The Institute of Aeronautics and Applied Mechanics
Department of Aerodynamics

subject: Advanced Computational Fluid Dynamics
Komputerowa Analiza Przepływów

Exercise 2

Flow through a valve

Purpose

To practice the simulation of the unsteady turbulent fluid flow using the moving (rotating) mesh.

Duration: 2h

The mesh should be created as shown in Fig 1 (coarse mesh). The zones A and C correspond to the inlet and outlet channels while the zone C corresponds to the main valve passage.

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. Set the unit length to mm.
2. Sketch the circles and rectangles as shown in fig 2.
   - Sketch 1: Make a circle with diameter 40 mm (xy-plane) at point (0 0 0)
   - Make two rectangles 10 x 80 mm (sketch 2) and 80 x 10 mm (sketch 3)
   - Make two circles (sketch 4). One with D_1=50 mm and the other with D_2=30 mm. Both circles are placed at point (-20, -20, 0).
3. Use the sketches as shown in fig 2 to make a number of surfaces. Go to **Modeling** mode select all sketches. **Concept/Surfaces from sketches**. Change **Add Material** to **Add frozen** under **Operation**.
4. Next, the surfaces have to be split into the number of surfaces. This can be done using **Extrude** option (**Create/Extrude**). Select all sketches and provide them under **Geometry**. Select **Slice Material** under **Operation**. Change **As Thin/Surface** from **No** to **Yes**. Put **Inward** and **Outward Thickness** to 0. Push on the **Generate** button. A number of surfaces will pop up under **Parts**. Apply the same **Extrude** operation once again selecting the sketch1 and sketch2. Do it once more selecting the sketch1 and sketch3. Some surfaces will be split into smaller surfaces.
5. The aim is to obtain as many small surfaces as possible (fig. 3). Still some surfaces have to be split into smaller ones. One can use **Create/Boolean/Subtract** option to subtract some surface(s) from the other (change **Preserve Tool Bodies** from No to Yes).

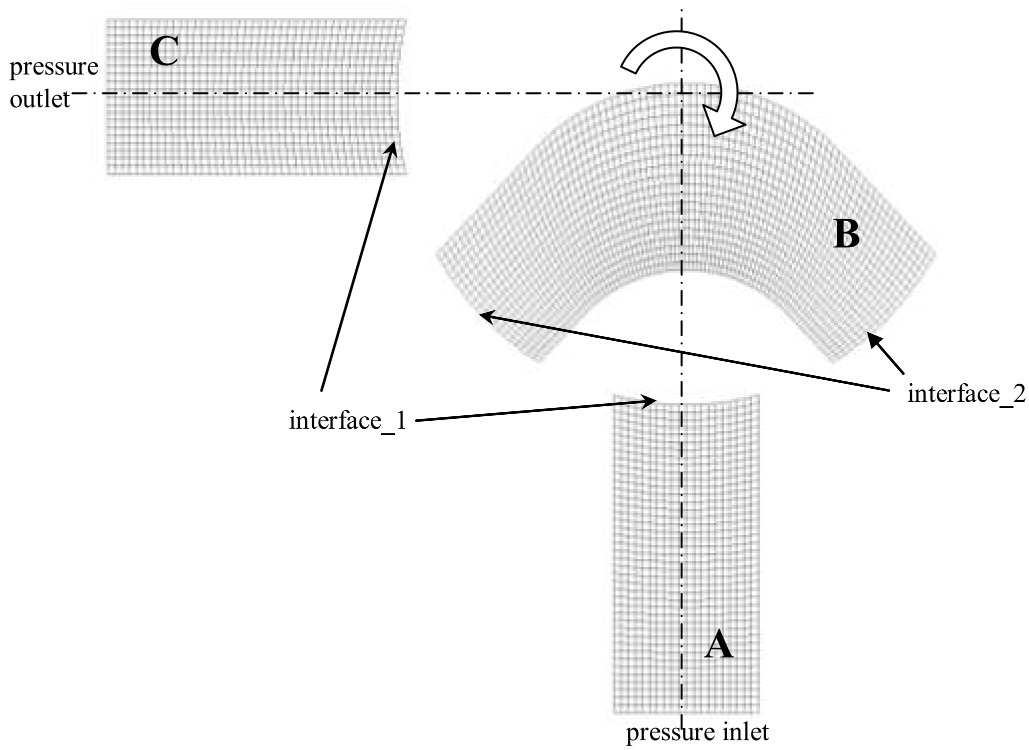

Figure 1: Computational mesh (coarse). Here the main passage is rotated by -45 deg.

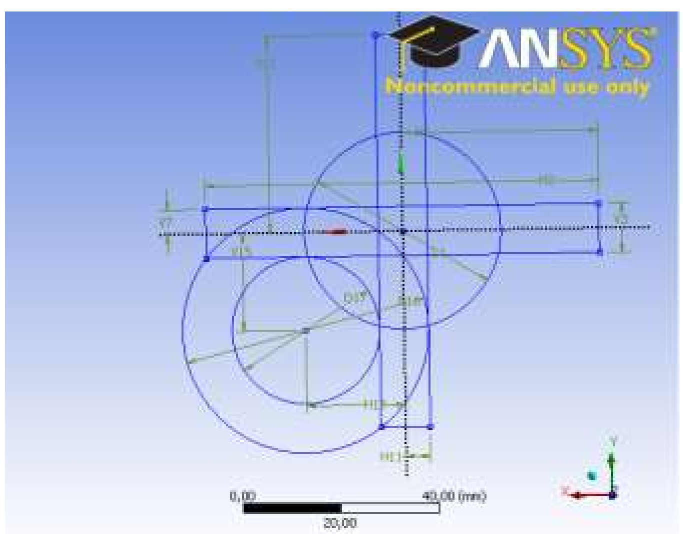

Fig. 2. The circles and rectangles used for preparation of valve.

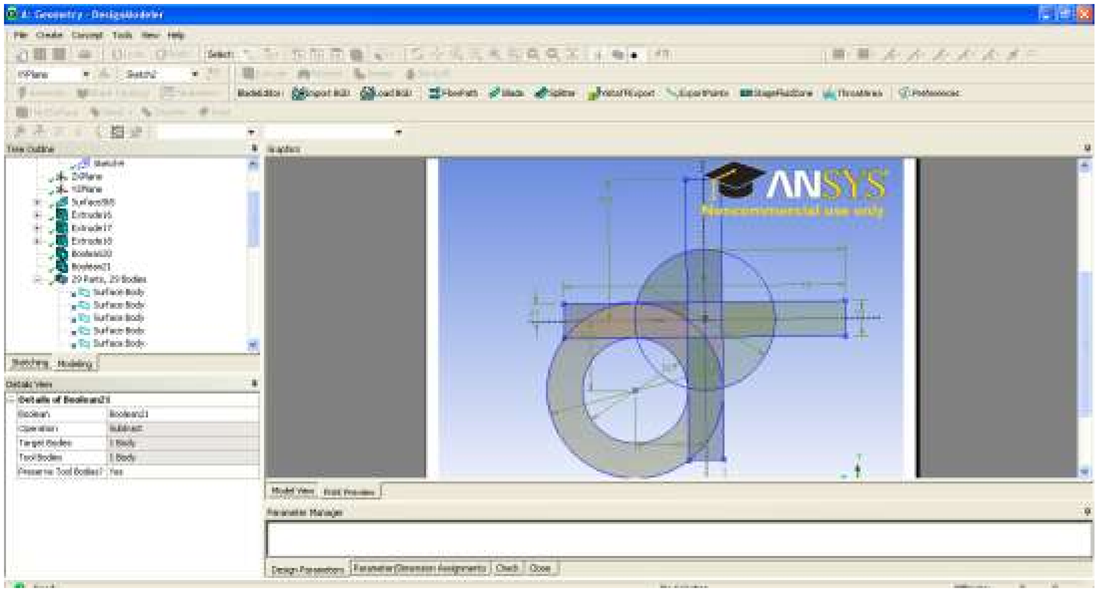

Fig. 3. Split of surfaces into the smaller ones.

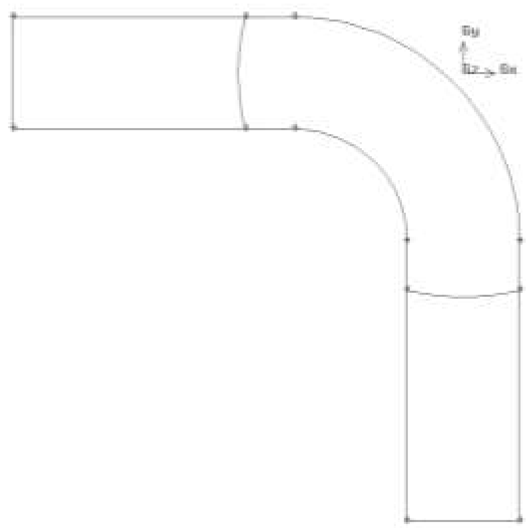

Fig. 4 Three surfaces.

6. Next some surfaces have to be merged in order to obtain the three surfaces as shown in fig. 4 **Create/Boolean/Unite.**
7. Put the unused surfaces to **Suppress** under **Parts** (fig. 5). Right click and suppress.
8. Make a **New Part** from the three active surfaces under **Parts**.
9. Rotate the main passage by 30 deg (fig. 6). **Create/Body Operation/Rotate**. Click on the zx-plane before choosing the axis perpendicular to the body under **Axis Selection**. Specify -30 deg under Angle.
10. Specify the name of boundaries and zones as shown in fig. 1. Select the line, right click and select the **Named Selection**. Specify the two zones A (inlet) and B (rotating_passage). Right click, **Named Selection**.

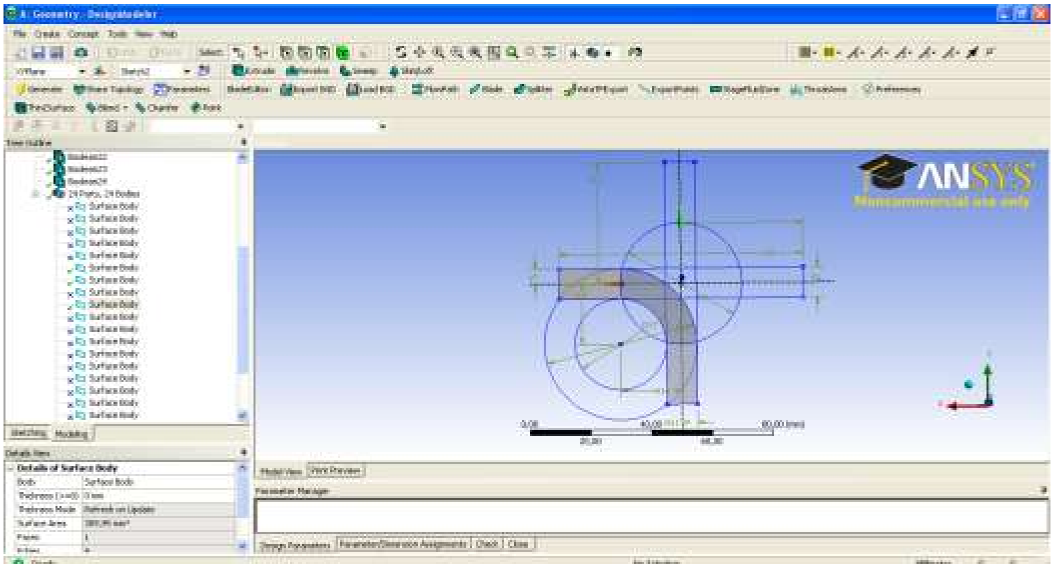

Fig. 5. Suppression of unused surfaces under Parts.

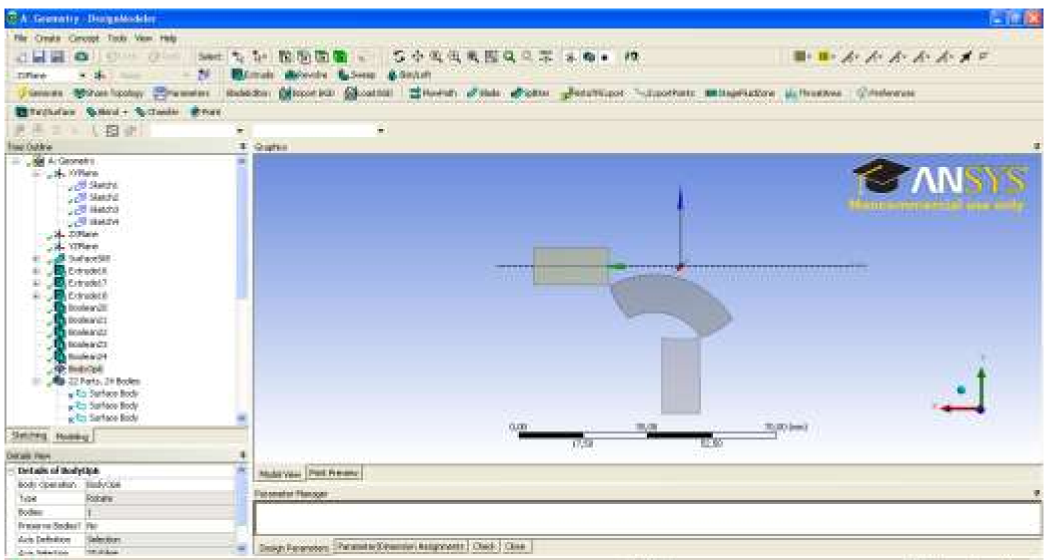

Fig. 6. Rotation of main passage by -30 deg.

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 the preliminary mapped mesh (see fig. 1). This will be the coarse mesh. Make the second fine mesh, refined near to the walls. Export both meshes to Fluent *.msh file. **File/Export/Fluent input file**

Fluent

1. Read the coarse mesh. **File/Read/Mesh**
2. Set the Pressure Based solver and activate Transient in **Define/General**
3. Check the scale. The mesh was created in mm.
4. Set-up the interface in **Define/Mesh Interfaces/Create Edit.** All the interfaces/interface with the name 'interface_1' should be assigned to all remaining interfaces/interface denoted by 'interface_2'. Provide the name under Mesh Interface.
5. Activate the Spalart-Allmaras turbulence model in **Define/Models/Viscous**
6. Create a new fluid (liquid water) in **Define/Materials/Fluid/Create Edit** . Select it from the Fluent Database.
7. Set the liquid water in all zones 'a', 'b' and 'c' (fig. 1) in **Define/Cell Zone Conditions/Edit.** Once the liquid water is set for the one zone/face the settings can be copied to the other zones/faces.
8. Change the angular velocity from rad/sec to rpm. **Define/Units**
9. Set the moving mesh in the motion type in **Define/Cell Zone Conditions/Edit** for the face 'b' (rotating_passage). The rotational velocity could be set to 20 rpm.
10. Specify the following boundary conditions:
    - Pressure inlet: Set the Gauge Total Pressure and Supersonic/Initial Gauge Pressure to 100000 Pa and the turbulent to molecular viscosity ratio to 10.
    - Pressure outlet: Set the turbulent to molecular viscosity ratio to 1.

11. Initialize the solution. **Solve/Initialization** with gauge pressure, velocity components set to zero and the <u>turbulent viscosity set to low values 0.01</u>. Verify the velocity magnitude in **Display/Graphics and Animations/Contours/Set up**. To adjust the view use **Display/View/front**
12. Coming back to the **Solve/Initialization.** Chose the **Patch** button and set the pressure equal to 100000 Pa for the zone/face 'A' (inlet).
13. Set up the animation sequences in **Solve/Calculation Activities** under **Solution Animation.** <u>The animation should be set up carefully.</u> If the animations are not working they should be removed in **Display/Graphics and Animations/Solution Animation Playback** under **Animation Sequences.** The *.hmf and *.cxa files should be removed from working directory too. The animation should be set up once again**.** The animations in **Solve/Calculation Activities /Solution Animation/Create Edit** should be set as follows:
    - Under **Animation Sequences** specify 1. Under **When** choose Time Step. <u>In **Define** set the Window to 3 and press the **Set** button.</u> In the **Display Type** take Contours of Static Pressure. Deactivate the **Auto Range** under Options and set –100000 and 110000 Pa for min and max pressure, respectively. A new window 3 will pop up. Click **Ok** button.
    - Under **Animation Sequences** specify 2. Under **When** choose Time Step. <u>In **Define** set the Window to 4 and press the **Set** button.</u> In the **Display Type** take Contours of Velocity Magnitude. Deactivate the **Auto Range** under Options and set the scale to proper values. Click **Ok** button.
    - One can set-up the other animations (velocity vectors, pathlines ..)

14. Compute the force acting in x or y direction on rotating_passage. First, the density and velocity has to be set to 1 and area to 2 under **Report/Reference value**. Next, specify the monitoring of force in **Solve/Monitors/Create**. Activate write to console and write to file.
15. In **Solve/Run Calculation** set the time step to 0.001s and the number of time steps to 10. Run the calculations. Next increase the number of time steps and complete the calculation (rotate the channel by 30-45 deg).
16. Read the fine mesh and perform the computations (steps 11-15). Compare the coarse and fine grid results.