

# Exercise 1: Introduction to ANSYS Workbench ver. 19

---

## *Introduction*

Figure 1 shows a flow chart where it is possible to observe the general steps to generate a simulation using any CFD software package. **ANSYS Workbench** assists the user in carrying out the tasks involved in an analysis process; typically the software is divided in three main parts:

- A pre-processor.
- A Solver.
- A post-processor.

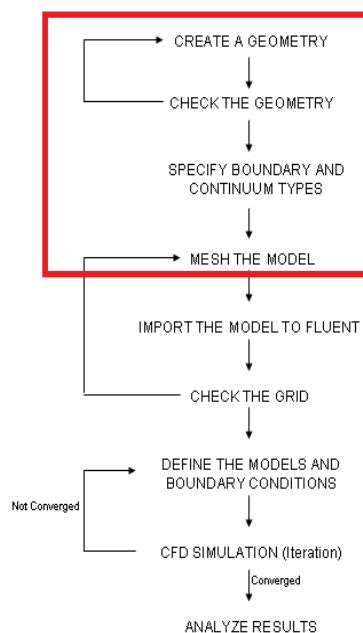


Figure N°1. The Analysis Process.

Exercise number 1 consists in a pre-processor task, and it is necessary to take into account the steps marked in red square in the figure 1.

## *Purpose*

Get familiar with selected components of **ANSYS Workbench**: **ANSYS DesignModeler** (geometry definition) and **ANSYS Meshing** (mesh generator) and **ANSYS Fluent** (flow solver).

### Project start-up

Before starting a solution of any flow problem using the ANSYS Fluent we will need to create geometry and make the computational mesh. Next the Fluent solver has to be properly set-up. The numerical results can be later analysed using either the Fluent or the external post processing tool. This is called the data post processing. During the labs we will use also Fluent for flow simulation and for data analysis.

*Drag and drop a **Fluid Flow (Fluent)** component from **Analysis Systems** into the **Project Schematic** (white window). As result, all necessary components will be shown altogether.*

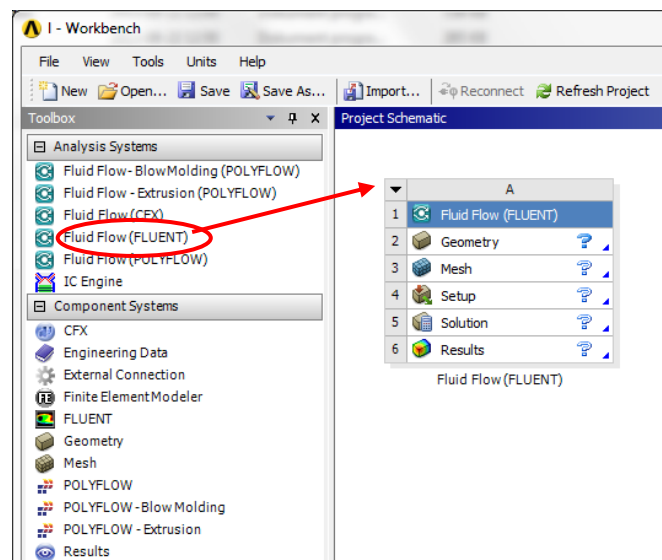


Figure N°2. ANSYS Workbench

Alternatively, each component can be shifted separately from **Component Systems** window into the **Project Schematic** window. We could start by adding the **Geometry** box, later **Mesh** and finally the **Fluent** box, as shown below.

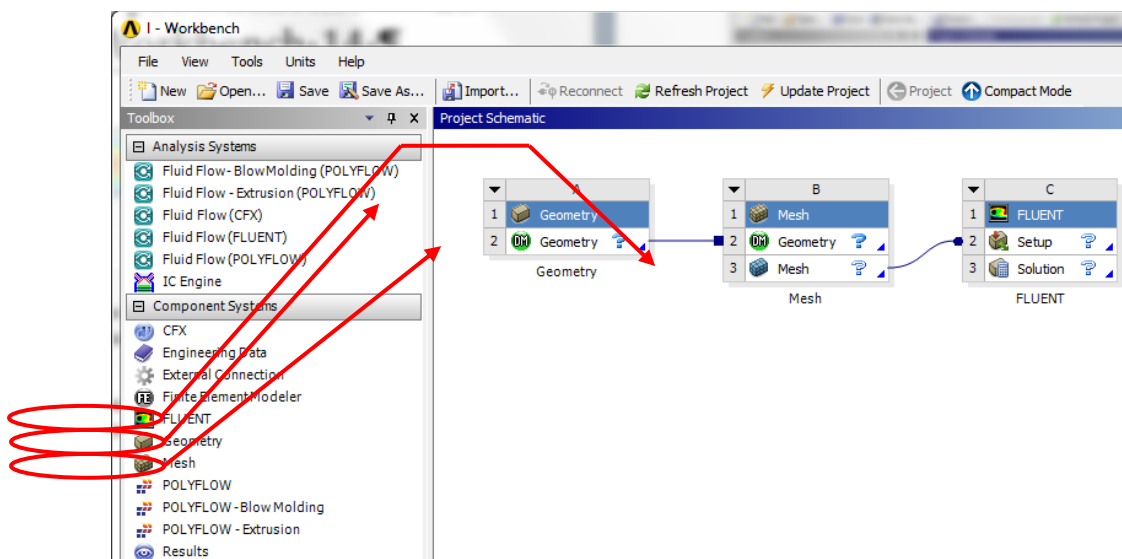


Figure N°3. ANSYS Workbench

Symbol ? means that this component is undefined yet. The symbol ? changes to ✓ if the component problem has been properly specified/updated. Each project component in the **Project Schematic** has to be properly defined before going to the next one. The meaning of some symbols is given below:

? - Problem is undefined. User has to introduce some changes into the given project component.

⚡ - Allows to update the data so they can be used in the remaining part(s) of the project loop.  
(**Update Project**)

🔄 - Project refreshing is required. (**Refresh Project**)

## Geometry

Start the **ANSYS Workbench**. Drag and drop **Geometry** box into the **Project Schematic**. Right click on the **Geometry** cell to select and run **DesignModeler**.

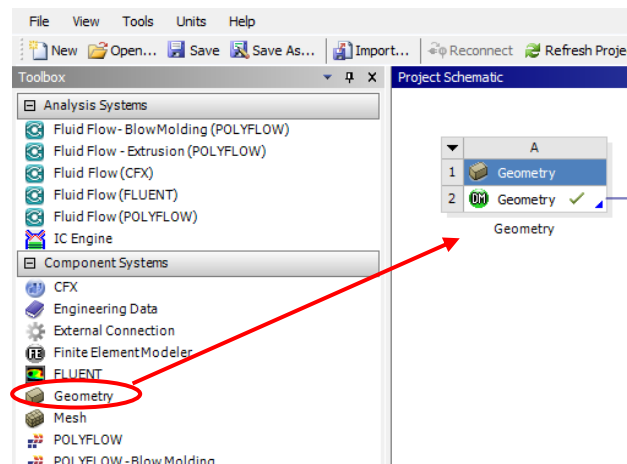


Figure N°4. ANSYS Workbench – Geometry Design Modeler

Select **Centimetres** as a desired unit length in **Units**.

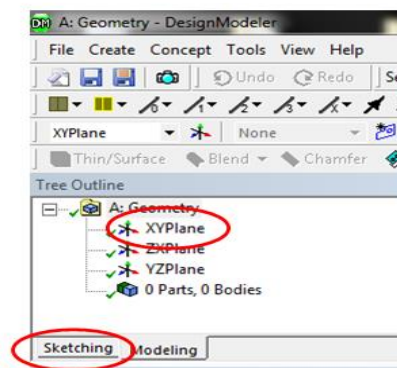
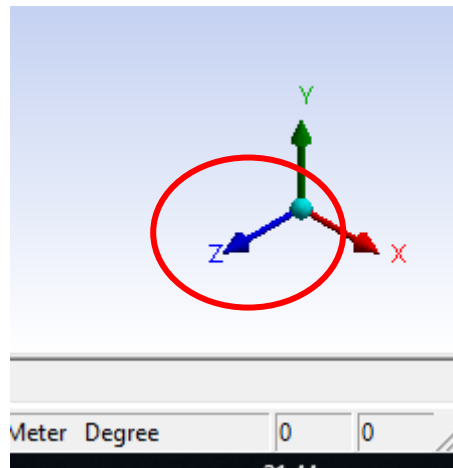


Figure N°4. ANSYS Workbench – Geometry DesignModeler

In the **Tree Outline** (on the left) select the **XYPlane** and click on the **Sketching** button. The **Sketch1** will pop-up automatically when we start our drawings.



Click on the Z axis (the triad in the bottom right in **Graphics** window) to position the drawing window in the X-Y plane. We are going to create a following sketch (Note that the fluid flow will be from left to right):

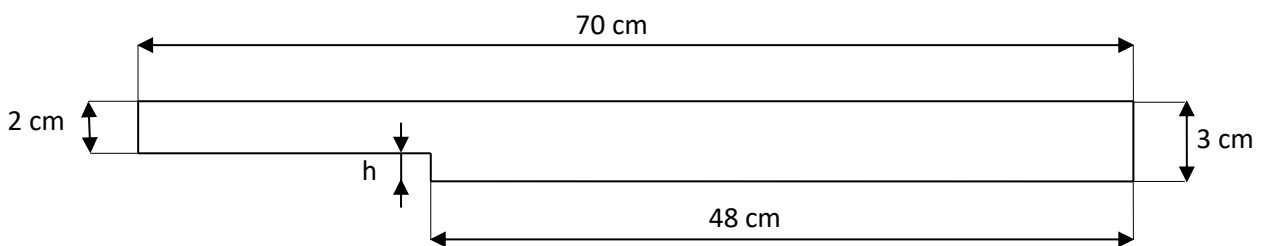


Figure N°5. Exercise N°1 Geometry

Select the **Draw/Rectangle** toolbox. Sketch the arbitrary rectangular in the **Graphics** window (it will be used for creating right side of our geometry – outflow section).

Change to **Constraints/Coincident** under the **Sketching Toolboxes** and make a link between the left bottom corner of the rectangle and the origin of the coordinate system (select rectangle's corner and respectively X and Y axis). The left bottom corner of the rectangle will be shifted/make coincident with the origin of the coordinate system. Activate **Show Constraints** (specify YES) under **Details of Sketch** to verify all constraints.

Specify dimensions of the rectangle. Select **Dimensions\General**. Select the vertical side of the rectangle and specify a proper size (Figure N°5) under **Details View\Dimensions**. Do the same for the horizontal side. The blue colour means that the dimensions/constaints have been correctly specified.

Switch to **Modeling**. Create the second sketch (smaller rectangle to the left). Select XYPlane and add the **Sketch** symbol as shown below.

Make sure that the proper sketch is selected when working on new geometry components.

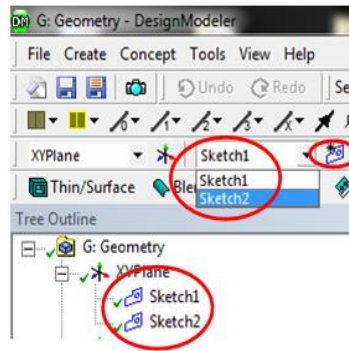


Figure N°6. ANSYS Workbench – Geometry DesignModeler

Create the second rectangular (inlet section), specify dimensions and constraints. Right upper corner of the new rectangular has to be linked with the left upper corner of the previously generated rectangular (**Constraints\Coincident**). See figure below. Specify dimensions of the new rectangle. Select **Dimensions\General**. Select the vertical side of the rectangle and specify a proper size (Figure N°5) under **Details View\Dimensions**

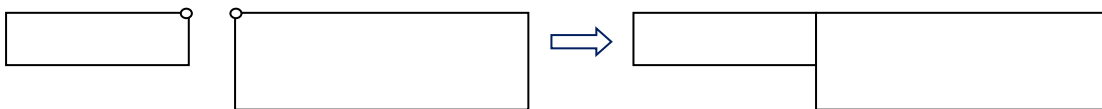


Figure N°7. Exercise N°1 Geometry Design

Use these two sketches for creating surfaces **Concept/Surfaces from Sketches**. Click on a newly generated **SurfaceSk** under **Geometry**. Hold the **Ctrl** button and select both sketches from the tree, confirm their selection by clicking **Apply** next to **Base Objects** cell. Right click and push on **Generate** to make new surfaces.

Change the properties of the newly created bodies from Solid to Fluid (**Part, Body: Surface Body**) under **Details of Surface Body/ Fluid/Solid**.

Note that any surface can be hidden in order to allow an easier manipulation on the remaining components of the geometry (select surface body, right click and use **Hide Body**). One can completely deactivate the given surface so it will not be used in the process of mesh generation (select surface body, right click and use **Suppress Body** to see how it affects the geometry).

The present object consists of two Parts. This means that both parts are seen as two separate elements and that there is no link between them (it means that physically there will be no flow from one part to the other). One part has to be made to enforce the continuity. Select both surfaces (holding shift button) under **2 Parts, 2 Surfaces** right click and select **Form New Part**. Check whether both surfaces are defined as Fluid.

One can verify coordinates (or dimension) of selected components (points, lines, surfaces etc). Go to **Tools/Analysis Tools/Entity Information** and select point selection filter (see below). Select arbitrary

point in the Graphics window and verify the coordinates (in the exercise you will need to know the coordinates of inlet, so write them down somewhere).

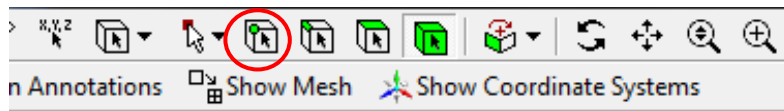


Figure N°8. ANSYS Workbench – Geometry DesignModeler

Save the project and close DesignModeler.

### *Mesh*

Drag and drop the **Mesh** cell into the **Geometry** box in the **Project Schematic**. Double click on the Mesh to open the meshing program. Select the **Mesh** under the **Outline**.

In **Details** specify the following under **Sizing**:

- Size Function: Adaptive
- Element Size: type 5 mm instead of Default (pay attention to the selected unit)

**Attention:** In Workbench the decimal mark (symbol separating the integral from the fractional part of the real number) is comma „ , “. In Fluent the decimal mark is dot “ . ”.

Click on **Generate Mesh** to make the first mesh.

Now we are going to make a finer mesh. This requires a specification of a proper size of the cell elements on selected edges. Activate the edge selection filter option in the upper menu panel. Select all vertical edges (use Ctrl to select multiple edges) right click on the **Mesh** in the **Outline** panel and select **Insert/Sizing**. Provide the proper **Size** of the cell elements (on a 30 mm edge we expect to have 10 divisions) or simply provide the **Number of divisions** on selected edge.

The specification from the edges has to be transformed to the other edges and to the interior of the domain. Select both surfaces (first activate surface selection filter), right click and insert **Insert/Face Meshing**. Make sure, that **Definition/Mapped mesh** is turned on. Generate the grid.



Figure N°9. ANSYS Workbench – Meshing

Insert the specific names at the edges corresponding to the inlet and outlet. Select **Geometry** under the **Object** window. Switch selection mode to edges. Select the leftmost vertical line (Graphics

window), right click and select **Create Named Selection**. Specify the name “velocity\_inlet”. Specify the edge name “pressure\_outlet” at the outlet from the domain (rightmost vertical edge). Specify the “Wall\_bottom” name on the bottom horizontal line after the step. The default boundary condition on the remaining (unnamed) edges will be wall.

Export the mesh using **File/Export** (select file extension as Fluent Input Files: \*.msh).

Open Fluent program (Menu Start/ANSYS 19.0/Fluent 19.0). Set: 2D, serial, double precision, specify the working directory. Run Fluent. Read the mesh (\*.msh file), **File/Read/Mesh**. Select option **Display Mesh After Reading**.

Verify a correctness of the boundary conditions specification (inlet-blue colour, outlet-red, walls-white). Save the \*.cas file (**File/Write/Case**) and close Fluent.

We go back to Workbench. Drag and drop the **Mesh** cell into the **Geometry** box in the **Project Schematic**. We will employ an automatic mesh generation technique now.

Specify the following **Mesh** settings.

Defaults:

Physics Preference: CFD

Sizing:

Use Advanced Size Function: On Proximity and Curvature

Generate the new mesh. Change the settings under **Physics Preference** and verify their effect on quality of generated mesh.

One can introduce the boundary layer mesh near to walls. The aim is to make the grid denser in the high velocity gradient flow regions. Right click on **Mesh** cell and introduce **Insert/Inflation**. Activate the **Face** symbol in the geometry selection filter in-upper menu panel and select all surfaces under **Geometry** selection cell. Select all edges corresponding to walls under **Boundary**. Use the Ctrl button to select multiple edges. Set the number of layers to 10 (**Maximum Layers**) and generate the mesh.



Figure N°10. ANSYS Workbench – Meshing

Save the Project.

### *Additional problem*

Make the geometry shown below and generate the mesh

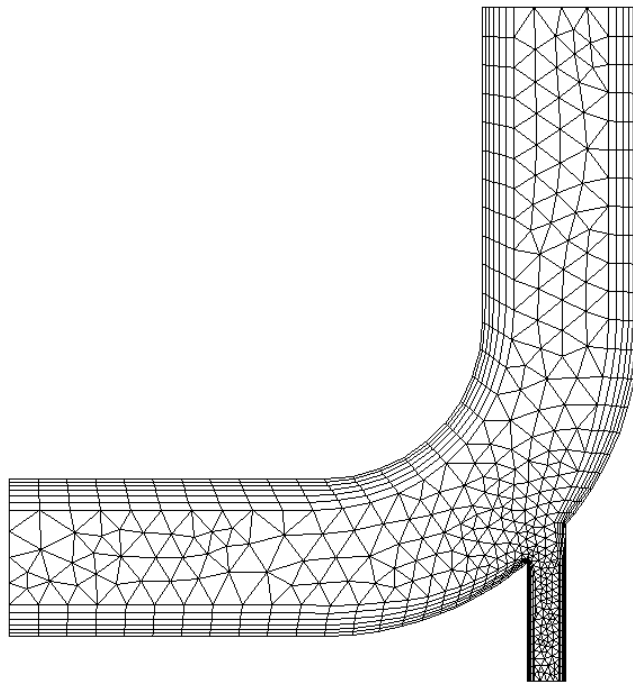
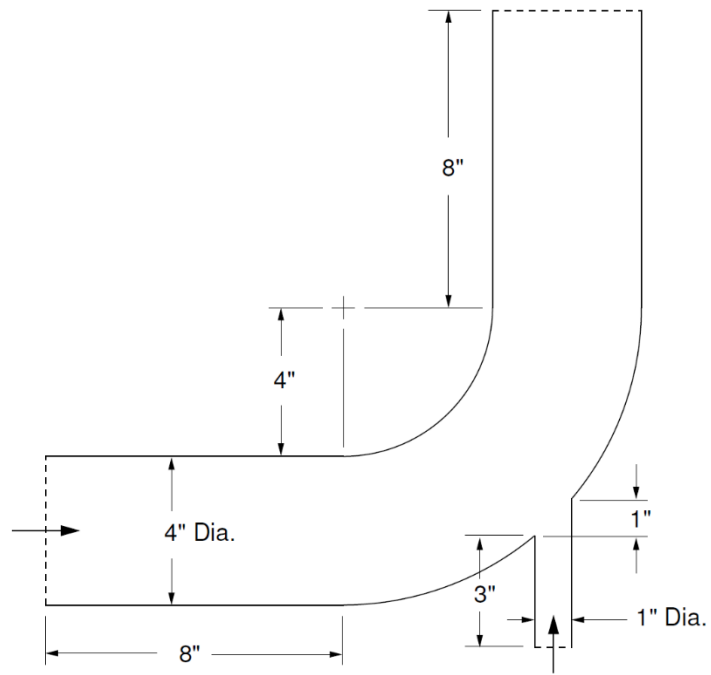


Figure N°11. ANSYS Workbench – Additional Project