AE 433 CFD – HW1

DUE DATE: Beginning of the next lecture (10.12.2021 Friday, 8:30)

Parameters for the flow in a pipe are given as follows:

Pipe radius: 0.02619 m

Pipe length: 7.62 m

Cases are as follows:

Case-1: Laminar flow, Re = 655, 2D

Case-2: Turbulent flow, Re = 111569, 2D

Case-3: Laminar flow, Re = 655, 3D

Case-4: Turbulent flow, Re = 111569, 3D

Instructions:

1. Use boundary conditions of inlet, outlet and wall for your solution.
2. Specify uniform flow at the inlet, no slip boundary condition at the wall and constant pressure at the outlet (gage pressure 0 for case-1 and case-3, gage pressure 400 Pa for case-2 and case-4).
3. Use “laminar” as the solution model for case-1 and “standard k-epsilon turbulence model with standard wall functions” as the solution model for case-2.
4. Take air density 1.17 kg/m^3 and viscosity $1.872x10^{-5}$ kg/m-s.
5. Take turbulence intensity 0.01% and turbulence length scale 0.000294 m at the inlet for case-2 and case-4.
6. Take backflow turbulent kinetic energy as 1 m^2/s^2 and backflow turbulent dissipation rate as 1 m^2/s^3 at the outlet for case-2 and case-4.
7. Take sand grain roughness height $2.5x10^{-5}$ m and roughness constant 0.5 for case-2 and case-4.
8. Assume an operating condition of 97725.9 Pa.
9. Use SIMPLE method with 2nd order accuracy.
10. Use residuals to be 1e-06 for all.

Consider the following reference values:

Area (m^2)	0.002154869
Density (kg/m^3)	1.17
Enthalpy (j/kg)	0
Length (m)	0.05238
Pressure (Pa)	0
Temperature (K)	288.16
Velocity (m/s)	0.2 (case-1 and case-3), 34.08 (case-2 and case-4)
Viscosity (kg/m-s)	$1.872x10^{-5}$
Ratio of specific heats	1.4

Point Name	x0	y0
point-1	7.62	0.000
point-2	7.62	0.005
point-3	7.62	0.010
point-4	7.62	0.015
point-5	7.62	0.020
point-6	7.62	0.021
point-7	7.62	0.022
point-8	7.62	0.023
point-9	7.62	0.024
point-10	7.62	0.025

Surface Name	x0	y0	x1	y1
x=10d	0.5238	0	0.5238	0.02619
x=20d	1.0476	0	1.0476	0.02619
x=40d	2.0952	0	2.0952	0.02619
x=60d	3.1428	0	3.1428	0.02619
x=100d	5.2380	0	5.2380	0.02619

For all cases:

a) Show your geometry/mesh/solution steps on CFD by figures in your report.
b) Show your residual output in a figure.
c) Export and show the axial velocity profile at the points and surfaces given in figure above. Compare cases with each other and experimental results.
d) Plot static pressure change in the axis, compare them with the experimental data.
e) Export wall shear stress values and report them.
f) Show axial velocity profile vectors in the surfaces above.
g) Comment on analytical solutions of CFD study. Is it possible to analytically solve cases? Answer for each case separately.

NOTES:

1. Report cannot be longer than 20 pages.
2. No additional submissions are needed other than **one printed report with a cover page**.
3. Experimental data are shared in AE433 page.