Department of Material Science and Engineering
Faculty of Natural Science
NTNU

NTNU

TMT4208: Project

Hand out: 05.03.2021
Hand in: 12.04.2021 before 12:00
in .pdf format on INSPERA

The project deals with various problems related to CFD simulations of practical problems, related to tasks given in exercises throughout the semester and examples discussed during the lectures. Three cases are presented below with some specific tasks/questions which you are asked to perform/answer.

In addition to the specific tasks you should - for each *case*:

i. Describe the mesh you are using - i.e. the number of divisions used.

ii. Assess the influence of the mesh for minimum one of the tasks/settings in the case. As a minimum, key results should be compared for two different meshes (on the same geometry).

iii. For each case, for minimum one of the tasks - investigate the influence of changing at least one numerical setting (i.e. some setting under the *solution* or *discretization*-tabs.) Use the internet to determine what the setting does and give a brief description of how it changed your results.

iv. Report the time which the simulation took to complete.

Your report should should describe the essensce of each casee you are studying along with any additional assumptions you have made as an introduction to the specific tasks. Maximum 20 pages are allowed for the report, with 12 pt font and margins as in the provided LaTeX template - corresponding to this document.

Case 1: The cavity revisited

a. Repeat the cavity case for three different velocities of the upper wall surface. How does this influence the *flow* within the cavity? *Hint:* Velocity probes, glyphs and plot-over-line-profiles are useful tools in order to describe the flow.

b. For one of the velocities investigated in task a., introduce a baffle and a corner for the cavity as sketched in figure 1. Describe how these features influence the flow in the cavity.

c. Which of the three cases (cavity, cavity with baffle and cavity with corner) has the largest (average) shear stress on the leftmost wall? *Hint:* The monitors menu has a tab which is called *Forces* - some of which are proportional to the wall shear stress.

Case 2: The heated cylinder

a. Repeat the heated cylinder case for three different velocities on the inlet. How does this influence the flow and temperature in the wake of the cylinder? Note that we are assuming laminar flow for this case - make sure your Reynolds number always corresponds to this assumption. To be safe, choose $Re_D = u_{in}D/\nu < 2000$.

b. For the smallest velocity chosen in task a), repeat the simulation with air as your working fluid. Explain any differences in flow- and temperature fields.

c. Estimate the heat tranfer cofficient/Nusselt number for the cases you have simulated. How well does this correspond to the values you expect from your own experience? *Hint:* The difference in heat going in and and out (at statistically steady conditions) must correspond to whatever amount of heat is transferred from cylinder to fluid.

Case 3: Rising bubbles

a. Change the setup of case 3 in the lecture to simulate a 2 cm diameter air bubble rising in water - starting 20 cm below the water surface. Estimate the terminal velocity of the bubble from your simulations - how does the terminal velocity and bubble shape compare to that given in the diagram we studied in the lecture on bubbles, drops and particles?

b. Run minimum three additional cases where you change some properties of the problem - for example surface tension or bubble size - and assess whether your simulations are in (at least qualitative) agreement with the diagram from Clift et al. *For example* - increasing parameter x appears to make bubbles more shaped like y and have terminal velocities of order z.

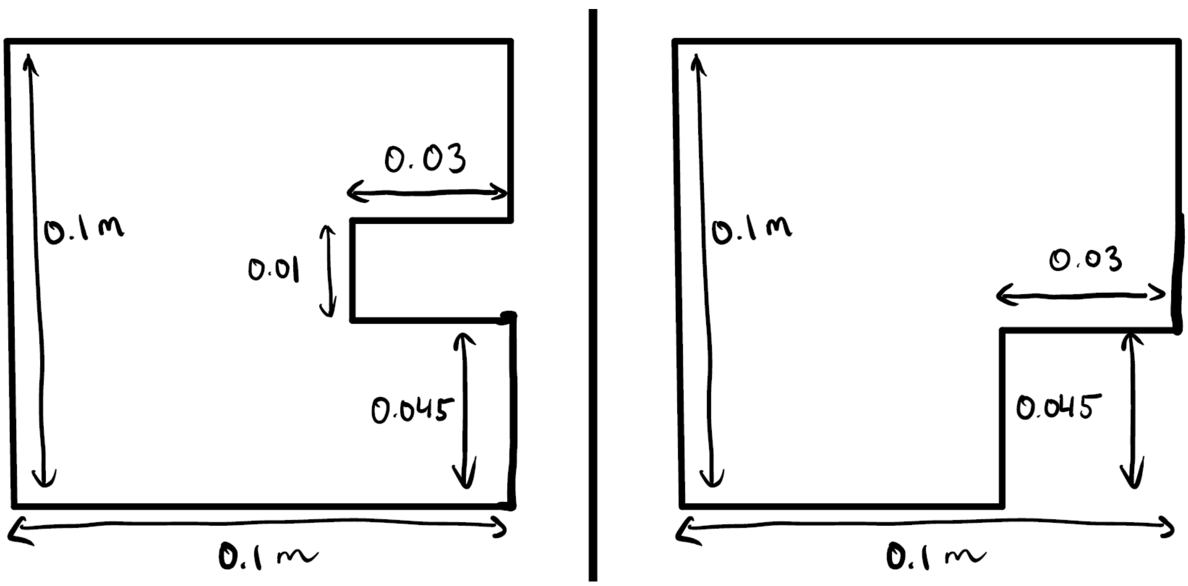

Figure 1: Sketch of domain for Case 1b.