Flow over an Airfoil - Geometry

Author: Benjamin Mullen, Cornell University

Geometry

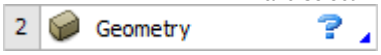
For users of ANSYS 15.0, please check this link for procedures for turning on the Auto Constraint feature before creating sketches in DesignModeler.

Download the Airfoil Coordinates

In this step, we will import the coordinates of the airfoil and create the geometry we will use for the simulation. Begin by downloading this file here and saving it somewhere convenient. This file contains the points of a NACA 0012 airfoil.

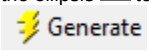
Launch Design Modeler

Before we launch the design modeler, we need to specify the problem as a 2D problem. Right click 

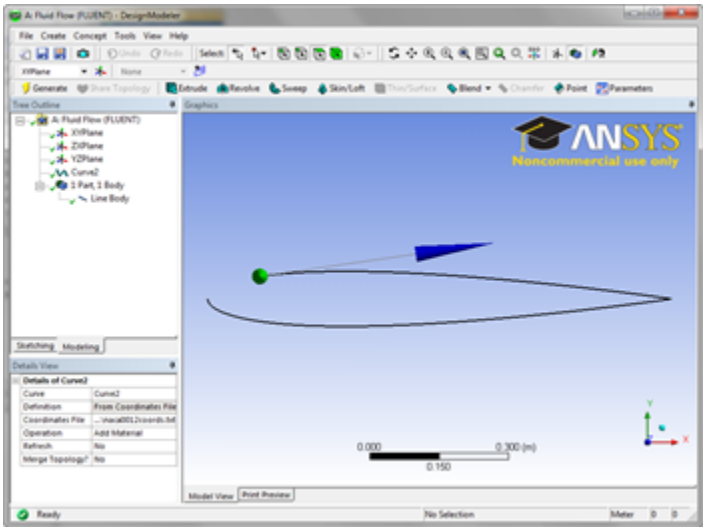
and select ***Properties*** . In the *Properties of Schematic A2: Geometry* Window, select ***Analysis Type > 2D*** . Now, double click 

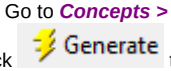
to launch the Design Modeler. When prompted, select ***Meters*** as the unit of measurement.

Airfoil

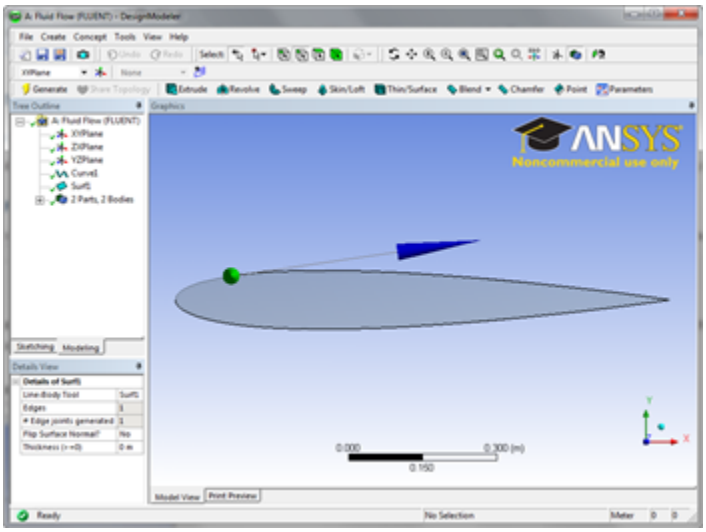
First, we will create the geometry of the airfoil. In the menu bar, go to ***Concept > 3D Curve***. In the *Details View* window, click ***Coordinates File*** and select the ellipsis  to browse to a file. Browse to and select the geometry file you downloaded earlier. Once you have selected the desired geometry file, click 

to create the curve. Click  to get a better look at the curve.



Next, we need to create a surface from the curve we just generated. Go to ***Concepts > Surfaces from Edges***. Click anywhere on the curve you just created, and select ***Edges > Apply*** in the *Details View* Window. Click 

to create the surface.

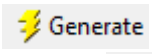


Create C-Mesh Domain

Now that the airfoil has been generated, we need to create the meshable surface we will use once we begin to specify boundary conditions. We will begin by creating a coordinate system at the tail of the airfoil - this will help us create the geometry for the C-mesh domain. Click  to create a new coordinate system. In the *Details View* window, select ***Type > From Coordinates*** . For ***FD11, Point X*** , enter 1.

Details View

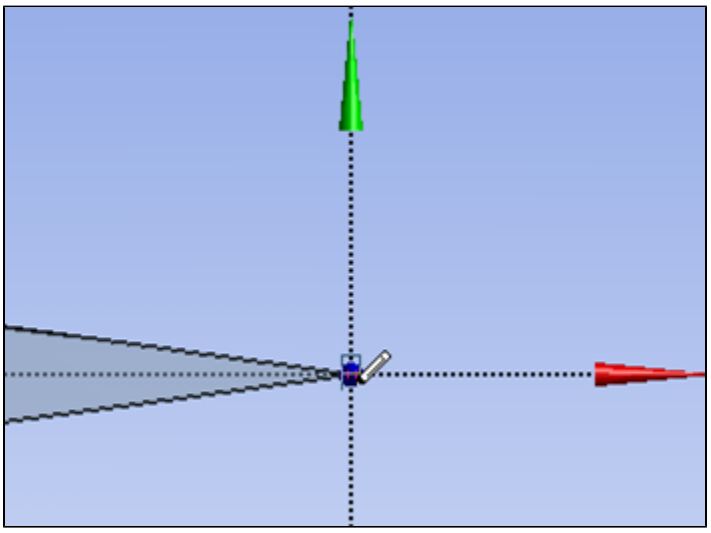
Details of Plane4	
Plane	Plane4
Sketches	0
Type	From Coordinates
FD11, Point X	1 m
FD12, Point Y	0 m
FD13, Point Z	0 m
FD14, Normal X	0 m
FD15, Normal Y	0 m
FD16, Normal Z	1 m
Transform 1 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

Click 

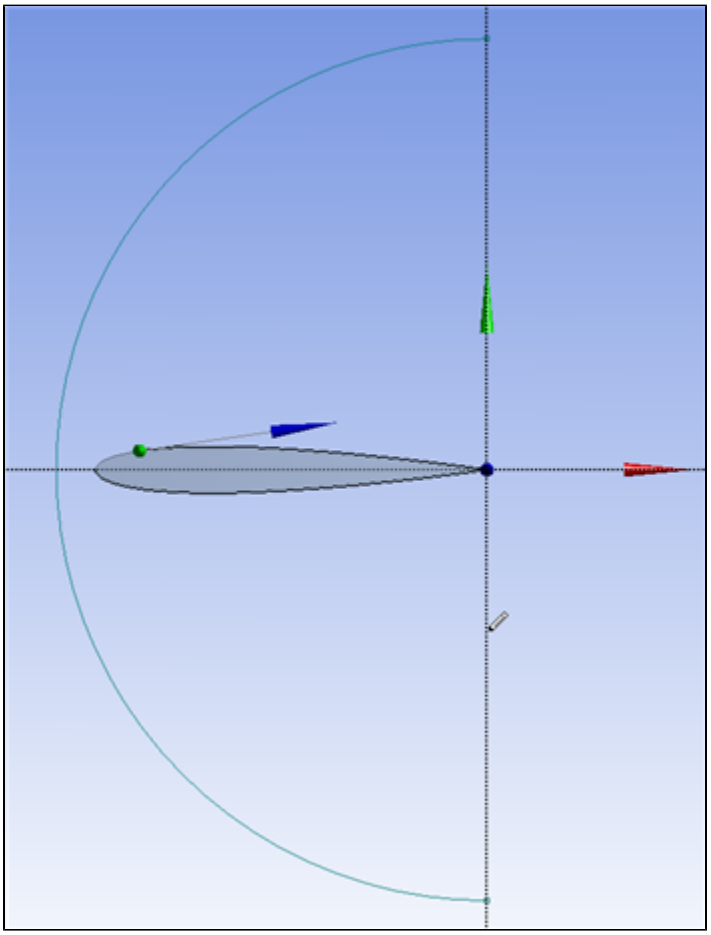
to generate the new coordinate system. In the *Tree Outline* Window, select the new coordinate system you created (defaulted to ***Plane 4***), then click  to create a new sketch. This will create a sketching plane on the XY plane with the tail of the airfoil as the origin. At the bottom of the *Tree Outline* Window, click the ***Sketching*** tab to bring up the sketching window.

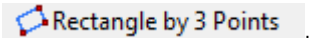
The first action we will take is create the arc of the C-Mesh domain. Click 

. The first click selects the center of the arc, and the next two clicks determine the end points of the arc. We want the center of the arc to be at the tail of the airfoil. Click on the origin of the sketch, making sure the P symbol is showing

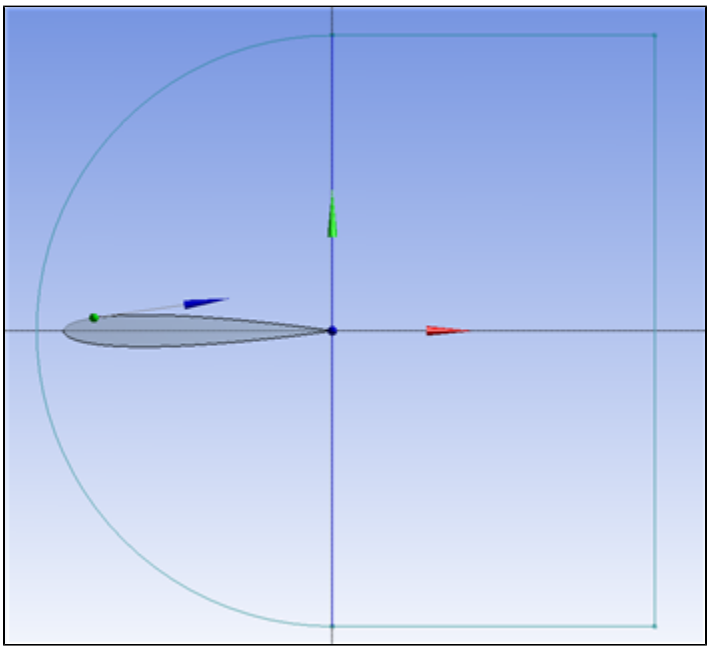


For the end points of the arc, first select a point on the vertical axis above the origin (a C symbol will show), then select a point on the vertical axis below the origin. You should end up with the following:

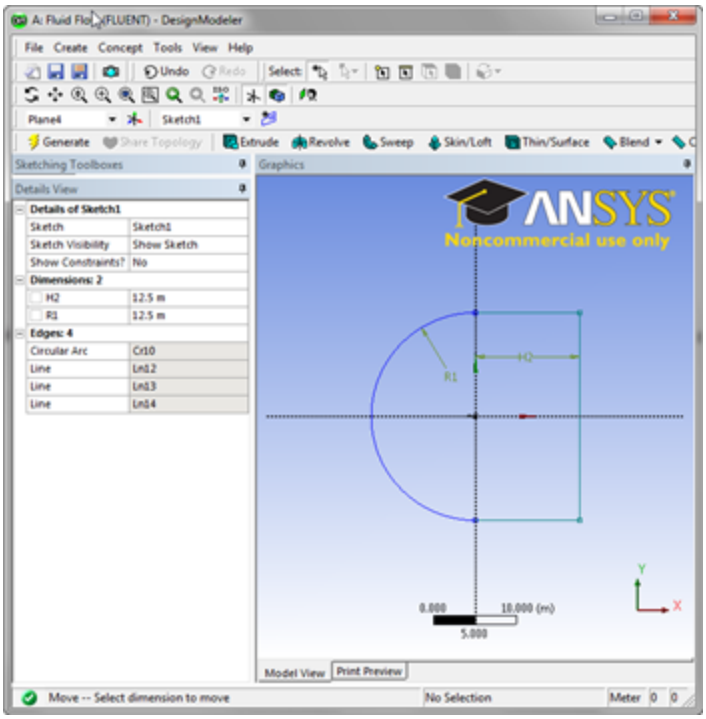


To create the right side of the C-Mesh donain, click 

. Click the following points to create the rectangle in this order - where the arc meets the positive vertical axis, where the arc meets the negative vertical axis, then anywhere in the right half plane. The final result should look like this:



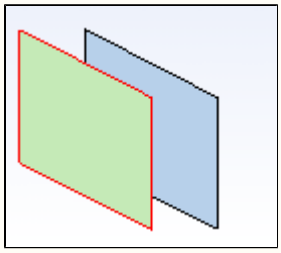
Now, we need to get rid of necessary lines created by the rectangle. Select ***Modify*** in the *Sketching Toolboxes* window, then select Trim. Click the lines of the rectangle the are collinear with the positive and negative vertical axises. Now, select the ***Dimensions*** toolbox to dimension the C-Mesh domain. Click Radius, followed by the arc to dimension the arc. Assign the arc a value of `12.5`. Next, select Horizontal. Click the vertical axis and the vertical portion of the rectangle in the right half plane. Also assign the horizontal dimension a value of `12.5`.



Click here to enlarge the image

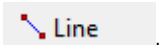
Next, we need to create a surface from this sketch. To accomplish this, go to ***Concept > Surface From Sketches***. Click anywhere on the sketch, and select ***Base Objects > Apply*** in the *Details View* Window. Also, select ***Operation > Add Frozen*** . Once you have the correct settings, click Generate. The final step of creating the C-Mesh is creating a surface between the boundary and the airfoil. To do this, go to ***Create > Boolean***. In the *Details View* window, select ***Operation > Subtract*** . Next, select ***Target Bodies > Not selected*** , select the large C-Mesh domain surface, then click ***Apply*** . Repeat the same process to select the airfoil as the ***Tool Body*** . When you have selected the bodies, click Generate

Selecting the Airfoil Body
Because the C-Mesh domain and the airfoil overlap, once you click in the vicinity of the airfoil ANSYS will select the C-Mesh domain but give you the option of selecting multiple layers

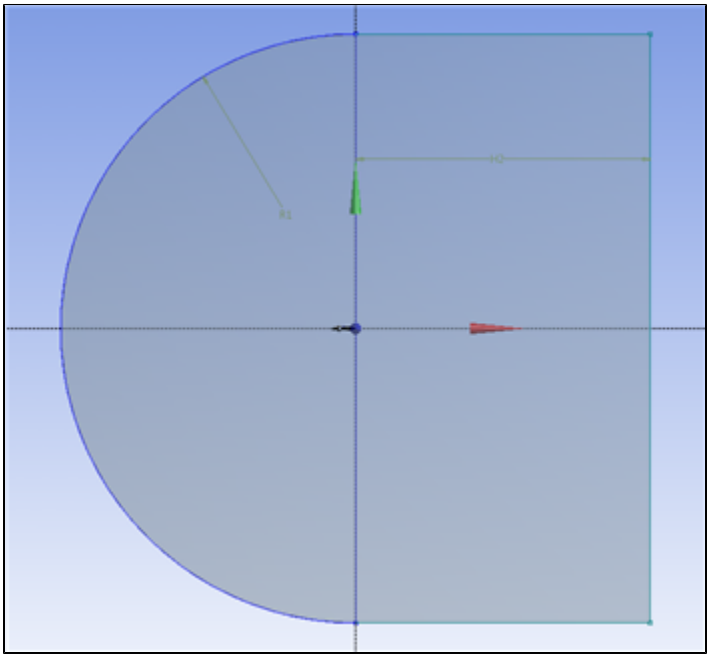


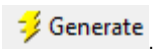
Select the layer that corresponds to the airfoil and the airfoil will be highlighted.

Create Quadrants

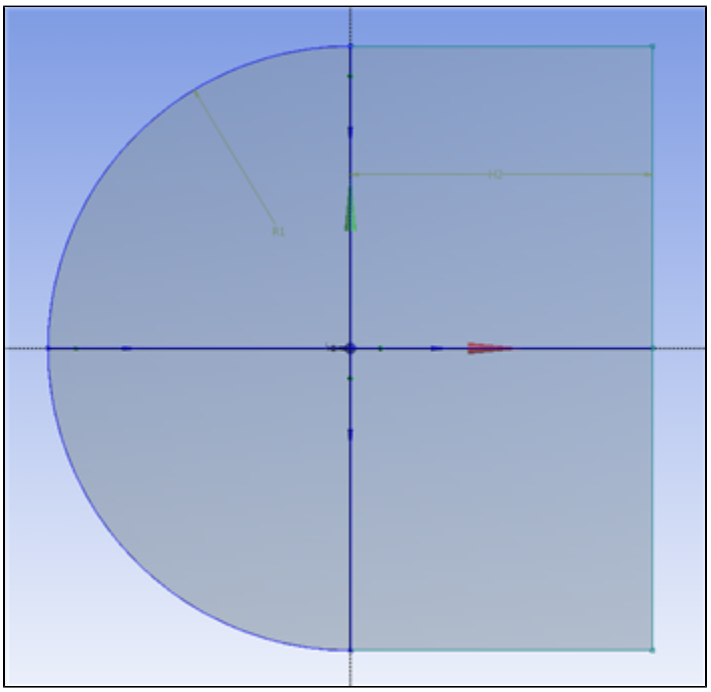
In the final step of creating the geometry, we will break up the new surface into 4 quadrants; this will be useful for when we want to mesh the geometry. To begin, select ***Plane 4*** in the *Tree Outline* Window, and click  . Open the sketching menu, and select 

. Draw a line on the vertical axis that intersects the entire C mesh. Trim away the lines that are beyond the C-Mesh, and you should be left with this:

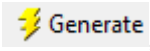


Next, go to ***Concepts > Lines from Sketches***. Select the line you just drew and click ***Base Objects > Apply*** , followed by 

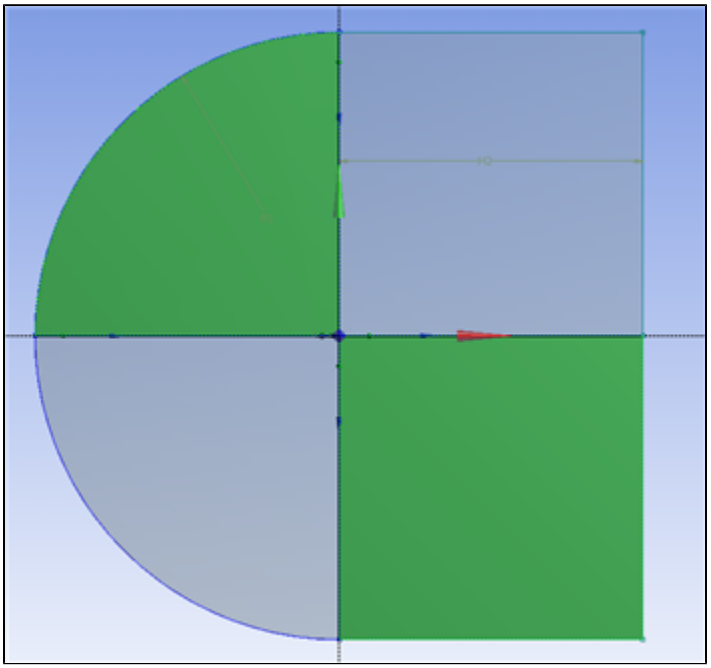
. Now that you have created a vertical line, create a new sketch and repeat the process for a horizontal line that is collinear to horizontal axis and bisects the geometry.



Now, we need to project the lines we just created onto the surface. Go to ***Tools > Projection*** . Select ***Edges*** press Ctrl and select on the vertical line we drew (you'll have to select both parts of it), then press ***Apply*** . Next, select ***Target*** and select the C-Mesh surface, then click ***Apply*** .

Once you click 

, you'll notice that the geometry is now composed of two surfaces split by the line we selected. Repeat this process to create 2 more projections: one projection the line left of the origin onto the left surface, and one projecting the right line on the right surface. When you're finished, the geometry should be split into 4 parts.



Suppress the line bodies by right clicking in the tree. You only need the surface body to be transferred to the mesher.

The geometry is finished. Save the project and close the design modeler, as we are now we are ready to create the mesh for the simulation.

Go to Step 3: Mesh

Go to all FLUENT Learning Modules