

STRUCTURAL WIND ENGINEERING

Roland Wüchner

Máté Péntek

Anoop Kodakkal

Presentation material from internal and external sources have been used either directly, modified or adapted to fit the purpose. Effort is continuously being made to accurately reference these. Nonetheless, check referencing in both the script as well as slides for completeness. In case of inconsistencies or mistakes please contact us!

The aim of this tutorial is to do another exercise on solving a fluid problem with GiD and Kratos. This time we will work with a 3D-problem.

Covered topics:

- Preprocessing
 - Geometry
 - Input data and conditions
- Postprocessing

You should be able to run this 3D example on your own computer in 15-45 minutes.

Disclaimer: This example serves the sole educational purpose of demonstrating how to setup a basic 3D CFD problem, run the simulation and do some postprocessing. A mesh and time step refinement should be carried out for a qualitative check of the results.

Technical note: Tested on 13.12.2020, works with GiD 15.0.1 and the Kratos problemtype (8.1) on Windows 10 and Ubuntu 18/20 64 bit.

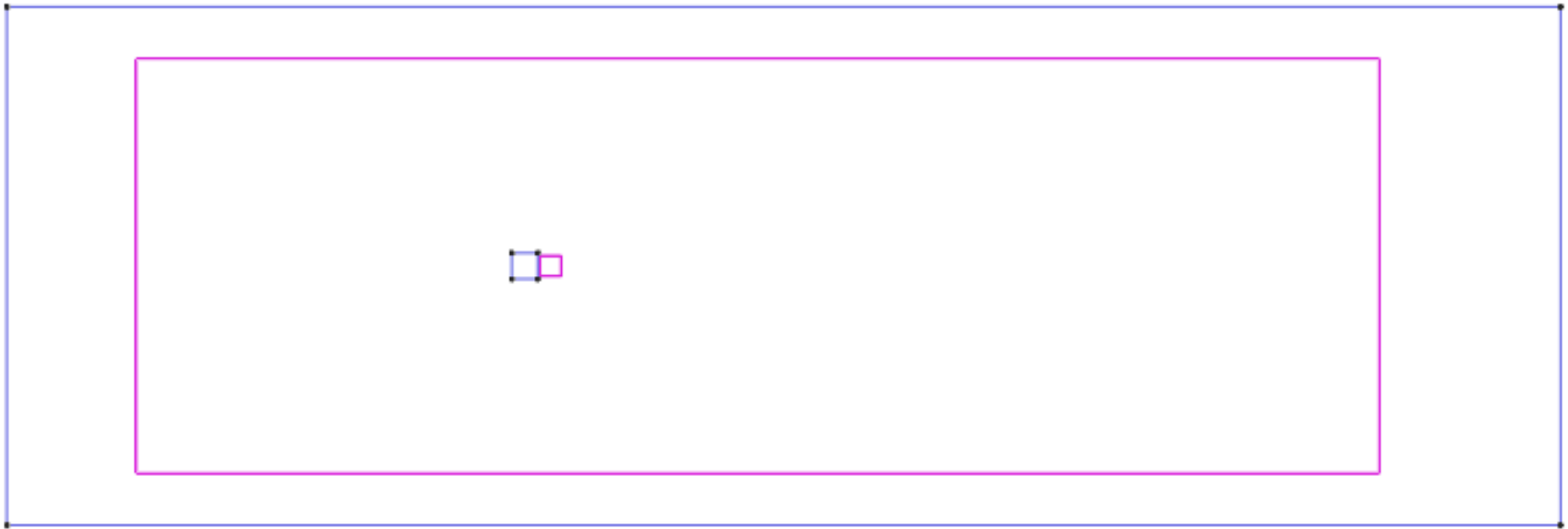
Defining the Geometry

- Create the geometry in the XY-plane using the following points to describe it:

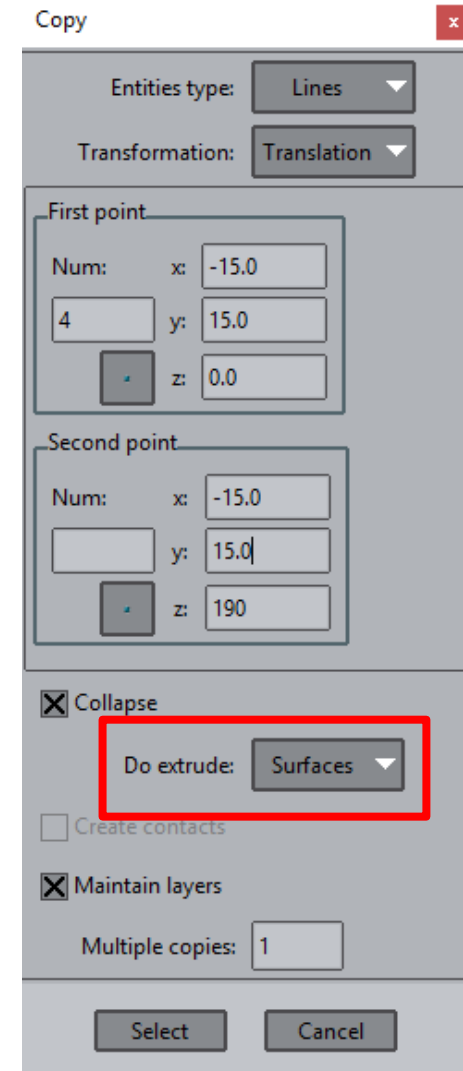
Structure			Boundary		
X	Y	Z	X	Y	Z
15.0	15.0	0.0	-600.0	-300.0	0.0
15.0	-15.0	0.0	-600.0	300.0	0.0
-15.0	-15.0	0.0	1200.0	-300.0	0.0
-15.0	15.0	0.0	1200.0	300.0	0.0

- Create the points first, followed by the lines and the surface

- The final geometry should look like this:



- Now we add the third dimension
or *Utilities* → *Copy*
Ctrl + c
- Select the following options in the Copy window:
 - *Entities type:* Lines
 - *Transformation:* Translation
 - *First Point:* One of the structure corners
 - *Second point:* Enter the same point but set the *z-coordinate to 190.0*
 - *Do extrude:* Surfaces
- Don't change any other options
- Click *Select* and mark the structure base (4 lines)
- Press *Esc* after marking the base



The image shows a 'Copy' dialog box with a red close button in the top right corner. The dialog is divided into several sections. At the top, 'Entities type:' is set to 'Lines' and 'Transformation:' is set to 'Translation'. Below this, there are two sections for point selection. The 'First point' section has 'Num:' set to 4, and coordinates x: -15.0, y: 15.0, and z: 0.0. The 'Second point' section has 'Num:' set to an empty box, and coordinates x: -15.0, y: 15.0, and z: 190. Below these sections, there are checkboxes for 'Collapse' (checked), 'Create contacts' (unchecked), and 'Maintain layers' (checked). The 'Do extrude:' dropdown menu is highlighted with a red rectangle and is set to 'Surfaces'. At the bottom, there is a 'Multiple copies:' field set to 1, and 'Select' and 'Cancel' buttons.

Copy

Entities type: Lines

Transformation: Translation

First point

Num: 4 x: -15.0 y: 15.0 z: 0.0

Second point

Num: x: -15.0 y: 15.0 z: 190

☒ Collapse

Do extrude: Surfaces

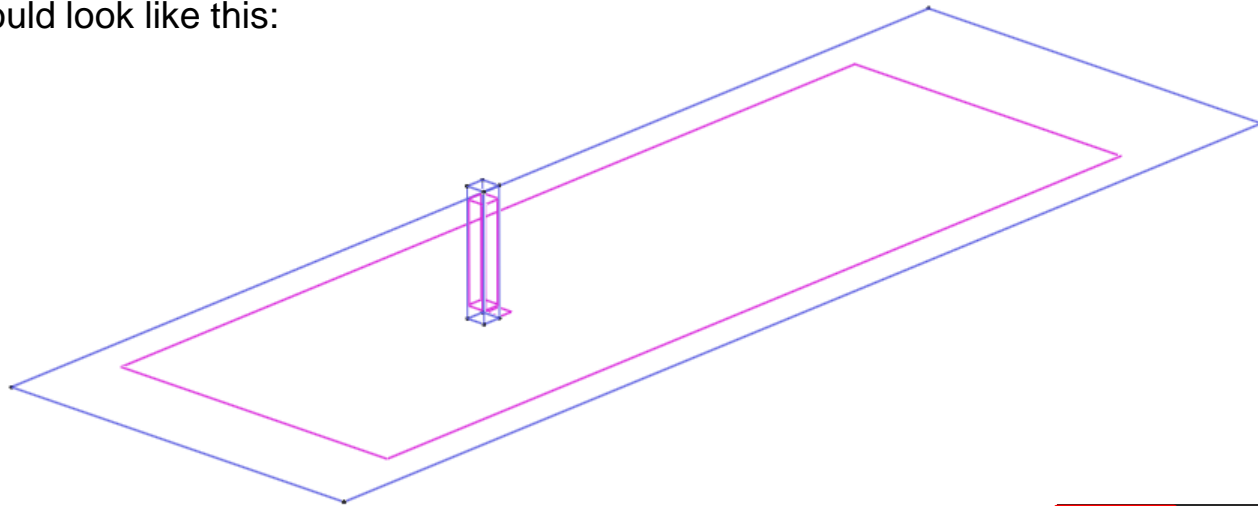
☐ Create contacts

☒ Maintain layers

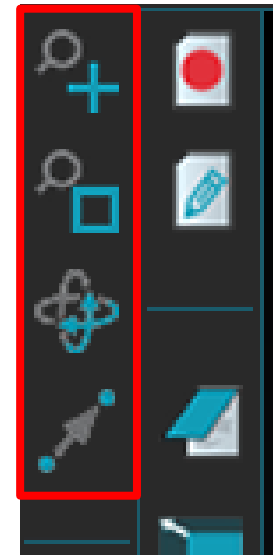
Multiple copies: 1

Select Cancel

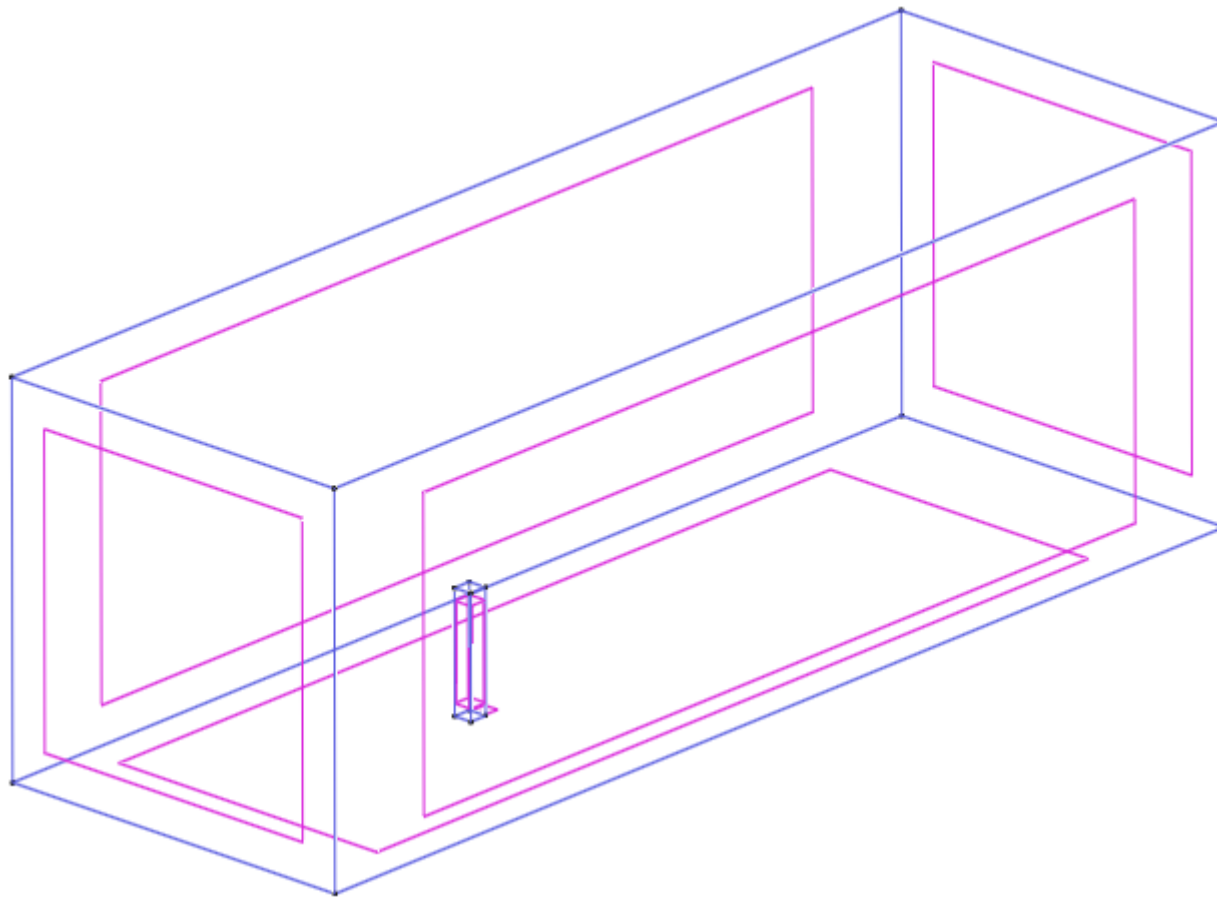
- The result should look like this:



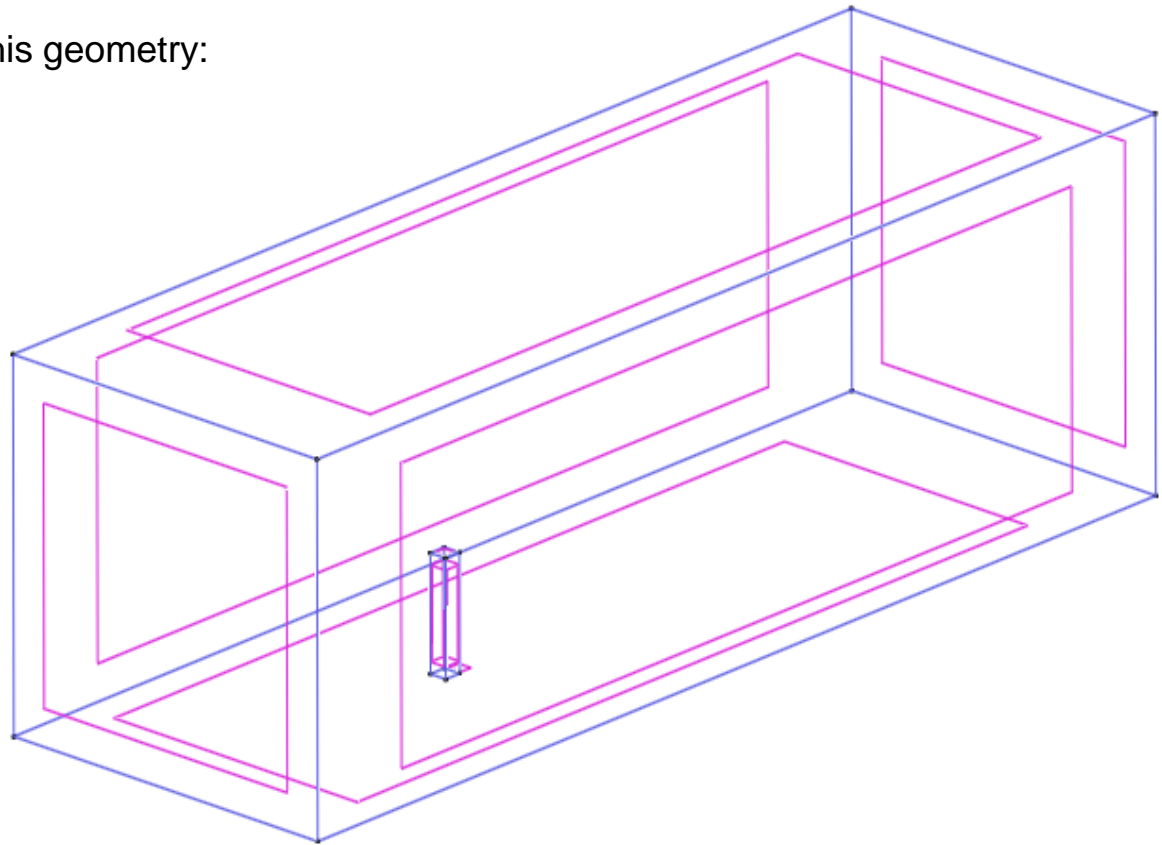
- Adjust/rotate the view to check if the transformation worked



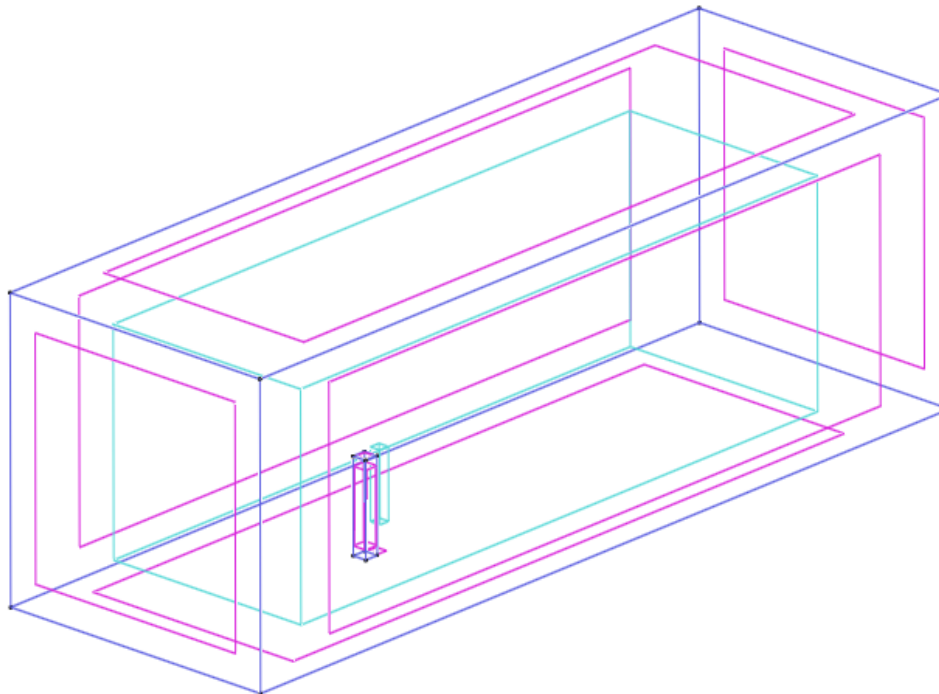
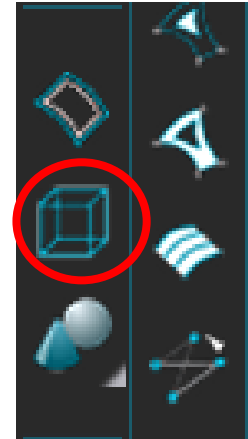
- Repeat the same steps for the outer boundary. Set the *z-coordinate to 600*
- *The result looks like this:*



- Clearly there are still two surfaces missing
- One *on top of the structure* and one *on top of the boundary*
- Creating them you will get this geometry:



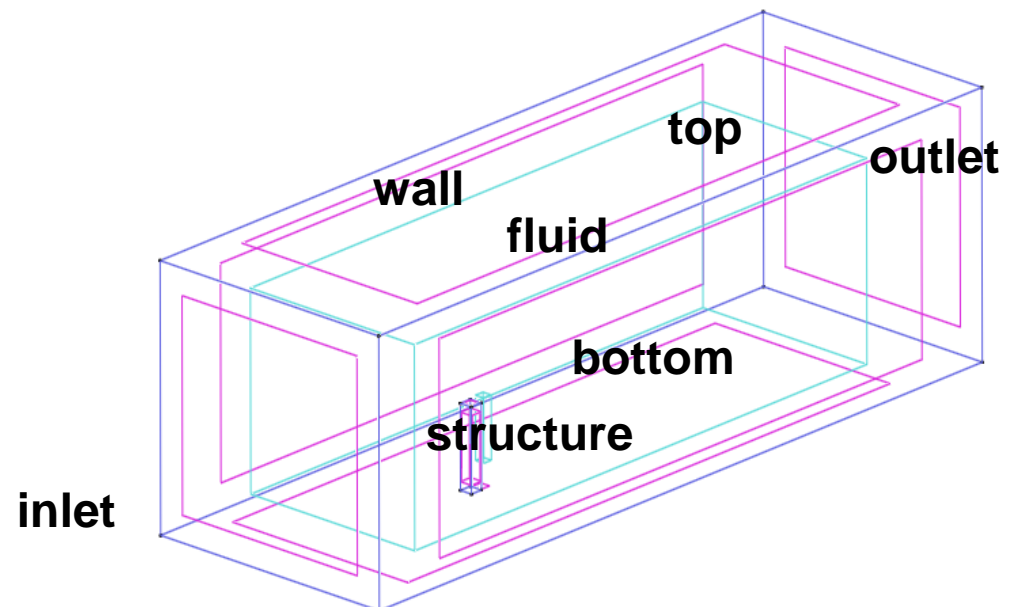
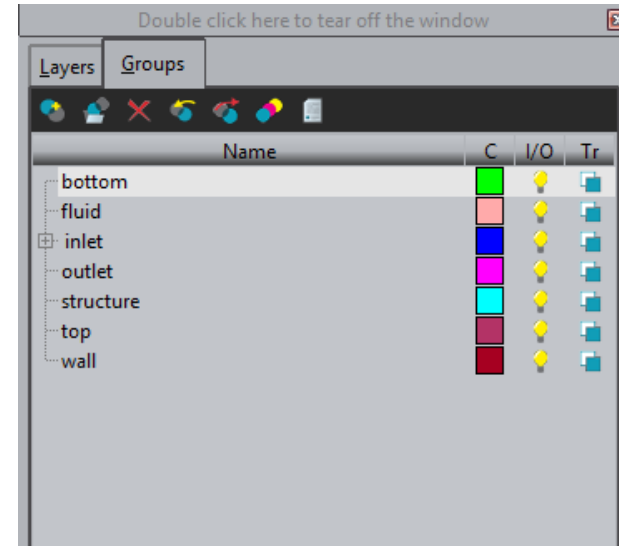
- The final step is to define the volume
Geometry → Create → Volume → By contour
- Select all surfaces and press *Esc*
 - Hint: You can select all surfaces by drawing a box around the entire model



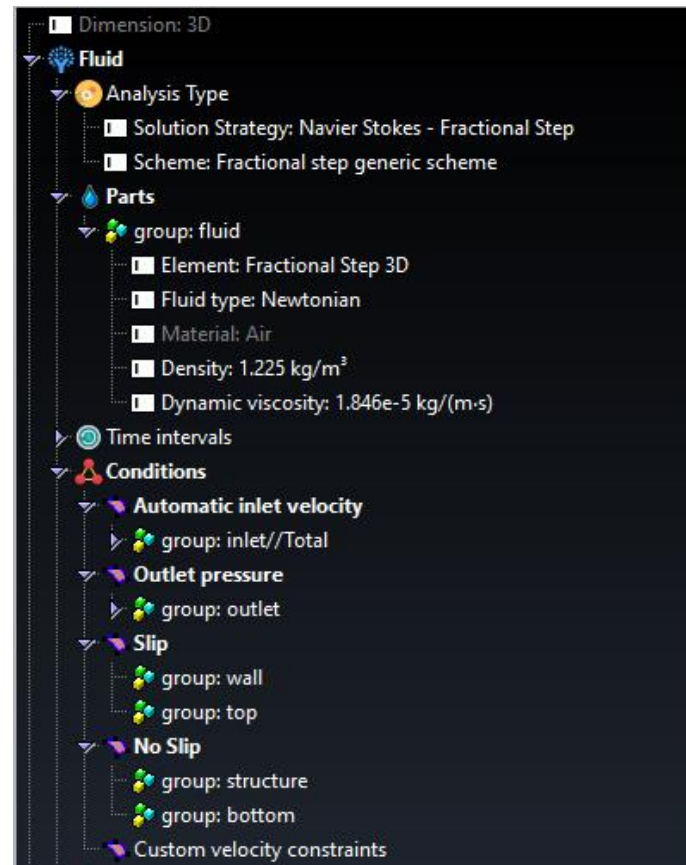
Problem Input

Define the entity groups

- *bottom* group
 - Select surface
- *fluid* group
 - Select volume
- *inlet* group
 - Select surface
- *outlet* group
 - Select surface
- *structure* group
 - Select surfaces of the structure
- *wall* group
 - Select surface on the sides
- *top* group
 - Select surface on the top

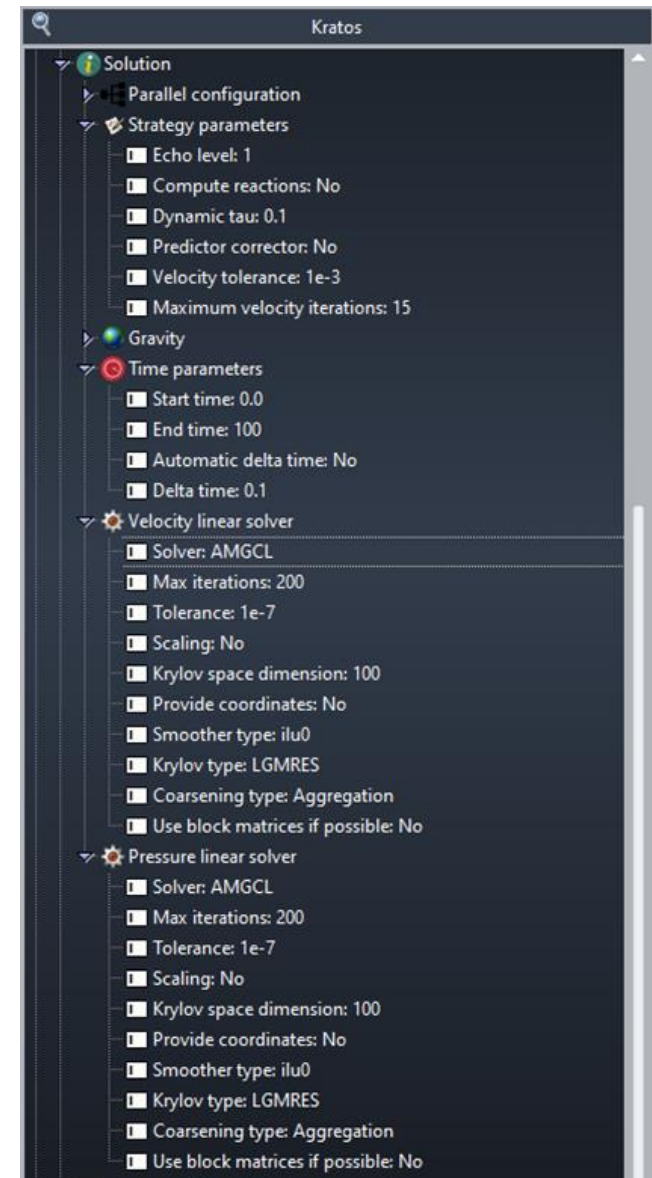
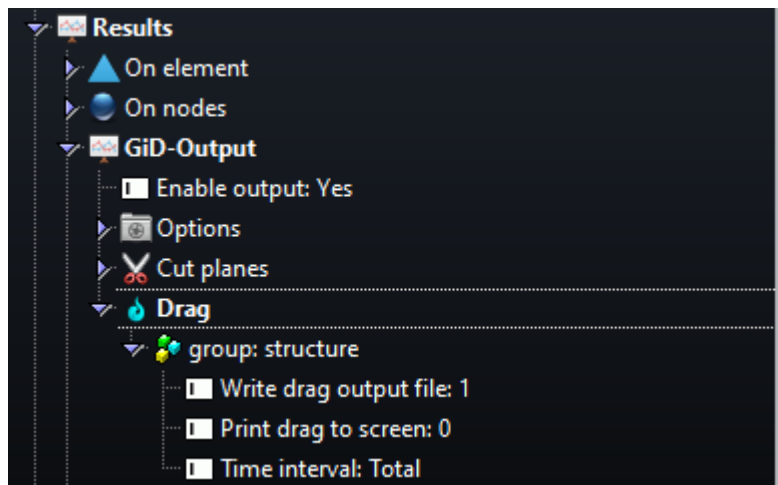


- Set *Solution Strategy* to *Navier Stokes – Fractional Step*
- Set the same *fluid properties* and *elements* as in the first tutorial
- Assign the following *boundary conditions*:
 - inlet: Inlet velocity ($X = 25.0$)
 - outlet: Outlet pressure (0.0)
 - wall: Slip
 - top: Slip
 - structure: No-slip
 - bottom: No-slip

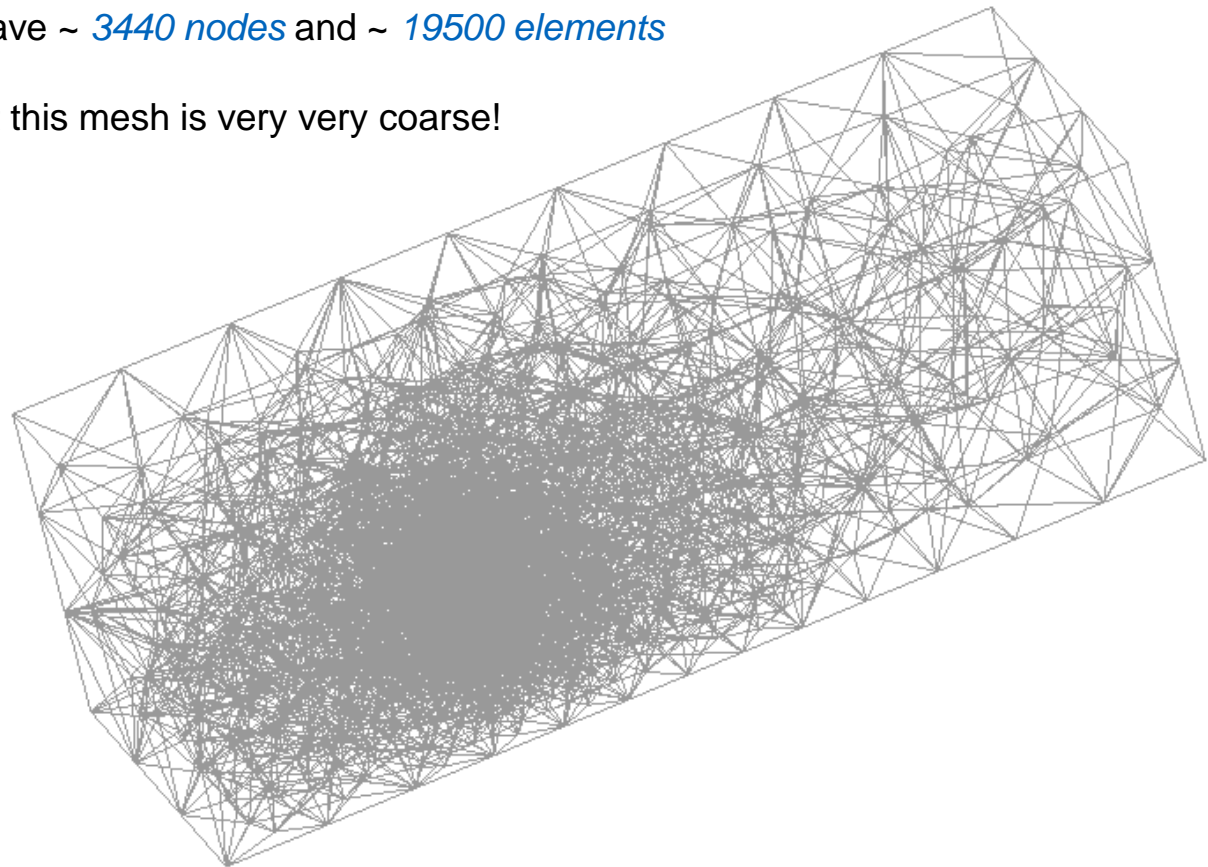


Model properties & boundary conditions

- Set the same *Solution parameters* as in the first tutorial
- Set *End time of the analysis to 100* and *Delta time to 0.1*
- Set *Write drag output file to group structure*

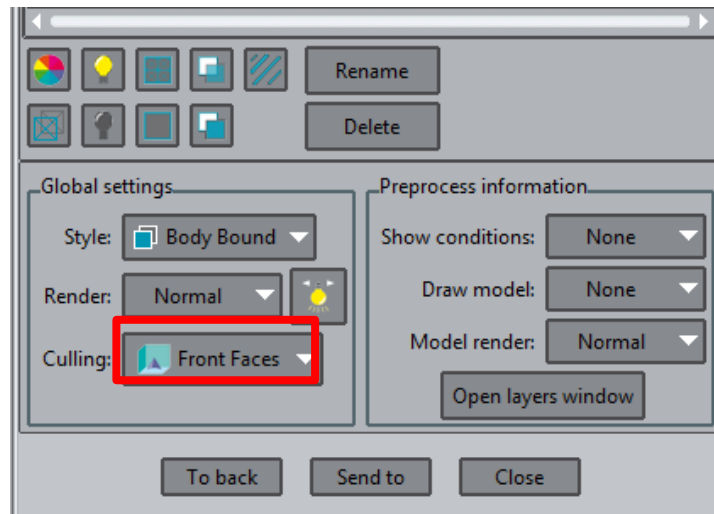


- Assign *size = 10 to the five surfaces of the structure*
- Click generate mesh and set the *element size to 200*. Don't forget to tick the box *get meshing parameters from model*
- The final mesh should have ~ *3440 nodes* and ~ *19500 elements*
- Please keep in mind that this mesh is very very coarse!

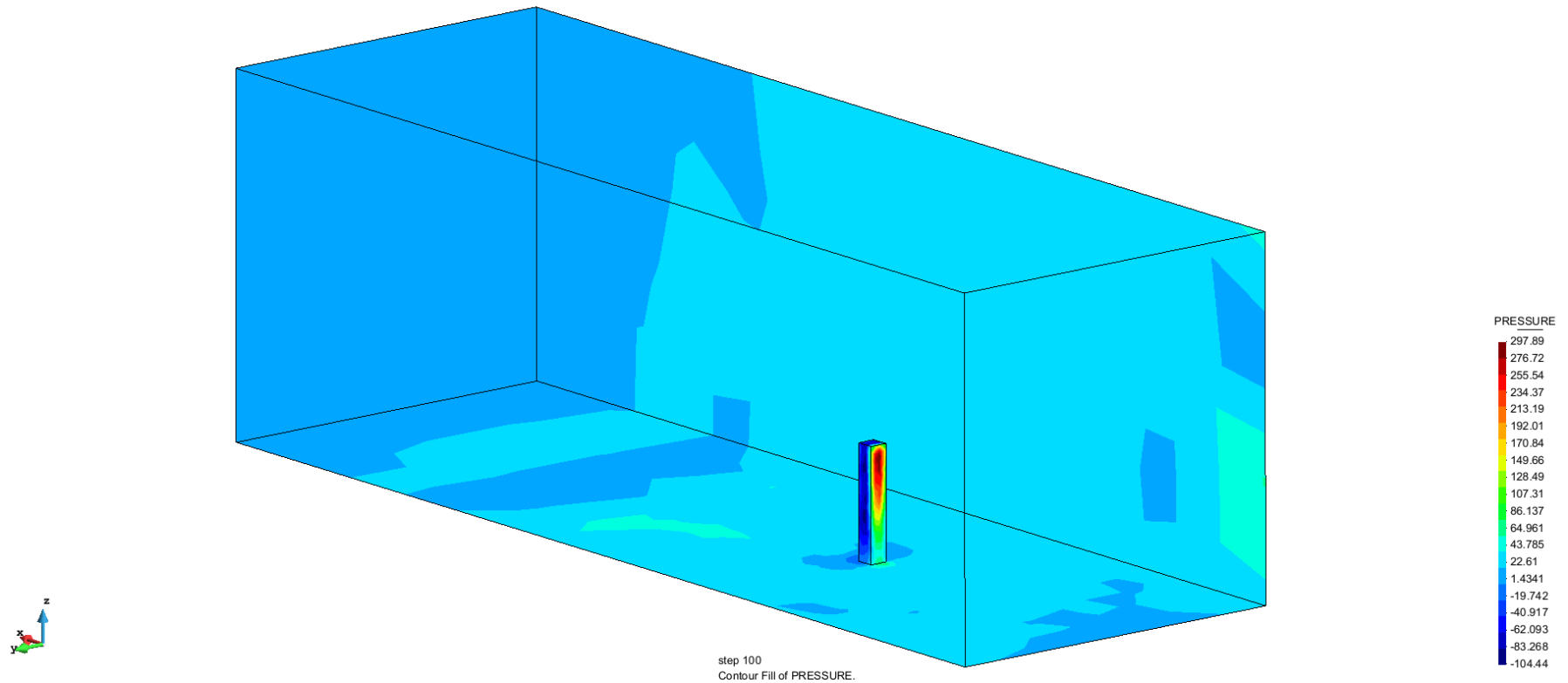


Postprocessing

- Play around with the results and the visualization
- Plot and animate the results for the velocity and the pressure and compare them
- To view the interior of the model
Window → View style
- In the **Select & Display Style** window, select **Culling: Front Faces**

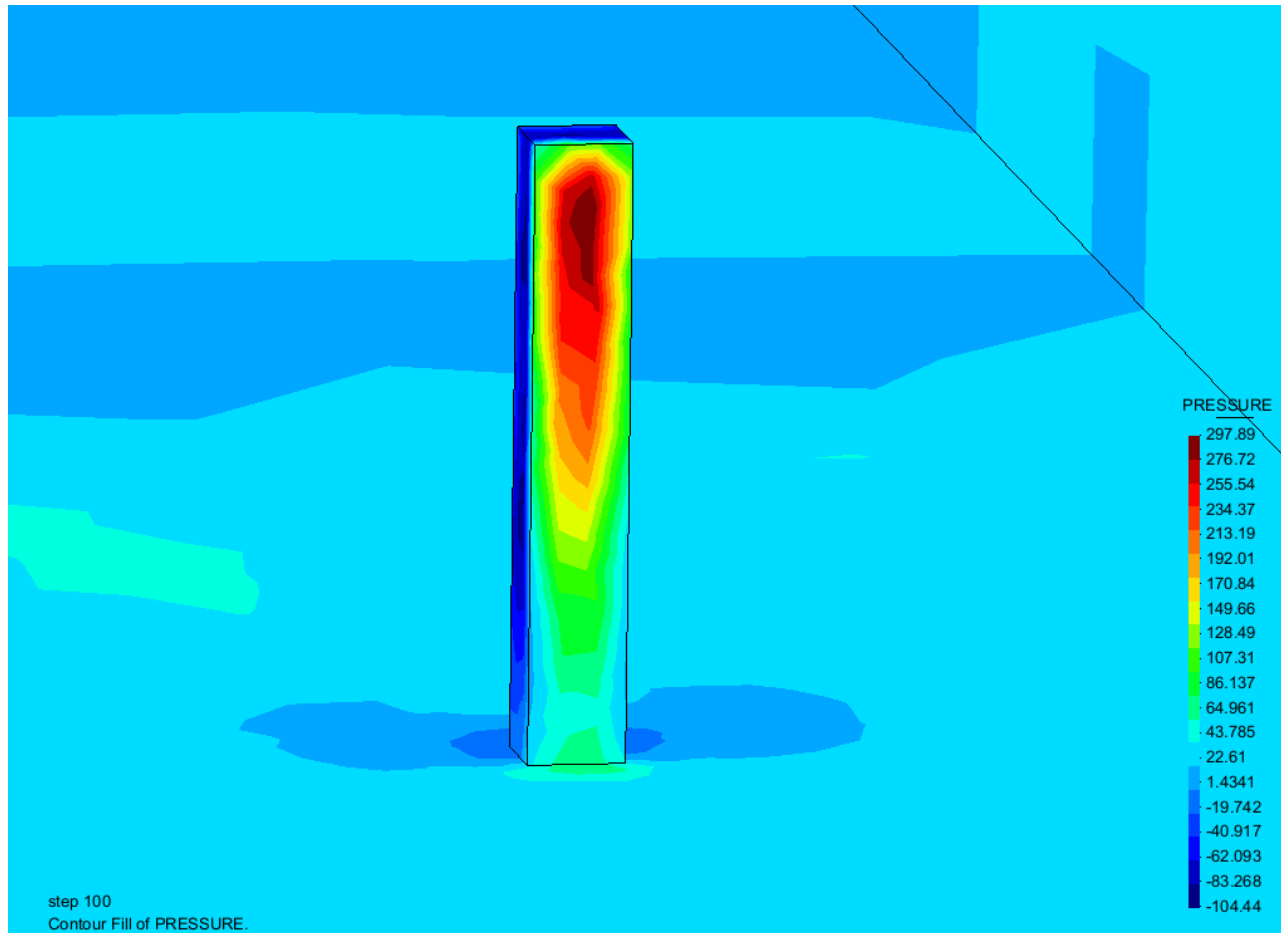


- Results for *pressure* in the last timestep:



Post processing

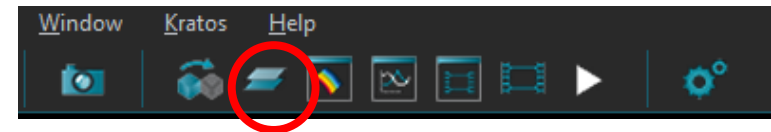
- Close-Up of the structure:



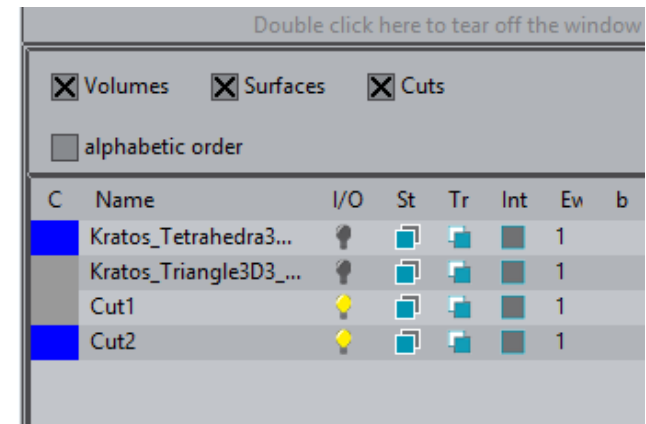
Using cuts

- **View** the results of an arbitrary plane using a cut
- Define a cut using
Geometry → Cut plane → 3 points
- **Use** these points (type this in command line one by one):

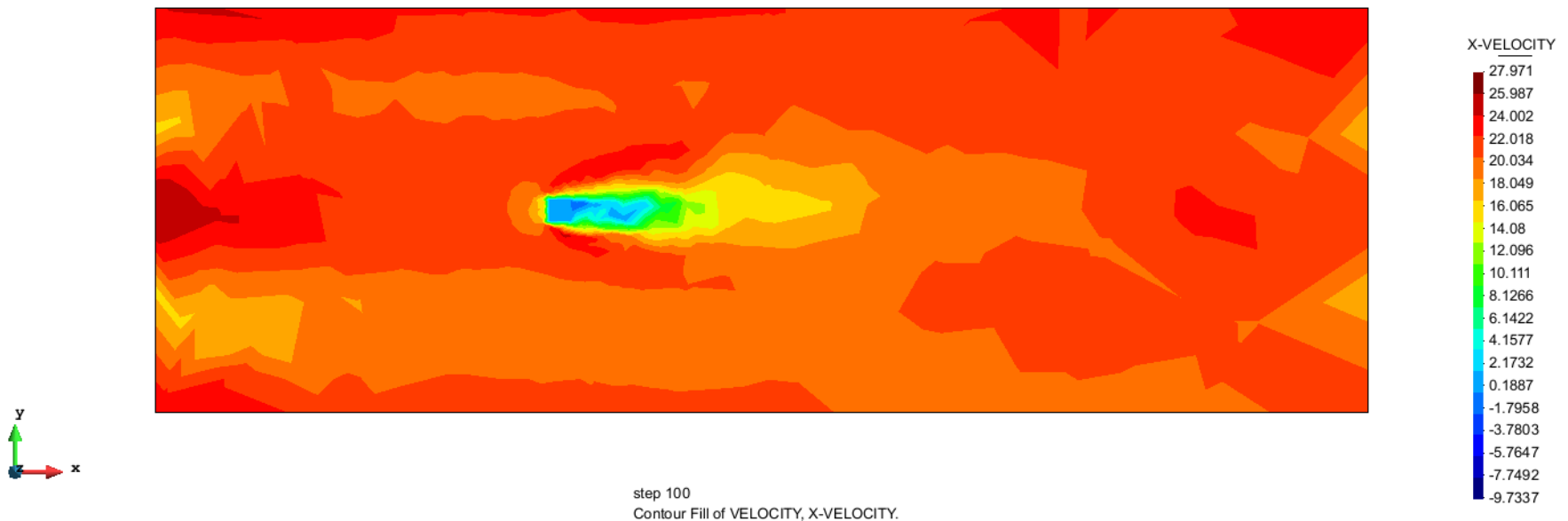
Cut plane		
X	Y	Z
0.0	0.0	190.0
1.0	0.0	190.0
1.0	1.0	190.0



- To **view** just the cut:
 - Click on **View Style** option
 - Turn off the light bulbs next to the Kratos entries
 - Set Culling: No, to show results on cut

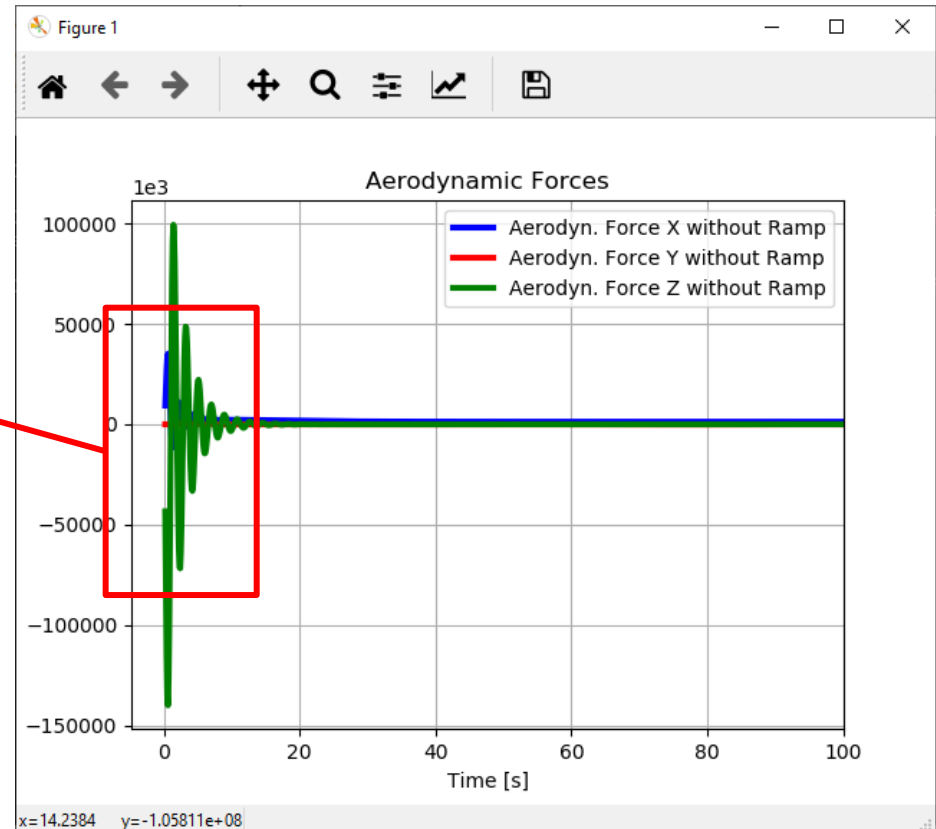
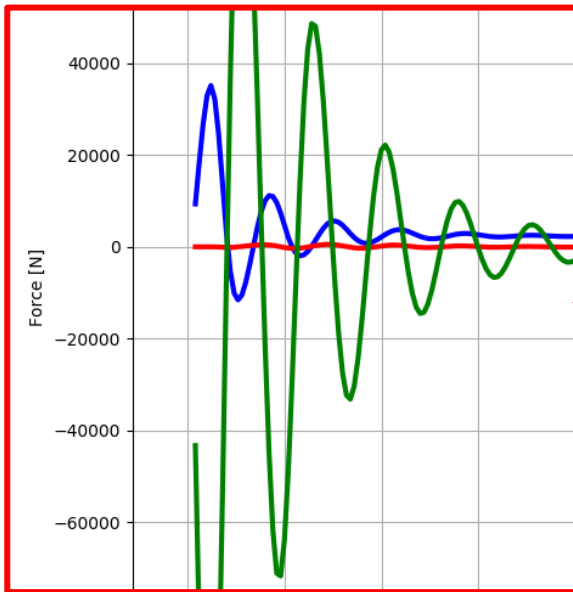


- Visualize the time evolution of the velocity using *Ctrl + m*
- The figure shows results for the *velocity in X direction* in the last timestep:

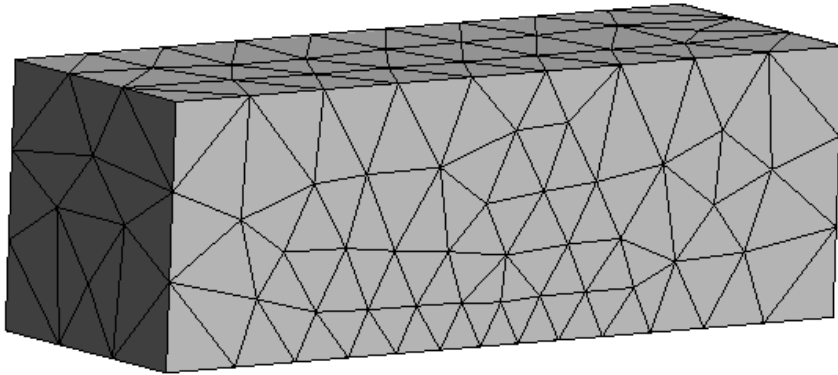


Aerodynamic results

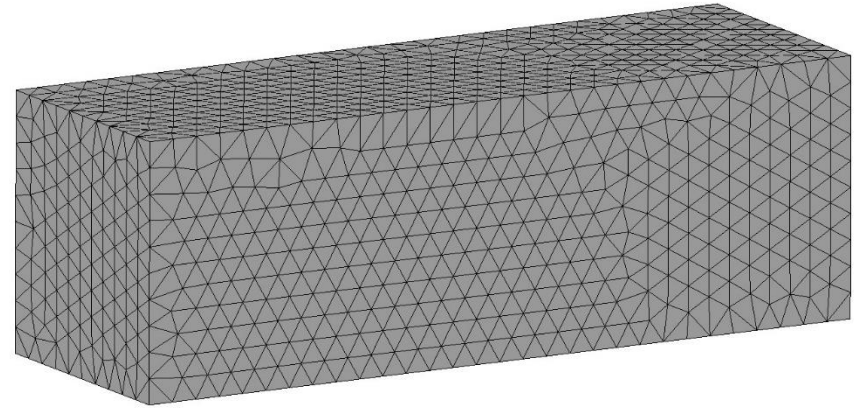
- Copy “[plot_aerodynamic_force_results.py](#)” into your GID project folder and run the python file
- The results should look like this:



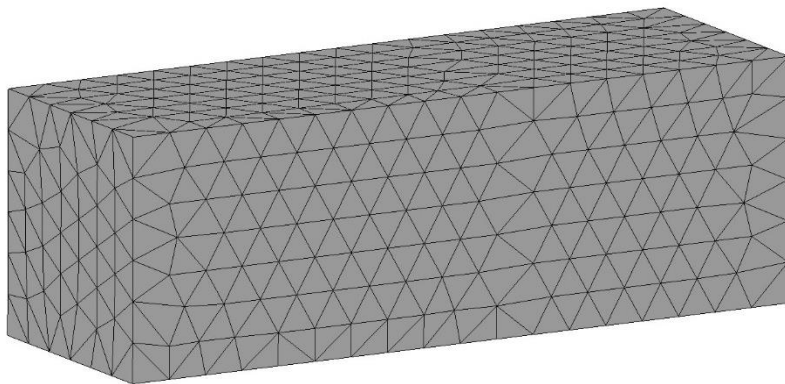
Very coarse



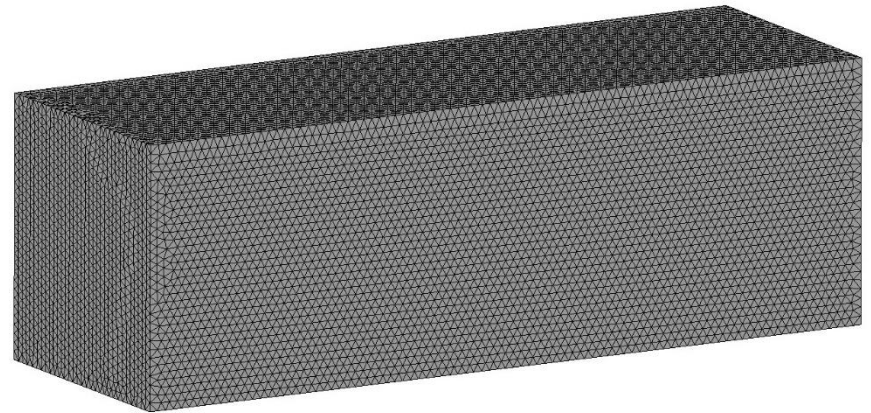
Medium



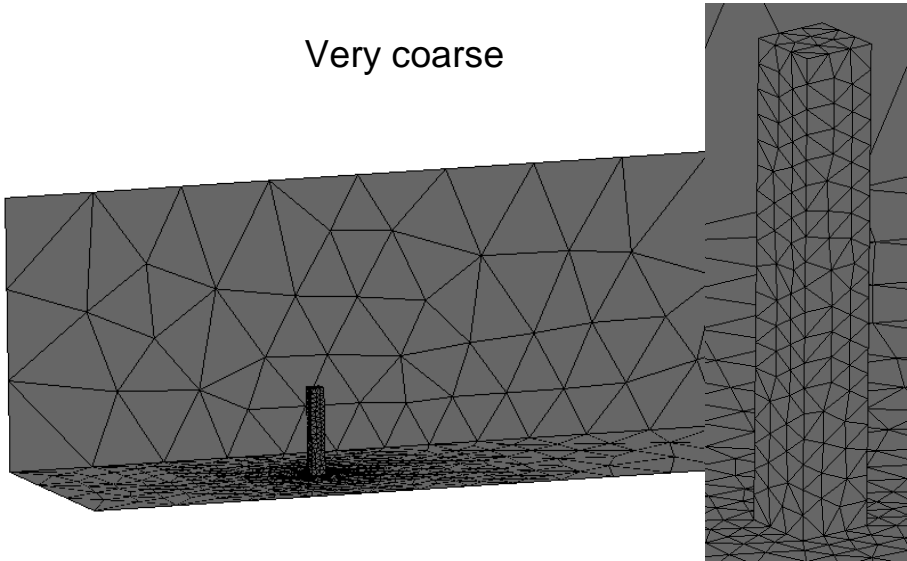
Coarse



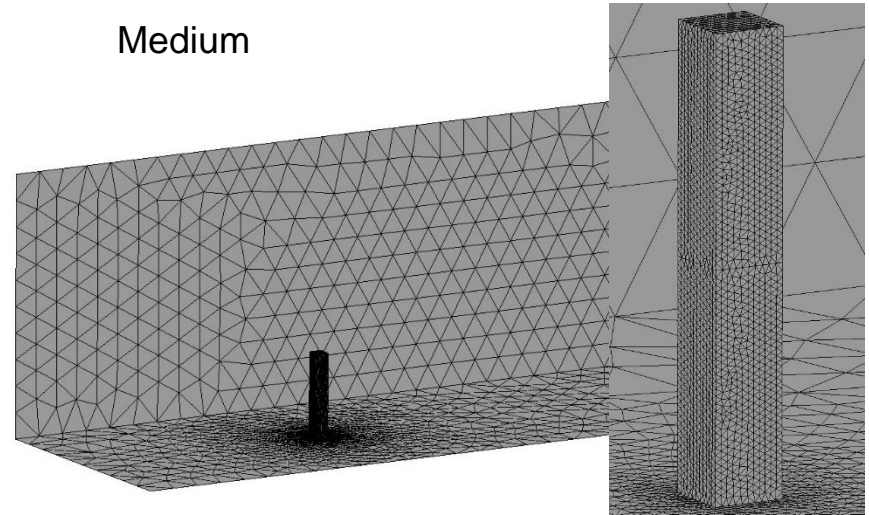
Fine



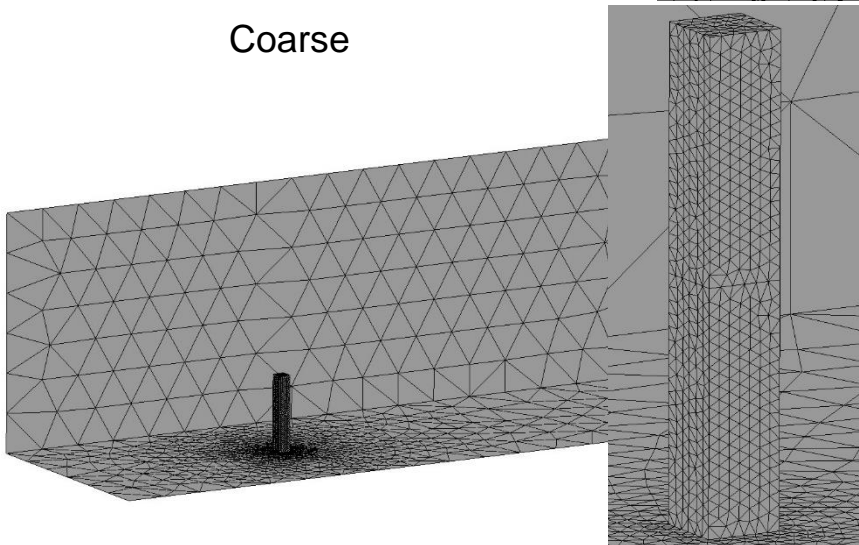
Very coarse



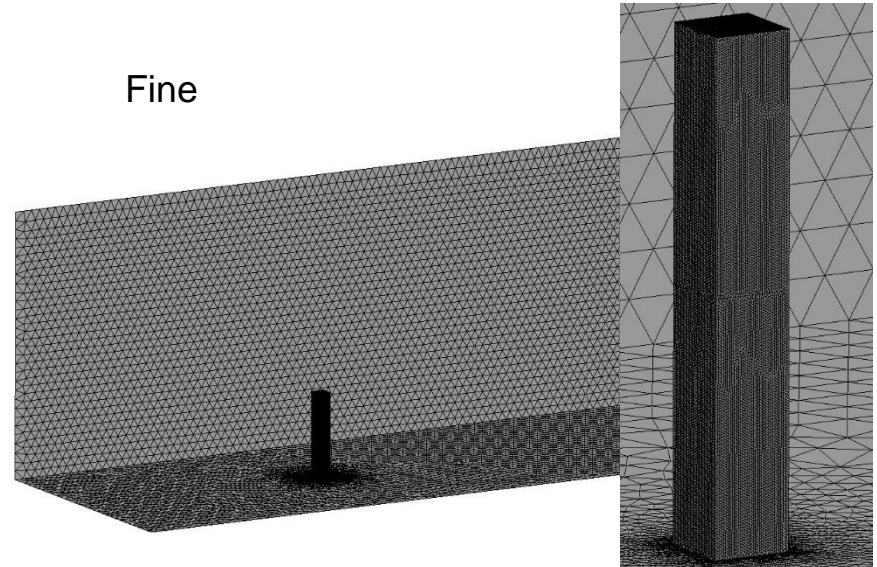
Medium



Coarse

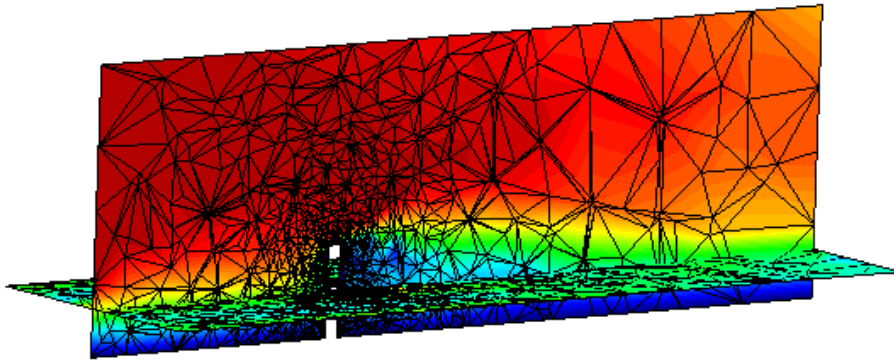


Fine

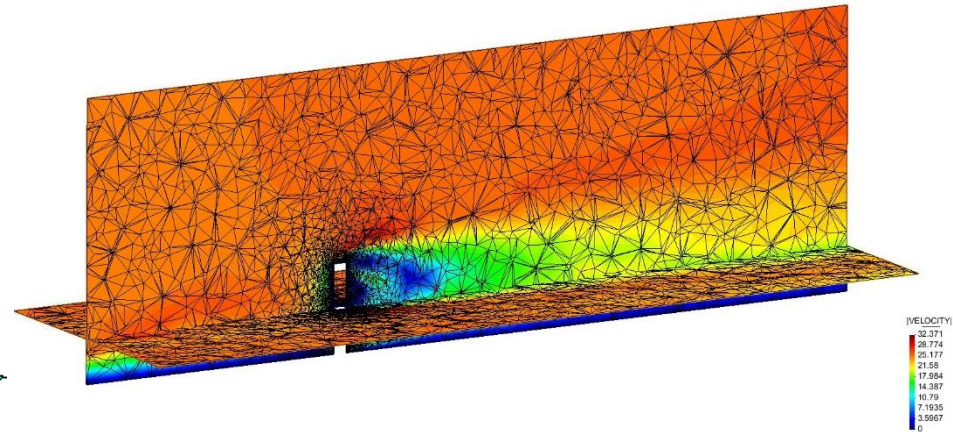


Result visualization – velocity magnitude

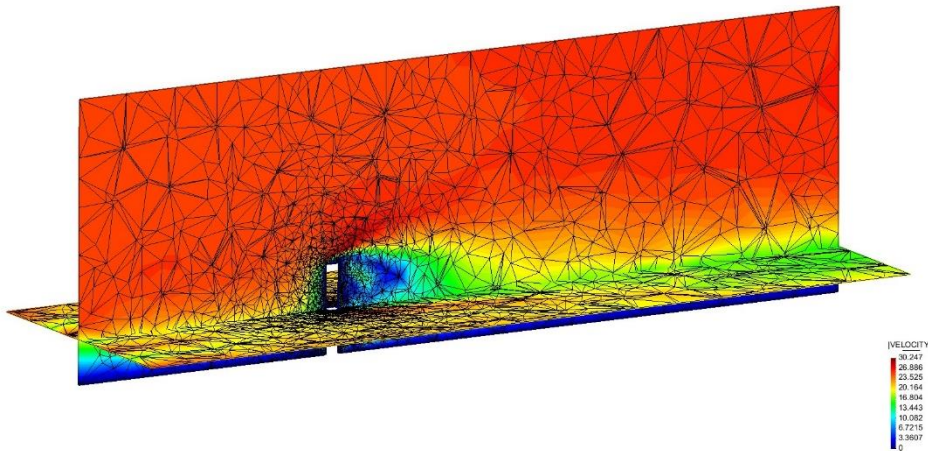
Very coarse



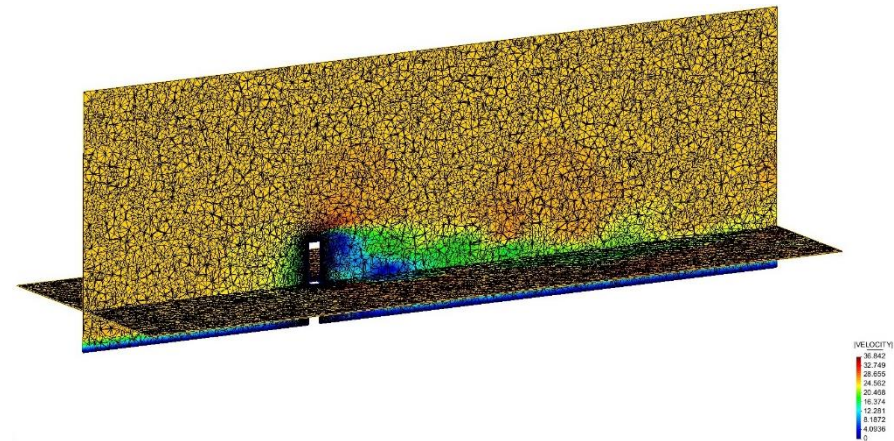
Medium



Coarse



Fine



Refinement plot

