The Institute of Aeronautics and Applied Mechanics
Department of Aerodynamics

subject: Advanced Computational Fluid Dynamics
Komputerowa Analiza Przepływów

Exercise 3

Adiabatic compression of air

Purpose

To simulate the compression of the air in the combustion chamber (combustion process is not taken into account) with the dynamic mesh technique.

Duration: 2h

Design Modeller and Mesher

The computational mesh is shown in Fig.1. The computational domain should be divided into the equidistant cells.

Fluent ver 12
1. Read the mesh. **File/Read/Mesh**
2. **Define/General**. Set the pressure-based solver and activate the transient under **Time** and axisymmetric under **2D-Space**.
3. **Define/Materials/Fluid.** The medium is treated as an ideal gas.
4. In **Define/Dynamic Mesh/Parameters** activate the **Dynamic Mesh**. In **Dynamic Mesh** activate **Options/In-Cylinder**.
5. In **Mesh Method** activate **Leyering**, **Smoothing,** and **Remeshing.**
6. In **Mesh Method** /**Settings/Leyering** set the **Split Factor** and **Collapse Factor** to 0.4 and 0.04, respectively.
7. Switch to the **Options/Settings**
8. Apply the following settings
   - **Crank Shaft Speed**: 1000-2000 rpm
   - **Starting Crank Angle**: 180 deg
   - **Crank Period**: 720 deg
   - **Crank Angle Step Size**: 0.5 deg
   - **Piston Stroke**: 0.08 m (here the distance should be less than 0.1, see Fig 1)
   - **Connecting-Rod Length**: 0.14 m
   - **The remaining settings might be left unchanged** (they are not used)
9. In the Fluent command line type (one by one)
   define <enter>
   models <enter>

```
dynamic-mesh-controls <enter>
in-cylinder-parameter > ppl <enter>
Lift Profile: (1) [()] **piston-full**
Lift profile: (2) [()] <enter>
Start: [180] 0 <enter>
End [720] <enter>
Increment: [10] 5  <enter>
Plot lift? [yes] <enter>
```
10. Move to **Define/Dynamic Mesh/Create Edit**. In **Zone Names** select moving_wall, In **Type** select **Rigid Body**. Move to **Motion Attributes**. In **Motion UDF/Profile** select ****piston-full****, in **Valve/Piston Axis** set X=-1, Y=0. In **Meshing Options** set **Cell Height**: 0.006-0.008m. Create the moving wall, pressing **Create** button. Press **Close** to finalize.
11. **File/Write/Case**. Save the project in your working directory.

One can verify the correctness of the dynamic mesh definition. First, the mesh can be displayed in **Dispaly/Mesh.** The movement of the dynamic mesh can be controlled in **Define/ Dynamic Mesh/Preview Mesh Motion.** If the dynamic mesh is set-up properly the case file might be read in **File/Read/Case**. Initialise the solution with 0 Pa and 0 m/s for gauge pressure and velocity, respectively.

The following animations/postprocessing have to be set-up:
1. Contour plots of pressure (0-900000Pa). Set window to 3
2. Contour plots of temperature (290-600K) Set window to 4.
3. Monitoring of the volume averaged pressure, density and temperature (**Solve/Monitors/Volume Monitors/Volume Average**). Set windows to 5,6, and 7.

Run the computations for the 720 time steps. The size of the time step is computed automatically, based on the settings above (p. no. 8). One can verify the temperature T_2 obtained by Fluent at top death centre (leftmost position of the piston -moving wall) with the analytic result for adiabatic process

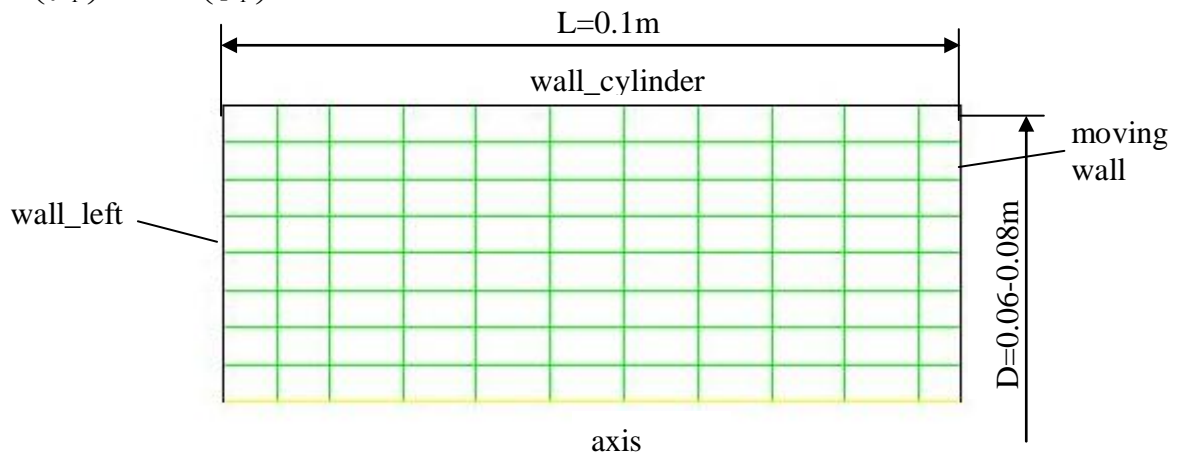$$T_2 = T_1 \left(\frac{\rho_2}{\rho_1} \right)^{\chi-1} = T_1 \left(\frac{p_2}{p_1} \right)^{\chi-1/\chi} \quad \text{for } \chi=1.4. \text{ where } p_1=101325\text{Pa}.$$

Fig.1. Computational mesh.