Cameron Hjeltness

5/4/11

How to creat and define a geometry within Star CCM+

File>>New Simulation.

Accept basic configurations.

Expand Geometry, right click on 3D CAD Models and click create new.

Note: More than likely, you will simply import an existing model from CATIA or Solidworks. To do this, just go File>>Import>>Volume Mesh. The following is information on how to create geometries within Star CCM+.

Right click on one of the planes, and click creat ew sketch. From here you can create geometries in the familiar fashion that you are used to from Solid Works or CATIA. The only difference is that if you want to want to create something new, more often than not, you have to right click to open up a function.
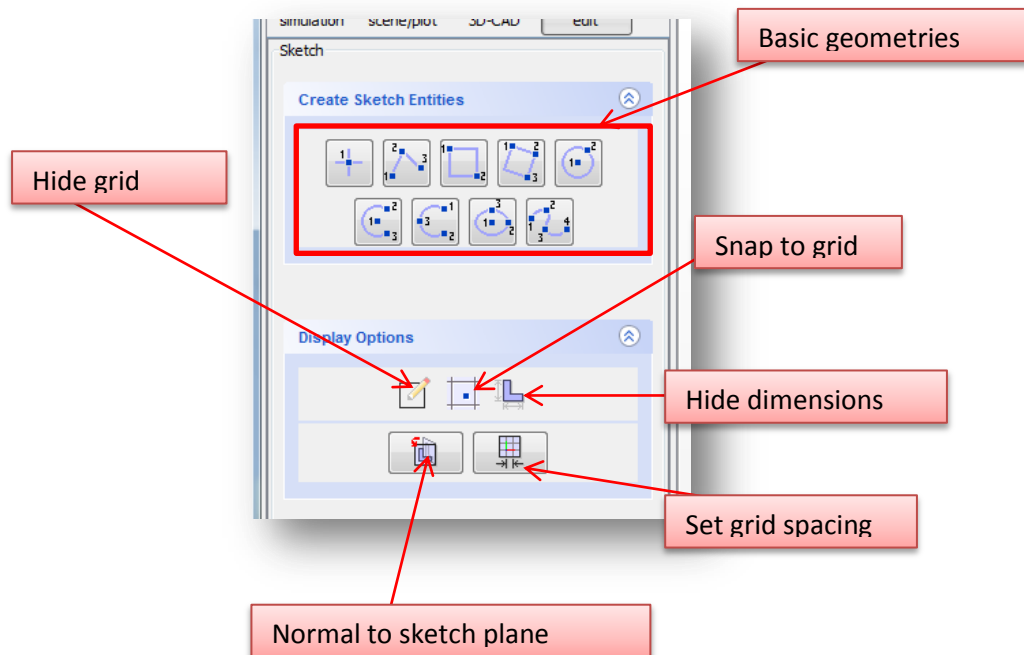

For demonstrational purposes I will generate a simple part with an angled extrude.
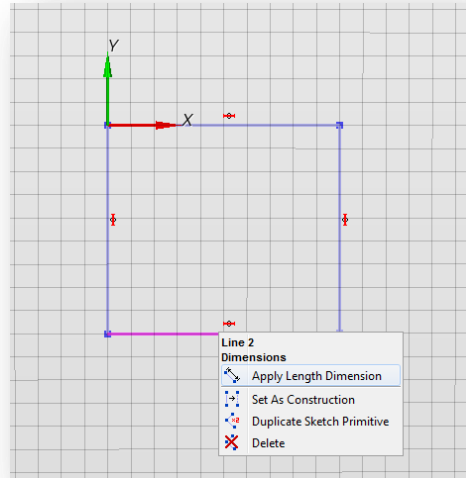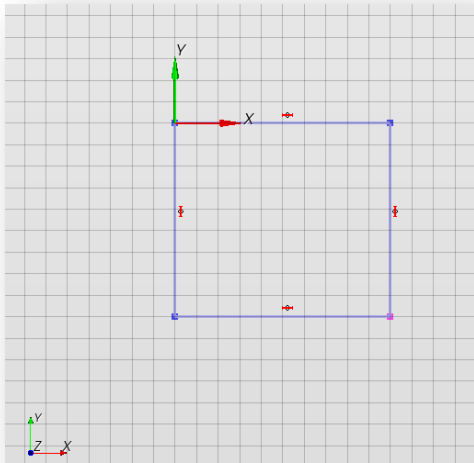
First create the profile. If you want to arbitrarilly redfine the size of the shape, just click and drag. Alternatively, right click and set the dimensions.

Right click on the newly created sketch and create extrude. Note all of the other possibilities in the same menu.

To create another feature on this part like an extrude cut, click on the face you wish to start the sketch from and create a new sketch.
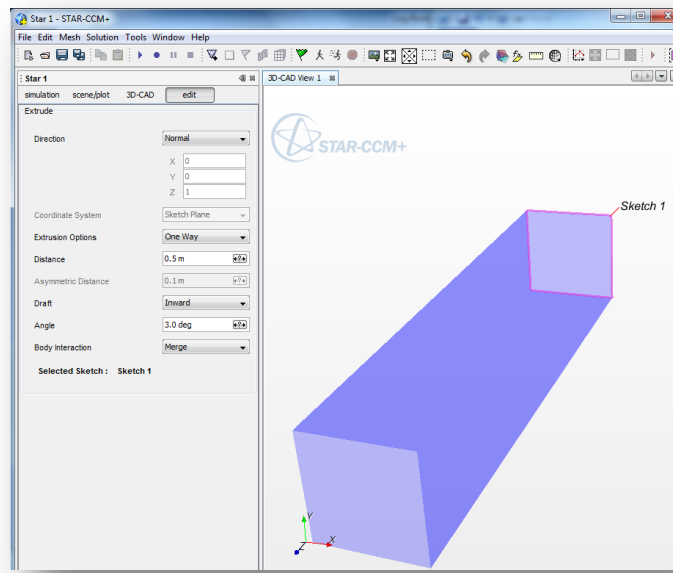
Click OK. Close the 3D-CAD.

Right click 3D-CAD Model1 and select new geometry part. To view this new part, RC (right click) scenes and select new geometry scene. This is where all visualizations will be located. This includes scalar, vector, geometry, mesh and empty scenes.

Expand "Surfaces" and RC on the default surface. Currently there is only one part surface. We want to break this up so that we can define things like the wall and the inlet/outlet ports. Select Split by angle. Define the angle as 45 degrees. This means that any angle that Star CCM detects that is over this value, it will create a new boundary at that point.

You should now have 6 surfaces.

.
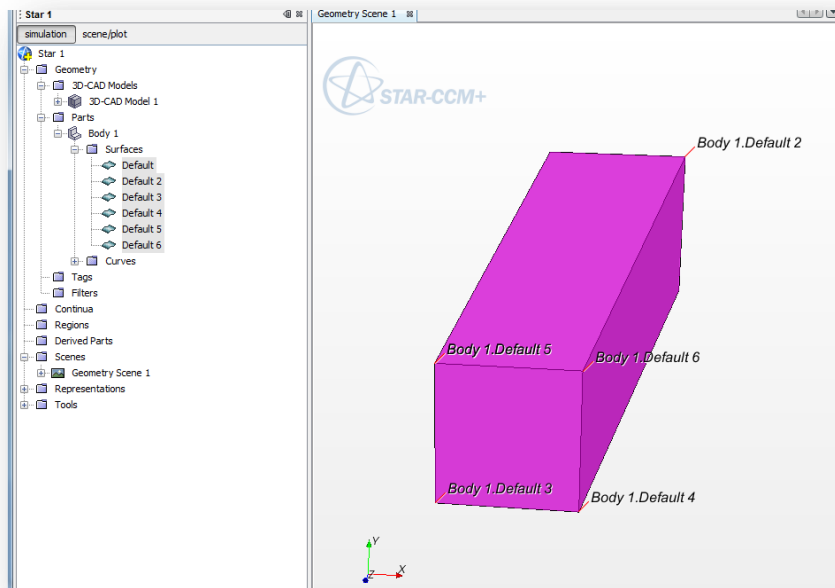
RC regions >> create new.

RC Body 1 >> Set region >> Region 1

RC Boundaries and then create 2 more boundaries. We need 3 total. 1 for the inlet, 1 for the outlet, and 1 for the wall constraining the fluid.

RC Boundary 1, then click the button with 3 dots. Select the 1st, 3rd, 4th, and 5th surfaces. To ensure that you are selecting the correct surfces, click on the individal surfaces and they will highlight in the geometry scene. The objective here is to create the wall for our boundary.

The default selection for this is a wall, which is what we want.
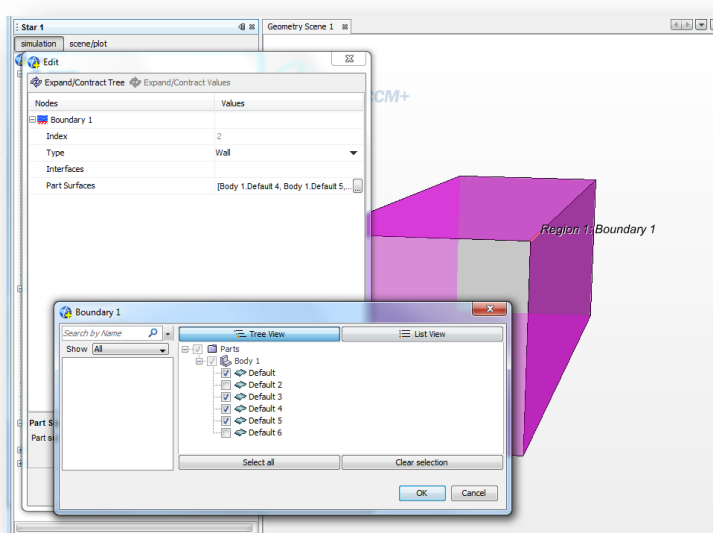
Boudary 2 >> Properties >> select the 2nd surface.

In the properties, make this a velocity inlet.

Default >> Properties >> Select the 6th surface.

In the properties, make this a pressure outlet.