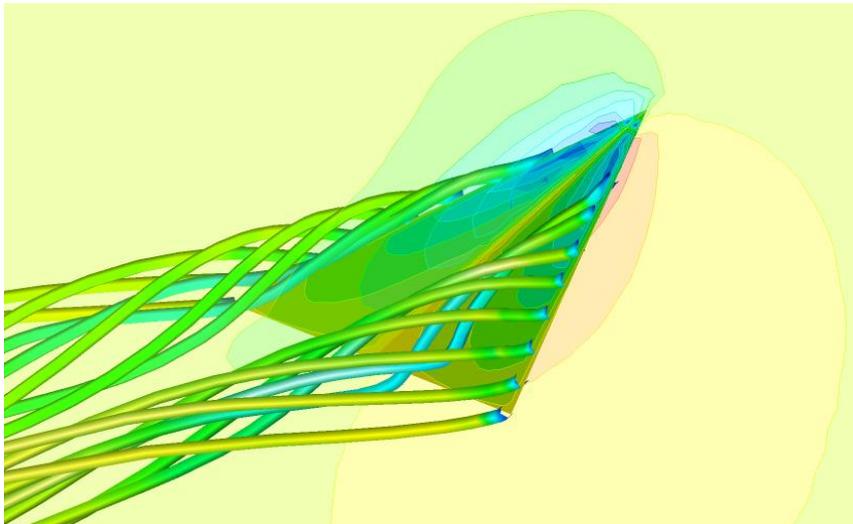


Exercise 4: 3D steady flow simulation over a delta wing

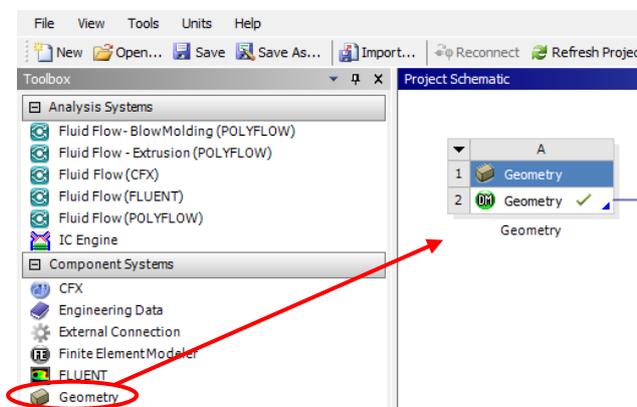


Purpose and description of the exercise

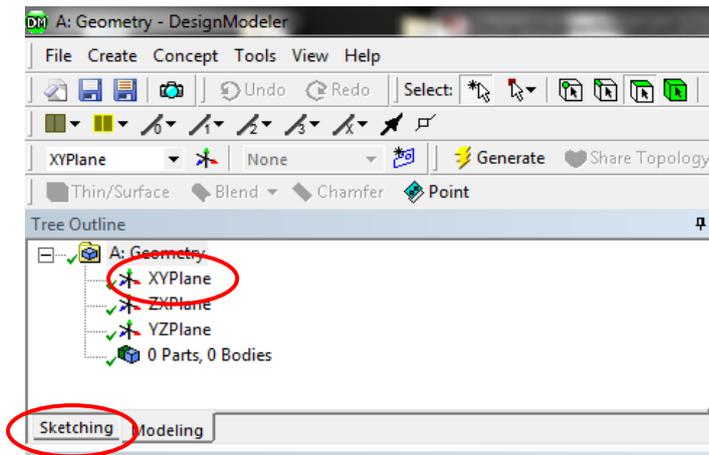
The purpose of this exercise is to perform a simulation of the three-dimensional (3D) flow over delta wing and processing of the numerical results. The wing (equipped with a propeller) is set at a high angle of attack (30°). Flow takes place in a closed space of the wind tunnel and has a plane of symmetry. Dimensions of the wind tunnel are $150 \times 200 \times 600$ cm (in x, y and z directions). The medium is air. The mean inlet velocity is $V = 10$ m/sec and the operating (ambient) pressure is $p = 101325$ Pa. The flow is assumed to be symmetric with respect to the x-z plane.

Preparation of the model

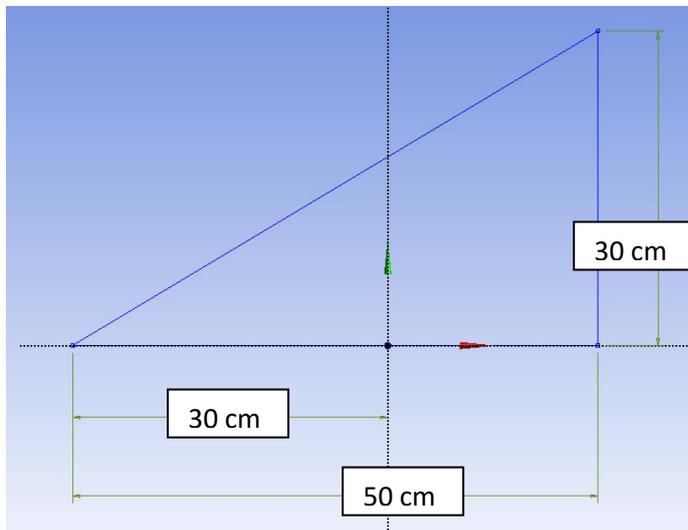
Open the ANSYS Workbench 19. Drag the **Geometry** icon from side menu panel and drop it into the **Project Schematic** window. Open the program by double clicking on the **Geometry**.



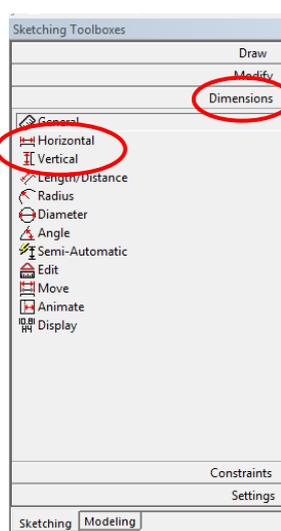
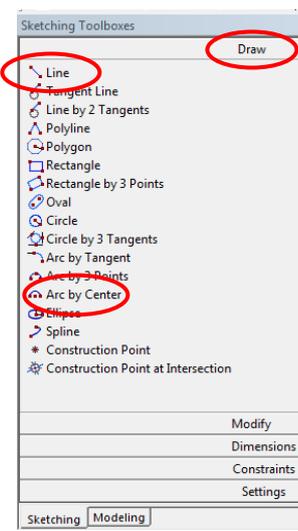
Set the unit length to centimeters in **Units**.



Select the **XY-Plane** and click on the **Sketching**.



Make the three line, see figure on the left):



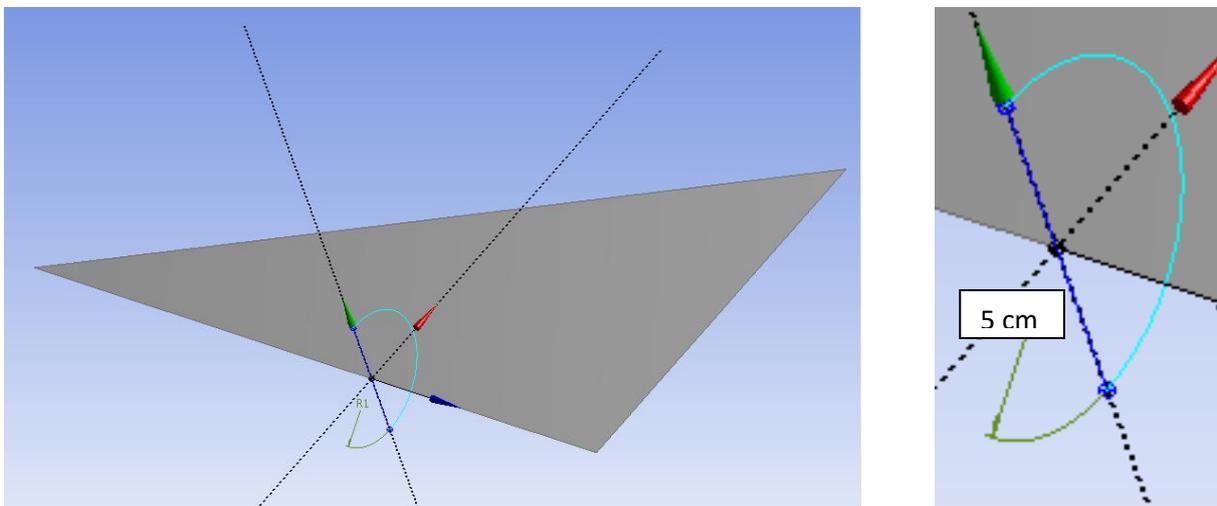
Options for making the geometry and specifying the size of the line segments

After finishing the sketch select the **Modeling** tab.

The previously generated sketch will be used to create surface. Select **Concept / Surfaces from Sketches** from top menu and choose the sketch (just select any of its edges, or select it in the side menu and click **Apply**), confirm the operation by clicking the Generate

Select the plane YZ and click on the **Sketching** tab.

Create an arc (180 deg) with the center placed at the origin of coordinate system. The arc has to be revolved along the Z-axis. Connect the extreme points with the line segment and specify the arc radius (5 cm).



Again, we need to create a surface, **Concept / Surfaces from Sketches**, select the proper sketch from the plane ZY, expand the list under **Operation** and select **Add Frozen** instead of **Add Material**. The propeller surface has to appear.

The calculations will be performed for the angle of attack set to 30 °, so we must now turn the wing together with propeller by 30 ° about the y-axis. Switch to **Create / Body Transformation/Rotate**, select the Bodies: wing and propeller hold Ctrl button when selecting the second surface. Select the y-axis under **Axis Selection** in the graphics window and enter the rotation angle to 30 degrees, and confirm by clicking on **Generate**.

Finally, the volume surrounding the wing and propeller has to be generated. It is a box (wind tunnel). The settings are specified below:

Create / Primitives / Box

Coordinates of the starting point:

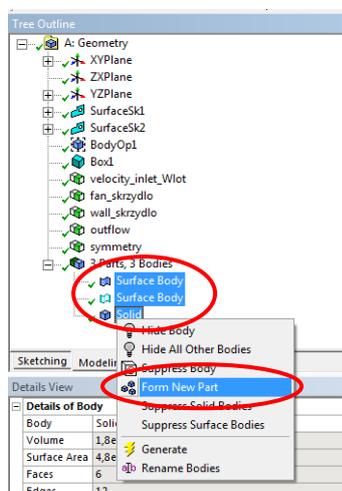
X=-250 cm, Y =0, Z = -80 cm

Diagonal:

X=600 cm, Y = 150 cm, Z= 200 cm

Click on **Generate** to finish the definition of the geometry.

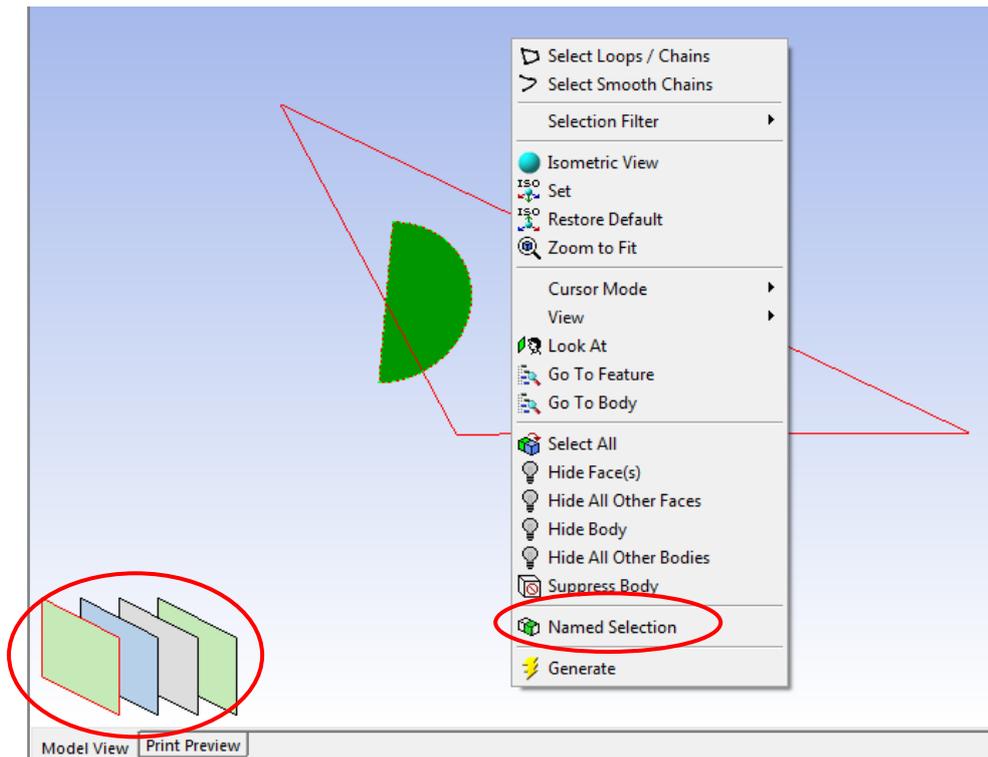
We have to create one part from the three objects listed under **3 Parts, 3 Bodies**. Expand the list of parts, select all surfaces holding the shift button, right-click and choose **Form New Part**



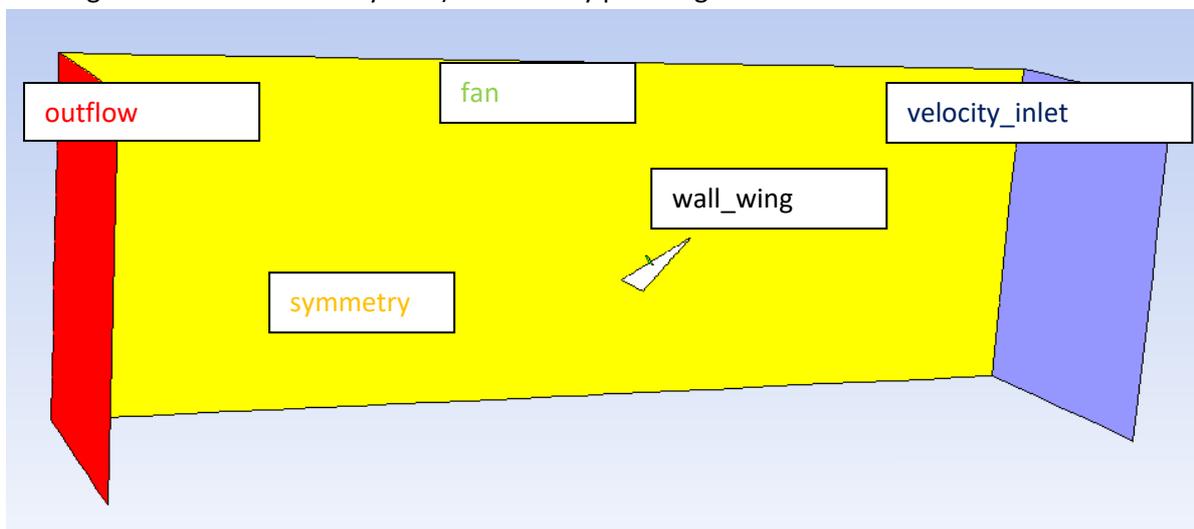
Select **Fluid** under **Fluid/Solid** option.

Select **Wireframe** in the top menu panel under **View**.

Specify the types of the boundary conditions by selecting the proper surface from the screen. Use the left mouse button, right-click and select the **Named Selection**. Remember to have selection filter set properly. Specify the name of the surface under **Named Selection**. To select more than one surface, one has to make the selection by holding the Ctrl button. To select the specific surface one has to verify the list of surfaces appearing in the left-bottom corner of the window.



Specify the names of the surfaces as shown below. The velocity_inlet_wlot corresponds to the surface which is placed on negative side of the x-axis (shorter distance from the selected surface to the origin of the coordinate system). Confirm by pushing the **Generate** button.



Save the project and close the DesignModeler.

We are going to make the mesh. Drag the **Mesh** icon and drop it over the **Geometry** in the **Project Schematic**. Open the mesh generation program by double-clicking on the **Mesh**.

Specify the following global settings for the Mesh (side menu):

Defaults:

Physics Preference: CFD

Solver: Fluent

Sizing:

Use Advanced Size Function: On: Proximity and Curvature

Max Face Size: 215 mm

Create the mesh by clicking on **Generate Mesh** in the top menu panel. Click Wireframe icon to display only the edges of the grid and verify the grid. (activate the mesh on the left menu panel).

We will change the grid sizing on the wing surface. Click the right mouse button, select **Insert \ Sizing** and set:

Scoping Method: Named Selection

Named Selection: wall_ wing

Item Size: 20 mm

Behavior: Hard

Apply the same mesh settings for „fan”.

Generate the mesh again and verify the number of elements on selected surfaces.

Info. One can select between: points, edges, surfaces, volumes (as shown below) in order to apply a specific settings related to geometry and mesh.



Click the right mouse button, select **Insert \ Sizing** and set:

Scoping Method: Geometry Selection

Geometry: Select the volume calculation (previously set volume selection mode)

Type: Sphere of Influence

Sphere Center: Global Coordinate System

Sphere Radius: 500 mm

Element Size: 50 mm

Generate the grid again, save the project and close the mesh generation program.

Project Schematic. From the left menu, drag the **Fluent** icon and drop it over the **Mesh** icon.

Refresh the project by clicking on **Project Update** and run FLUENT by double-clicking on the Setup icon (leave the default settings).

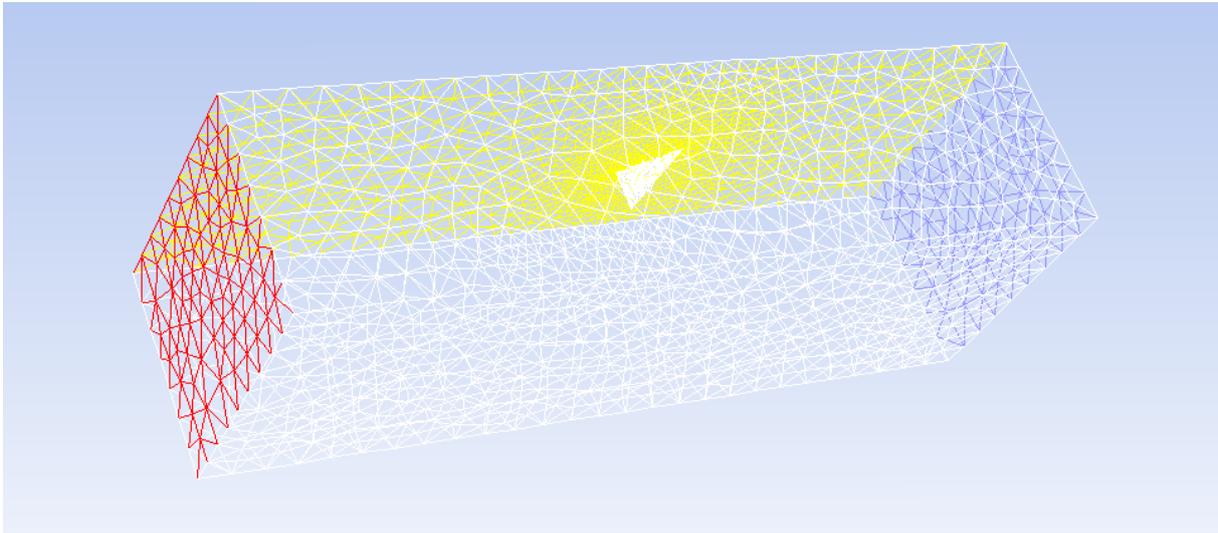


Fig. View of the computational mesh in Fluent program.

Fluent program

General - leave the default settings, select the **Check** to verify the grid and the **Scale** (the domain size should be of the order of several meters).

Models \ Viscous – the flow is treated as turbulent – activate the Spalart – Allmaras model

Materials - verify the Mach number (see the specification of the inlet velocity below) set the density to constant if $Ma < 0.3$

Boundary Conditions – the reference pressure has to be equal to 101325 Pa in **Operating Conditions**. Verify the types of the boundary conditions and specify the settings as below:

velocity: inlet velocity : 10 m/s and turbulence to molecular viscosity ratio 10 (do not mix it with modified turbulent viscosity).

fan: set the pressure jump to 0 (Pressure Jump = Constant = 0 Pa) . This settings are for the first part of the exercise (means that propeller is not working). In the second part a certain pressure jump will be applied.

outflow: leave without any modifications (Flow Rate Weighting = 1)

symmetry : no action required.

The remaining (unspecified earlier) boundaries are walls (default boundary type in Fluent).

Settings for the flow solver :

Solution Methods : change the pressure-velocity treatment scheme from SIMPLE to Coupled, and the discretization scheme to the first-order upwind.

Do not change the settings in **Monitors \ Residuals**

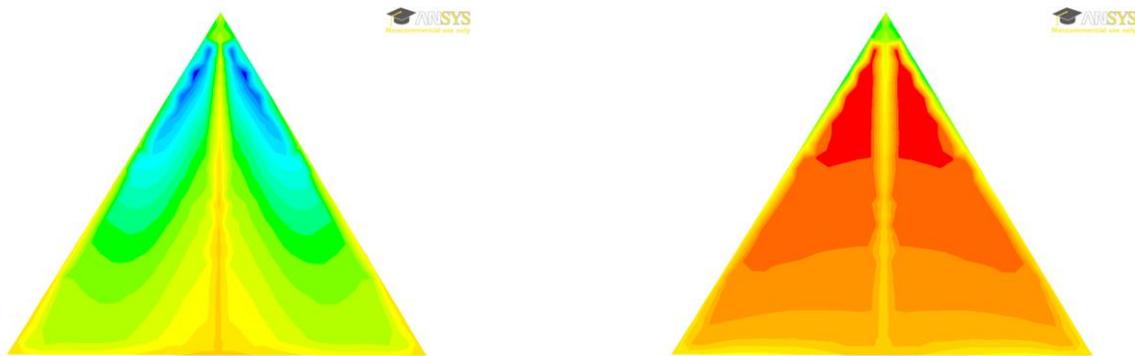
Initialization of the solution, **Solution\Initialization \ Standard Initialization**, select the **Compute from** the inlet to the computational domain. Confirm by clicking **Initialize** .

Run the calculation. Specify a certain number of iterations and continue the simulations until converged solution is obtained.

Analysis of the results (Results tab)

We will reflect the results over the symmetry plane. Switch to **Results\Graphics\Views** in the **Mirror Planes** select the surface called "symmetry" and confirm **Apply**

Visualize and save (JPG file) the contours of pressure **Graphics \ Contours** and velocity vectors **Vectors** on the upper and lower surfaces of the wing. The velocity vectors are obtained from the cell centers at the wall (so slightly above the physical wall). Determine which surface (wall_wing or wall_wing_shadow) corresponds to the top and bottom sides of the wing. One can adjust the **Scale** and **Skip** values to make the visualization clearer.



Contours of static pressure on top and bottom sides of the wing.



The velocity vectors on top and bottom sides of the wing (note that the velocity vectors are obtained at the cell centers not at the physical wall).

Visualization of the vortex structures

Create two planes of constant values of the X coordinate: From the top menu select **Setting up Domain \ Create \ Surface \ Iso-Surface** and specify the fixed values of the x-coordinate: Set x to 0.05, and specify the name under New Surface Name, and **Create**. Next, specify the plane at distance x=0.2.

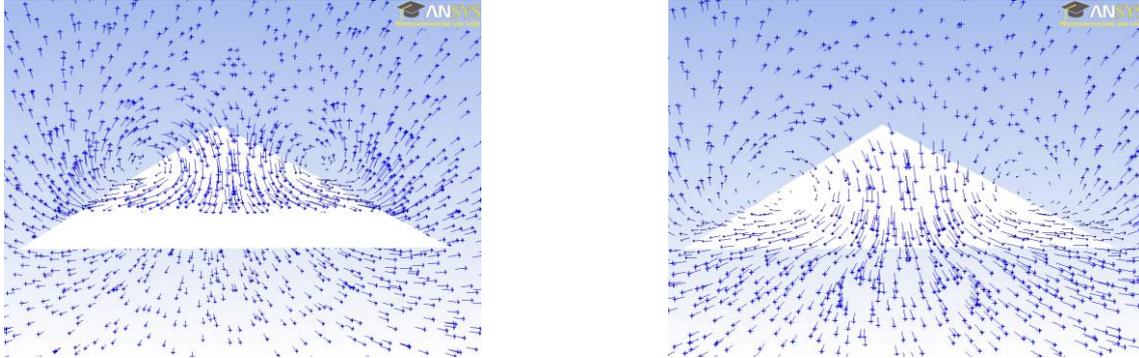


Fig. Velocity vectors In the planes $x = 0.05$ and $x = 0.2$

Definition of Pathlines

Now we will create a special kind of line on leading edge of the wing, which will be used for releasing path lines. First you have to display geometry of the wing:

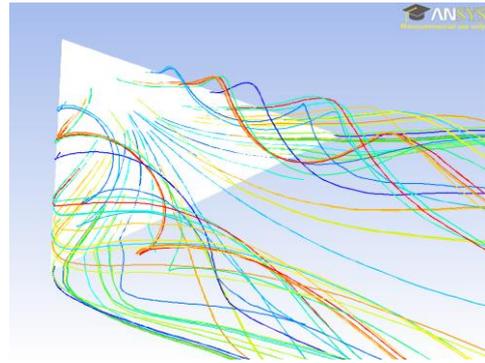
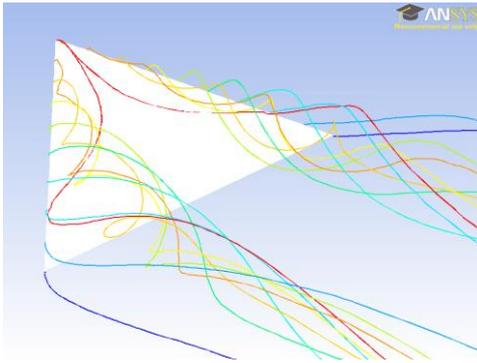
Display \ Mesh (**Warning:** make sure to turn off mirror display, so you won't confuse real geometry with it's reflection!). In the window of mesh display under **Options** choose **Faces** and under **Surfaces** select "wall_wing". Now, you can create line on the leading edge:

Surface \ Create \ Line / Rake Leave **Line Tool** ticked off and choose **Rake** as a **Type**. **Number of Points** should be set to 10. Now click on **Select Points With Mouse**. In the graphic window, where wing is displayed you must point to its beginning and end (right click accordingly on the first and second point), this way you define the leading edge, in **New Surface Name** field input name of geometry you are creating (i.e. leading_edge) confirm by clicking on **Create**. In a similar way you can add line defining the trailing edge. (**Warning:** Mouse buttons settings can be checked using **Display \ Mouse Buttons**)

Path lines visualisation

Results \ Graphics \ Path Lines

Under **Release from Surfaces** select leading edge you just created (you can also use wing surface).

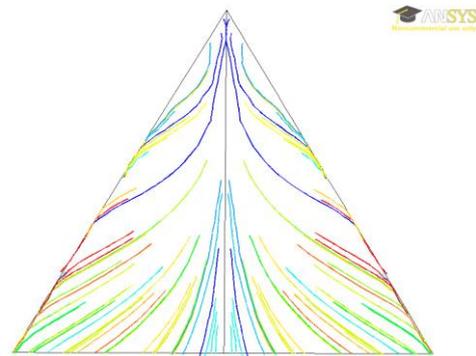
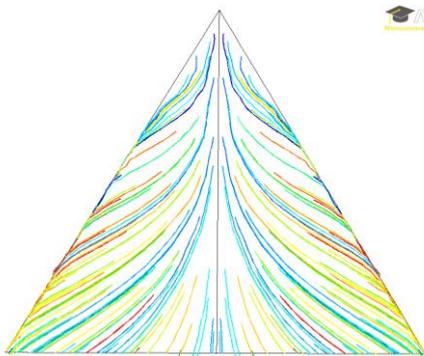


Example of path lines visualisation, released respectively from leading edge and wing surface.

(Warning: when you want to release path lines from a big area, or small one but consisting of significant amount of nodes, you must insert **Path Skip value to cut down number of path lines displayed – if their number will be too high Fluent may crash!)**

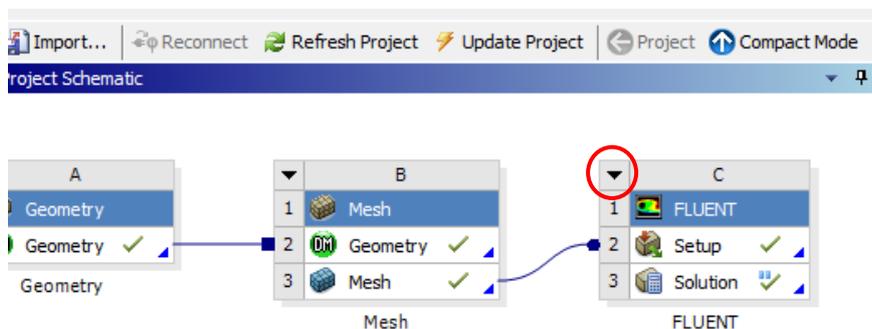
Oil flow visualisation

Results\Graphics\Path Lines Under **Options** select **Oil Flow**, in fields **On Zone** and **Release from Surfaces** select „wall_wing”. Confirm by selecting **Display**.



Oil flow visualisation of respectively upper and lower wing's surfaces.

Save Project and close Fluent, in the Project Schematic window unroll options menu of the Fluent cell and select **Duplicate**, then run Fluent from the newly created cell by clicking on **Setup**.

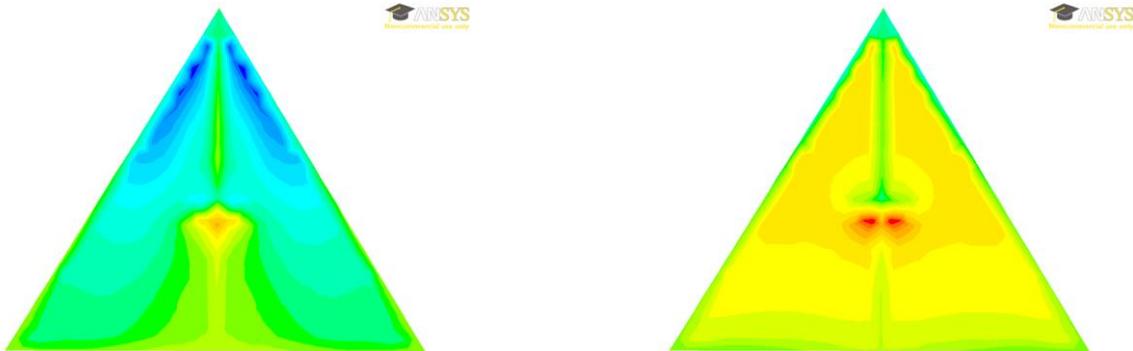


Aim of next part of the exercise is to solve and make visualisations of the case with fan turned on:
Setup\Boundary Conditions\fan>Edit

Pressure Jump = 200 Pa (Constant), if needed select **Reverse Fan Direction**

Other boundary conditions are not modified. Initialize the solution and calculate until convergence is achieved. Analyse this case similarly to the first one. Check if fan is working in the right direction (check pressure contours), to change it use option **Reverse Fan Direction** .

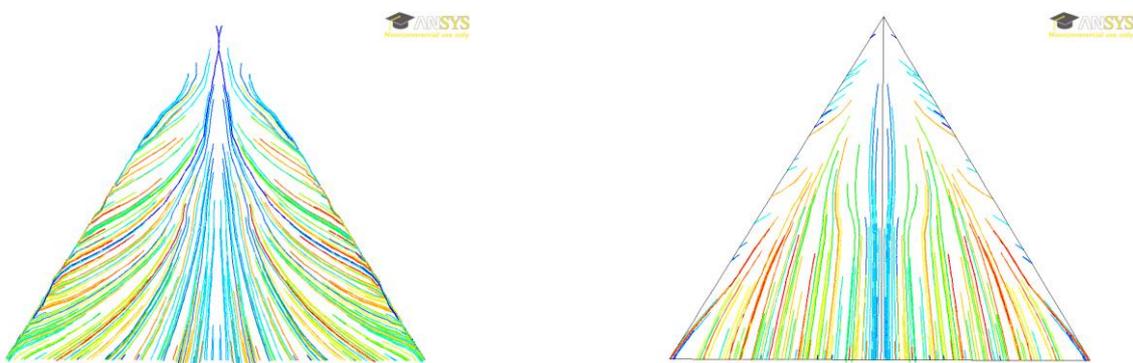
Analyse the case with fan turned on:



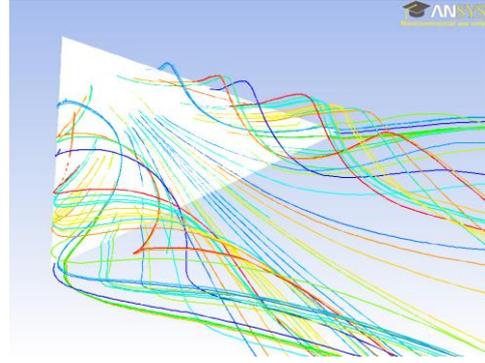
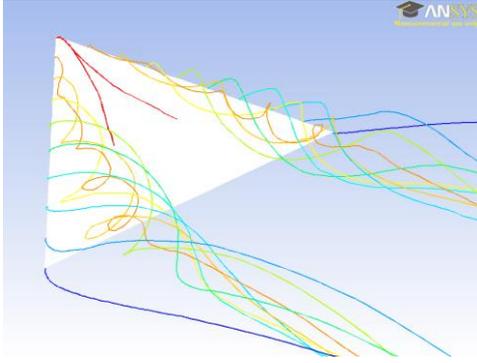
Contours of static pressure on top and bottom sides of the wing.



The velocity vectors on top and bottom sides of the wing



Oil flow on top and bottom sides of the wing



Example of path lines visualisation, released respectively from leading edge and wing surface.