

**subject: Advanced Computational Fluid Dynamics**  
**Komputerowa Analiza Przepływów**

---

Exercise 1


**Flow over an airfoil**

Purpose

To practice a generation of block-structured mesh and perform a simulation of steady turbulent fluid flow over NACA0012 profile at angle of attack set to 1.55 deg and freestream Mach number 0.7.

Duration: 2h

Design Modeler

1. Open Workbench 14 (Start Menu/Programy/ANSYS 14...). Drag and drop the **Geometry** component system into the **Project Schematic**. Open the **Geometry** cell. Make a NACA0012 profile using the text ASCII file 'naca0012-3d-curve.txt': Open Design Modeler. Specify **m** as unit of length. Select **Concept/3d-Curve**. Provide a link to above file under **Coordinates File**. Click on **Generate** button.
2. Mirror the airfoil curve in the **zx**-plane (**Create/Body Operation/Mirror**, select **zx**-plane).
3. Make a surface. Select **Concept/Surfaces from Edges** (select the airfoil, use Ctrl button to select multiple elements).
4. We are going to generate a fluid areas surrounding the airfoil. Add new **Sketch**  in the **XY**-Plane. Go to the **Sketching** mode. First, a half of the cylinder will be made in front of the airfoil (fig 1). Make a circle with origin at point (0,1 0,0 0,0). Set a circle diameter to 15m. Use **Dimensions** to specify the above parameters. Make a straight vertical line with length larger than the circle diameter. Place the vertical line at distance 0,1 from y-axis (use **Parallel** option to make the line parallel to y-axis). Use **Modify/Trim** option to remove the right half of the cylinder and the line segments above the circle.

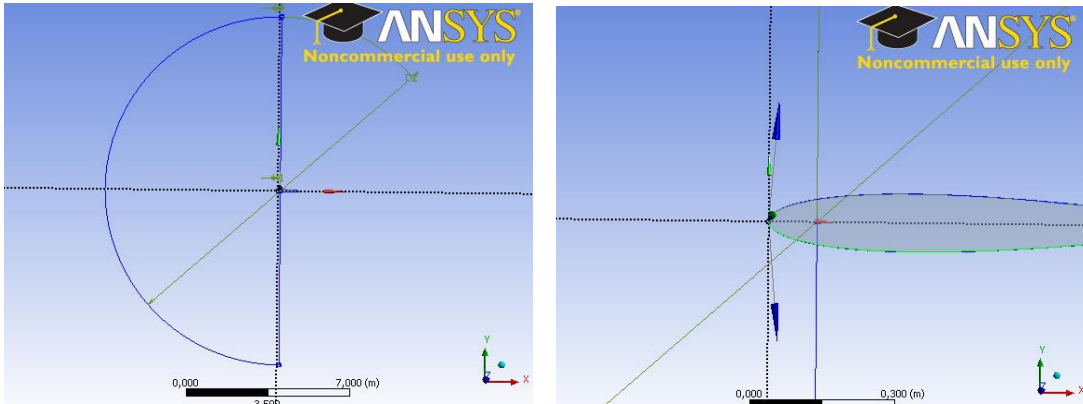


Fig. 1. Half of the cylinder (**Sketching mode**).

5. Make a surface using the prepared sketch. Go to **Modeling** mode, select **Concept/Surfaces from sketches**. Change **Add Material** to **Add frozen** under **Operation** (new fluid element will be made).
6. Add new **Sketch** in the XY-Plane. Go to the **Sketching** mode. Make a rectangular 0,9 x 7,5m. The end points of the bottom side of rectangular are : (0,1 0), (1 0). Use **Coincident** in order to attach some lines with the other. Fig. 2 shows the rectangular in the sketching mode.

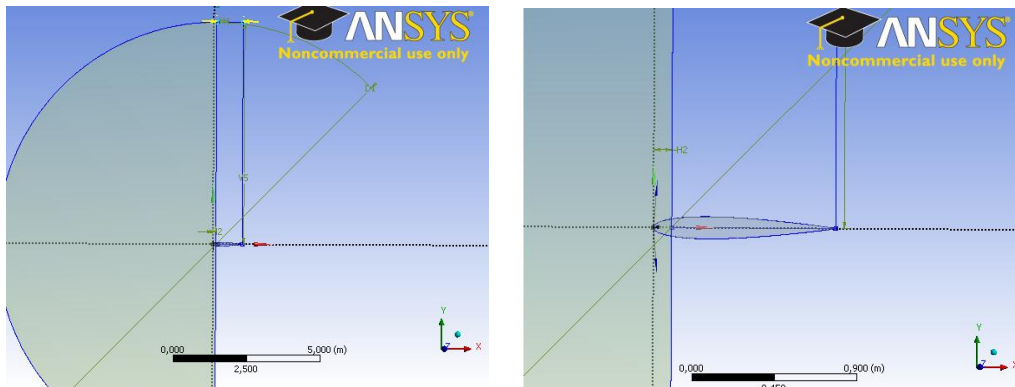


Fig. 2. Rectangular placed next to half of the cylinder.

7. Add new **Sketch** in the XY-Plane. Make a rectangular 6 x 7,5m. The end points of the bottom side of rectangular are : (1 0), (7 0). Use **Coincident** option to connect some lines together.
8. Make the two surfaces. Switch to **Modeling** mode (**Concept/Surfaces from sketches**), select both rectangles, Change **Add Material** to **Add frozen** under **Operation**.
9. Subtract the airfoil surface from fluid areas surrounding the body (half of circle and rectangle). **Create/Boolean/Subtract**. **Preserve Tool Bodies** set to No. See fig. 3

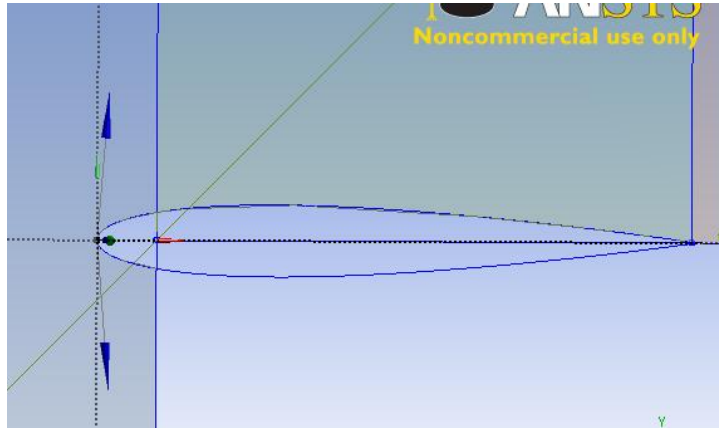


Fig. 3. Airfoil body subtracted from half of cylinder and rectangular.

10. Mirror the two rectangles (one with subtracted airfoil) in the  $zx$ -plane. (**Create/Body Operation/Mirror**, select  $zx$ -plane)
11. Connect the five faces together to ensure that there are linked together (**Tools/Joint**). See fig. 5

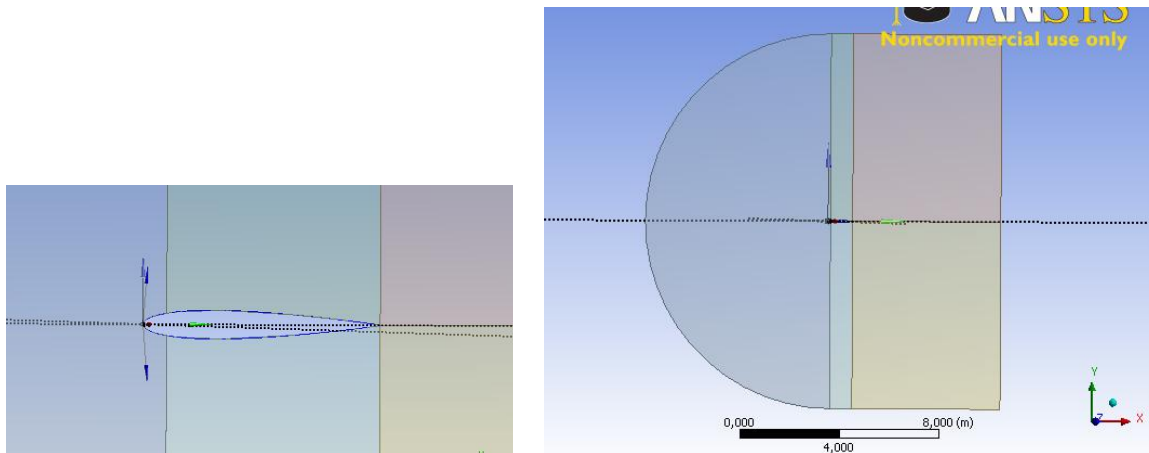
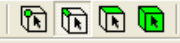


Fig. 4. The five faces surrounding the airfoil.

12. Make the fluid volume (surface). This will be done by making a one part from 5 surfaces listed under **Parts**. First, deactivate the **Line Body**, right click and select **Suppress** (This body will not be used for meshing). Select the remaining five bodies (use Ctrl or Shift buttons to select multiple bodies), right click and select **Form New Part**.
13. Provide the names indicating the airfoil surface (wall) and boundary in the freestream (pressure far field). Select the line option in the top menu , select the curves corresponding to the airfoil, right click, select **Named Selection** and put the curves under **Geometry**. Specify the name 'wall\_airfoil' under **Named Selection**. In a similar way select the curves at the boundaries of the computational domain, provide the name 'pressure\_far\_field'. Save the project (Drive D, folder Student, make your own folder there, if needed). Close **Design Modeler** (green project part).

## Meshing

1. Add the **Mesh** button (drag and drop) in the **Project Schematic** (Workbench) and link it to the present **Geometry**.
2. Click on the **Mesh** button. Change the settings under the **Physics reference** from **Mechanical** to **CFD**, use **Fluent** as **Solver Preference**. Generate a preliminary (unstructured) mesh (see fig. 5). The mesh is very coarse near to the surface and it cannot be used for viscous flow simulations.

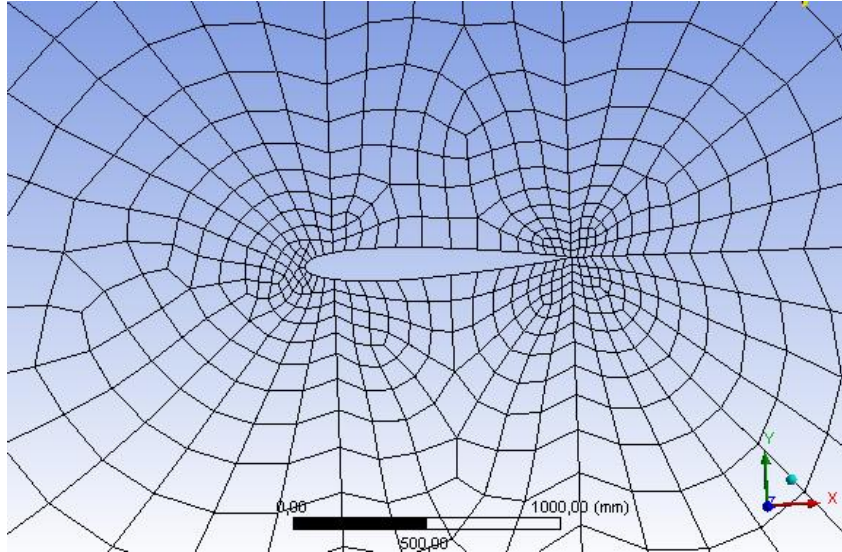
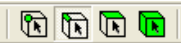


Fig. 5. Preliminary mesh generated over airfoil using the global mesh settings.

3. The block-structured grid can be generated. Note that each surface has to be mapped virtually onto the rectangular. In order to do this one has to specify a proper number of elements on selected edges.
    - Specify about ~100 of nodes on edges perpendicular to surface, with bias factor ~4000.

The mesh has to be refined near to the wall. First, select line option  in the top menu, next select the edge(s) from the screen, click on **Mesh**, right click, select **Sizing**. Specify the element size under **Type/Element Size** or change to **Number of Divisions** under **Type** and specify the number of elements.
  - keep the same number of elements on edges corresponding to opposite sides of virtual rectangular. Otherwise, the mapped scheme cannot be applied for making the block-structured mesh.
  - the boundary layer mesh (inflation) cannot be used now.
4. Right click on the **Mesh**, select **Mapped Mesh Meshing**. Add all faces under **Geometry**. Select **Quadrilaterals** under **Method**. Click on Generate button. Fig. 6 shows the view of the computational mesh.
  5. Save the meshing project. Export the mesh to \*.msh Fluent file. **File\Export**.

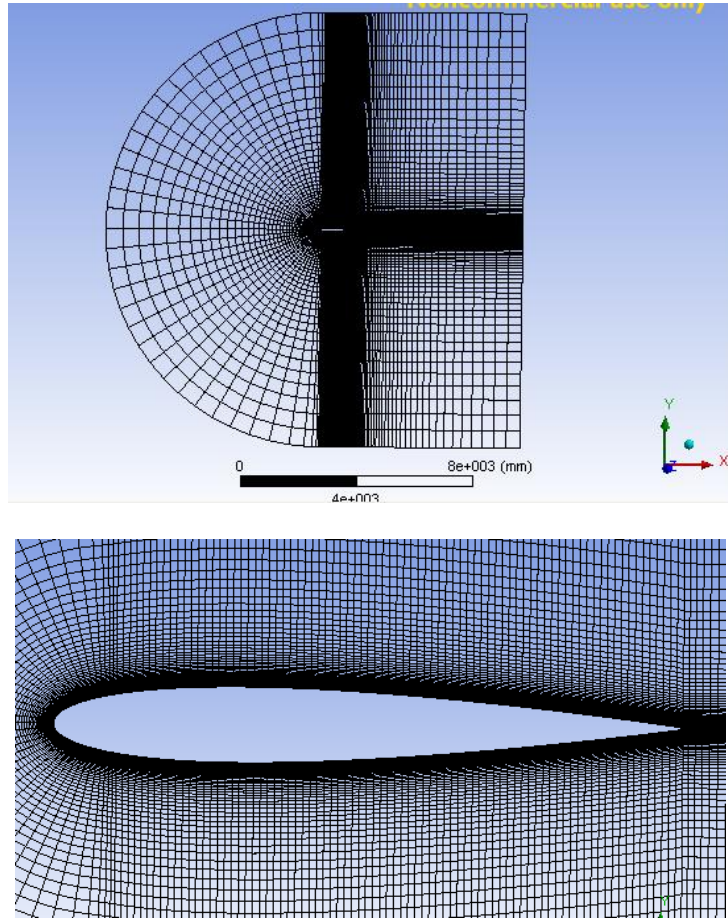


Fig. 6 Final block-structured mesh

## Fluent

1. Read the mesh file. Check the scale. Keep the length in m. Rotate the mesh by 1.55 deg. **Mesh/Rotate/** set -1.55 deg.
2. Set-up the computations in Fluent. The inlet Mach (M) number is 0.7, the Reynolds number based on chord length of the airfoil and inlet velocity is  $1.5 \cdot 10^6$ . Change the fluid viscosity  $\mu$  to specify the proper Reynolds number.
3. Select appropriate turbulence model (Spalart-Allmaras or SST k-omega) and specify the boundary conditions on pressure far field boundary. The numerical results will be compared with experiment performed at conditions  $p_0 = 101325\text{Pa}$ ,  $T_0=311\text{K}$ . The static pressure and static temperature has to be specified in the boundary condition tab.
  - Set the operating pressure to 0 Pa **Define/Boundary Conditions/Operating Condition**.
  - Set the static pressure to 73048 Pa (**Define/Boundary Conditions/pressure far field**).

$$\frac{p_o}{p} = \left[ 1 + \left( \frac{\gamma - 1}{2} \right) M^2 \right]^{\frac{\gamma}{\gamma - 1}}$$

- Set the static temperature to 283 K. For air  $\gamma=1.4$ .

$$\frac{T_0}{T} = 1 + \left( \frac{\gamma - 1}{2} \right) M^2$$

- Turbulence quantities. Ratio of turbulent to molecular viscosity =10, Turbulent intensity = 1%.
- 4. Run the computations. The residuals should fall down below  $10^{-5}$ . The numerical scheme should be set to the second order upwind.
- 5. Verify the pressure, temperature and velocity fields. Verify the lift and drag coefficients  $c_d = F_d / (0.5 \rho u^2 A)$ , (A is equal to 1) and compare them with literature data (activate the freestream conditions in **Report/Reference values**). For this flow the measured lift and drag coefficient are equal to  $c_l = 0.241$ ,  $c_d = 0.0079$ . Note that the forces have to be computed with respect to direction of the incoming flow. This is why the mesh has been rotated in the beginning by 1.55 deg. It means that the lift and drag coefficients can now easily be obtained in **Report/Forces/Direction vector** by setting 1 and 0 factors under Direction vector.  $x=1$   $y=0$  for drag,  $x=0$ ,  $y=1$  for lift.
- 6. Compare the measured and computed pressure coefficient profiles. Display/Plots/XY Plot. The experimental profiles are stored in files: test-data-top.xy and test-data-bottom.xy (**Load File** under **File Data**)