

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!

In this tutorial we will solve a simple example using GiD and Kratos

Covered topics:

- Pre-processing
 - Geometry
 - Input data and conditions
- Post processing of results

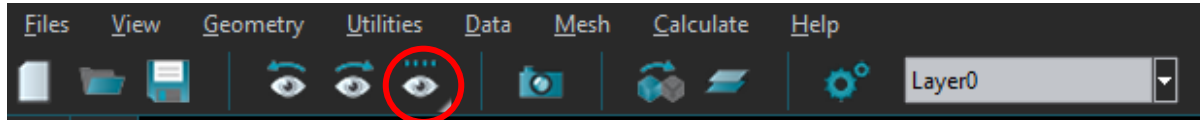
Disclaimer: This example serves the sole educational purpose of demonstrating how to setup a basic 2D CFD problem, run the simulation and do some postprocessing. Although the setup is chosen to realistically capture the drag coefficient and shedding frequency, for any real case in wind engineering a 3D setup should be adopted accompanied with detailed mesh and time step study.

Technical note: Tested on 11.12.2020, works with GiD 15.0.1 and the Kratos problemtype (8.1) on Windows 10 and Ubuntu 18/20 64 bit.

Defining the Geometry

Necessary steps

- Rotate the view to plane XY
View → Rotate → Plane XY



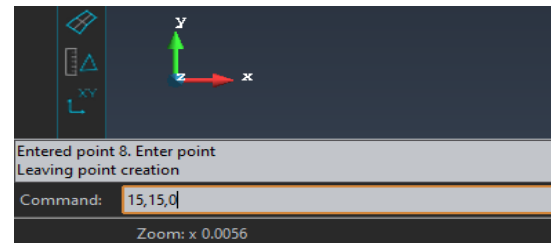
- Create domain and beam base
Geometry → Create → Point

Points to be created ()

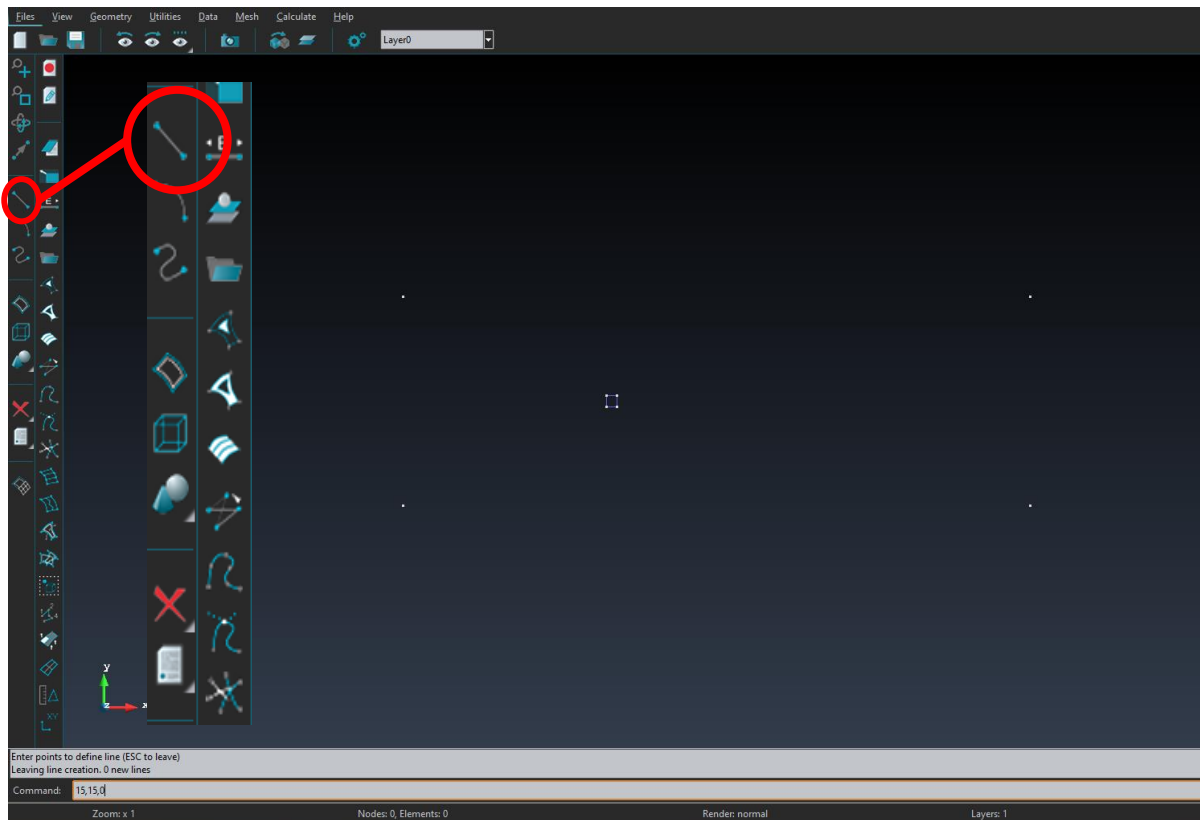
Structure X	Y	Z
15.0	15.0	0.0
15.0	-15.0	0.0
-15.0	15.0	0.0
-15.0	-15.0	0.0

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

- Enter the coordinates in the *Command* line

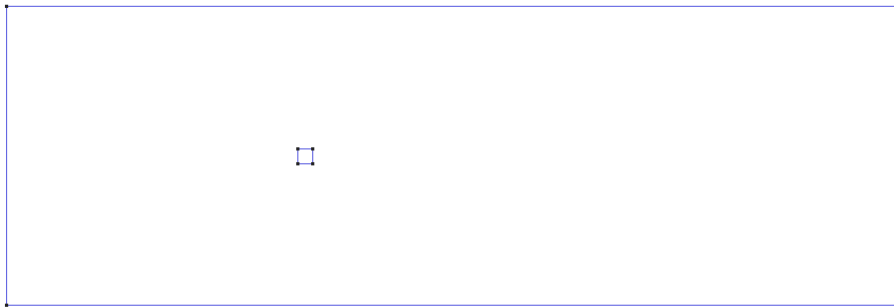


- Now we define the lines
Geometry → Create → Straight line
Using snap on points *Ctrl + a*
- Join the points of the structure

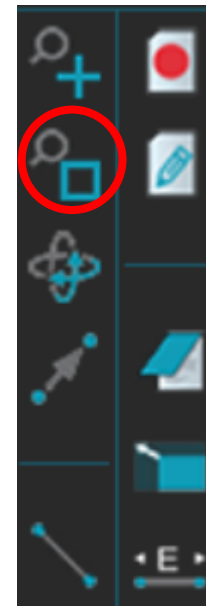


Geometry

- Once finished, press *Esc* or *Mouse wheel*
- Create additional lines to define the boundary

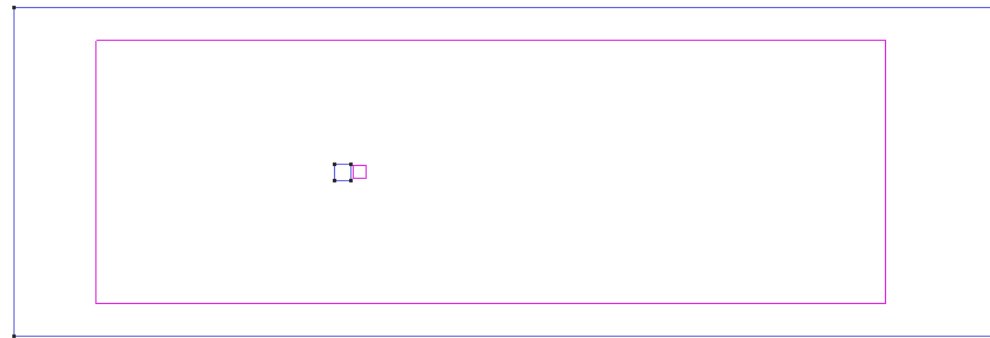


- Set your zoom frame so the whole geometry is shown







Geometry

- The final step is to define the surface
Geometry → Create → NURBS surface → By contour
- Select all lines and press *Esc* or *Mouse wheel*



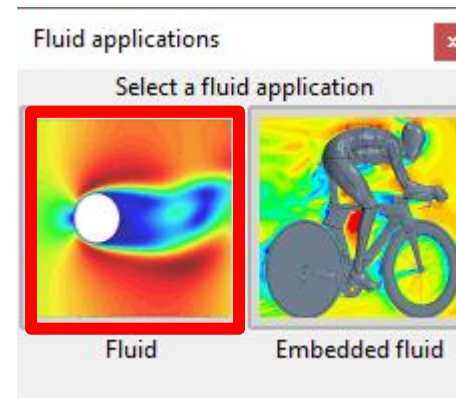
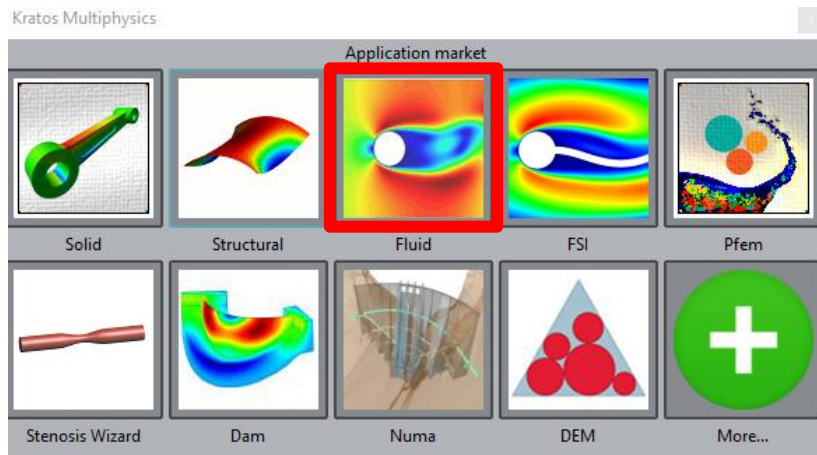
- Save the project at the current state
- You have the following files generated in the folder *Tutorial1_2D_CFD.gid*

	Tutorial1_2D_CFD.geo	04.12.2019 09:44	GEO-Datei
	Tutorial1_2D_CFD	04.12.2019 09:44	PNG-Datei
	Tutorial1_2D_CFD.tree	04.12.2019 09:44	TREE-Datei
	Tutorial1_2D_CFD.vv	04.12.2019 09:44	VV-Datei

Problem Input

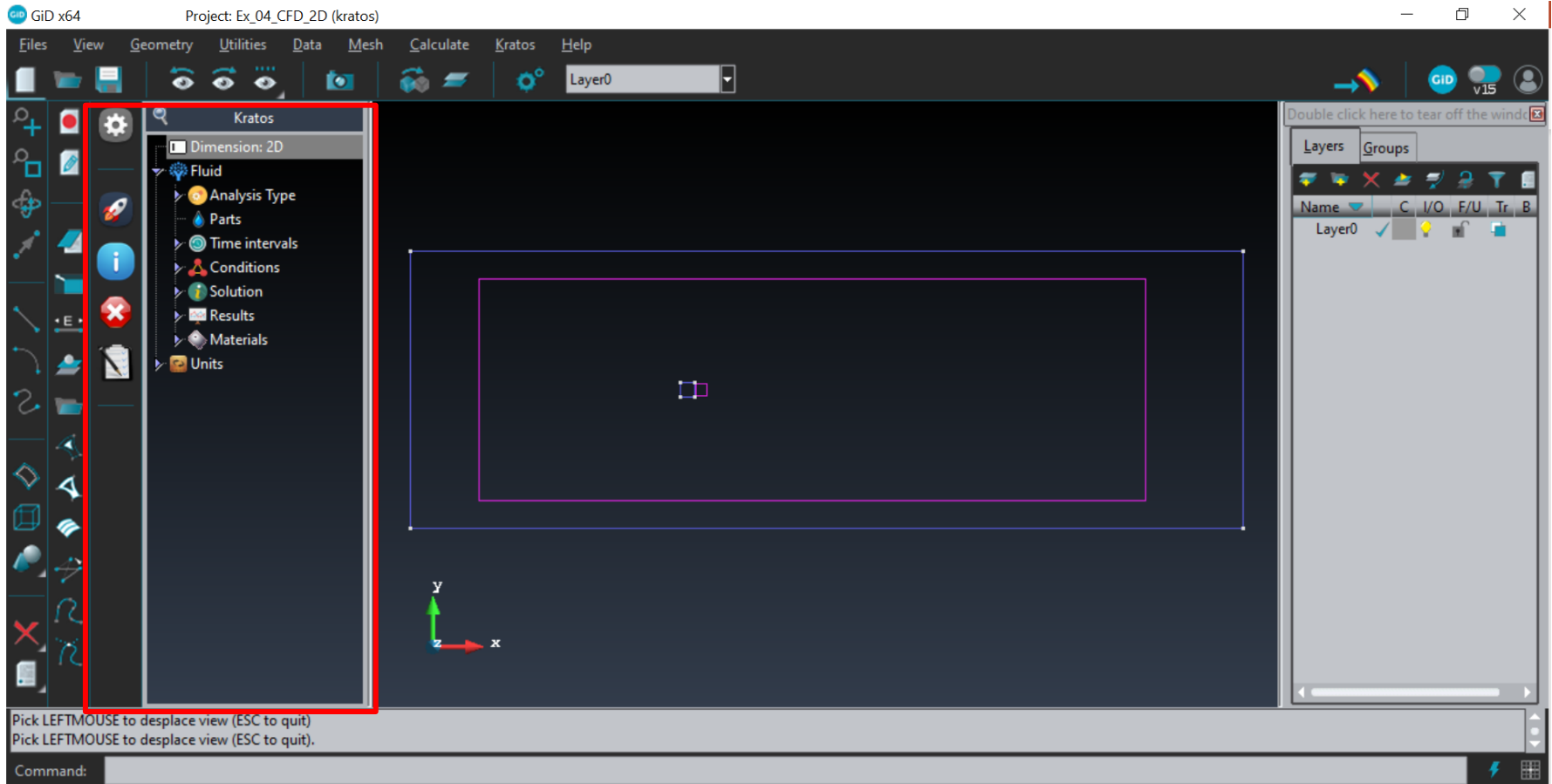
Input data

- Load the Kratos problem type
Data → Problem type → Kratos
- Select *Fluid* in the first window (Application Type) and *Fluid* in the second window
- Select *2D* in the next window (Analysis Type)



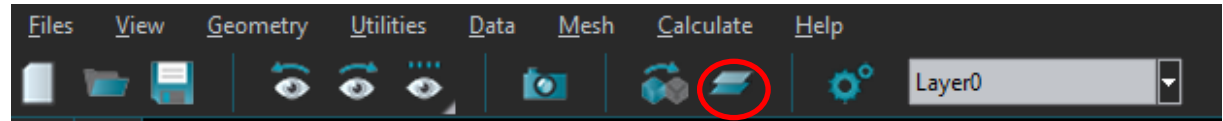
Input data

- A new column will appear in the left side.

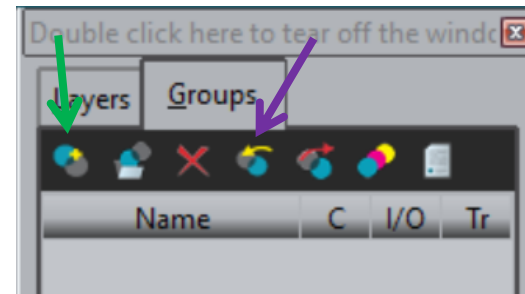


Define the entity groups

- Click the *Groups* or *Layers* button to open the group window
- Each group will define a part of the model
 - Fluid body
 - Inlet line
 - Outlet line
 - Solid boundary

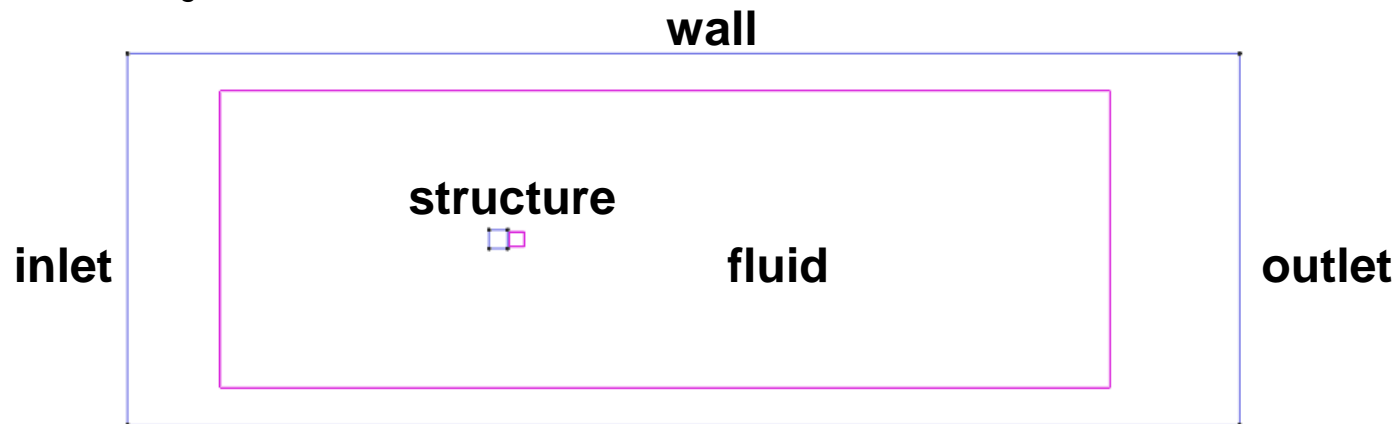
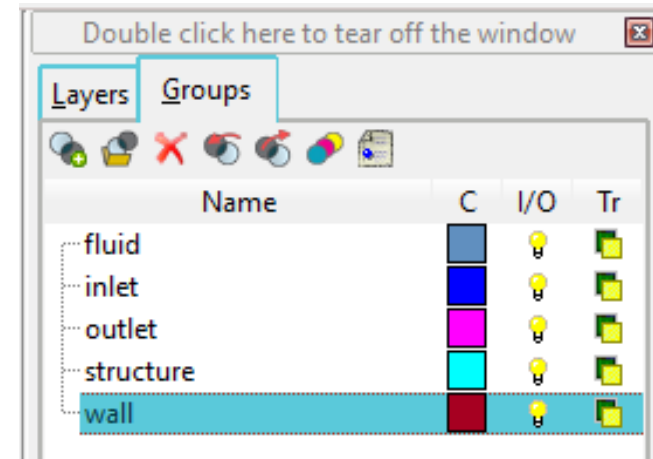


- Create a new group
 - Click the *New* button
- Assign entities to a group
 - Click the *Assign* button
 - Choose an entity type
 - Select the respective entities by clicking them
 - Klick *ESC* or *mouse wheel* to finish



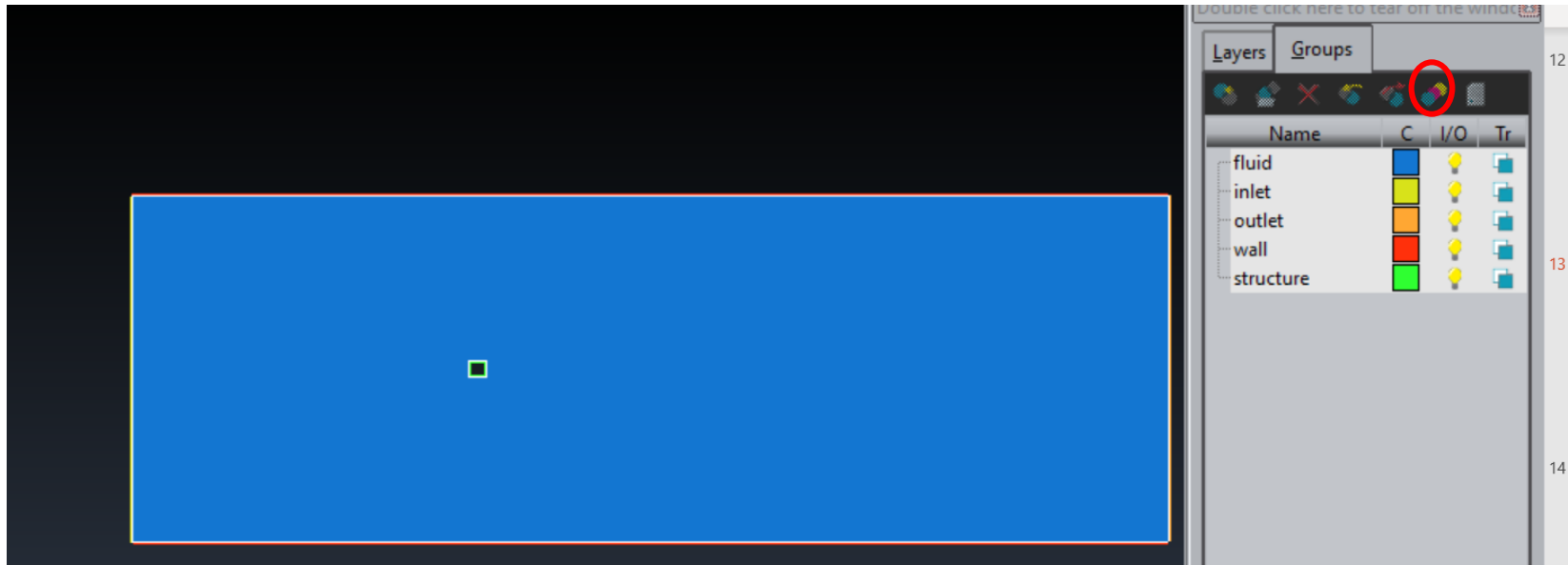
Define the entity groups

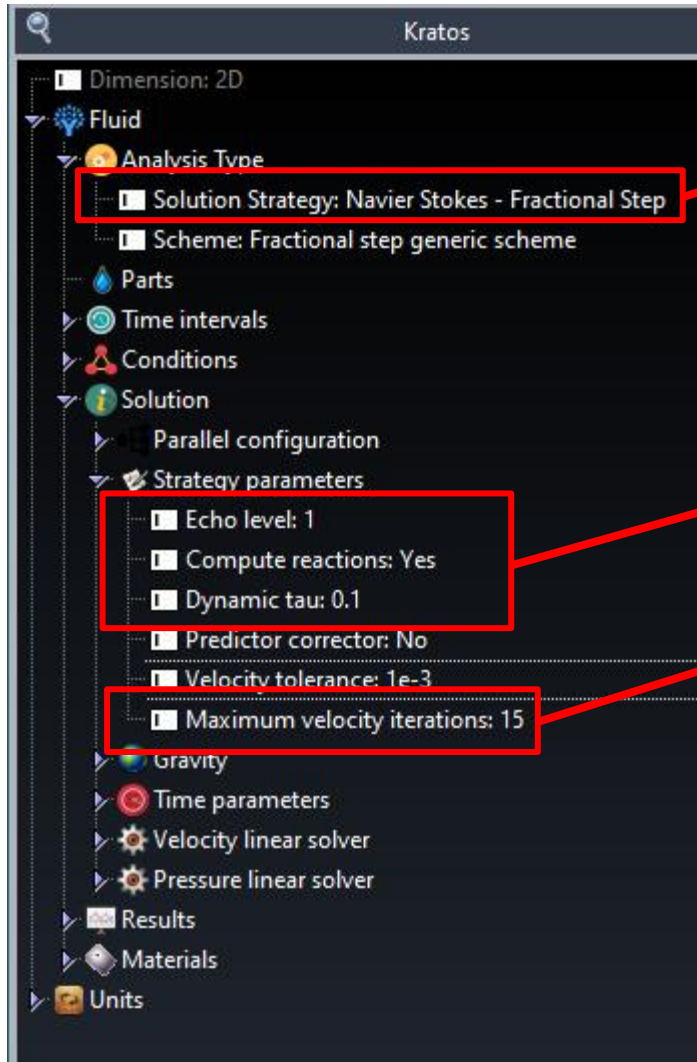
- *fluid* group
 - Select surface
- *inlet* group
 - Select left line
- *outlet* group
 - Select right line
- *structure* group
 - Select lines of the structure
- *wall* group
 - Select all remaining lines



Define the entity groups

- Group entity selection can be visualized → check if you assigned correctly:
 - Select the entities you want to visualize and click the *Draw groups by color* button



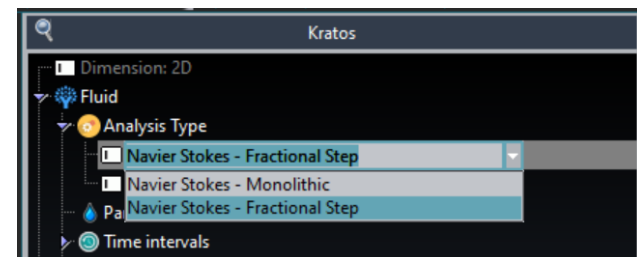


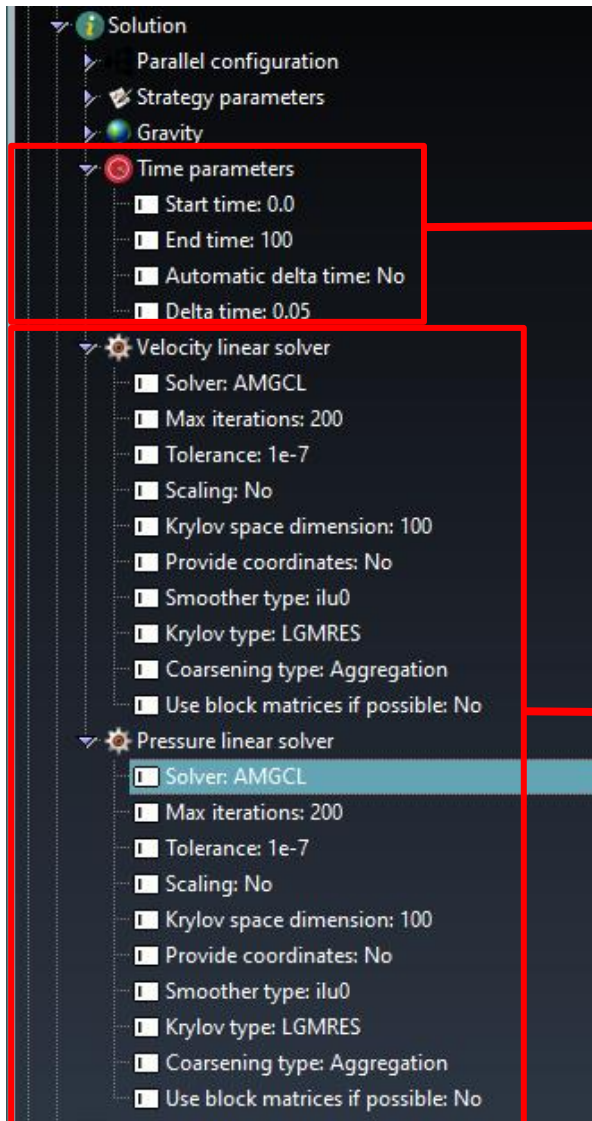
Set the *Solutions Strategy* to *Fractional Step*

Set the *Echo level* to *1*
Dynamic tau to *0.1*
Compute reactions to *Yes*

Set the maximum number of *velocity iterations* to *15*

Note: double click to change a property





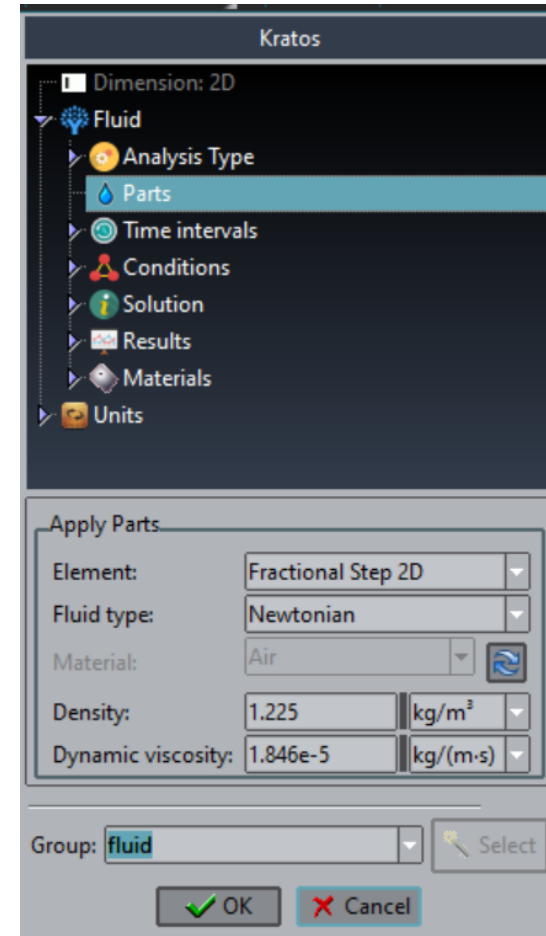
Set the time parameters:

- Start time: 0.0
- End time: 100
- Time step: 0.05

Select an *iterative* solver for velocity and pressure
-> e.g. AMGCL

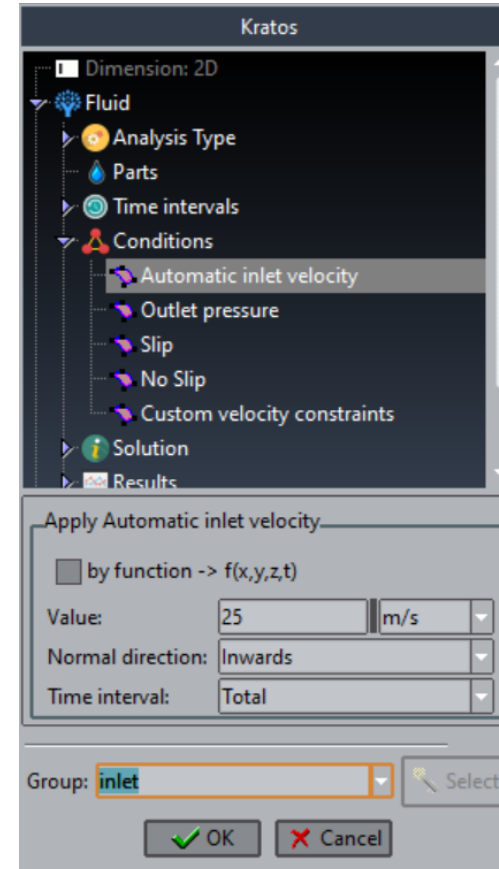
Fluid properties

- Create a new property for your fluid by selecting *Parts*
 - Define the *fluid type* (e.g. Newtonian)
 - Specify the material properties or click „Refresh Button“ for
 - Choose *Fluid* for *Group*.
 - Click *Ok*



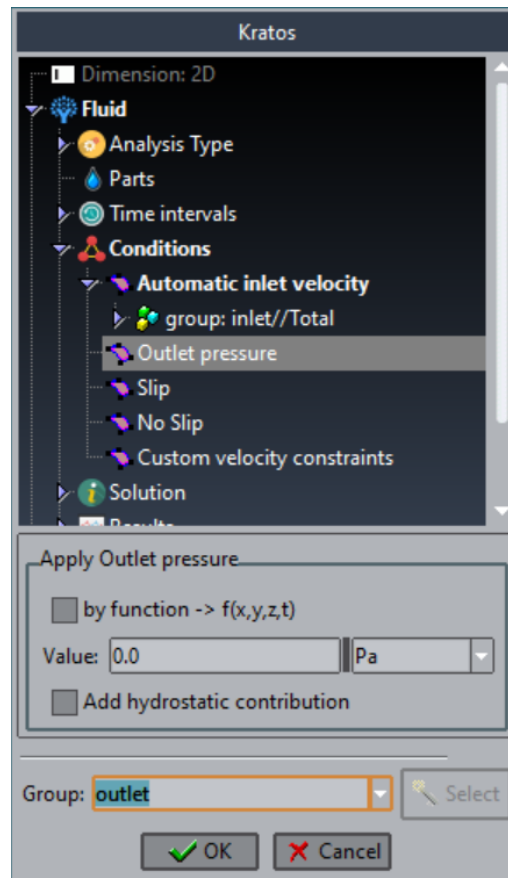
Boundary conditions I

- Assign the boundary conditions using the entity groups
- First, assign the inlet condition
 - Choose the *Inlet velocity* condition
 - Assign it to the *inlet* line group
- Set the *value* to 25.0 and *Normal direction* to Inwards

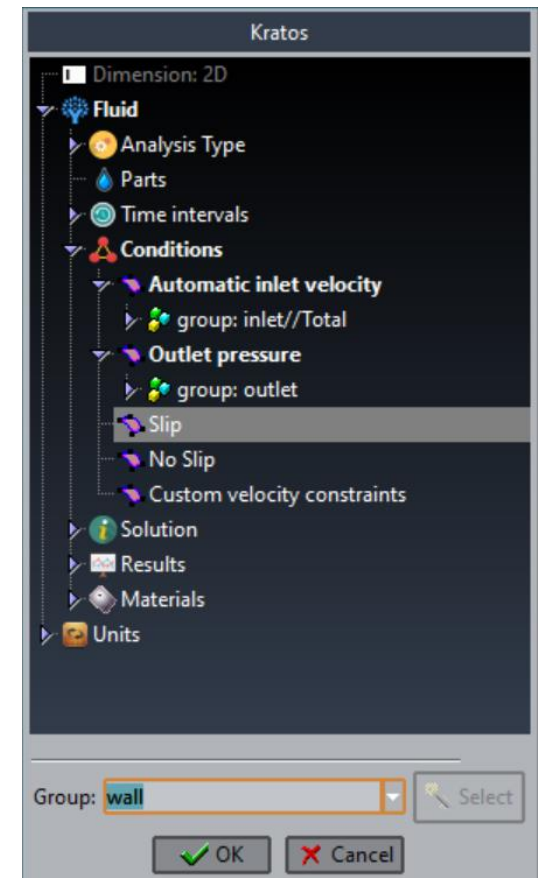


Boundary conditions II & III

- Assign the *outlet pressure* condition
 - Set the pressure of the *outlet* entity group to 0.0
 - Click *OK*

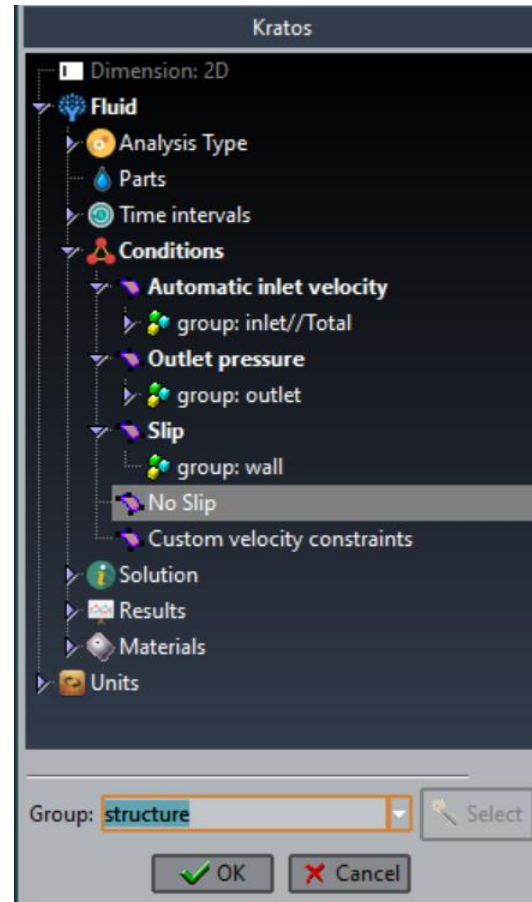


- Assign the *slip* condition
 - Choose the *wall* entity group
 - Click *OK*



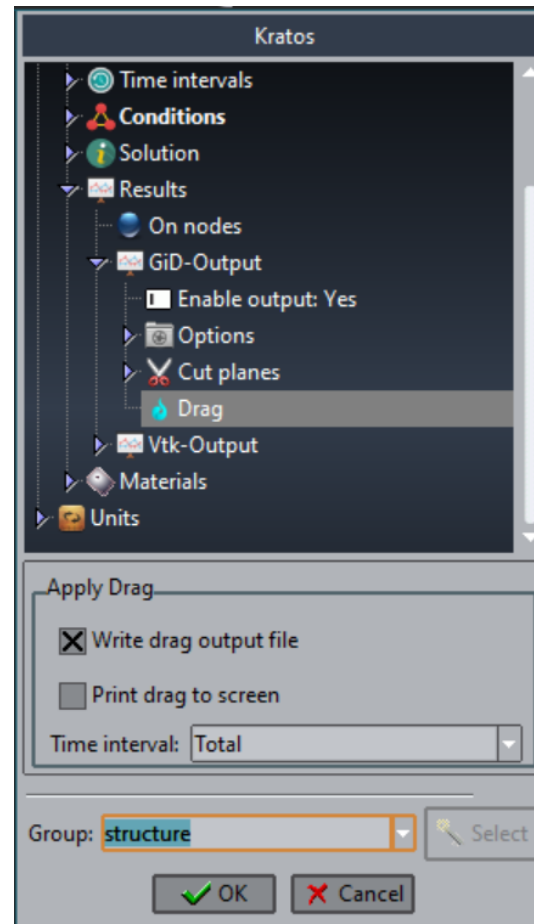
Boundary conditions IV

- Finally, assign a *No-Slip* condition to the structure boundary
 - Select the *structure* entity group
 - Click *OK*



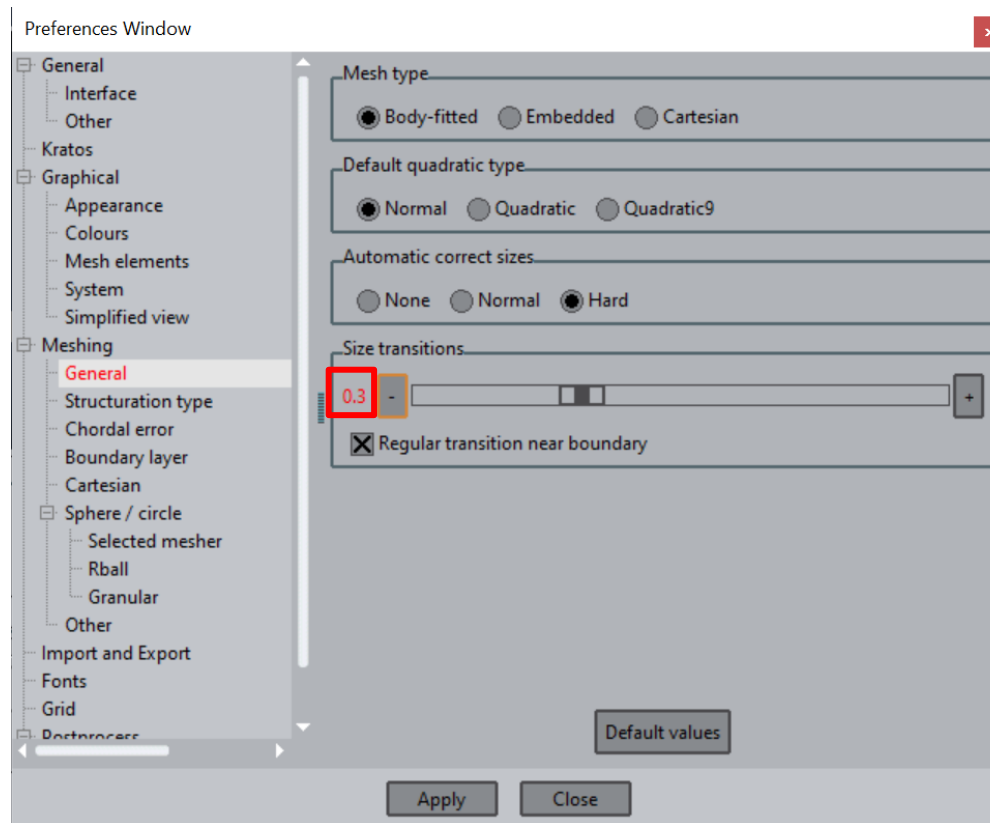
Compute drag force

- In order to compute the aerodynamic load on the body:
 - Select under **Results/GiD-Output/Drag** the *structure* entity group
 - After calculation a text file with drag forces will be in the project folder

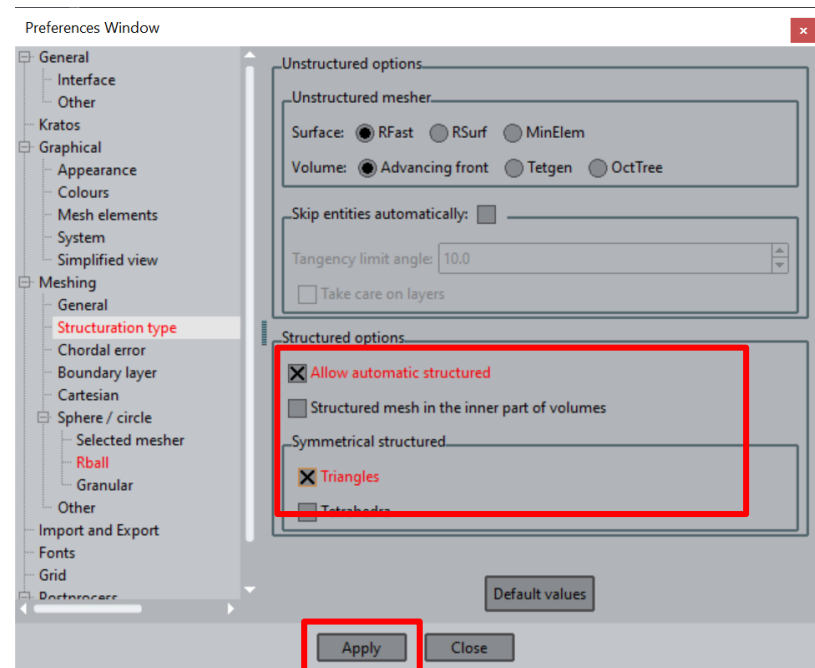
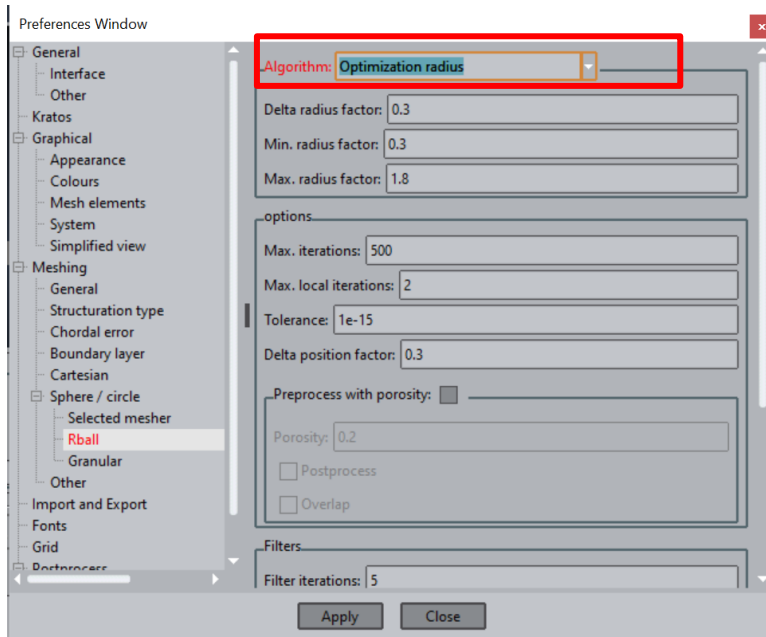


Mesh settings

- For changing the mesh settings go to
Utilities → *Preferences*
- Under the Meshing tab change the settings according to the pictures shown below

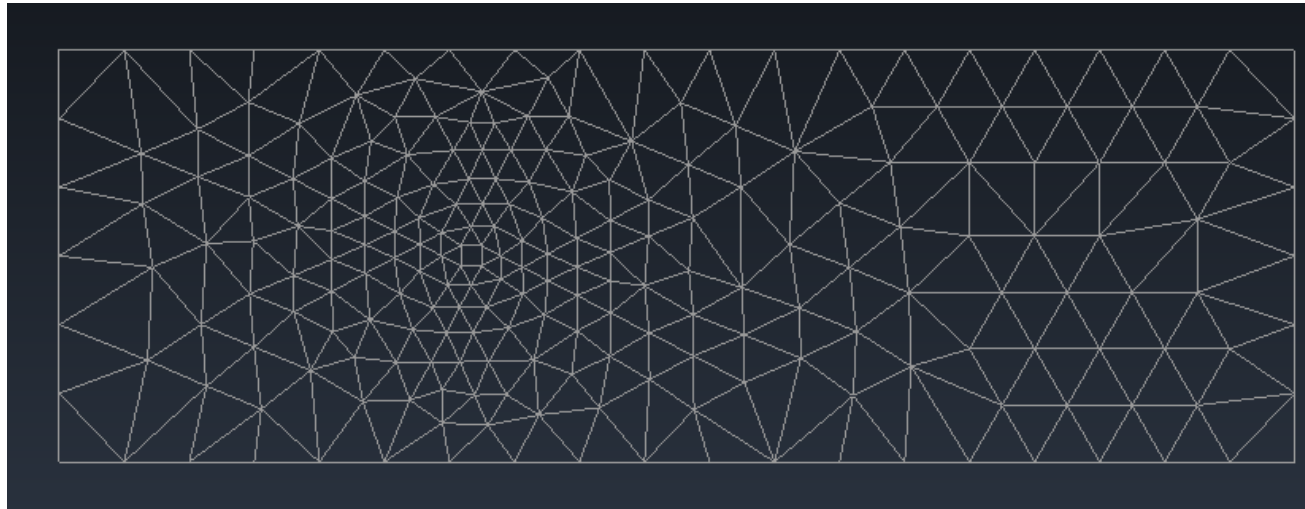
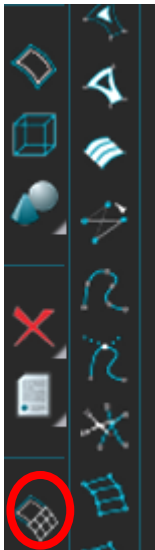
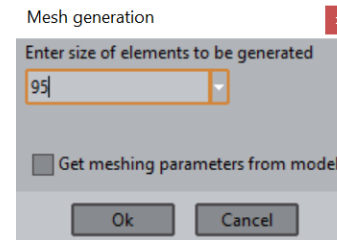


Mesh settings



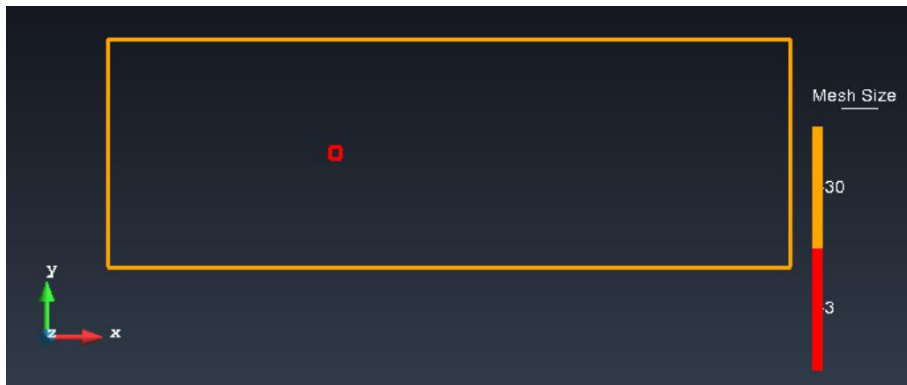
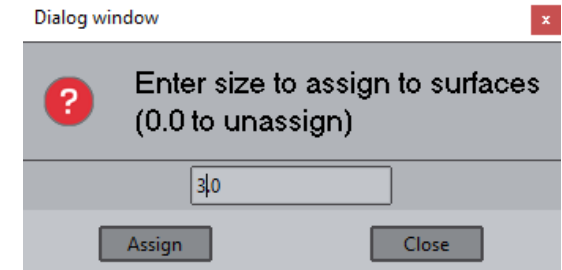
→ Click *Apply*

- Mesh by selecting the menu option
Mesh → *Generate mesh*
or *Ctrl + g*
- In the box that appears, set size to 95 and click *Ok*
- You can view the mesh using the *View mesh* button (Toggle mesh-geometry view)



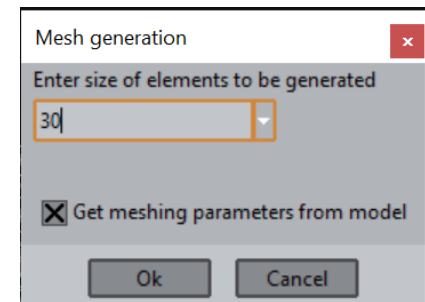
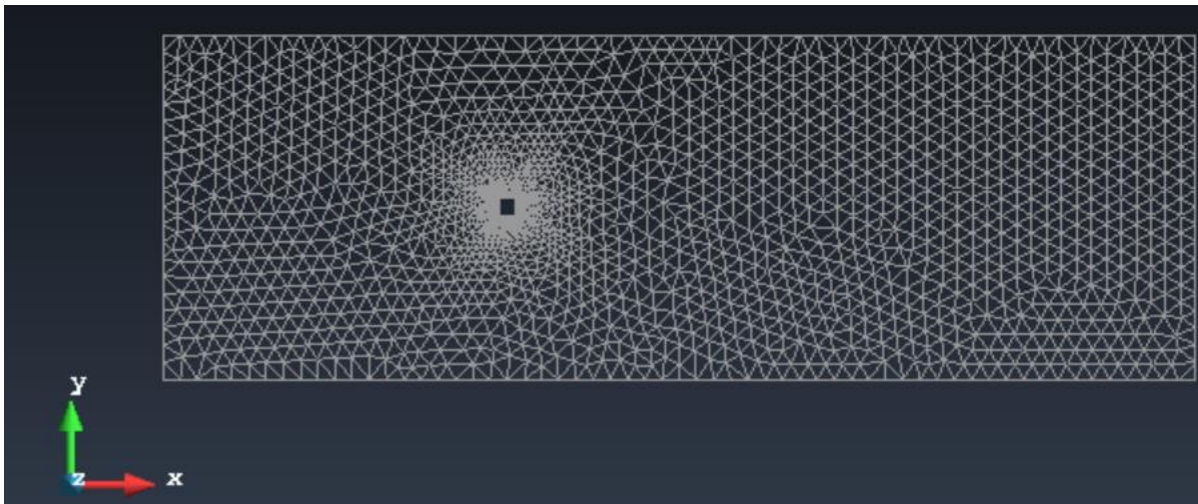
Refine the mesh

- As the mesh is quite coarse, we are going to refine it
Mesh → Unstructured → Assign sizes on lines
- In the box that appears, set size to 3 and click the *Assign* button.
Then select all lines of the structure and press *Esc*
- Repeat it for the Boundary Box with size 30
- The assigned sizes can be viewed by
Mesh → Draw → Sizes → Lines



Refine the mesh





- Now mesh the domain again by pressing **Ctrl + g** (*old mesh will be erased*)
- In the box that appears, set element size to **30**, tick the **Get meshing parameters from model** box and click **Ok**.
- The generated mesh should have ~ **1950 nodes** and ~ **3900 elements**












Save the project

- Save the project at the current state
- You have the following files generated in the folder *Tutorial1_2D_CFD.gid*

Folder structure – after the definition of the geometry

 Tutorial1_2D_CFD.geo	04.12.2019 09:44	GEO-Datei
 Tutorial1_2D_CFD	04.12.2019 09:44	PNG-Datei
 Tutorial1_2D_CFD.tree	04.12.2019 09:44	TREE-Datei
 Tutorial1_2D_CFD.vv	04.12.2019 09:44	VV-Datei




















Folder structure – after the solver, BCs, meshing

 Tutorial1_2D_CFD.cnd	04.12.2019 10:03	CND-Datei
 Tutorial1_2D_CFD.geo	04.12.2019 10:03	GEO-Datei
 Tutorial1_2D_CFD	04.12.2019 10:03	SOFiPLUS-X-Linientypdefinition
 Tutorial1_2D_CFD	04.12.2019 10:03	MSH-Datei
 Tutorial1_2D_CFD	04.12.2019 10:03	PNG-Datei
 Tutorial1_2D_CFD.prj	04.12.2019 10:03	PRJ-Datei
 Tutorial1_2D_CFD.spd	04.12.2019 10:03	SPD-Datei
 Tutorial1_2D_CFD.tree	04.12.2019 10:03	TREE-Datei
 Tutorial1_2D_CFD.vv	04.12.2019 10:03	VV-Datei

Solve the problem

- Save your model
or *Files → Save*
or *Ctrl + s*
- Launch Kratos with
Calculate → Calculate
or *F5*
- The input data will be checked for errors
- The calculation should not take more than 5 minutes

Folder structure – after clicking *Solve*

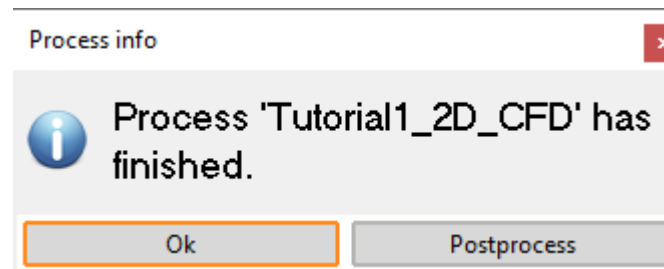
 vtk_output	04.12.2019 10:10	Dateiordner
 FluidMaterials	04.12.2019 10:04	JSON-Quelldatei
 FluidModelPart.Drag_structure_drag	04.12.2019 10:10	SOFiStiK Input File
 MainKratos	27.03.2019 13:17	Python File
 ProjectParameters	04.12.2019 10:04	JSON-Quelldatei
 Tutorial1_2D_CFD.cnd	04.12.2019 10:03	CND-Datei
 Tutorial1_2D_CFD	04.12.2019 10:04	Fehlerprotokoll
 Tutorial1_2D_CFD.geo	04.12.2019 10:03	GEO-Datei
 Tutorial1_2D_CFD.info	04.12.2019 10:10	INFO-Datei
 Tutorial1_2D_CFD	04.12.2019 10:03	SOFiPLUS-X-Linientypdefinition
 Tutorial1_2D_CFD.mdpa	04.12.2019 10:04	MDPA-Datei
 Tutorial1_2D_CFD	04.12.2019 10:03	MSH-Datei
 Tutorial1_2D_CFD	04.12.2019 10:03	PNG-Datei
 Tutorial1_2D_CFD.post.bin	04.12.2019 10:10	BIN-Datei
 Tutorial1_2D_CFD.post	04.12.2019 10:04	SOFiStiK List File
 Tutorial1_2D_CFD.prj	04.12.2019 10:03	PRJ-Datei
 Tutorial1_2D_CFD.spd	04.12.2019 10:03	SPD-Datei
 Tutorial1_2D_CFD.tree	04.12.2019 10:03	TREE-Datei
 Tutorial1_2D_CFD.vv	04.12.2019 10:03	VV-Datei

Solve the problem

- You can check if the solution is running properly using
Calculate → View process info

```
output info for 'current' Thu Dec 10 17:23:25
FractionalStepStrategy: Calculating Pressure.
FractionalStepStrategy: Updating Velocity.
Fluid Dynamics Analysis: STEP: 1639
Fluid Dynamics Analysis: TIME: 81.94999999999749
FractionalStepStrategy: CONVERGENCE CHECK:
FractionalStepStrategy: FRAC VEL.: ratio = 0.00667168; exp.ratio = 0.001 abs = 7.32271
FractionalStepStrategy: CONVERGENCE CHECK:
FractionalStepStrategy: FRAC VEL.: ratio = 0.000154482; exp.ratio = 0.001 abs = 0.169556
FractionalStepStrategy: Fractional velocity converged in 2 iterations.
FractionalStepStrategy: Calculating Pressure.
FractionalStepStrategy: Updating Velocity.
Fluid Dynamics Analysis: STEP: 1640
Fluid Dynamics Analysis: TIME: 81.99999999999748
FractionalStepStrategy: CONVERGENCE CHECK:
FractionalStepStrategy: FRAC VEL.: ratio = 0.00660848; exp.ratio = 0.001 abs = 7.25352
FractionalStepStrategy: CONVERGENCE CHECK:
FractionalStepStrategy: FRAC VEL.: ratio = 0.00015446; exp.ratio = 0.001 abs = 0.169536
FractionalStepStrategy: Fractional velocity converged in 2 iterations.
FractionalStepStrategy: Calculating Pressure.
FractionalStepStrategy: Updating Velocity.
Fluid Dynamics Analysis: STEP: 1641
Fluid Dynamics Analysis: TIME: 82.04999999999748
FractionalStepStrategy: CONVERGENCE CHECK:
FractionalStepStrategy: FRAC VEL.: ratio = 0.00654623; exp.ratio = 0.001 abs = 7.18537
FractionalStepStrategy: CONVERGENCE CHECK:
```

- A window will pop up once the solution process finishes

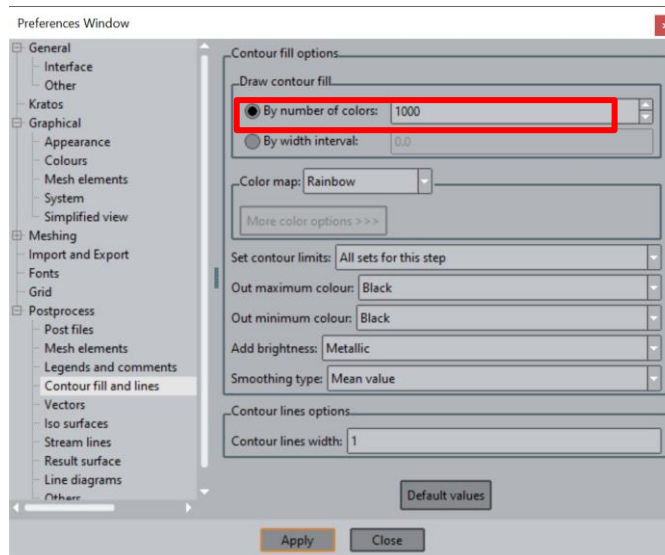


Solution Postprocessing

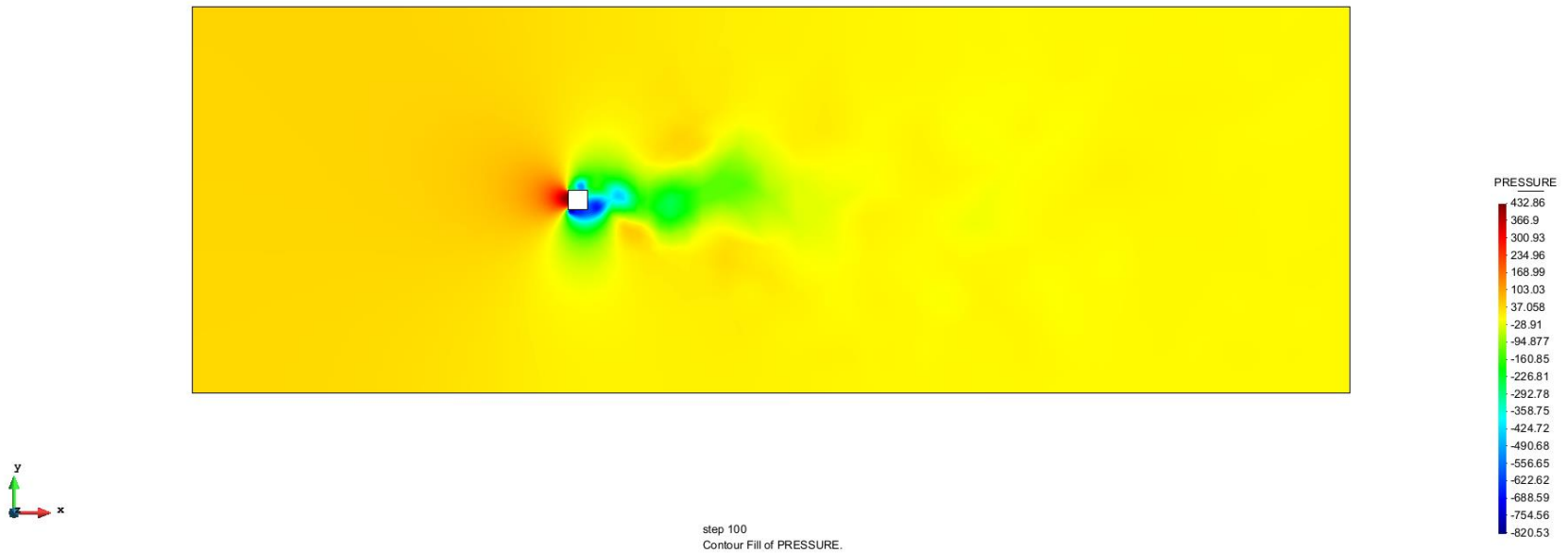
Post processing

- After switching to postprocess, GiD will load the results of the model by default
- Other results can be opened with
Files → Open...
 or *Ctrl + o*
- Common result formats are
.post.bin binary results
.post.res ASCII results
.post.lst A list file pointing to multiple result files
 +
.vtk (ASCII or binary) or even *.h5*

- View the pressure results with:
Files → View results → Contour fill → PRESSURE
- Different time steps can be viewed with:
Files → View results → default analysis/step → Kratos → „select step“
or *Ctrl + d*
- Change number of colors for Contour fill:
Utilities → Preferences → Postprocess → Contour fill and lines

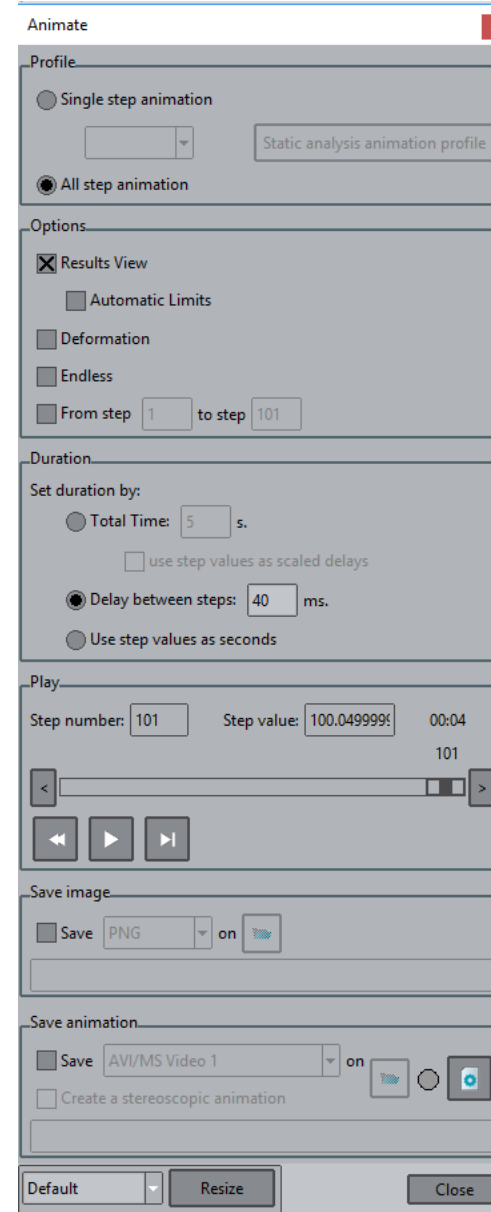


- Results for *pressure* in the last timestep:

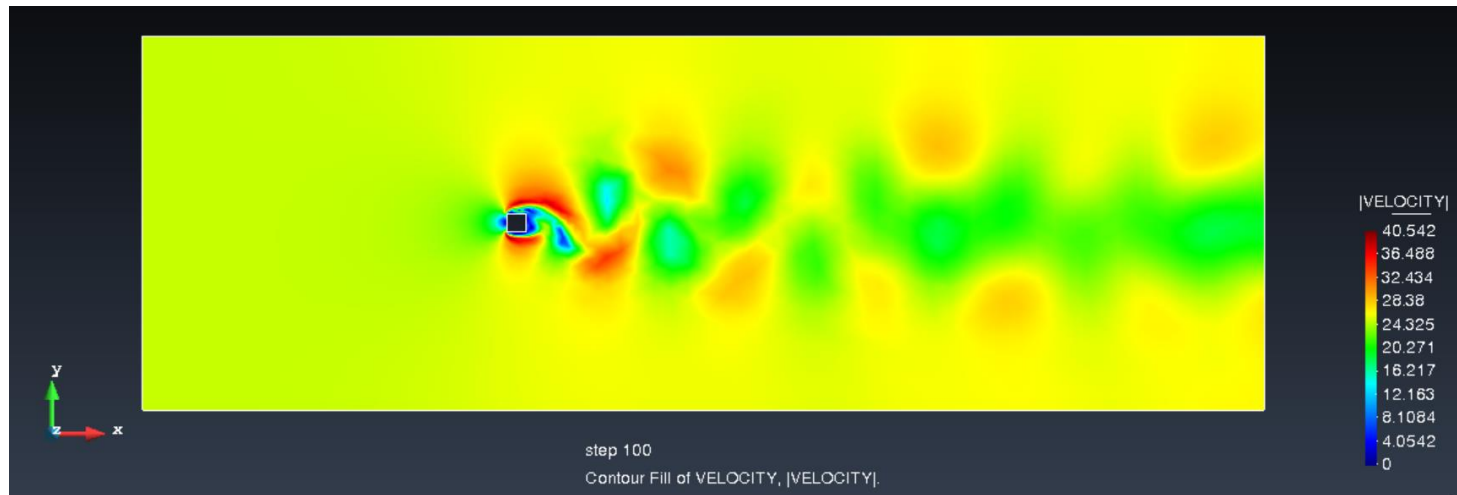


Post processing

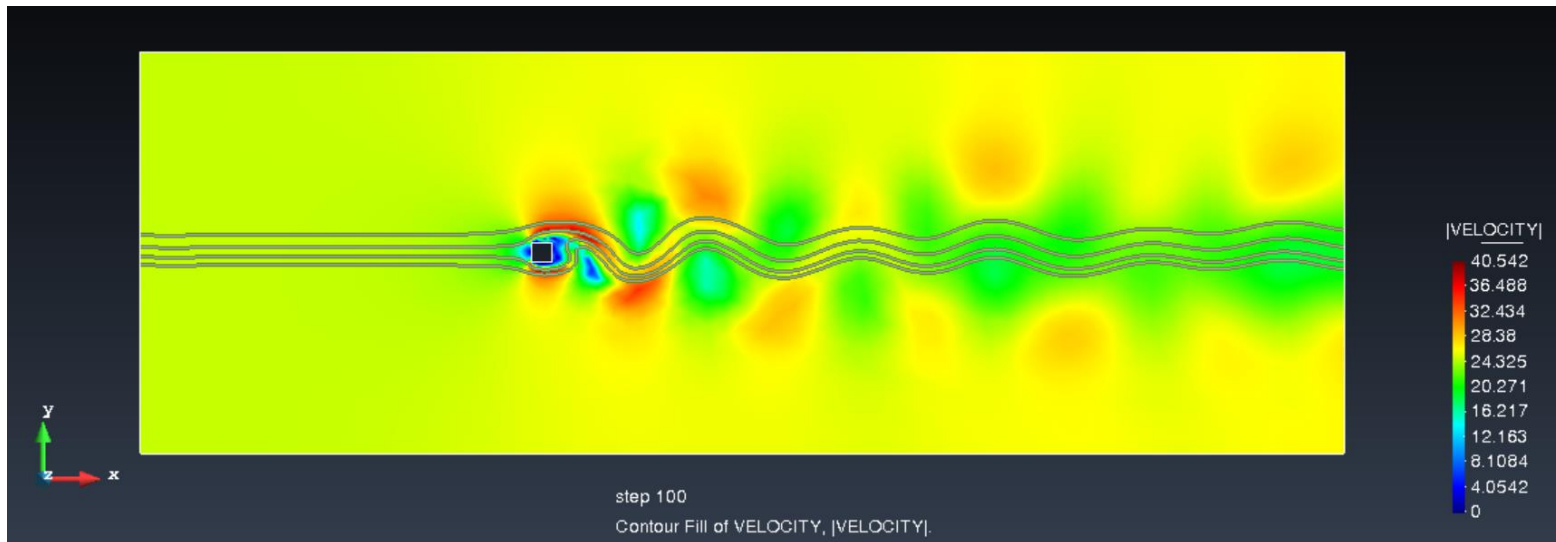
- Results can be animated using
Window → Animate...
or *Ctrl + m*



- Play around with the results and the visualization:
- E.g. view the velocity results
- Results for *magnitude of velocity* in the last timestep:

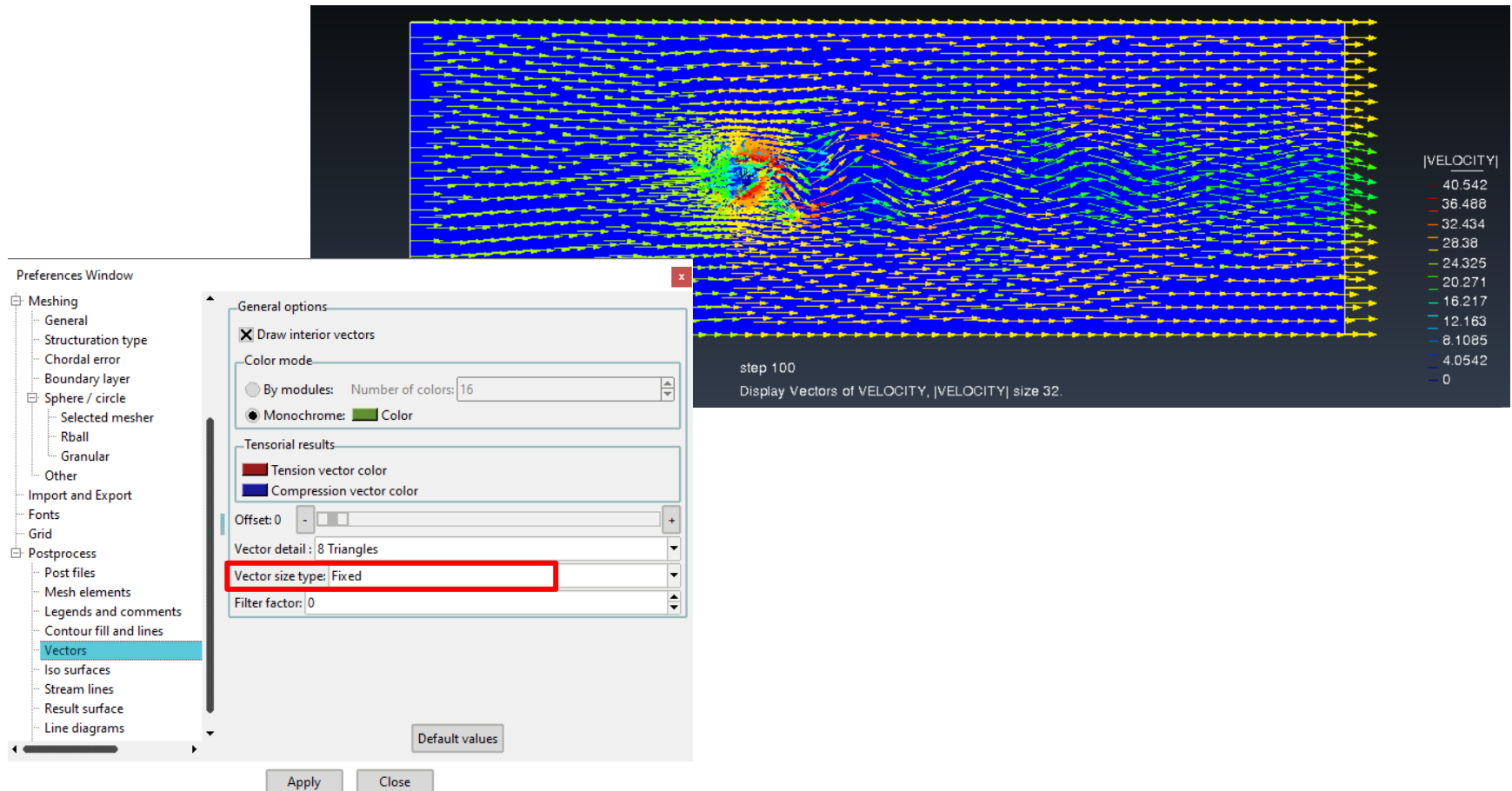


- View results → Stream lines → Single point → Velocity



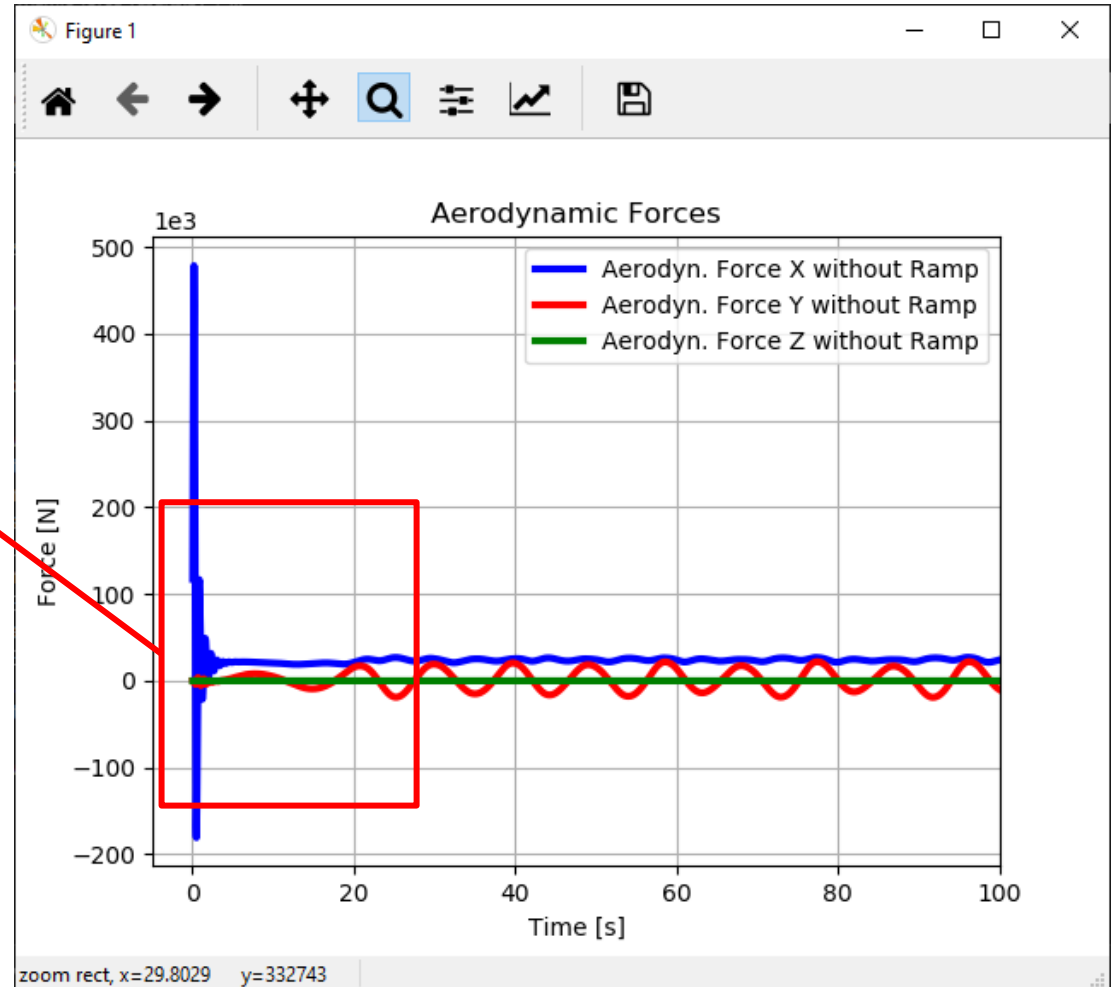
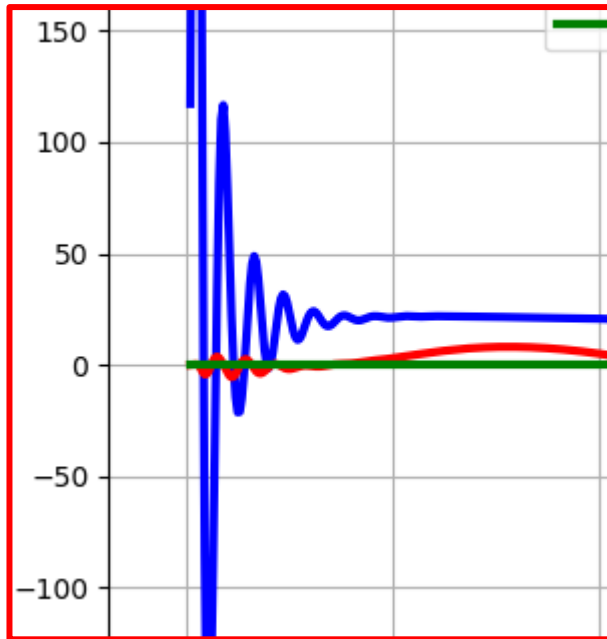
- View results → *Display vectors* → *Velocity* → $|Velocity|$

Change the vector size: *Utilities* → *Preferences* → *Postprocess* → *Vectors*



Aerodynamic results

- Copy “[plot_aerodynamic_force_results.py](#)” from *AdditionalFiles* into your GID project folder and run the python file
- Check the correct file name of the drag results in the python file
- The results should look like this:



- Calculate the Strouhal number St and the drag coefficient C_D
- f – frequency of the vortex shedding
- L – characteristic length
- U – freestream velocity of the flow

$$\Rightarrow St = \frac{f * L}{U} = (0.125 - 0.15)$$

- ρ – density of the fluid
- A – reference (projected) aread
- F_D – (time-averaged) drag force (from the previous plot)

$$\Rightarrow C_D = \frac{2 * F_D}{\rho * A * U^2} = (2.0 - 2.2)$$