OpenFOAM

This submission uses the tool *OpenFOAM* to generate different simulation scenarios

Scenario 1: Lid Driven Cavity

This solves the lid driven cavity problem similar to the first submission in this lecture. This serves as a initial plausibilization with OpenFOAM.

Scenario 2: Cavity with a Step

We inspect a different domain with an inflow and outflow condition at the left and right boundary, see following image

Geometry

Therefore we need to define in `blockMeshDict` 8 vertices (with additional 8 vertices since in openFoam everything is 3D). Then the three blocks I, II and III are defined with these vertices. Similar the boundary faces are defined. On the left there is an inflow BC and on the right there is a outflow BC. They are defined in the initial value file `0\p` and `0\U`.

Inflow For `U` is of type `fixedValue` with value `uniform (1 0 0)` For `p` of type `zeroGradient`

Outflow For `U` the type is `zeroGradient` For `p` of type `fixedValue` with value 0

Parameters

parameter	value
nu	0.005
deltaT	0.0005
write Interval	20
solver	icoFoam

Process

1. `blockMesh`
2. `icoFoam`
3. `paraFoam`

Resolution

The problem was solved for different resolutions: * 1 Unit = 10 Nodes -> 225 Cells * upper left square I is 5x5 * took 2.3s * 1 Unit = 100 Nodes -> 22 500 Cells * upper left square I is 50x50 * took 337s

Results

Note: Unfortunately the colorbar uses white font, so white background makes it invisble. See accompanying pdf.

Flow U

Start Value t = 0

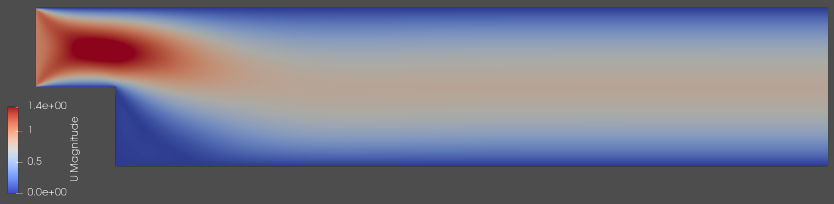

Figure 1: t=2

End Value t = 2

Pressure p

Streamlines

Glyphs (Arrows)

Low resolution 3D Visualization

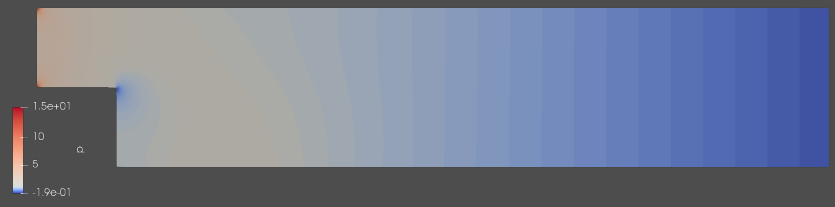

Figure 2: p

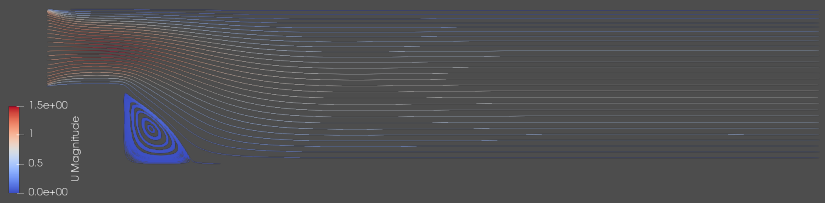

Figure 3: Streamline

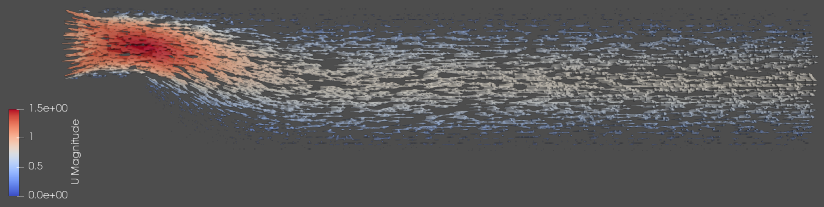

Figure 4: Glyph

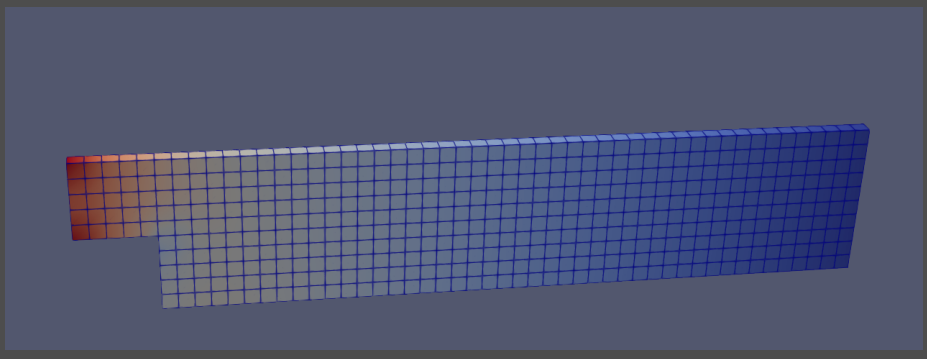

Figure 5: 3D initial

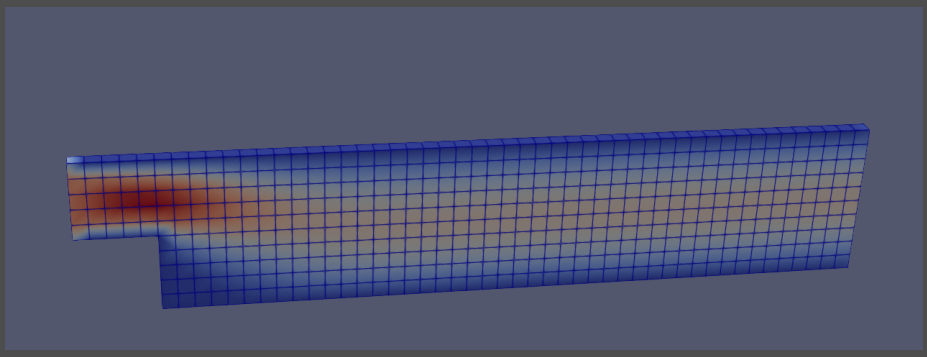

Figure 6: 3D end