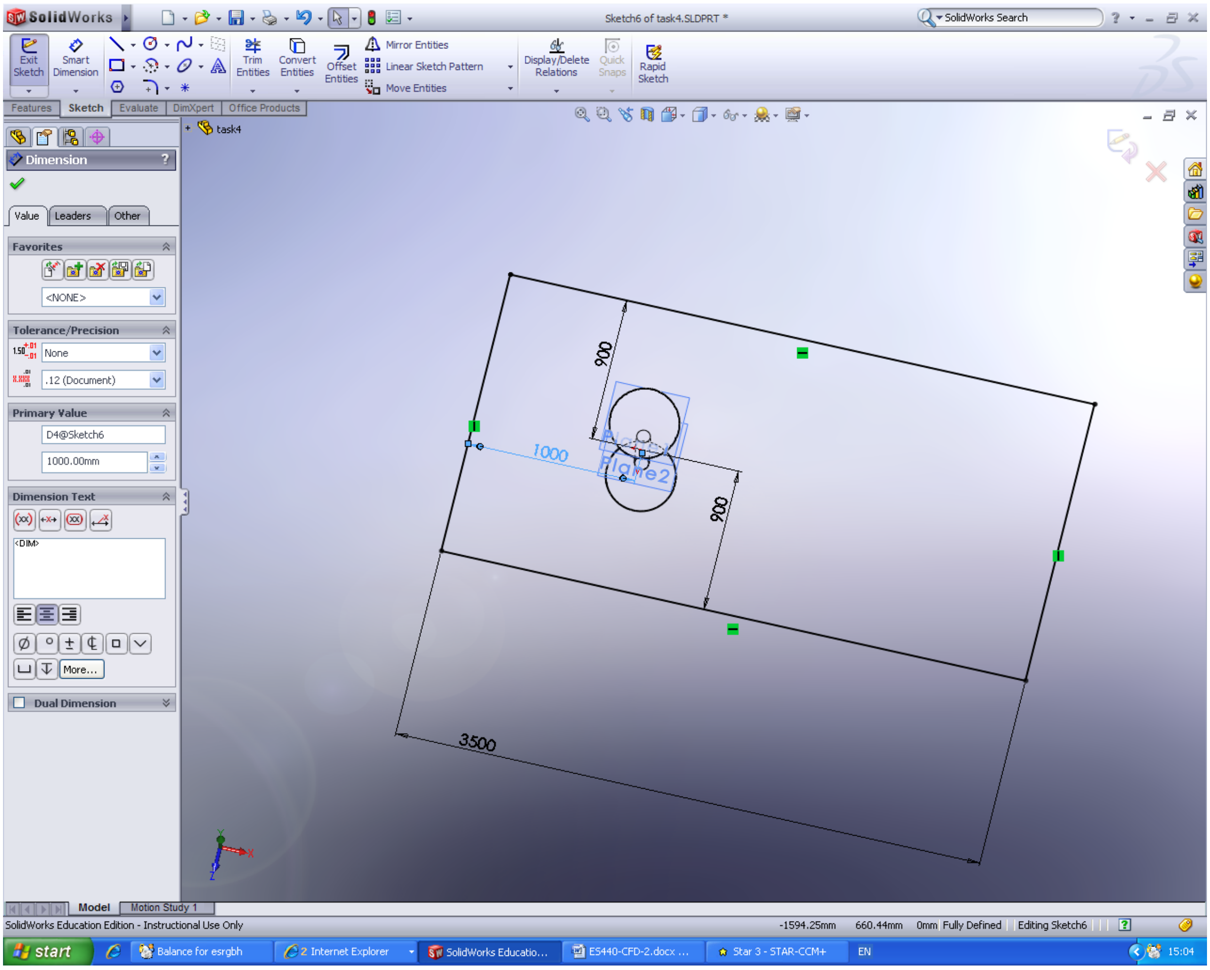
- **Go >insert>features>combine,** change the operation type **to combine,** and select the two spheres and the rod as the parts to combine, and press the green tick. The three solid bodies now merge to one with no conflicting geometry.
- Check that your part is **one solid body** by changing your display style to **hidden lines visible**. If you see the dashed lines of the rod visible in the spheres, then you have not merged your bodies correctly. Your screen should not look like below:



<u>Task 2:</u> creating the domain and modelling the fluid

- So now that we have are part, **we now need to model the fluid around the part**. To do this we create a block representing the computational box and then subtract the part from it.
- In this example, we will have the centre of the part **1m from the inlet** and **2.5m from the outlet**.

- Using plane 2 (middle plane) in the Solidworks model, **draw a rectangle, with an overall length of 3.5m, a width of 1.8m with the part in the middle width wise (900mm from the edge) and the part 1m from the inlet**. It should look something like below:

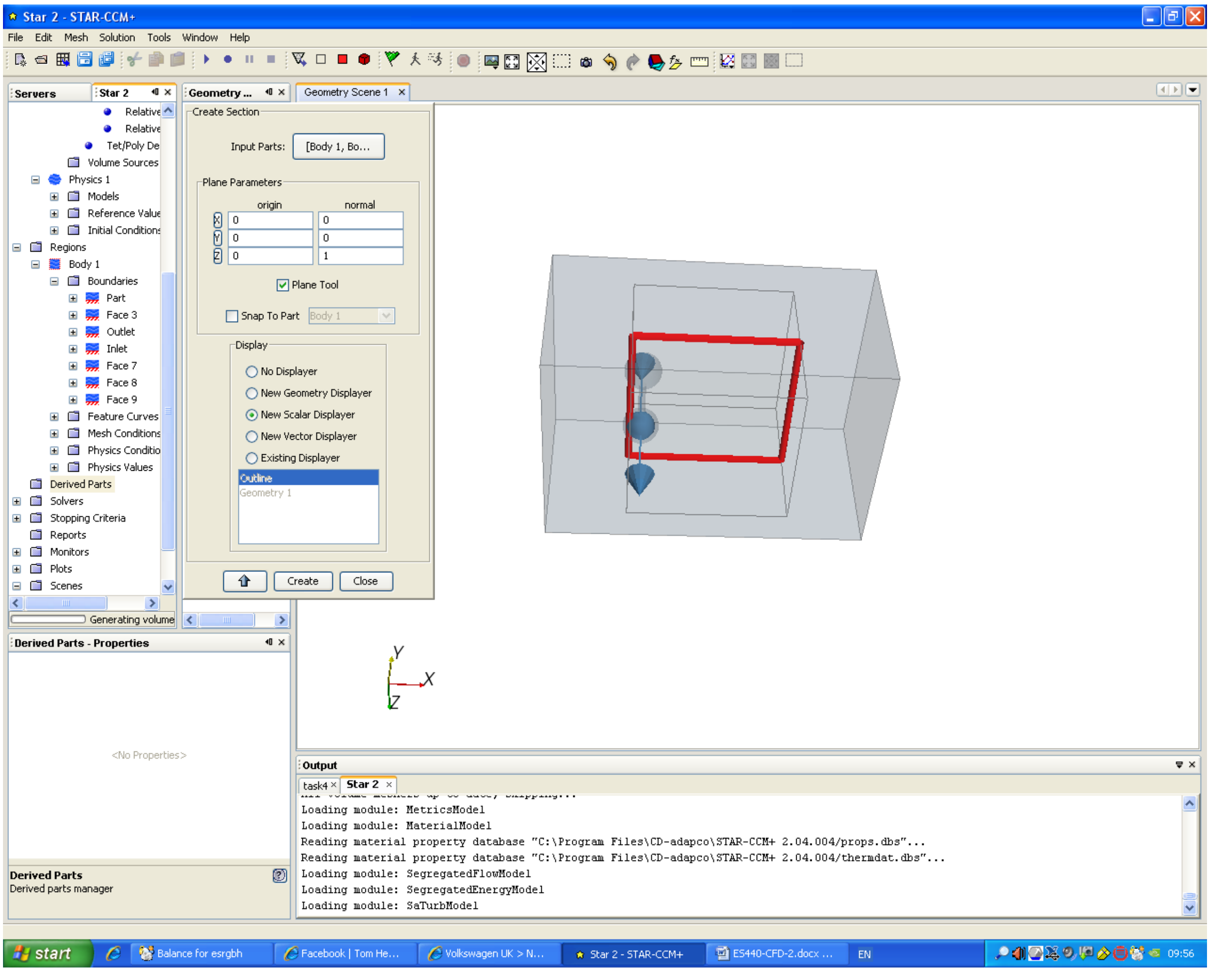


- Extrude the rectangle **both directions by 1200 mm to create a box**. This time it is important to **untick merge results so that the box is a separate body from the part**.
- Check using the hidden lines visible under the display style to make sure the **part is separate from the box**.
- Now to model the fluid, we remove the part from the box. Go >**insert>features>combine. Change the operation type to subtract** and **select the box as the main body with the part/spheres as the part to subtract and click the green tick**.
- Make sure the subtraction has worked by doing a **longitudinal cross sectional of the box (as below)**
- **Save the part as a parasolid**, exporting all bodies, and exit Solidworks.

2. Creating the simulation

Task3: Setting up an unsteady simulation

- Start a new serial simulation and import the geometry, on the cad import options, select **Boundary mode: one boundary per face, Region mode: one region per body, Sharp edge angle: 30 degrees** and click ok.
- Combine the part faces and rename all the faces appropriately e.g inlet, part
- For the mesh properties select:
  - Surface remesher
  - Polyhedral mesher (This is to show the different volume mesh to the tetrahedral mesher)
  - Prism layer mesher
- Leave all the mesh values as default, apart from:
  - Base size = 0.08m
  - Surface size> relative minimum size = 50%
- Create a volume mesh
- Create the physics continuum, you should have:
  - Three dimensional
  - Stationary motion
  - Gas material
  - Segregated flow
  - Ideal gas
  - **Implicit unsteady**
  - Turbulent viscous region
  - Spalart-Allmaras
  - standard Spalart-Allmaras model
- **Save at this point!**
- Change the inlet velocity to **10ms^{-1}** (side closet to the spheres) and have a **pressure outlet**
- Create a new scalar plane section under the derive parts, the plane going length ways (longitudinal) and half way through the width. Tick the box for the plane tool to help, then you should have a plane similar to the one below:



- **Create two new scalar scenes, one of pressure and velocity magnitude**, with the newly created plane as the only part in both.
- **Create a vector scene of velocity** again with the newly created part as the only part, with 3D head 3D tail as the vector style.
- At this point please **save the simulation** and from now on any other saves you do for this simulation, please do under a **different file name**!

3. Setting the run criteria

For implicit unsteady simulations three parameters have to be decided by the user, those being the time step size, the length of time, and the number of iterations per time step.

In the case of this simulation we are interested in the vortex shedding from the spheres, so using the Strouhal number, St=0.2 ($St = \frac{fD}{U}$, U=10ms^{-1} and D= 0.4m diameter of spheres), we can estimate that the frequency of shredding (f) is about 5Hz.

Therefore we will run the simulation for 0.6 seconds to get 3 cycles and have 20 time steps for each cycle, giving us 100 total number of time steps with a time step size of 0.01 seconds.

In the case of this simulation we will have the inner iteration number at 10, this is so that the simulation can be run in the two hour lab time that we have but in the case of the coursework you should run till the solutions have converged with the inner iteration number required worked out from practice runs. **The estimated running time of this simulation is 36 minutes in F210.**

Task 4: Setting the criteria

So the first thing is to set the total time of the simulation:

- Go Solvers>Implicit unsteady and under the properties, **change the time step to 0.01 s** and keep the temporal **discretisation as 1st order**

The second step is to set the stopping criteria

- Stopping criteria>Max inner iterations> change default from 20 to 10
- Stopping criteria>Max physical time change to 0.6 s
- Stopping criteria>Max steps > untick the enabled if this is not done then the simulation will stop after the default setting of a 1000 time steps (although not relevant for your simulation you must do this in your coursework)

4. Saving time step data

Now that the stopping criteria are set, the final part is to set up an automatic saving of data. Without this after every timestep the scenes are refreshed for the next time step, and the data is lost, Star-CCM+ provides a way of saving the scene automatically at the end of every time step.

- In your 'my documents folder' make three new folders names 'pressure', 'velocity' and 'vector'
- Now go Scenes>Scalar scene 1>Attributes>Update
- In the properties window change the update policy to time step, file format to jpeg, press **'the 3 dots in a small grey box'** button next to file path and select the newly created 'pressure' folder , change the output base filename to pressure, and tick the box next to output frame to file.
- Now on the 1st step it will save the scene as **Scene_1pressure00001.jpg** in the pressure folder and on the n-th step the scene will be saved as pressure_1pressure**nnnnnn**.jpg
- Do the same for the velocity scene and the vector scene saving them in their corresponding folders
- Run the simulation and wait for it to finish

Well done! You have now done an unsteady simulation, and in your three folders you should have images for each time step. I have rendered the vector images myself to create a video which is on the website. When it comes to your assignment investigate what plots and scenes you can record and what would be relevant to your problem!