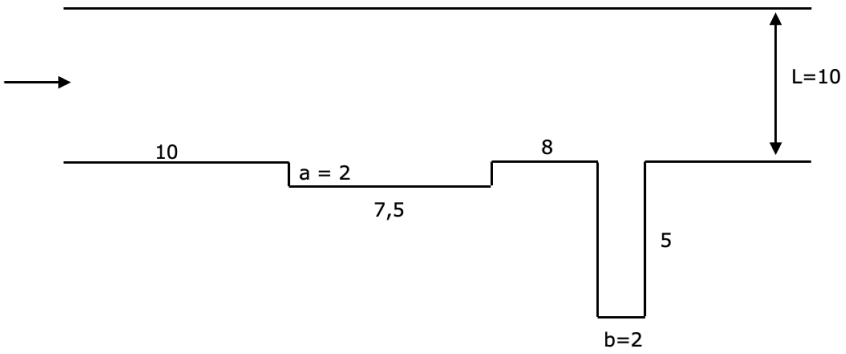
Project for the lecture Applied CFD

Project: CA5	
Name: Vittor Braide Costa	Name: Giovanna Girotto
Matrikel Number: 03783398	Matrikel Number: 03783374
E-Mail: vittor.costa@tum.de	E-Mail: giovanna.girotto@tum.de
Moodle Group: 75	Moodle Group: 75

Problem definition:

Investigate the following stationary cavity flow.



Inlet conditions:

Mach number: $Ma = 0.1$
Reynolds number: $Re_L = 5 * 10^4$

Create a structured 2D-Mesh with ICEM. Choose a reasonable domain length.

Select the SST turbulence model and conduct a simulation taking friction into account. Solve the RANS equations with CFX. Use the "high resolution" scheme for the spatial discretization. Evaluate the result and, if necessary, adjust the grid depending on the solution. Focus especially on refining the grid next to walls in poorly resolved areas. Repeat the calculation on the improved grid. Check the success of the grid refinement, and provide final key properties of this grid on the cover page of the report.

Now, double the depth of cavity a, and the width of cavity b. Perform the simulation again.

Special task:

Compare the results of the SST turbulence model with a non-algebraic Reynolds stress turbulence model of your choice for the cavities with the original dimensions. Pay special attention to the computing effort and convergence behavior.

Content of the report:

Cover page (1 page):

- Indication of the project number, names, and matriculation numbers
- Short presentation of all solver settings:
  - Advection scheme, timestep control, and timestep size
  - Fluid properties and fluid model
  - Boundary conditions
- Most important grid properties (grid size, grid dimension, and quality criteria)
- Do NOT provide any explanations or derivations. Save these for the main part.

Content of the main part (max. 10 pages!):

Boundary conditions and setup (about 20% of the points):

- Justify briefly and precisely the choice of your boundary conditions, fluid model, and discretization.
- State the advantages and disadvantages of the used turbulence model. What influence on the result do you expect it has?

Grid (about 30% of the points):

- Display your blocking, the whole grid, and the resolution in the boundary layer appropriately (zoom in when necessary).
- Display the "y+" value of the cells next to non-slip walls for the whole domain.
- Critically assess the quality of the grid.

Convergence behavior (approx. 10% of the points):

- Display the convergence (residuals) of all relevant quantities and evaluate their behavior.

Discussion of the results (approx. 25% of the points):

- Present relevant properties and peculiarities of the flow as well as characteristic quantities of the problem in a suitable way.
- Describe and interpret the influence of the size of the cavities.
- Discuss the special task

Annex:

- Add the results of the CFX Report Viewer for all relevant simulations.

Pay attention to a clear and concise presentation (approx. 15% of the points).

Please upload the final report as one single PDF file in moodle until September 15, 2024.

Contact: acfd.aer@ed.tum.de