Assignment for AE4202 – CFD for Aerospace Engineers

Introduction

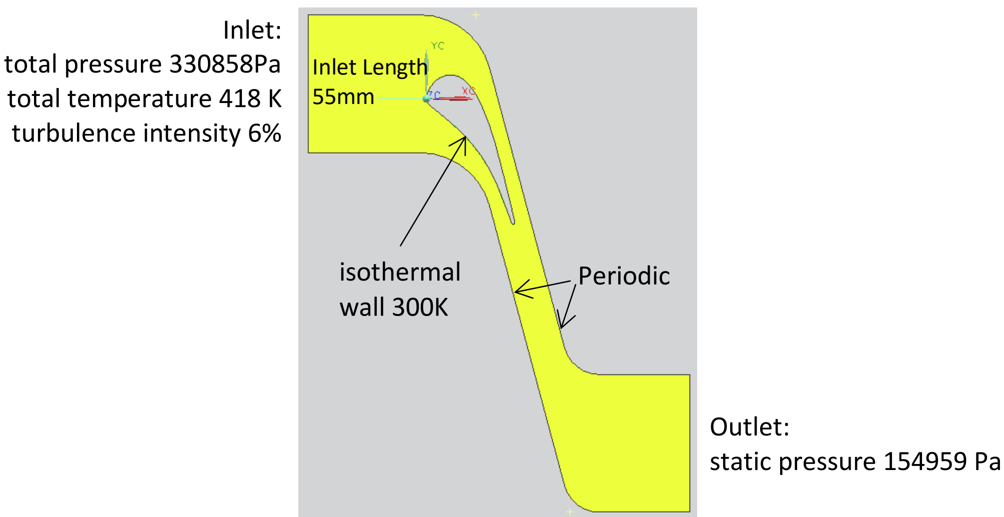
The assignment should be performed in groups of two students. You are free to choose your partner. Both team members should equally contribute to the work and submit a joint report. Though not recommended, you could also do the work alone or submit individual reports.

The deliverable is a short report with max. **10 pages**, excluding title page and list of references. The report should be submitted through Brightspace by **25 February 2020**. If you submit the report late, one point will be subtracted from the grade; a second point will be subtracted if you submit more than one week past the due date.

Problem description

Your task is to perform 2D steady state RANS simulations of a highly loaded transonic turbine nozzle guide vane, the so-called VKI S1 Nozzle.

Consider the compressible flow of air, which is considered to be an ideal gas with a dynamic viscosity given by Sutherland's law. The geometry and boundary conditions are sketched below, coordinates of the turbine blade with a chord length of 67.6 mm are provided in a separate file. The vertical domain height is 57.5mm such that the upper and lower domain boundaries are periodic. The inflow boundary condition with turbulence intensity 3% should be imposed 55mm upstream of the blade's leading edge, you may yourself choose a suitable distance for the outflow boundary.



Tasks

1. Load the blade coordinates in ICEM and create the blade geometry and a suitable computational domain.

2. Mesh the computational domain with a coarse, structured multi-block grid with about 20.000 cells (grid1). Pay attention to orthogonality, smoothness and resolution of the boundary layers. Visualize and describe blocking and mesh in the report and discuss the mesh quality.

3. As boundary conditions, impose an isothermal wall at the blade, total temperature, total pressure and turbulence intensity at the inflow and static pressure at the outflow. This is easier if you set the CFX reference pressure to zero. The other boundaries are periodic.
   Use the 'high-resolution' scheme for the spatial discretisation and the k-omega SST turbulence model. Choose a suitable fluid model and time-stepping scheme. Report the simulation setup and discuss the reasons for your choices.

4. Perform a steady state simulation. Plot the solver residuals to verify convergence of the simulation. State the Mach number measured near the inflow and at the outflow boundary.

5. Visualize the flow field: produce contour plots for the Mach number, the static pressure and temperature. Plot the non-dimensional pressure coefficient Cp along the blade surface. Extract the lift and drag forces and summarize them in a table. Interpret your results.

6. Compute and plot the y+ value of grid1 along the blade. Refine the grid such that y+ < 1 for the refined grid (grid2). Visualize and describe blocking and mesh, pay attention to orthogonality and smoothness. Perform the simulation and plot the solver residuals to verify convergence of the simulation. Plot y+ for grid2 to verify that the resolution target was achieved. Critically discuss the mesh quality.

7. Perform the same post processing and visualizations as done for grid1. Include the new Cp curve in the same figure and the lift and drag forces in the same table as the results for grid 1 (same holds for point 8 and 9). Describe and evaluate the results.

8. In order to quantify the effect of numerical diffusion, repeat the simulations on both grid1 and grid2 with a first-order upwind scheme. Plot the solver residuals and verify convergence of the simulation. Repeat the same post processing and visualizations as done before. Visualize and analyse the effect on the results.

9. In order to quantify the effect of the turbulence model, repeat the simulation on the fine mesh (grid2) with a turbulence model that can better account for anisotropic turbulence. Which model did you chose and why? Plot the solver residuals and verify convergence of the simulation. Visualize and analyse the effect on the results.

10. Summarize your results, discuss them critically and suggest possible improvements.