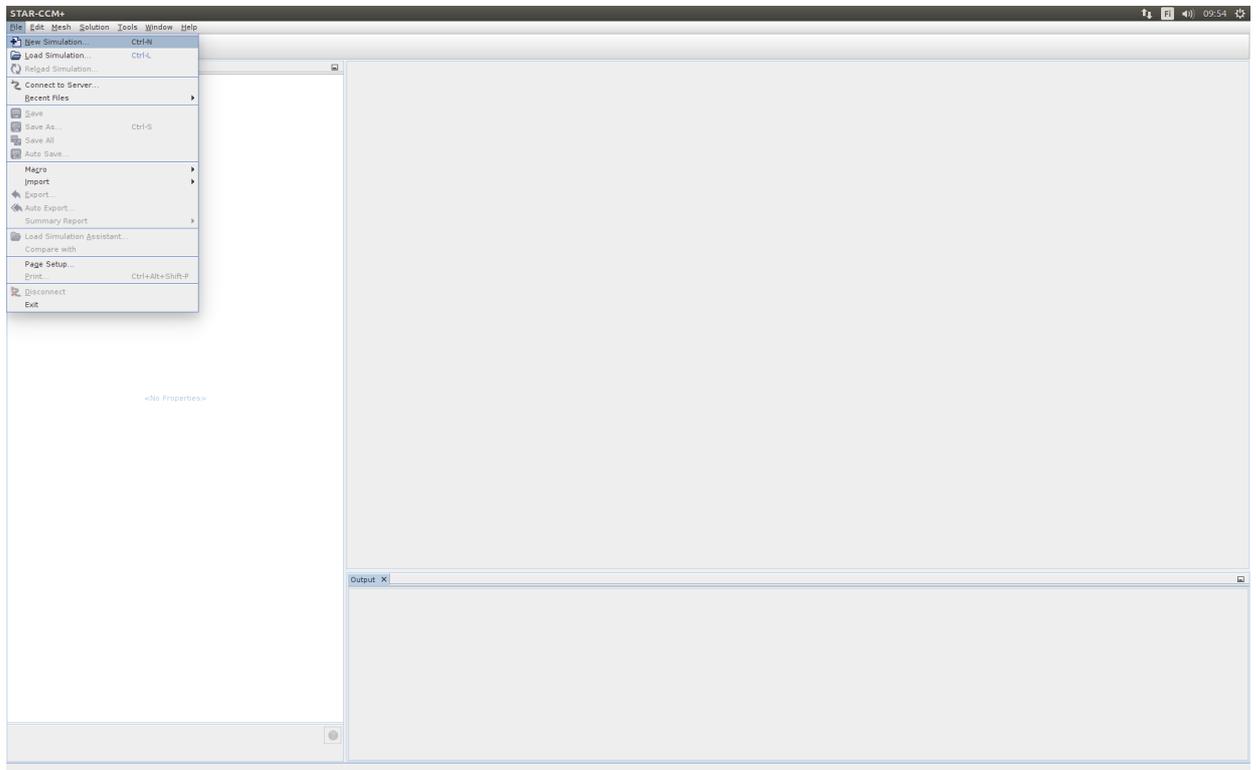


Fully turbulent flow around a sphere using StarCCM+

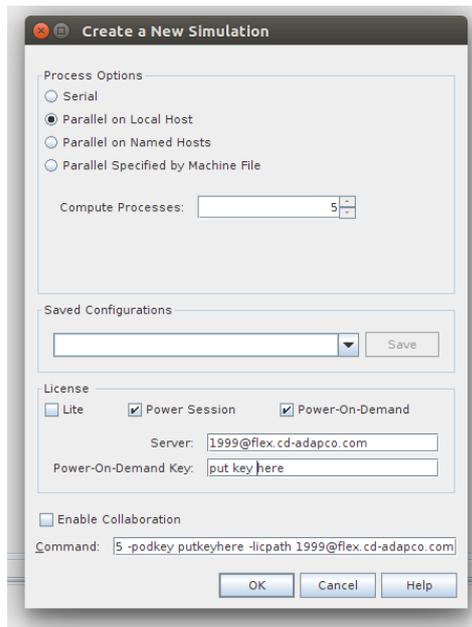
In this tutorial you will simulate a fully turbulent flow around a sphere using a given CAD geometry. It would be possible to generate the geometry directly in StarCCM+, but one of the objectives of the tutorial is to get you familiar with the import of a given geometry into the software. Remember to save the simulation project periodically at the end of each phase. You can do this either via the disk icon in the toolbar or through the *File* menu.

Starting StarCCM+

1. Launch StarCCM+. On linux the program starts with the *starccm+* command.
2. Choose *File* → *New simulation...* from the top menu.



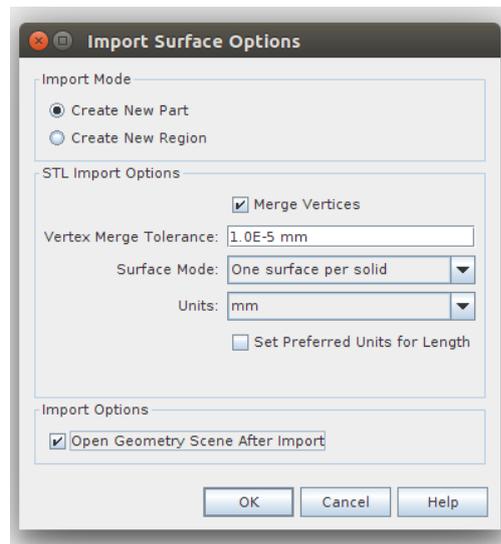
3. Setup the processor and licensing options for the current session.
 - a) Choose either *Serial* (single core) or *Parallel on local host* (multiple cores) option. For the parallel option choose the number of cores (it is advisable to choose one less than the number of physical cores, if you intend to do something else while the simulation is running).
 - b) Fill in the license details: choose *Power-On-Demand* option, fill in the *Server* details as shown below, unless they are automatically filled and type in the *Power-On-Demand Key* that was given on MyCourses.



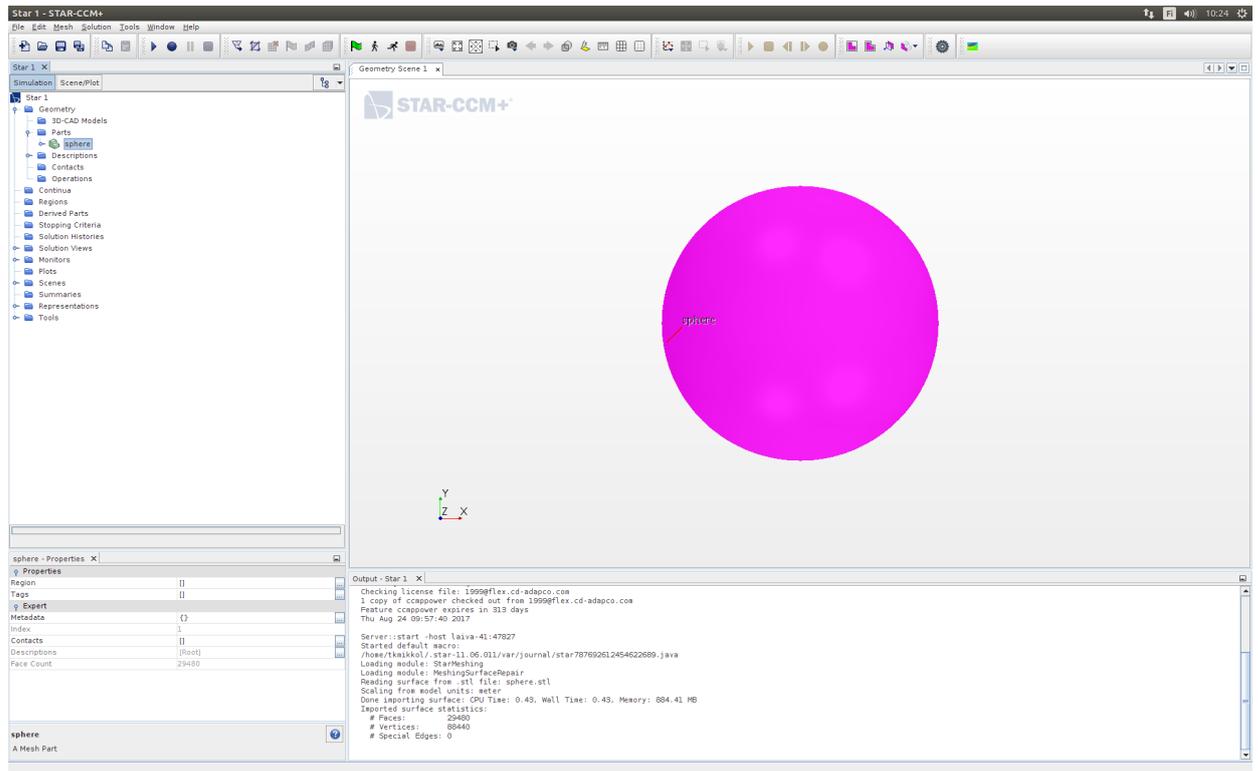
c) Click *OK*.

Setting up the geometry

1. Download the STL-file for the geometry from the Tutorial page on MyCourses.
2. Choose *File* → *Import* → *Import surface mesh...* from the top menu.
3. In the window that opens, change *Units* to mm (the stl-file is saved in mm) and select *Open geometry scene after import* option. This should load the geometry and open a scene where you can see a rendering of the sphere.



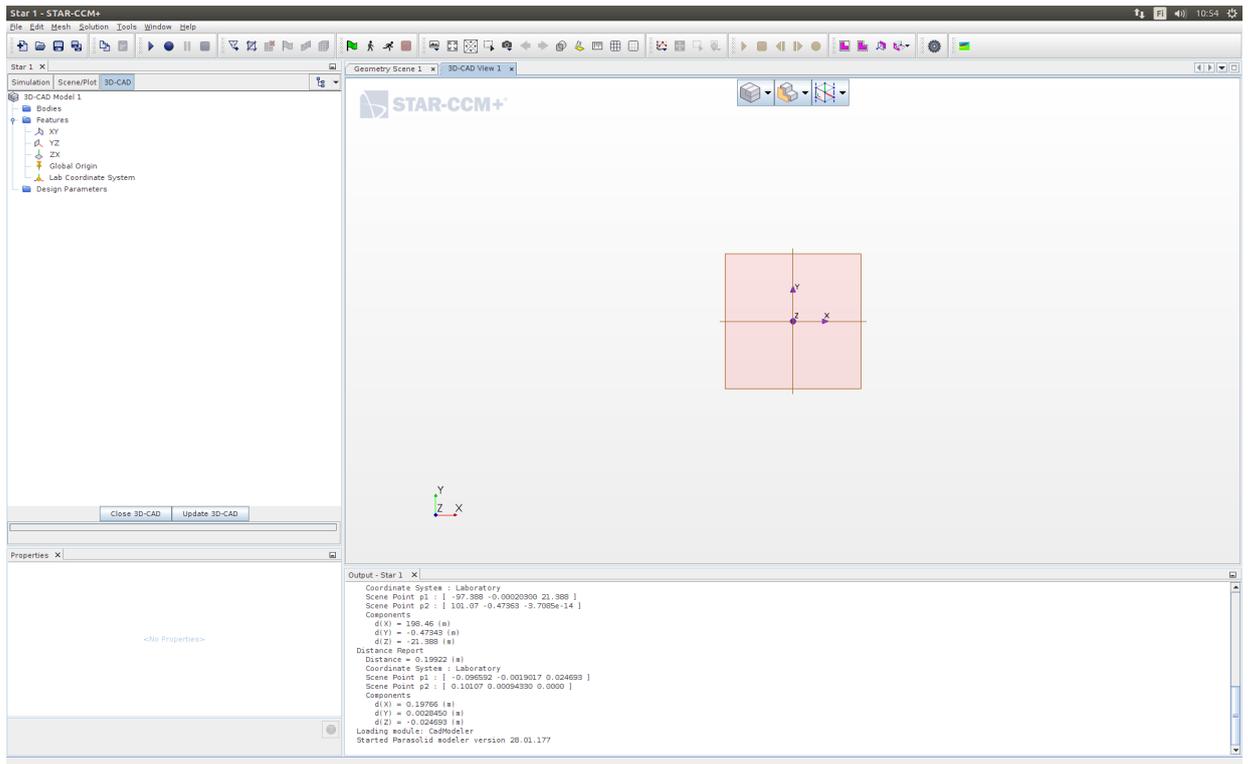
4. The tree on the left hand side of the main window is used to control all aspects of your simulation project. The imported sphere geometry should have appeared in the simulation tree under *Geometry* → *Parts*. You can open the sub-nodes of the hierarchy by clicking on the circle of the connecting lines. If you click the sphere in the tree, it is highlighted in the geometry scene on the right.



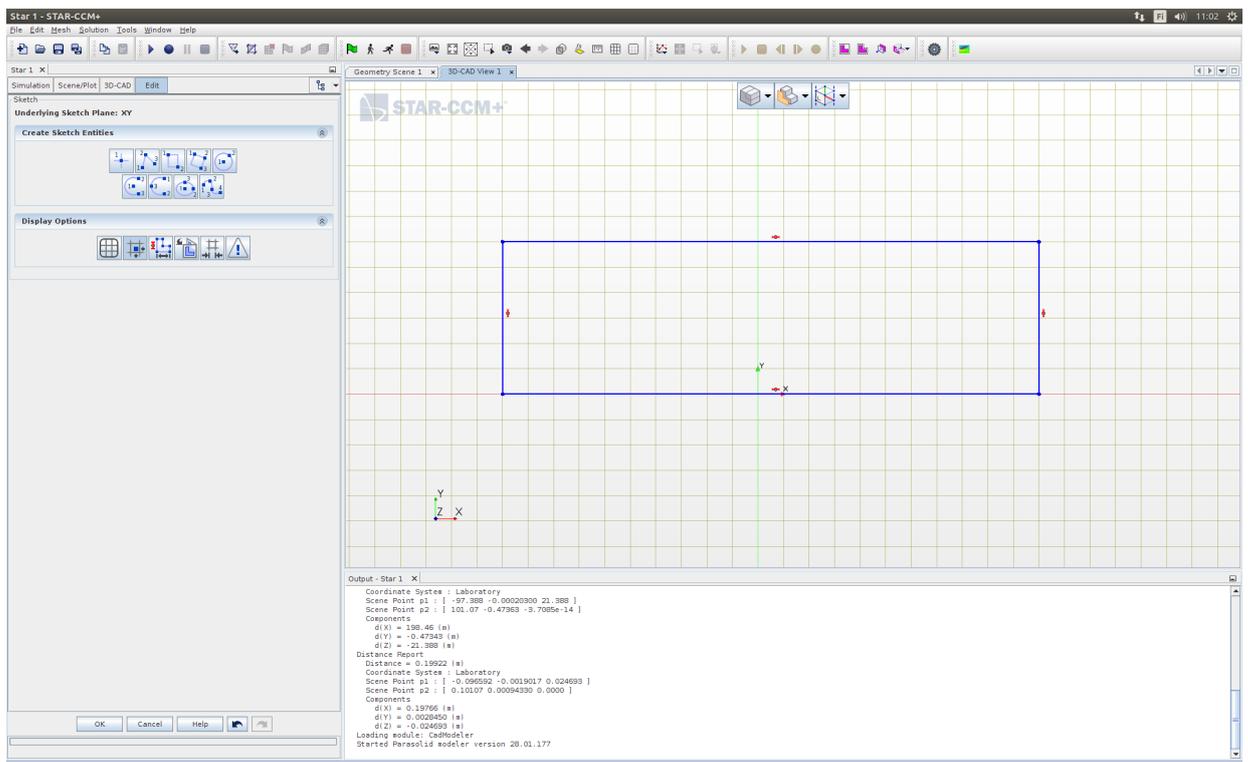
Setting up the simulation domain

We will specify a cylindrical domain, that covers only one quarter of the cylinder. We will do this by specifying the profile of the domain and then revolving this profile by 90 degrees around the longitudinal axis to form the volumetric domain.

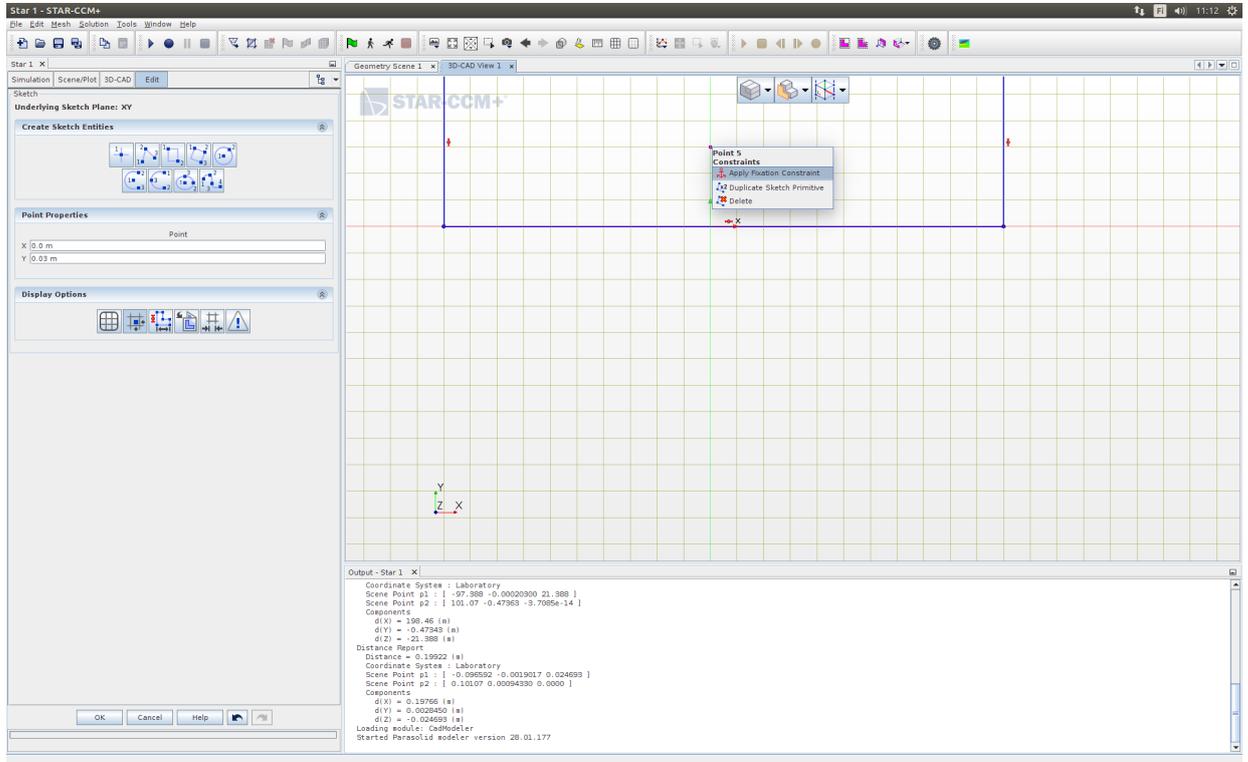
1. Right-click *3D-CAD Models* in the simulation tree and choose *New* from the context menu. This opens up 3D-CAD View.



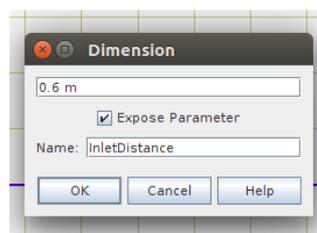
2. Right-click *Features* → *XY* and choose *Create sketch* in the object tree on the right. This changes to a sketch view, where we can draw elementary shapes on the XY-plane. You can rotate (left mouse button), zoom (scroll wheel) and pan (right mouse button) the view.
3. Click the rectangle icon under the *Create sketch entities* in the toolbar on the left.
4. Draw a rectangle by clicking first in the lower left hand corner (**on the x-axis**) and then in the upper right hand corner of the rectangle. The dimensions can be whatever at this point.



5. Add a reference point inside the rectangle on the y-axis by clicking on the point icon in the left hand toolbar and clicking at some point on the y-axis. We will use this point to specify the distance of the inlet and outlet boundaries from the sphere center.
6. Anchor the point by applying a fixation constraint. Right-click the point and choose *Apply fixation constraint*.

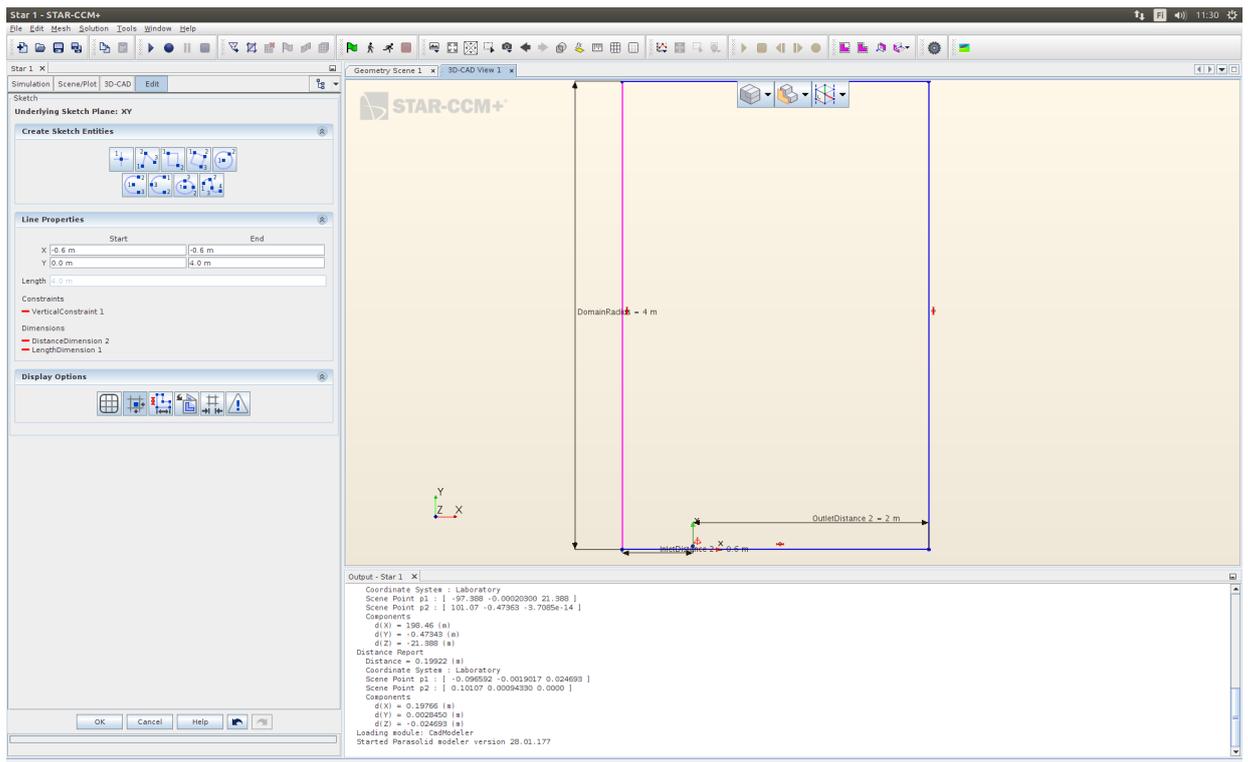


7. Specify the distance of the inlet boundary from the reference point. Click on both the reference point and the vertical edge on the left while holding down the Ctrl-key on the keyboard. Right-click the edge and choose *Apply Distance Dimension* from the context menu. Specify the *Distance* in the window, choose the *Expose parameter* option and give the parameter a meaningful *Name*. Click OK. By giving names to the dimensions, these can be modified easily afterwards.

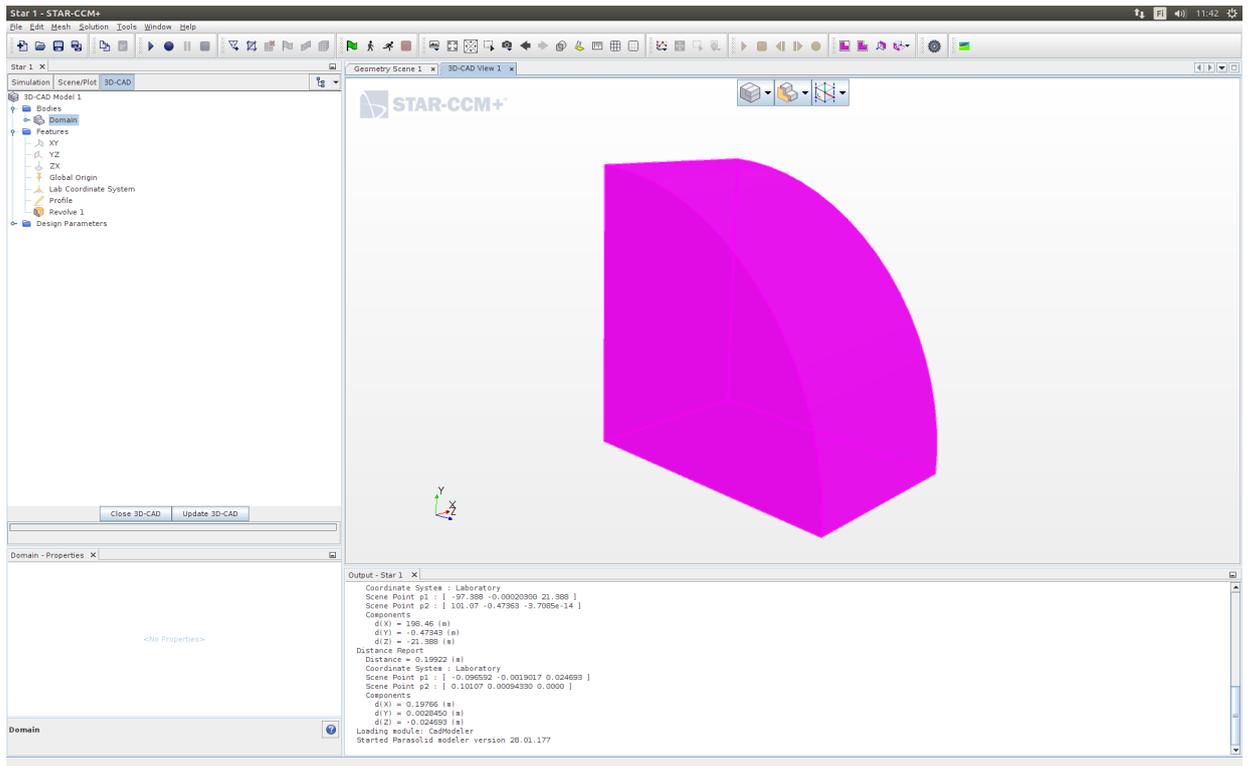


8. Specify the outlet distance similarly, i.e. choose the reference point and the right vertical edge and apply a distance dimension with a meaningful name. Make sure that the edge is on the right side of the reference point before you specify the distance dimension. If this is not the case, you can drag the edge by click-and-drag.
9. Specify the radius of the domain by right-clicking the left vertical edge and choosing *Apply length dimension*. Specify the radius, expose the parameter and give a sensible name.

10. Check the domain profile. It should look something like this with the dimension specifications.



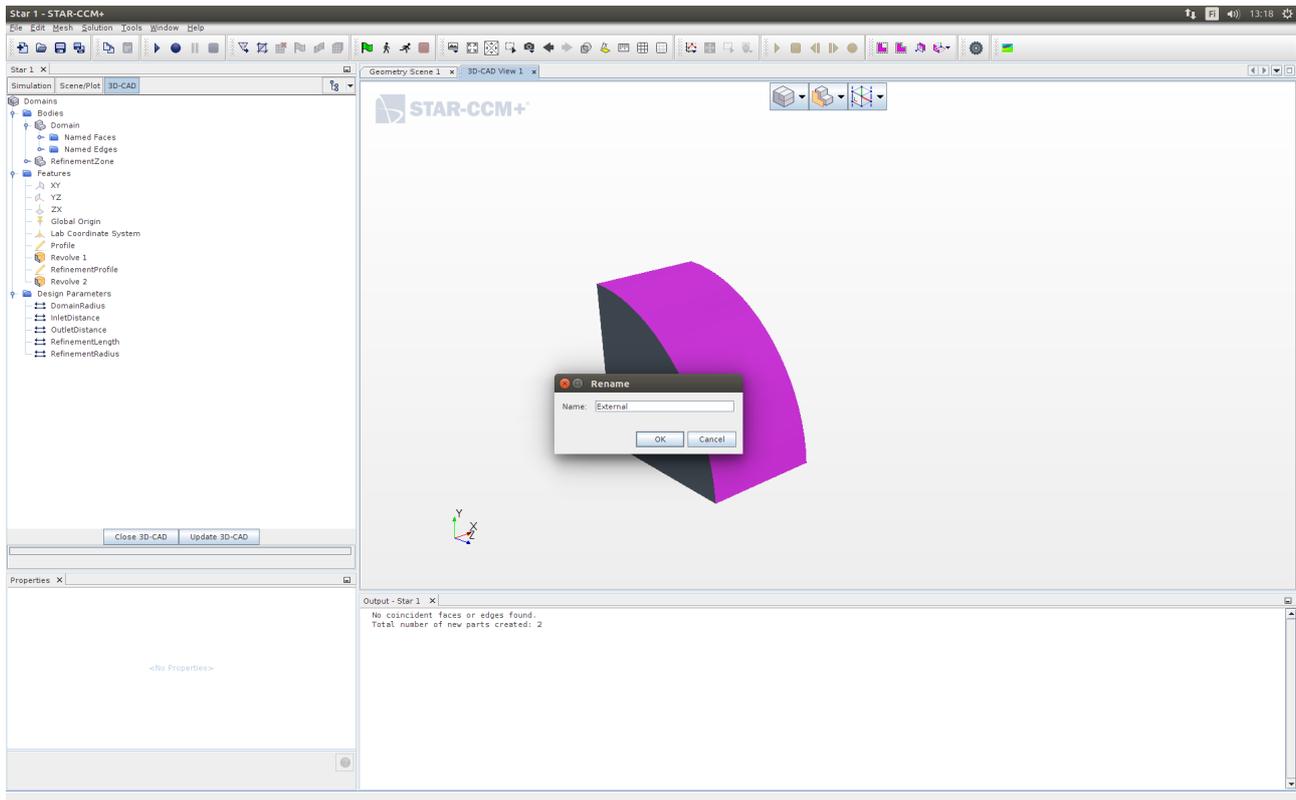
11. Close the sketch view by clicking *OK*. At the lower left hand corner.
12. Right-click *Sketch 1* in the object tree, choose *Rename* from the menu and give the object a sensible name (e.g. profile).
13. Right-click the profile object in the tree and choose *Revolve* from the menu. This opens a view, where you can modify the options for the operation. The defaults are fine, i.e. we want to revolve the profile around the x-axis by 90 degrees and generate a solid (rather than a sheet as this will be the fluid volume). Click *OK*.
14. This operation generated a body under the *Bodies* or *Body Groups* node in the tree. Rename this to something sensible (e.g. domain).



15. In order to specify appropriate boundary conditions on each face, we should give sensible names to each face of the body. You can choose faces by clicking them in the 3D-CAD view (multiple faces can be selected with Ctrl-clicking).

- Choose the faces on the XY- and XZ-planes simultaneously, right-click one of the faces and choose *Rename* under the faces heading in the menu. Give these faces a name *Symmetry*.
- Choose the face with the outer normal in the negative x-direction and rename this to *Inlet*.
- Choose the face with the outer normal in the positive x-direction and call this *Outlet*.
- Choose the curved face and call this *External*.

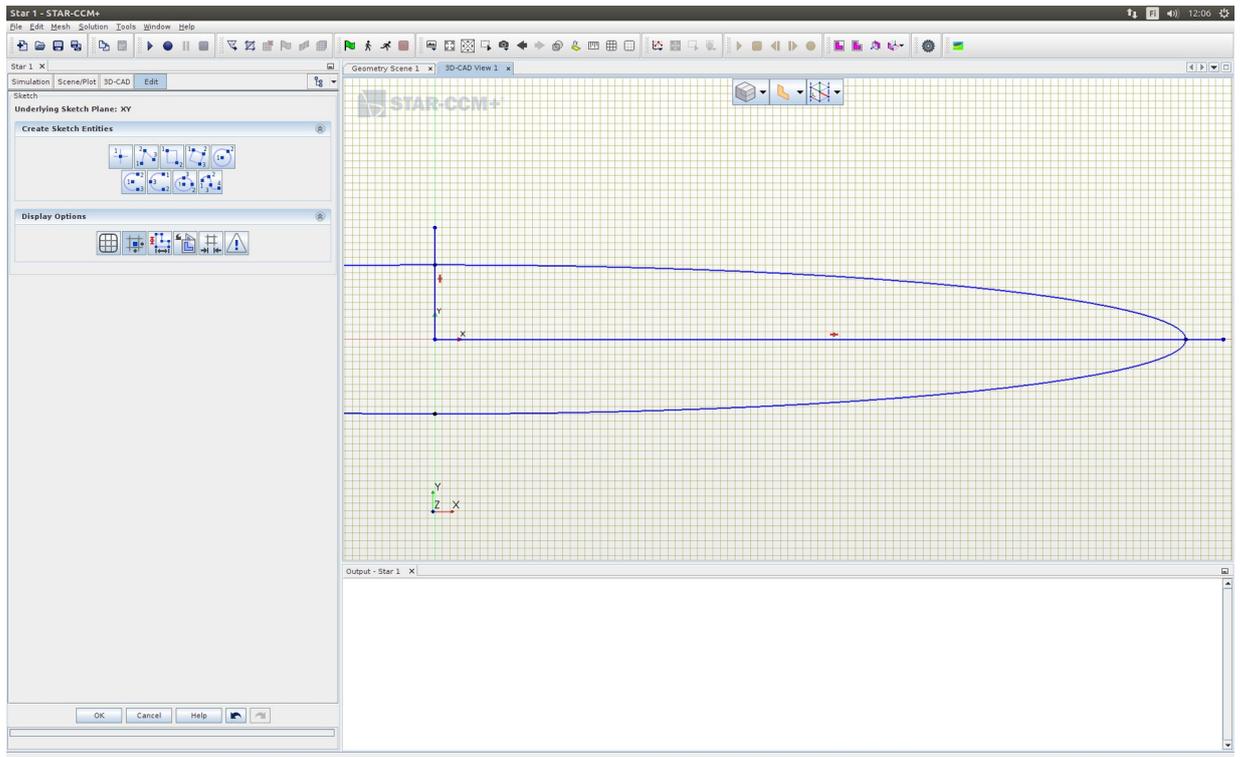
You can see the named faces in the tree under *Bodies* → *Body name* → *Named faces*.



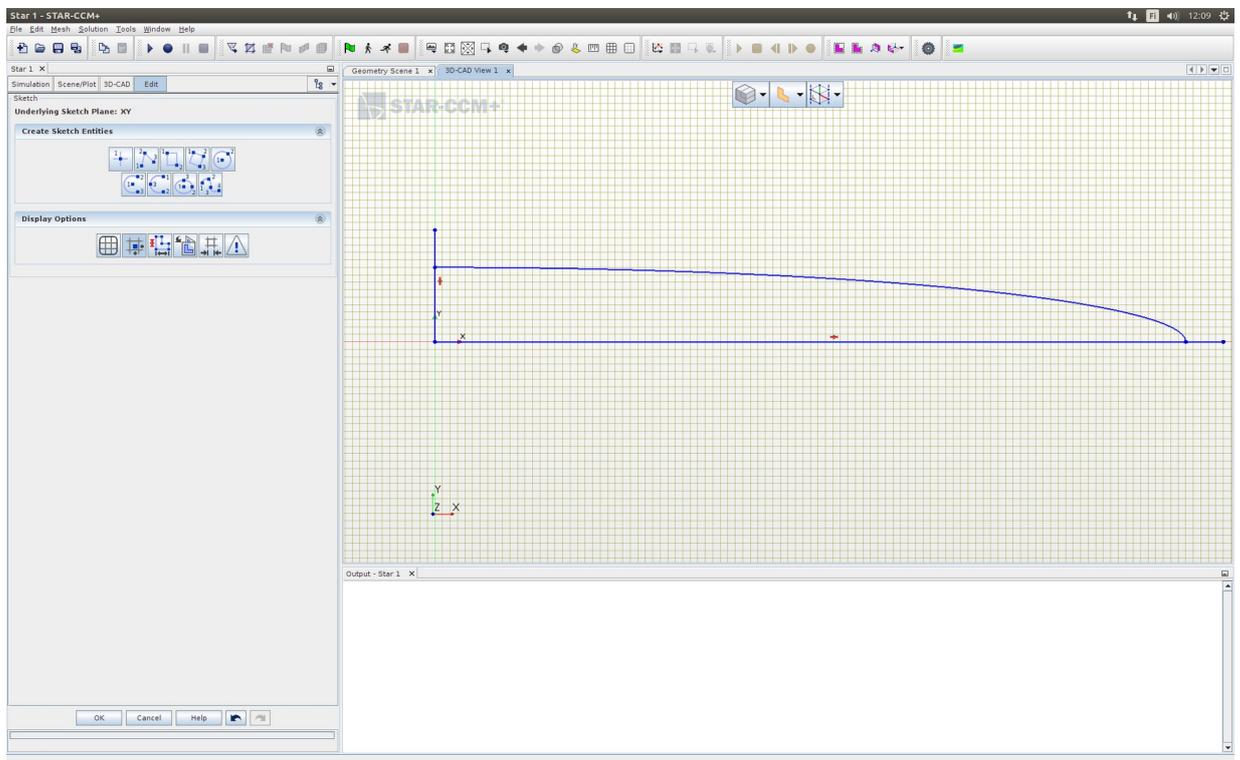
Setting up refinement volume

Let's specify an additional volume in the wake of the sphere, in which we will use a higher grid resolution. This will have a half-ellipse profile and will be generated similarly to the fluid domain by revolving the profile around the x-axis.

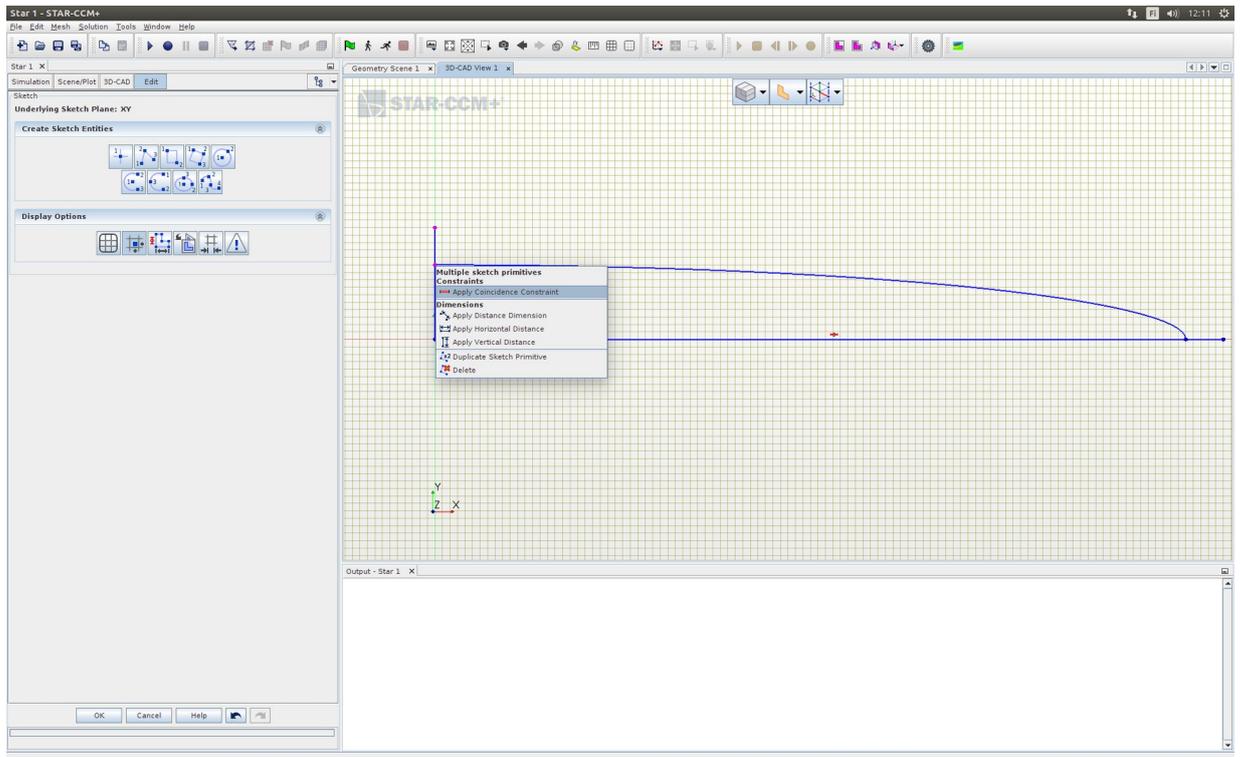
1. Right-click *Features* → *XY* in the tree and choose *Create sketch*. You can change the view through the camera icon in the top toolbar. *Standard views* give you easy access to different projections. Hide the existing model by deselecting *Solid bodies* in the menu that opens by clicking the Model visibility icon at the top of the 3D-CAD view.
2. Click the *Ellipse* icon and specify an ellipse with the centre at the origin.
3. Draw a vertical line and a horizontal line both starting from the origin and reaching outside the ellipse. The horizontal line should run from the origin towards positive x. You can stop the line drawing operation by pressing ESC key on the keyboard.



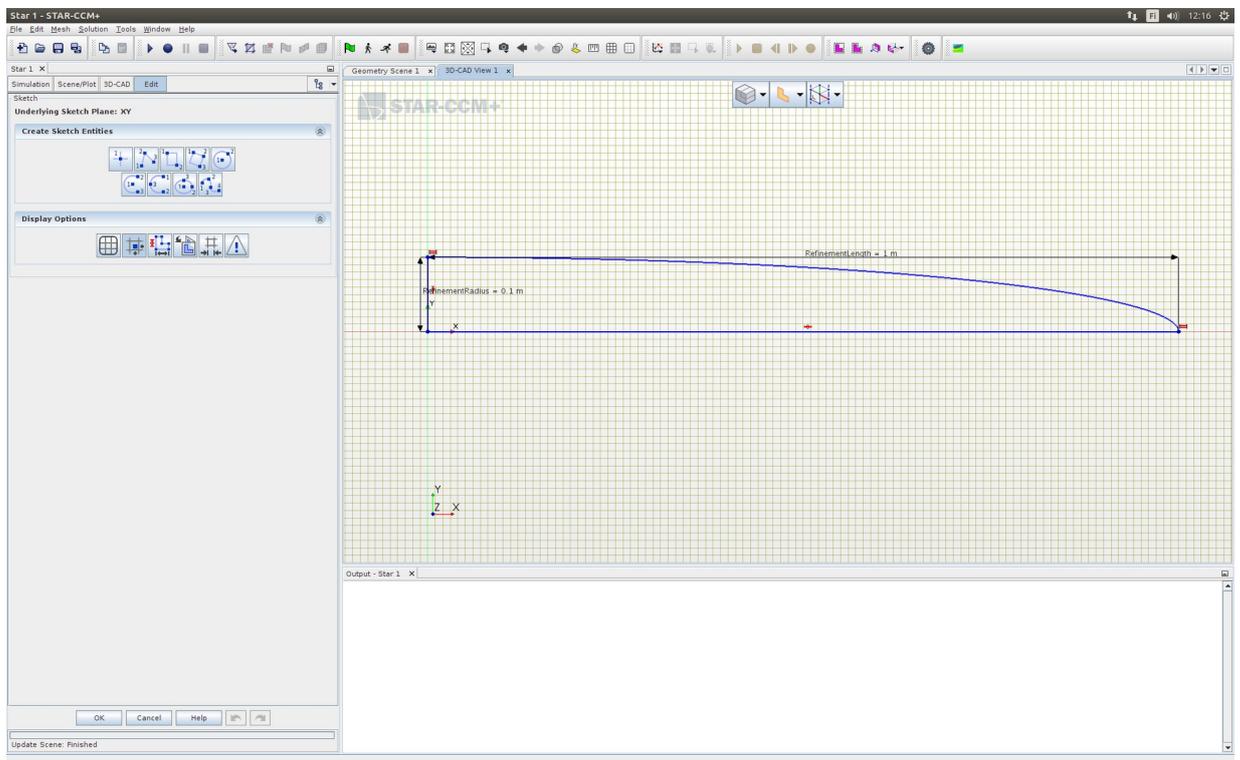
4. Choose the ellipse and the two lines by ctrl-clicking on them. Right-click the ellipse and choose *Split for trim*. This splits the ellipse into two parts.
5. Right-click the left lower half of the ellipse and choose *Delete*.



6. Choose the end point of the vertical line and the end point of the ellipse arc below this. Right-click one of the points and choose *Apply coincidence constraint*.

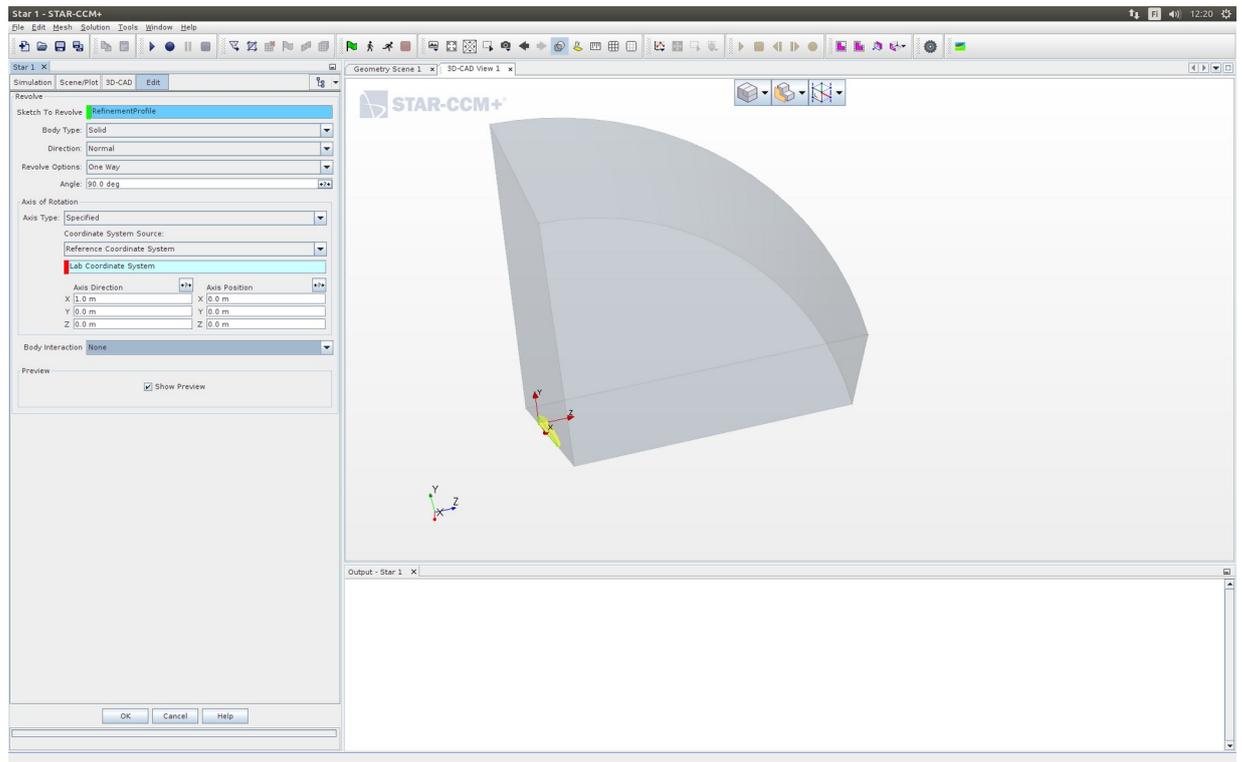


7. Repeat this for the right hand end points of the horizontal line and the ellipse arc.
8. Apply length dimension for the horizontal and vertical line and give appropriate names for the lengths.

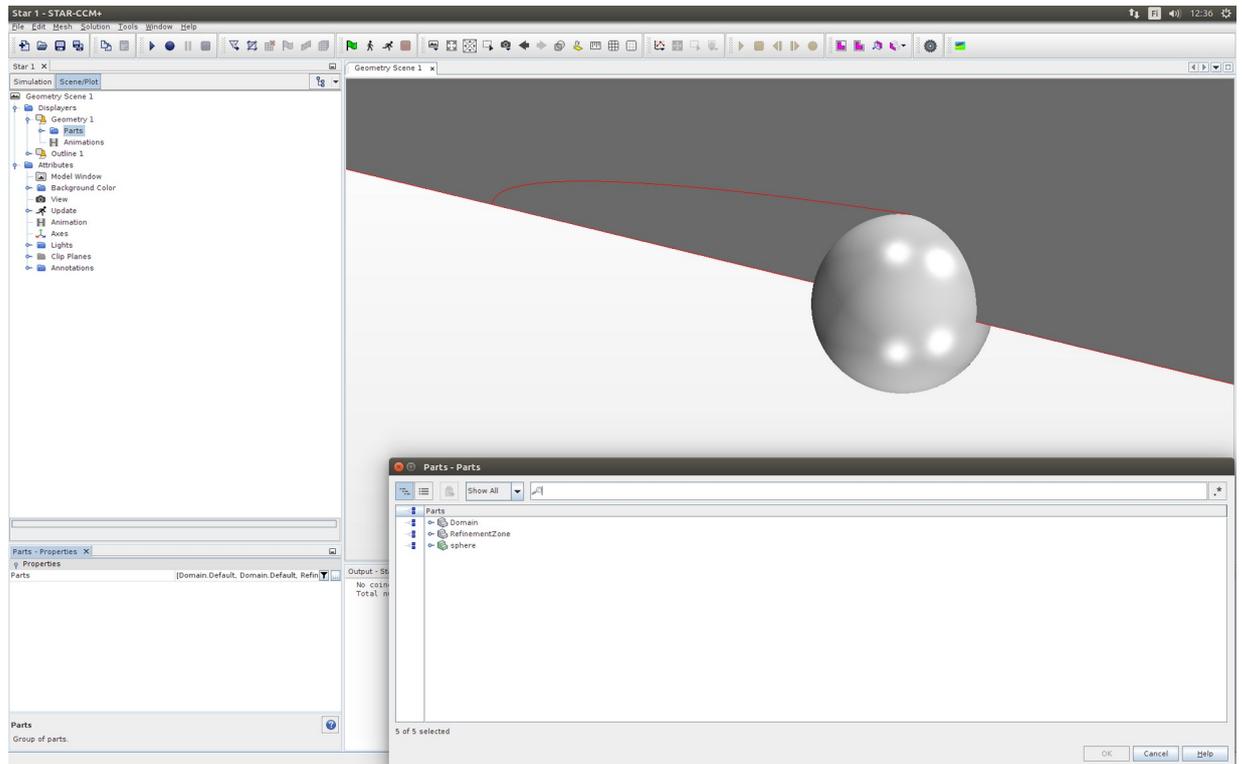


9. Close the sketch view.
10. Rename the sketch.
11. Right-click the refinement profile and choose *Revolve*.

12. Use the defaults, but for the *Body interaction* option choose *None*. Otherwise the fluid domain and the refinement bodies will be merged into a single body, which we do not want.



13. Rename the refinement body under the *Bodies* node.
14. Check that you can see the specified lengths under the *Design parameters* node. You can play around with these by clicking on a parameter and changing the value in the lower left hand corner. The design is updated by clicking the *Update 3D-CAD* button. You can see the bodies, if you reactivate the *Solid bodies* option under the Model visibility icon of the 3D view.
15. Close the 3D-CAD view.
16. Rename the *3D-CAD Model 1* under the *3D-CAD Models* into something sensible (e.g. domains)
17. Right click the renamed CAD model and choose *New geometry part*. Keep the default options in the window. This generates two new parts under the *Parts* node.
18. You can include these into the geometry scene. Choose the *Scene/Plot* tab above the simulation tree. In the geometry scene tree open up *Displayers* → *Geometry 1* node and click the *Parts* node. In the lower left hand corner you have the associated properties. Click the three dots on the *Parts* line under the properties. Activate all the parts by clicking on them. The items are activated, if there are blue marks next to them in the parts selection window.

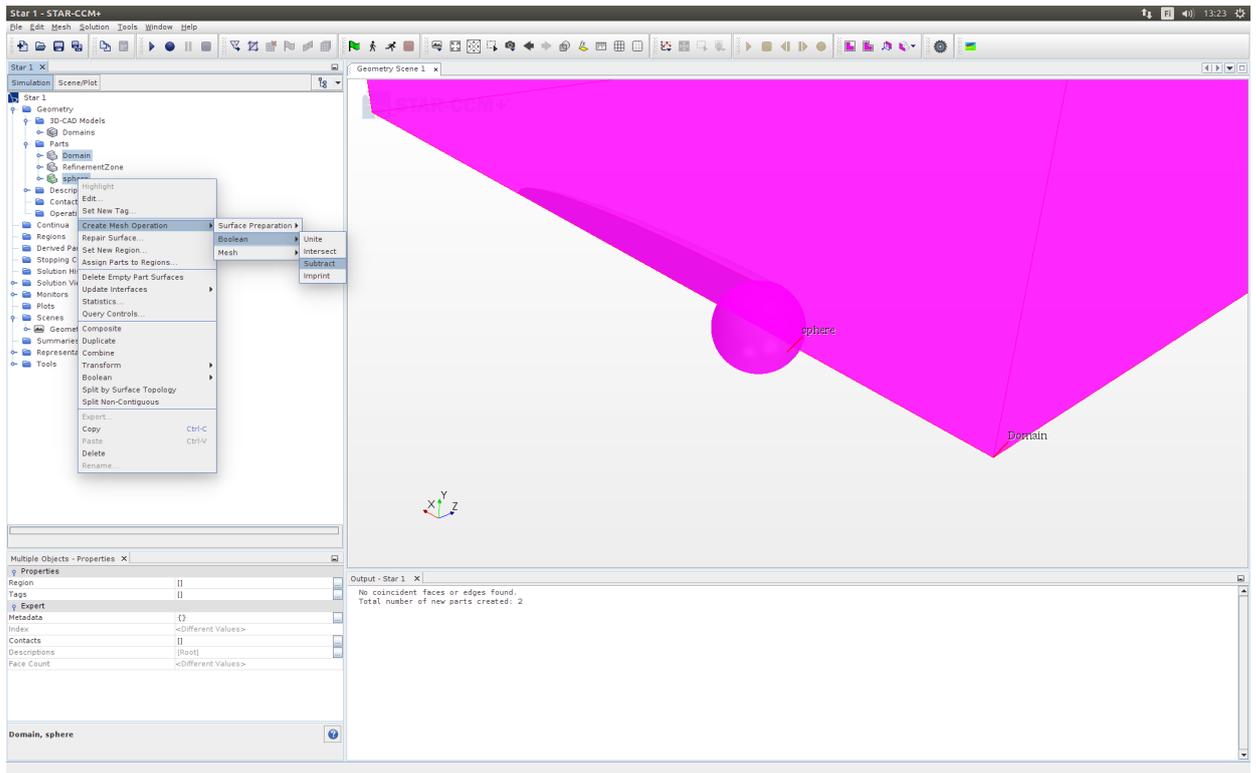


19. You can go back to the simulation tree by clicking on the *Simulation* tab above the tree.

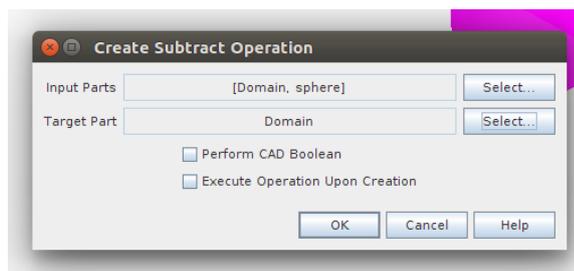
Setting up the fluid domain

The fluid domain should only include the volume occupied by the fluid. Therefore, we need to cut out the spherical volume from the domain that we have specified. This is a simple operation.

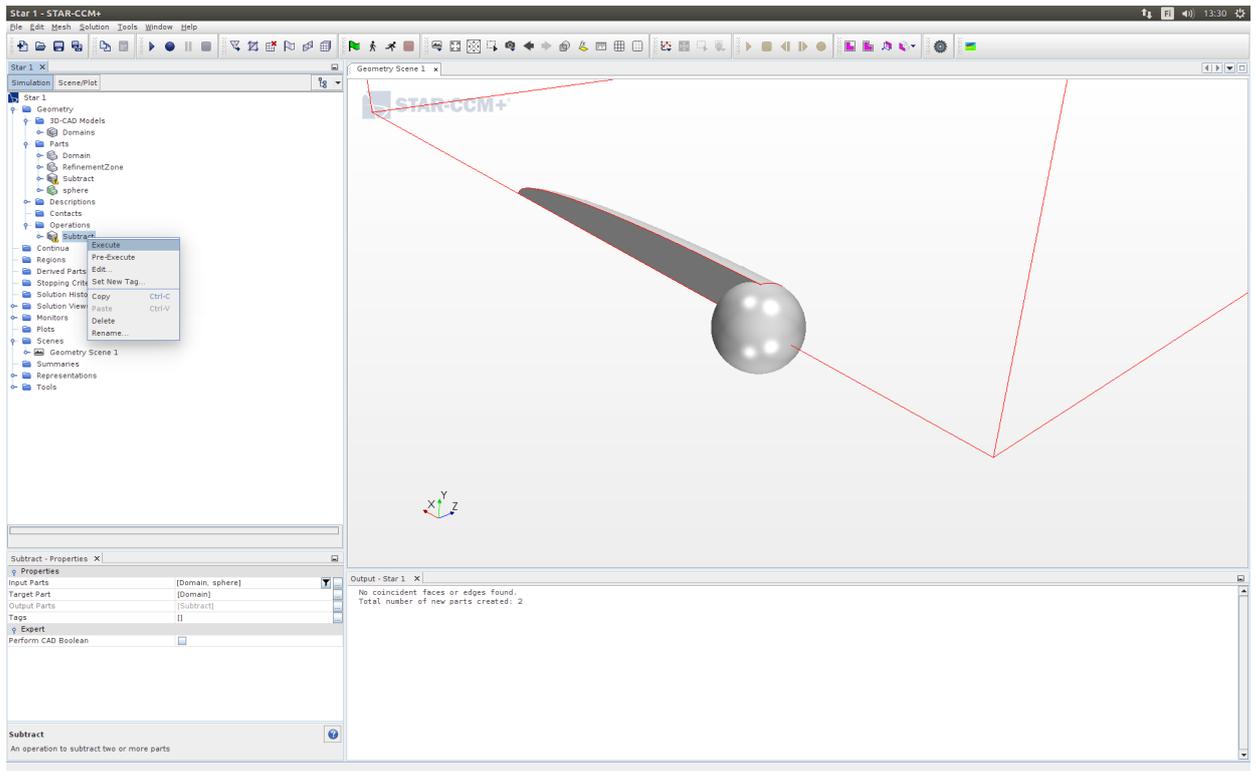
1. Choose both the domain and the sphere under the *Parts* node in the simulation tree (you can choose multiple items by pressing the ctrl-key while clicking with the mouse).
2. Right-click either one of these and choose *Create mesh operation* → *Boolean* → *Subtract*.



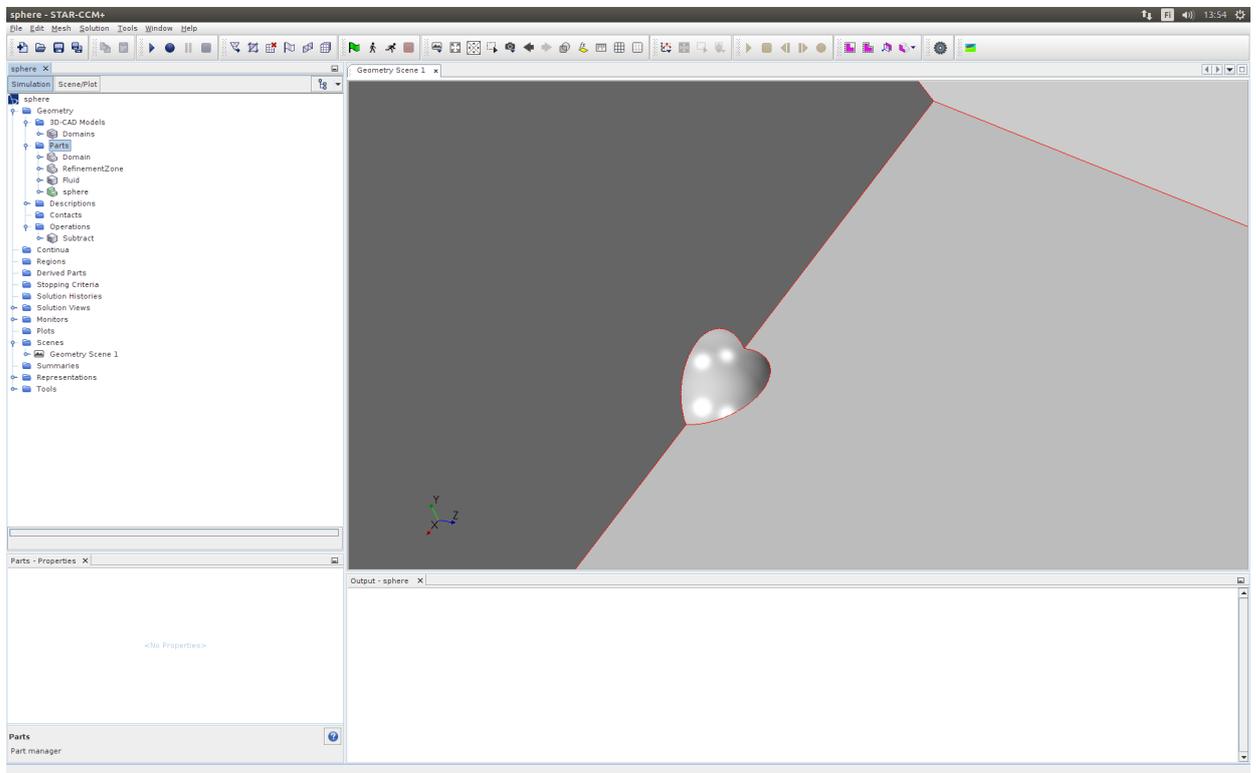
- In the window select the domain as the *Target part*. Click OK.



- This creates a new operation under the *Operations* node called *Subtract*. Right-click this operation and choose *Execute*.

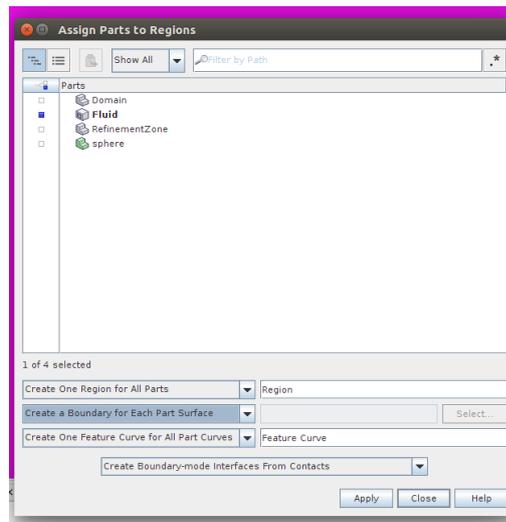


5. This cuts the spherical volume out of the domain. The resulting volume can be found under the *Parts* node. Rename the resulting part into something sensible (e.g. fluid). Go to the *Scene/Plot* tab and activate only the fluid domain under the *Parts* node. Go back to the *Simulation* tab.



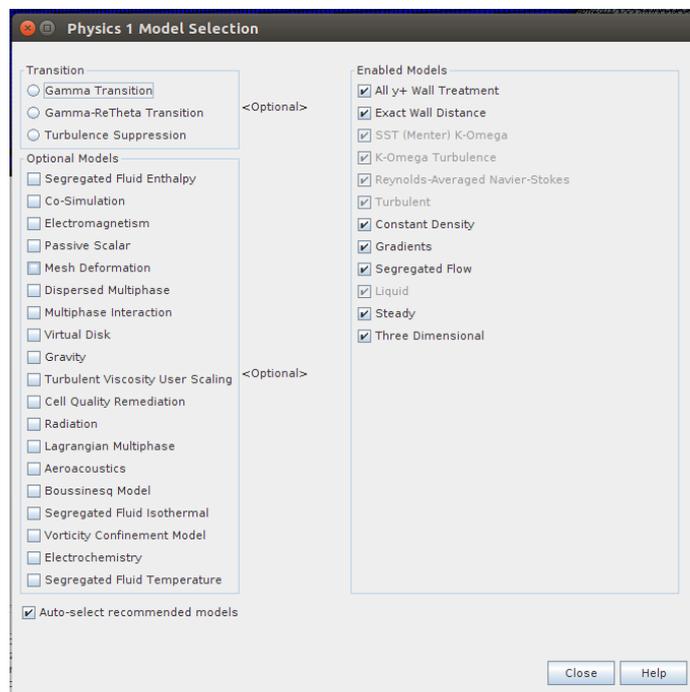
6. Turn the fluid part into a region, i.e. right-click fluid part under the *Parts* node and choose *Assign parts to regions...* from the menu.

- In the window change the second pull-down menu option to *Create a boundary for each part surface*. This will produce one boundary for each of the named faces defined above. Click Apply and close the window.



Setting up the models and boundary conditions

- Right-click *Continua* in the simulation tree and choose *New* → *Physics continuum*.
- Right-click the *Physics 1* node and choose *Select models*. Make the following choices. Selected models are shown in the right column. Note that the column on the left is dynamic, i.e. the list depends on the previous choices that you have done. Close the window.



- Open the *Regions* → *Region* → *Boundaries* node. Click each boundary one by one and set the *Type* under the properties according to the following table.

External

Velocity inlet

Symmetry	Symmetry plane
Inlet	Velocity inlet
Outlet	Pressure outlet
Sphere	Wall

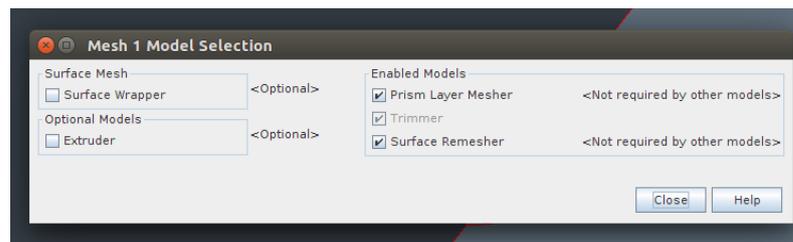
- Open the *Tools* node, right-click *Field functions* and choose *New* → *Scalar*
- Rename the field function under the tree as *Radius* and under the *Properties* part in the lower left hand corner specify the *Function name* as *Radius*. Click the three points next to the *Dimension* field and set the *Length* option as 1 (i.e. this has units of length). For the function *Definition* set the constant value 0.1 (the radius of the sphere). This function can now be used in any of the numeric fields related to length.
- We will set-up the velocity according to the Reynolds number referred to the sphere diameter. In order to do this we need to specify some additional variables. Under the *Tools* → *Field functions* node, specify the following new scalars.

Field function	Function name	Definition
Fluid density	rho	1000
Fluid dynamic viscosity	mu	1E-3
Reynolds number	Re	1.14E6
Inlet velocity	V_inf	$\frac{\text{Re} * \mu}{\rho * 2 * \text{Radius}}$

- Go through the boundary conditions under the *Physics conditions* and *Physics values* nodes under the *Regions* → *Region* → *Boundaries* node for each boundary. The default values for the turbulent quantities and pressure are OK. The velocity should be changed to refer to the *Inlet velocity* variable specified above. Velocity is used for both the inlet and the external boundary.
 - For the inlet boundary check that the *Velocity specification* under the *Physics conditions* node is set as *Magnitude + Direction* and that the *Flow direction specification* is set as *Boundary-Normal*. This means that the velocity is normal to the boundary and you will only give the magnitude of the velocity. Under the *Physics values* node set the *Value* for the *Velocity Magnitude* as V_{inf} .
 - For the external boundary set the *Velocity specification* as *Components* and under the *Physics values* node set the *Value* for the *Velocity* as $[V_{inf}, 0.0, 0.0]$.
- Check that the initial conditions for pressure and turbulent quantities under the *Continua* → *Physics 1* → *Initial conditions* node match the boundary conditions. Set the initial condition for *Velocity* as $[V_{inf}, 0.0, 0.0]$.
- Set the *Density* and *Dynamic viscosity* for the fluid under the *Continua* → *Physics 1* → *Models* → *Liquid* → *H2O* → *Material properties* node using the variables ρ and μ above.

Setting up the mesh

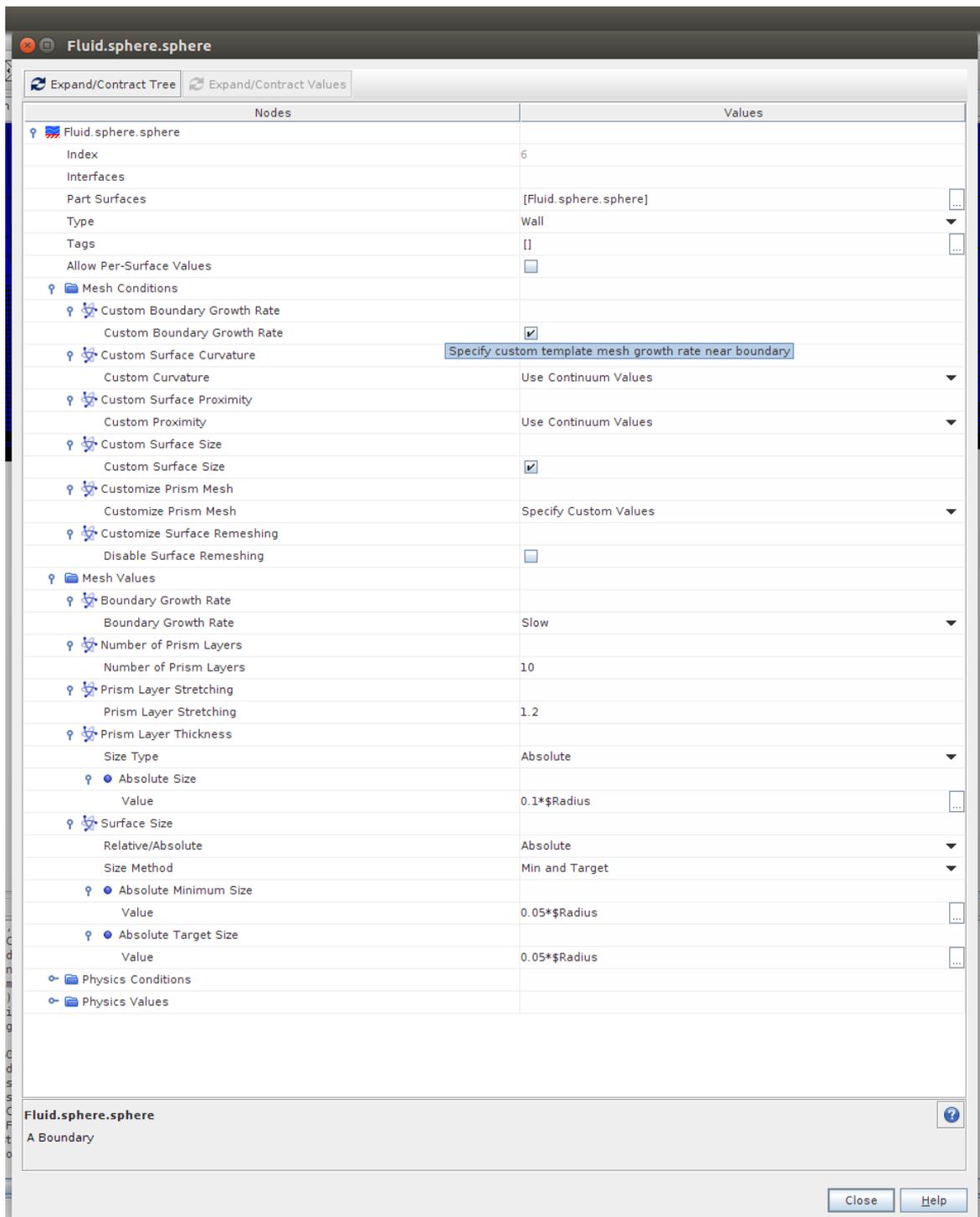
1. Right-click *Continua* in the simulation tree and choose *New* → *Mesh continuum*. This will generate a new mesh node under the *Continua* node.
2. Right-click *Models* under the *Mesh 1* node and choose *Select meshing models*.
3. In the window choose the following options in this order: *Surface remesher*, *Trimmer*, *Prism layer mesher*. We will use the surface remesher to remesh the sphere surface (stl-surface is already a mesh, but we do not want to use this). Trimmer will generate an unstructured cut-hexahedra mesh for majority of the volume. Prism layer mesher will be used to generate a boundary layer mesh close to the sphere surface.



4. Adjust the domain and refinement zone size based on the radius of the sphere. Find the *Design parameters* node under the *Geometry* → *3D-CAD models* node. In the properties section of each parameter give the following *Values*.

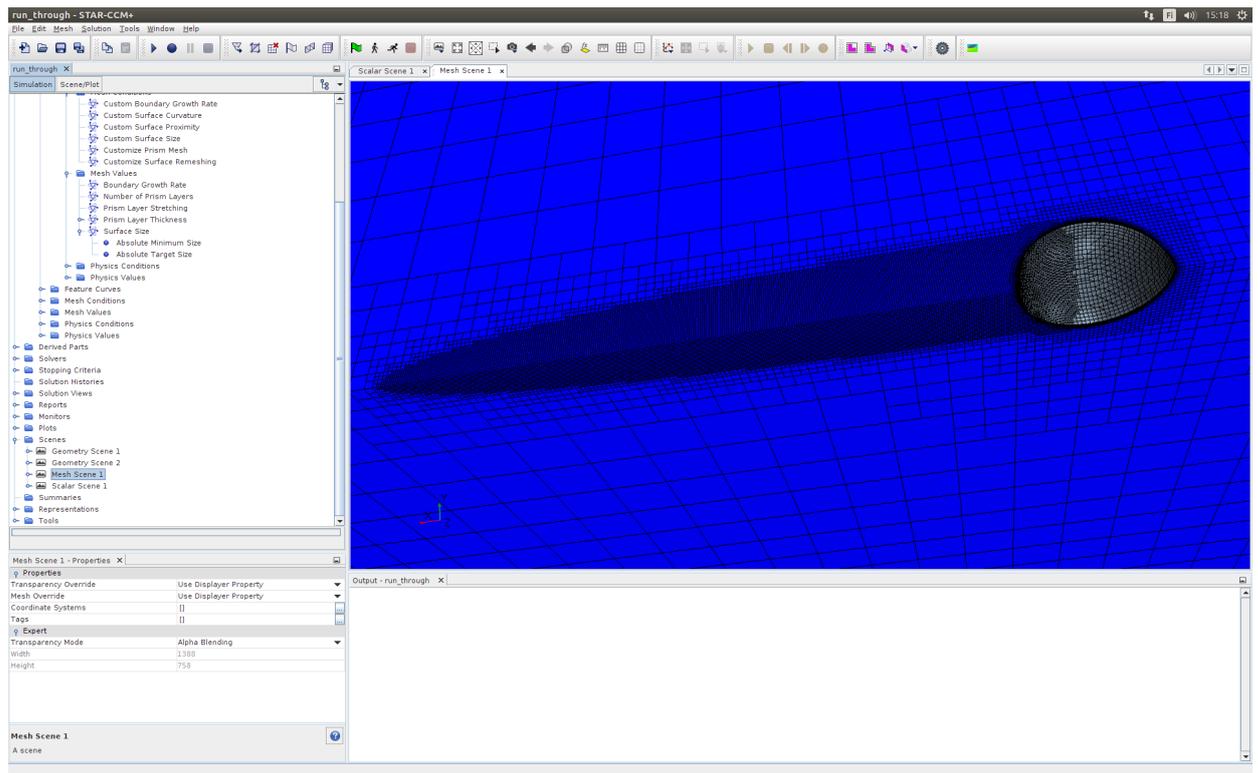
Domain radius	$60 * \$Radius$
Distance of inlet	$10 * \$Radius$
Distance of outlet	$40 * \$Radius$
Length of refinement zone	$10 * \$Radius$
Radius of refinement zone	$\$Radius$

5. You will notice that there are exclamation marks next to several of the geometry objects. This means that the specification has been changed, but has not been applied yet. Right-click the fluid domain node under *Geometry* → *Parts* node and choose *Update*.
6. Under *Continua* → *Mesh 1* → *Reference values* node set the *Base size* value equal to $\$Radius$. This will specify the base size of the cells in the volume grid. We will modify this with refinements on the sphere surface and inside the refinement zone in the wake.
7. The mesh generator will use the boundary condition information, when deciding which method to use on different surfaces. Surface remeshing and prism layer generation will be done only on wall type boundaries. Because we have just one wall type boundary, we could specify the options for these tools under the mesh continuum node. However, in the following we will specify the meshing options directly for the sphere surface. Under the *Regions* → *Region* → *Boundaries* node set the following options for the sphere boundary. You can modify everything at once by right-clicking the boundary and choosing *Edit*. You might get warnings, that the expressions using the sphere radius cannot be evaluated, but these can be ignored.



8. Right-click the *Continua* → *Mesh 1* → *Volumetric controls* node and choose *New* from the menu.
9. In the *Properties* choose the refinement part as *Part*.
10. Under the *Mesh conditions* → *Trimmer* node check *Customize isotropic size*.
11. Under *Mesh values* → *Custom size* choose *Absolute* as the *Size type* and set the *Absolute size* as $0.05*\$Radius$.
12. Generate the mesh by clicking the *Generate Volume Mesh* icon on the toolbar or by choosing the same option from the *Mesh* menu at the top of the window.

13. You can visualise the mesh by right-clicking on the *Scenes* node and by choosing *New scene* → *Mesh*. The result should look something like this.

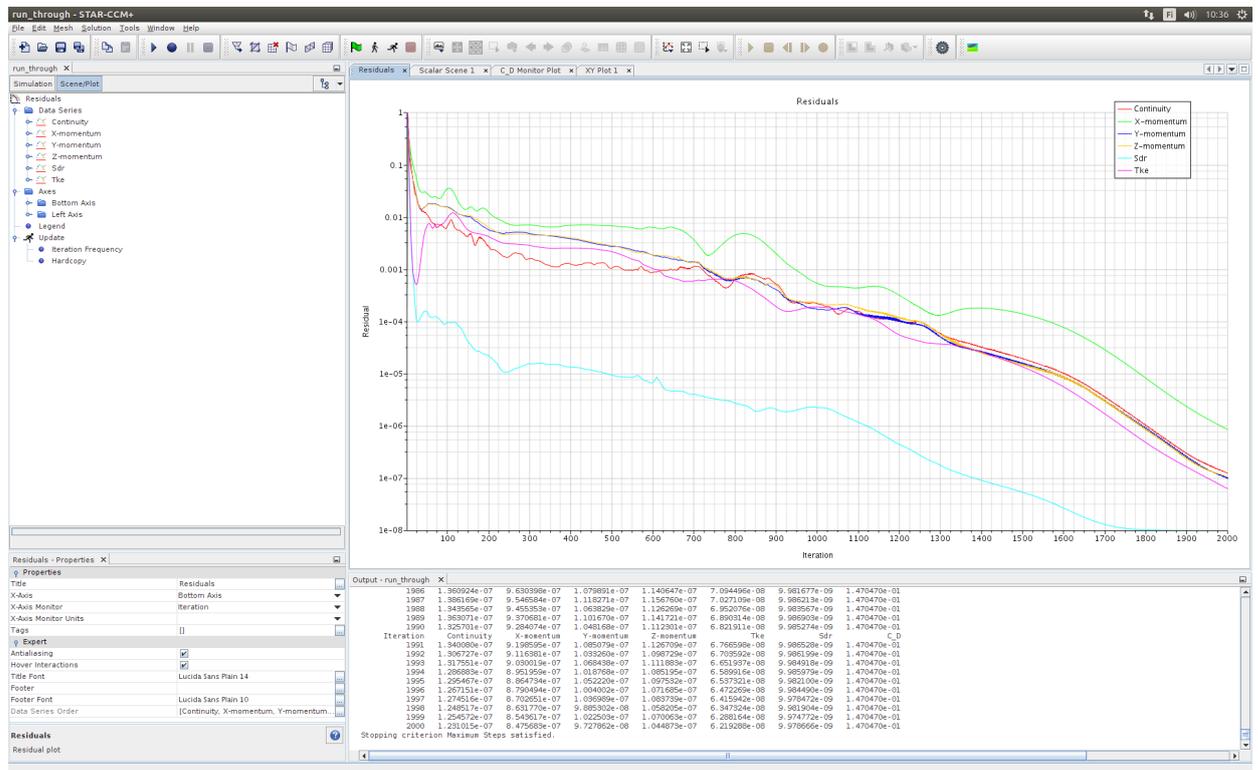


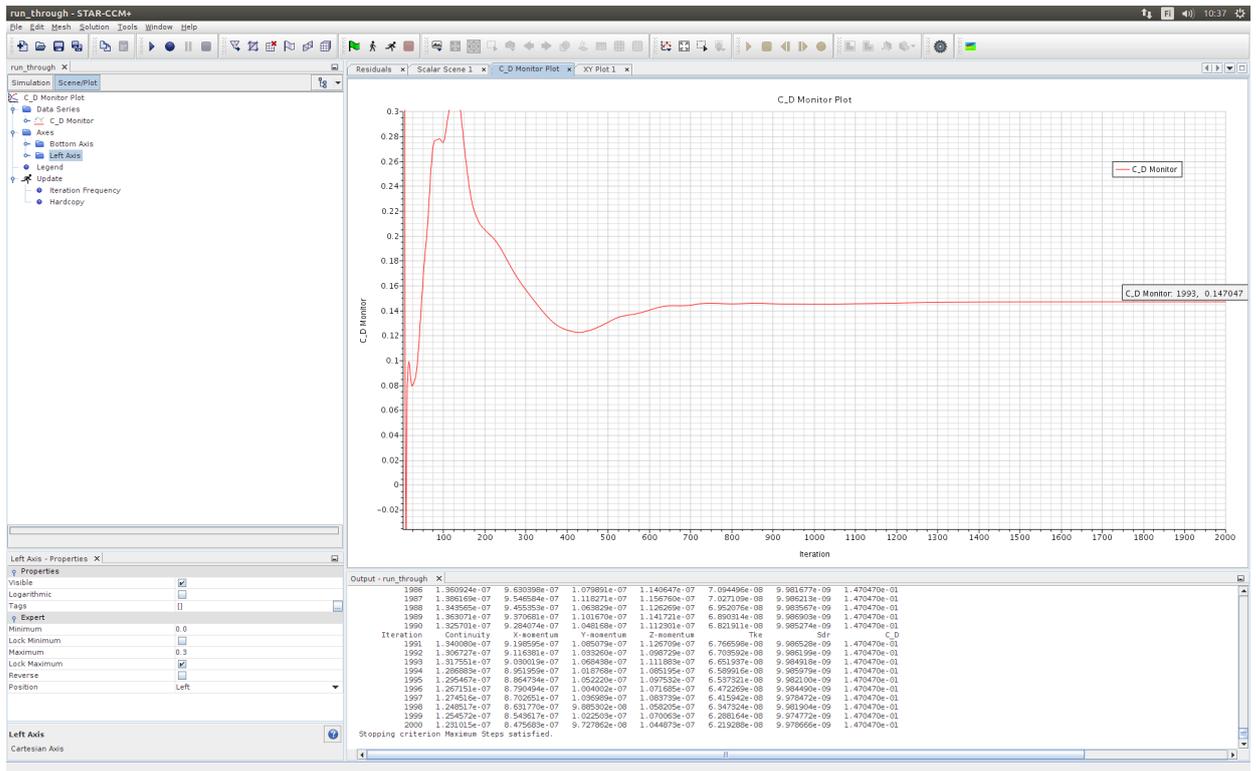
Running the simulation

We are now almost ready to run the simulation. The software has set default values for the discretisation operators, solvers for the linear systems and under-relaxation factors. You can find these under the *Continua* → *Physics 1* → *Models* and *Solvers* nodes.

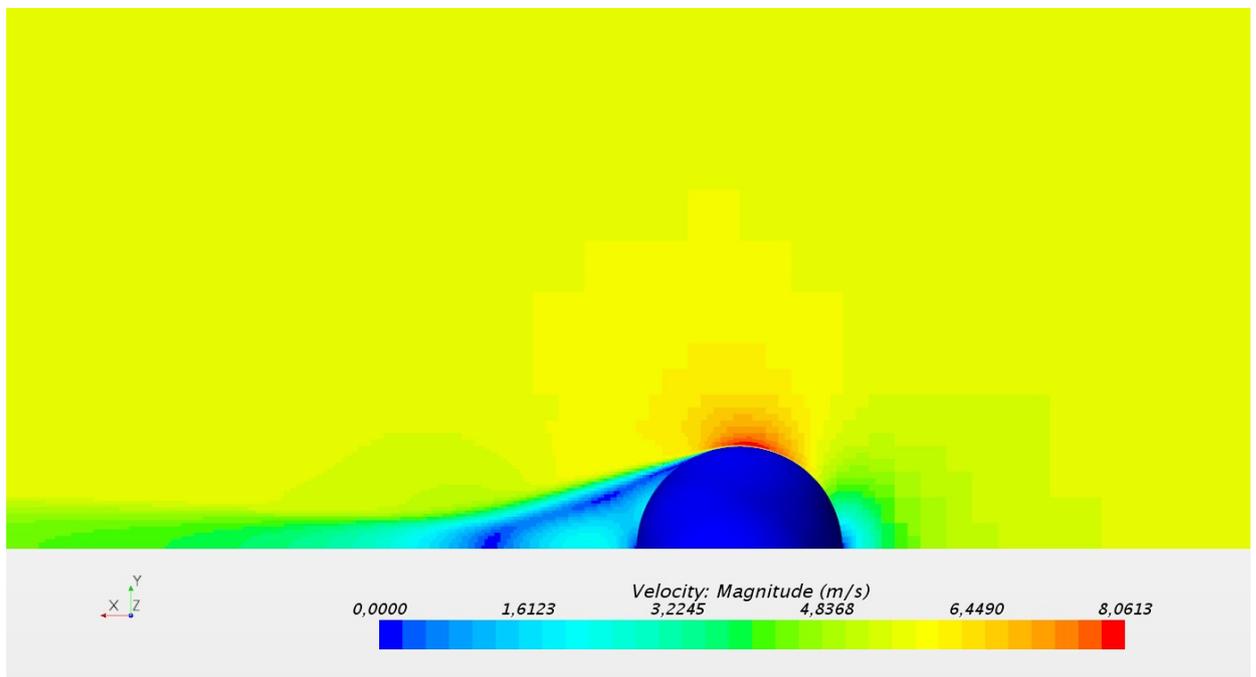
1. Set the number *Maximum steps* under the *Stopping criteria* node to 2000.
2. Right-click the *Scenes* node and choose *New scene* → *Scalar*.
3. Go to the *Scene/Plot* tab.
4. Click the *Outline 1* node and deactivate the *Outline*.
5. Open the *Scalar 1* node and choose the fluid region for the *Parts* property.
6. Click *Scalar field* under the *Scalar 1* node and set *Function* as *Velocity* → *Magnitude*.
7. Go to the *Simulation* tab.
8. Right-click the *Reports* node and choose *New report* → *Force coefficient*.
9. Rename the *Force coefficient 1* report as *C_D*.
10. Click the report and set the *Reference density* and *Reference velocity* using the specified variables ρ and V_{inf} . Set the *Reference area* as $\text{acos}(-1) * \text{pow}(\text{Radius}, 2) / 4$. This is the frontal area of the sphere. The acos-function gives pi and the division by four is required as we have only one quarter of the sphere in our simulation. For the *Parts* choose the sphere surface under the *Regions* → *Region* → *Boundaries* node in the window that opens.

11. In order to monitor the evolution of the drag coefficient during the simulation right-click the *C_D* report and choose *Create monitor and plot from report*.
12. This opens a new tab for the monitoring of the drag coefficient. Click the *Scene/Plot* tab and set the *Maximum* for the *Left axis* property as 0.3. The drag coefficient is very large at the beginning of the simulation and by default the axis is scaled based on the minimum and maximum values of the history, which will mask the evolution of the drag coefficient towards the end of the simulation.
13. Click the *Run* icon in the top toolbar (looks like a running man) to start the simulation. As the simulation starts, a new tab opens up showing the convergence of the residuals of the different equations. You can study the evolution of the solution by changing between the tabs. You can also stop the simulation at any point by clicking the Stop icon in the top toolbar. While the simulation is not running, you can save the simulation project and this will also store the current state of the solution and the convergence histories. The convergence histories for the residuals and the drag coefficient should look something like this.





The field for the velocity magnitude should look like this with a large area of separated flow behind the sphere. You can play around with the plotting options under the *Scalar 1* node in the *Scene/Plot* view.

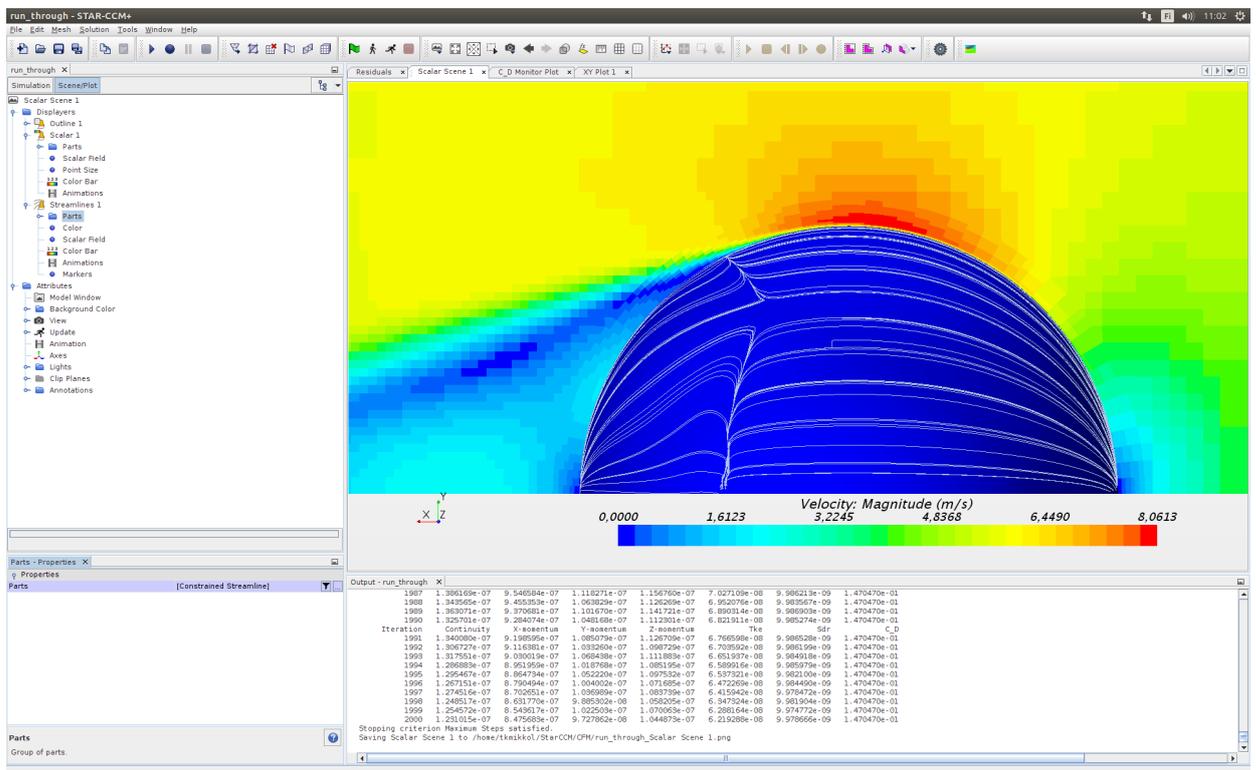


Analysing the results

There are various options to analyse the results within StarCCM+. You can study e.g. scalar and vector fields by generating additional scenes by right-clicking the *Scenes* node. Cross-sections and e.g. streamlines can be produced under the *Derived parts* node. As an example we will produce

streamlines on the sphere surface (limiting streamlines) and plots of the pressure and shear stress distribution on the surface.

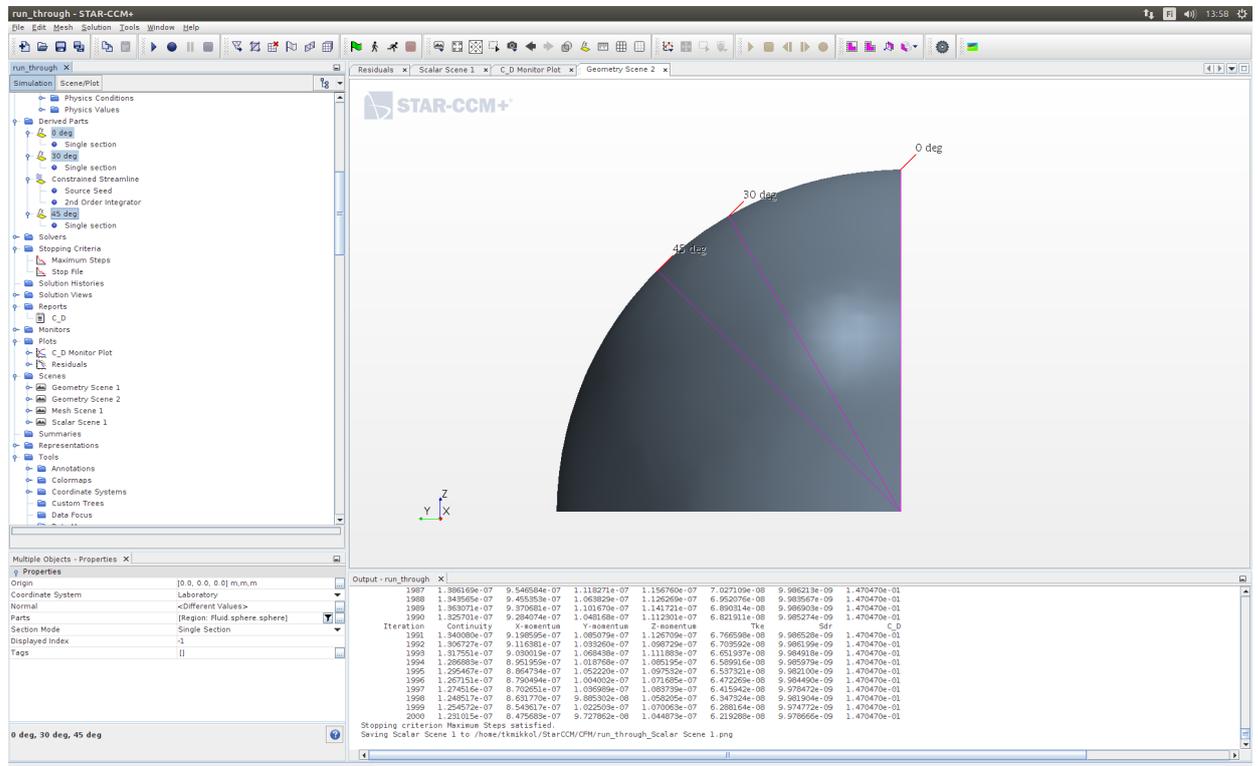
1. In the simulation view right-click the *Derived parts* node and choose *New part* → *Constrained streamline*. Select the sphere under the *Regions* node as both the *Input parts* and *Seed parts*. You can choose to produce a *New streamline displayer*, but this is not necessary as you can also include the new part in existing scenes. We will choose the latter option, so choose the *No displayer* option, click *Create* and *Close*.
2. You can modify the options of the streamline part under the *Derived parts* → *Constrained streamline* node. Click the *Source seed* node and change the *N grid points* option to *[10,10]*. This will increase the number of streamline seed points.
3. Let's make the streamlines visible in the existing scalar scene. Go to the scalar scene view and click the *Scene/Plot* tab. Right-click the *Displayers* node and choose *New displayer* → *Streamline*.
4. Under the properties of the *Streamlines 1* node set the *Color mode* property as *Constant*. This will use a constant colour for the streamlines rather than colouring them based on a field function.
5. Under the *Streamlines 1* → *Parts* node select the constrained streamline part. The results should look like this.



You can clearly see separated flow with streamlines curving up for the top part and curving down for the lower part. We can see that the symmetry conditions are affecting the results, as the condition is forcing the flow to be parallel to the plane, whereas the separation pattern indicates that there could be a flow component normal to the plane. In effect we are

enforcing some periodicity to the separation pattern and the influence of this could be studied by changing the flow domain.

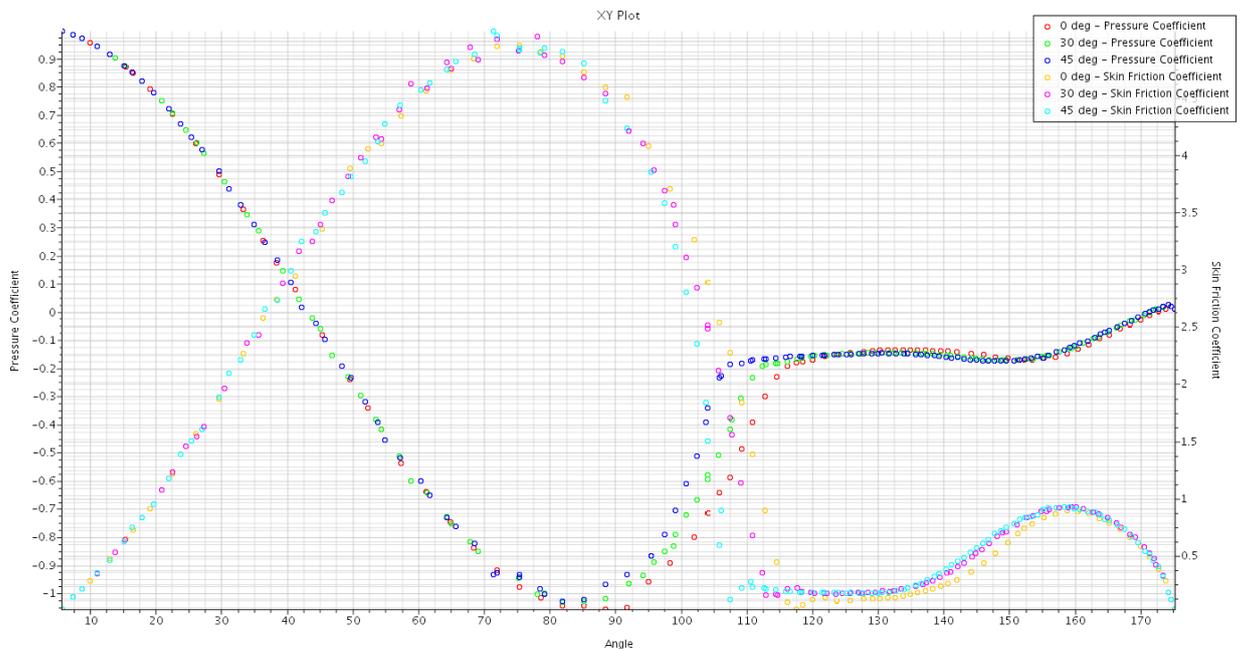
- In order to plot the pressure and skin friction coefficients on the sphere surface, let's first produce the following cross-sections through the sphere surface.



Right-click the *Derived parts* node and choose *New part* → *Section* → *Plane*.

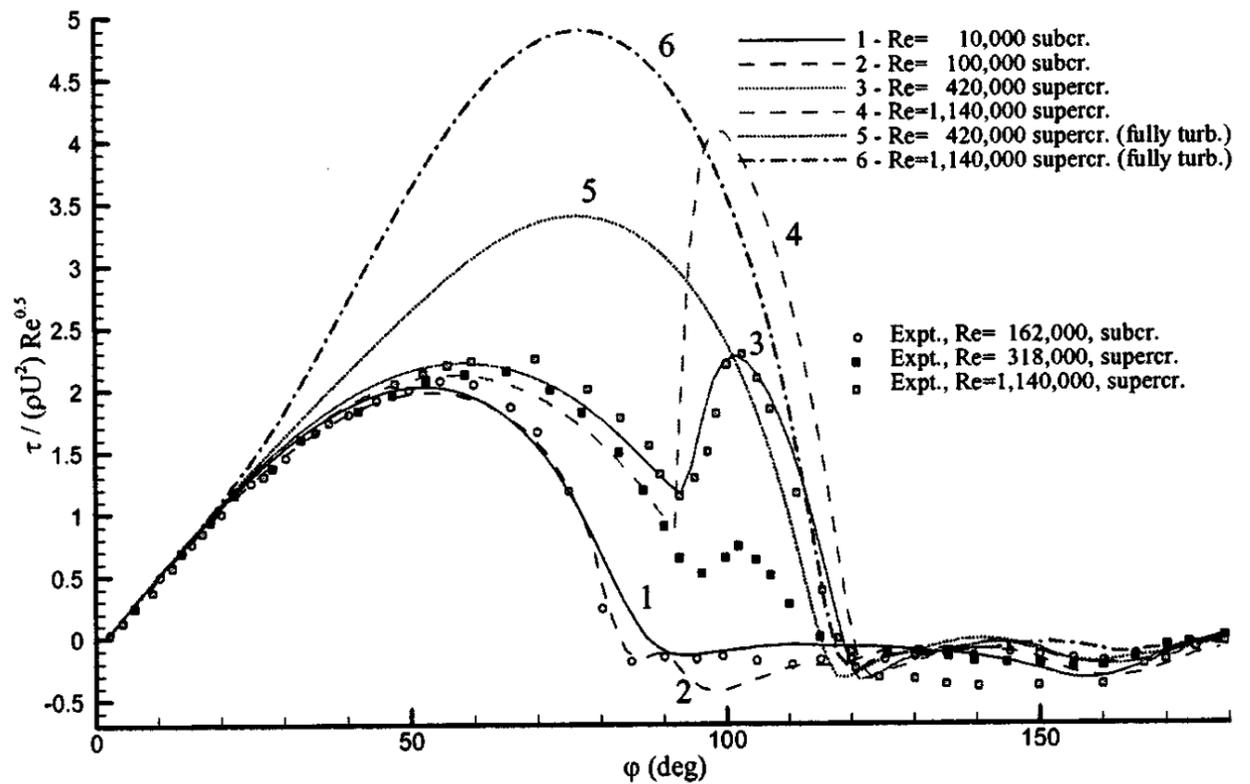
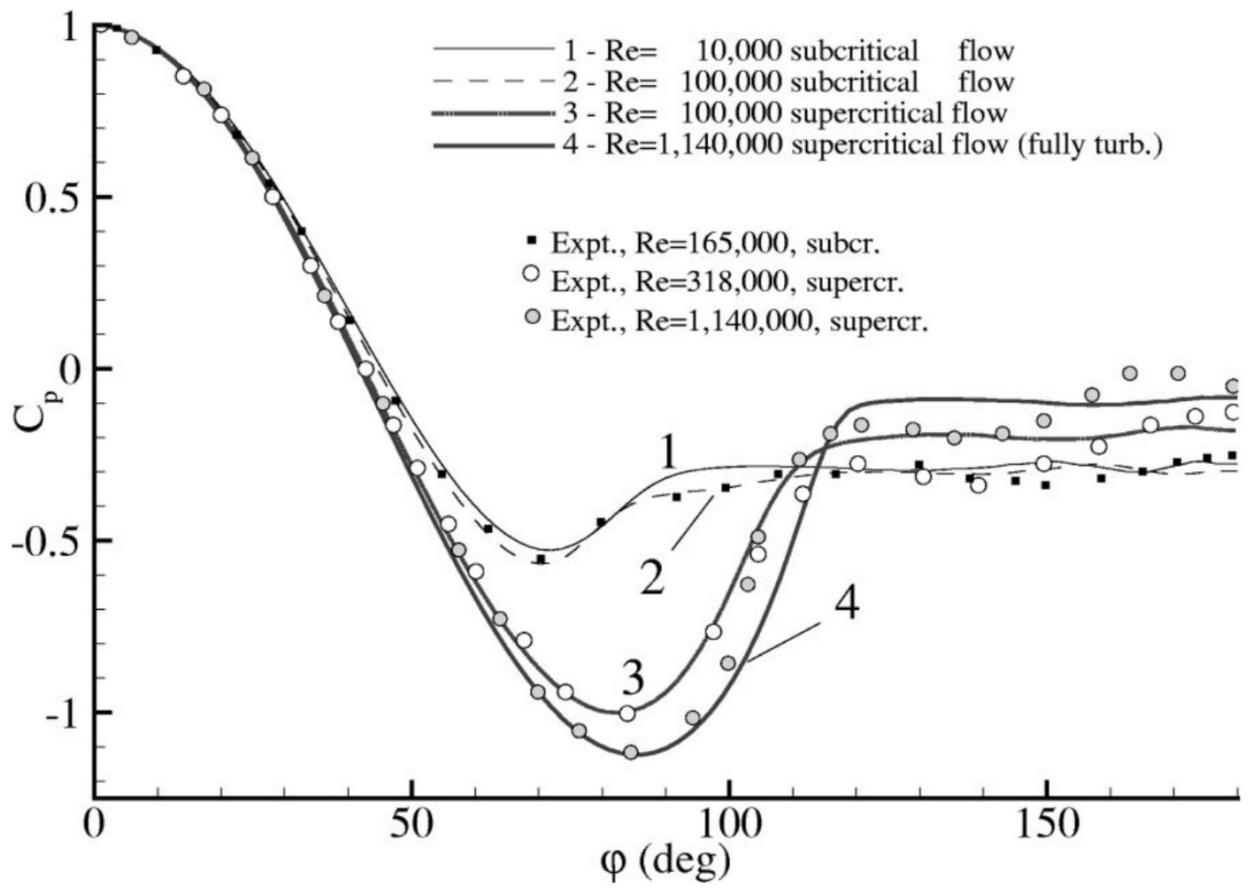
- For the plane set the *Origin* as $[0,0,0]$ and *Normal* as $[0, 1, 0]$ and for the *Parts* property select the sphere surface under the *Regions* node.
- Having the section run exactly at the boundary does not necessarily produce correct data, so let's move the section slightly to guarantee that it is running through the data. Click the *Single section* node under the section and set the *Offset* property as $1E-10$.
- Rename the section as *0 deg*.
- Repeat the procedure for two more sections using *Normal* values of $[0, -1.732, 1]$ and $[0, -1, 1]$ and renaming the sections as *30 deg* and *45 deg*. For these you do not need to set an offset as these sections run through the data in any case.
- Specify a new scalar field function called *Angle* with a *Definition* of $\text{acos}(-\text{\$}\{\text{Position}\}[0]/\text{\$}\{\text{Radius}\})/\text{acos}(-1)*180$. This gives the angle of the point on the surface of the sphere, measured from the stagnation point.
- Click the *Tools* → *Field functions* → *Pressure coefficient* node and set *Reference density* and *Reference velocity* using $\text{\$}\rho$ and $\text{\$}V_{inf}$.
- Click the *Tools* → *Field functions* → *Skin friction coefficient* node and set *Reference density* as $2*\text{\$}\rho/\text{sqrt}(\text{\$}Re)$ and *Reference velocity* as $\text{\$}V_{inf}$.

14. Right-click the *Plots* node and choose *New plot* → *XY plot*.
15. Click the *Scene/Plot* tab.
16. Under the *XY plot 1* node set the *Parts* to include all three cross-sections.
17. Click the *X type* node and set *Type* property as *Scalar*.
18. Click the *Scalar function* node and choose *Angle* as the *Scalar function*.
19. Set the *Scalar function* under *Y types* → *Y type 1* → *Scalar function* as *Pressure coefficient*.
20. Right-click the *Y types* node and choose *New*.
21. Set the *Scalar function* under *Y types* → *Y type 2* → *Scalar function* as *Skin friction coefficient*.
22. Right-click the *Axes* node and choose *New axis* → *Right*.
23. Click the *Y types* → *Y type 2* node and set the *Y-Axis* property as *Right axis*. The resulting plot should look like this.



The pressure distribution agrees quite nicely with the results presented in Constantinescu and Squires (2004), Numerical investigations of flow over a sphere in the subcritical and supercritical regimes, *Physics of Fluids*, Vol. 16, No. 5 shown below. The skin friction distribution is closer to the numerical fully turbulent curve of the same reference, which is understandable as the simulation does not include transition modelling. Based on the experimental results transition seems to occur at around 90 degrees and therefore the simulations are overpredicting the shear stress for the front half of the sphere. Constantinescu and Squires (2004) report that the separation angle for the fully turbulent condition should be 117 ± 2 degrees. The simulation results show that depending on the cross-section, the separation occurs between 107 and 117 degrees. Thus, the separation angle is slightly underestimated. The simulated drag coefficient is 0.147 (right-click the *C_D* report and choose *Run report*), which is overestimating the reference value of 0.102 ± 0.011 reported in Constantinescu and Squires (2004). This might be explained by the

underestimation of the separation angle and the consequent overestimation of the separation zone.



The study should then be extended with a grid resolution study, study on the influence of the choice of the turbulence model and study on the influence of the domain size and boundary conditions.