\_\_

## Virtual Wind Tunnel, Backward/Forward Facing Step and Concave/Convex Corner

### Brief description

This simulation acts as a “virtual wind tunnel” by allowing you to simulate the external aerodynamics on a forward and backward facing step in addition to convex and concave corners. The step or corner used, concave/convex corner angle, and backward/forward facing step height can be easily changed so that students can understand the relationship between freestream flow speed, step height, concave/convex corner angle, flow separation, and flow re-attachment.

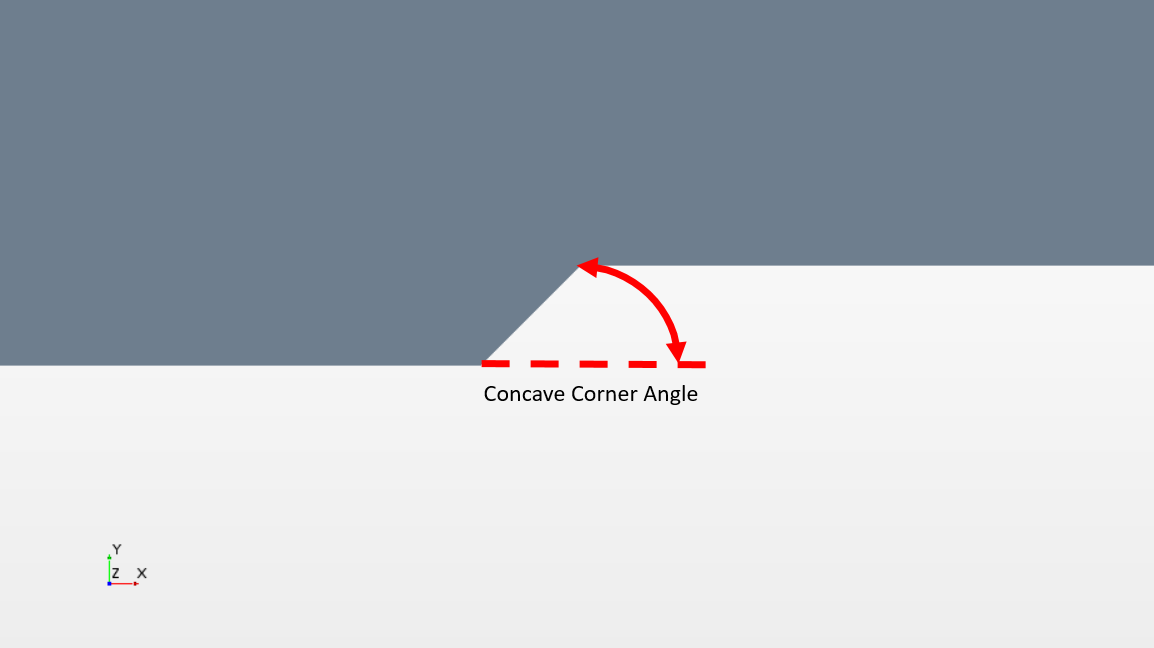
### Learning objectives

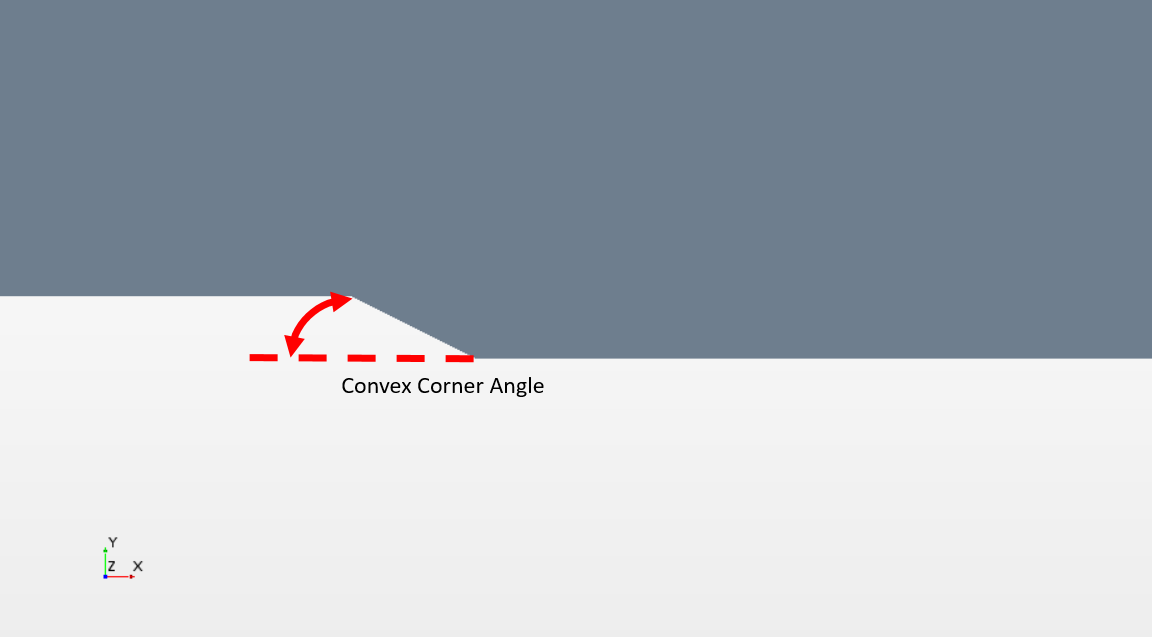
At the end of this exercise you should be able to ….

1. Better understand the relationship between freestream flow speed, step height, concave/convex corner angle, flow separation, and flow re-attachment.

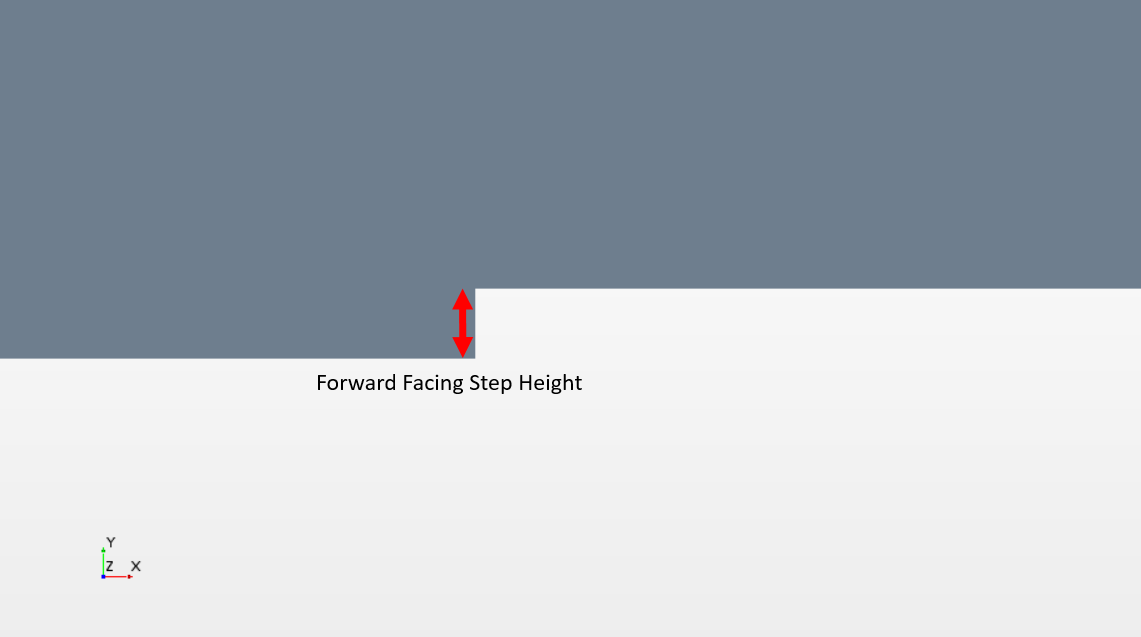
### Steps you need to take

1. Open STAR-CCM+.
2. File, load simulation.
3. Where appropriate, enter additional licensing options if needed/wanted (Power on Demand, put in your PoD key, select parallel, etc.).
4. Browse, go to the folder containing the sim file for this course, select virtual\_wind\_tunnel\_2d\_subsonic\_concave.convex.corners\_steps.sim. Select Open, OK.
5. Select the Create/Open Scenes icon at the top of the screen (), then select Open All Scenes. Note that multiple tabs will open above, you can click these tabs to switch between what is shown in the viewing window.
6. To manipulate the viewing of a scene:
   1. Panning: To pan the image in the visualization display, hold down the right mouse button and drag the mouse in the direction where you want to move the object. The image appears to change its view angle as you pan. This effect is because the view is in the Perspective projection mode, which depicts three-dimensional space. In the Parallel projection mode, the view angle stays the same.
   2. Rolling: To roll the scene, hold down the <Ctrl> key, and click and drag the mouse upward or downward. Using the <Ctrl> key rotates the object along an axis perpendicular to your screen.
   3. Rotating (not enabled for 2d simulations): STAR-CCM+ uses a point on a part to rotate. If you select a point somewhere other than on a part, STAR-CCM+ uses the depth to that last point; the depth is normal to the view plane. The scene can be rotated around various axes, vertical, horizontal, or arbitrary, depending on the direction you move the mouse. To rotate, choose a point on or near the object in the visualization display, and click and drag (holding down the left mouse button) from that point. This technique rotates the object around the axis perpendicular to the direction of your drag. You set the location of the axis when you choose a point to begin clicking and dragging.
   4. Zooming: There are two ways to zoom with the mouse:
      * 1. Choosing a focal point: Choose a point inside the display, then click and drag downward using the middle mouse button. This action brings your view closer to that point. Moving the mouse straight up has the opposite effect.
        2. Using the mouse wheel: Turning the mouse wheel toward you brings your view closer to the visualization display; the opposite wheel movement takes your view farther away.
      1. The mouse pointer must be within the visualization display for this operation to work. Its particular position within the display determines the direction of this type of zooming.
   5. Hot Keys:
      1. Side View, S key
      2. Top View, T key
      3. Front View, F key
      4. Flip vertical, C key
      5. Reset View, R key
   6. Animations: Click  to play and  to stop. In this simulation, animations are enabled for the vector scenes only.
7. Select the Create/Open Plots icon at the top of the screen (), then select Open All Plots. These plots will help you to compare drag, lift, Cd, and Cl numerical results for the body in use. When changes are made to the body or freestream velocity, you will also be able to use these plots to numerically compare the results of the new analysis compared to the old one. For the purpose of this exercise you can ignore the residuals tab.
8. With the plots tabs now open at the top of the screen, there are a number of tabs to sort through. To see the tabs that are hidden to the left/right of the screen, use the arrows () at the top right of your screen.
9. If you would like to change the geometry angle or height, please read steps 10 and 11. If you would like to swap the body (forward/backward facing step or convex/concave corner) being used, see step 12. If you would like to change the freestream velocity please read step 13.
10. To change the geometry angle or height, in the simulation tree expand Geometry, 3D-CAD Models, 3D-CAD Model 1, and Design Parameters. Here you will notice design parameters, the following images outline what each parameter is (keep in mind the white area in each picture highlights the solid volume, the gray area indicates the fluid volume). Note that we do not recommend changing the step height to below 0.15m. If values below this are used, the current mesh setup may not handle the wake features properly.

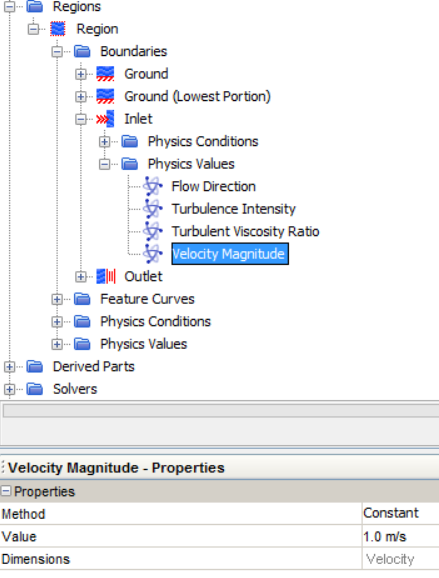








1. Once you have made the desired changes to the geometry, click the icon at the top of the screen. Browse to the java file called meshrun.java and select Open. This java macro will automatically mesh and then continue to run your simulation. Now skip to step 14.
2. To swap the body being used, expand the Operation node, select the node named “What body do you want to use? (Boolean Subtract)”, select the area to the right of “Input Parts”, deselect the body you do not want to use, and select the body you want to use. Click the icon at the top of the screen. Browse to the java file meshrun.java and select Open. This java macro will automatically mesh and then continue to run your simulation. Now skip to step 14.
3. To change the freestream velocity, please go to Regions, Region, Boundaries, Inlet, Physics Values, and select Velocity Magnitude. In the properties window below change the value to the desired value.



1. While the simulation runs you will see the current results in the scalar scenes, vector scenes, and plots. Let the simulation run until the force and/or force coefficient plots converge to a value. This should happen in approximately additional 500 solve iterations. To zoom in/out on the plots, use the middle mouse wheel or this icon . To reset the view hit R on the keyboard or the icon shown here in the center .

Disclaimer:

The focus of this lesson is on the approximate flow features. This simulation is meshed in 2d to allow a larger number of computers the ability to run these calculations.