

# STRUCTURAL WIND ENGINEERING

Roland Wüchner, Chair of Structural Analysis, TUM

Máté Péntek, Chair of Structural Analysis, TUM

**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 the postprocessing of an already existing result file and import an given geometry.

## Covered topics:

- Post processing of results
- Using a 3D Geometry from a CAD software

**Disclaimer:** This example serves the sole educational purpose of demonstrating how to postprocess a 3D CFD case. It also includes note how an example case could be set up from CAD data.

**Technical note:** Tested on 04.12.2019, works with GiD 14.1.7d and the pre-release of the Kratos problemtype (7.1) on Windows 10 and Ubuntu 18 64 bit.

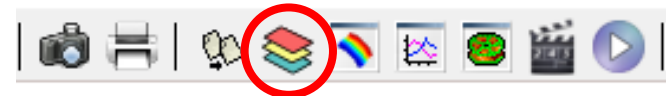
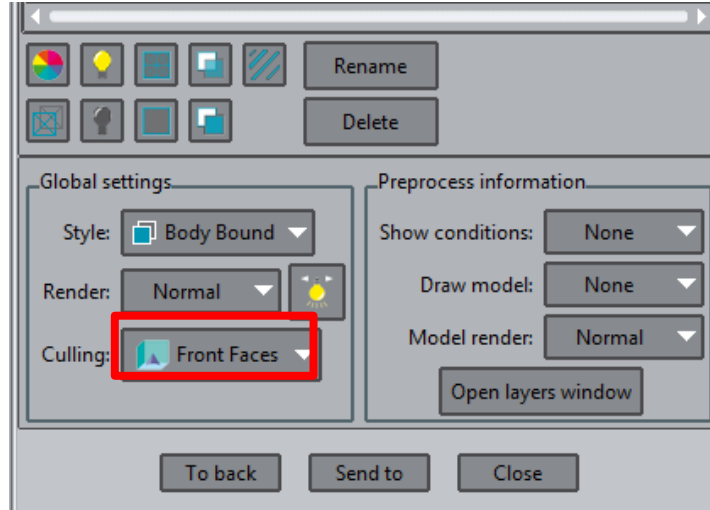
# Solution Postprocessing

- Previous Task 2 (3D CFD-analysis) takes too long to compute during the tutorials
- To see the results download the pre-calculated result file
- To load the provided results, go to [postprocessing](#)
- Load the provided ... *.post.bin* – file

## Alternative:

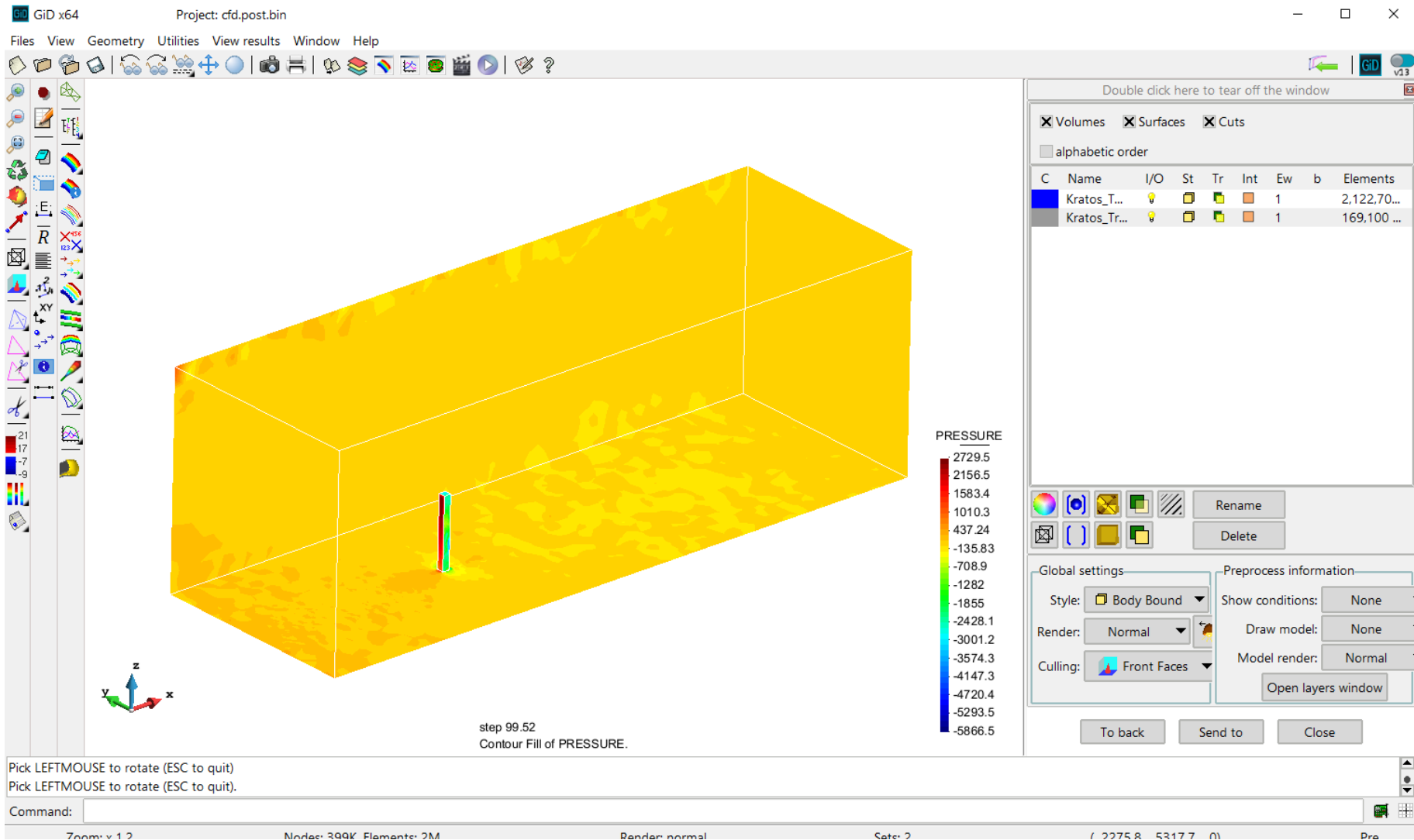
- Although you can solve the previous problem with the created mesh, it is by far too coarse to get reasonable results. But it can be done for the purpose to practice
- The following slides deal with results from a different structure (different values to your 3D calculation)
- The result files of a computation with a reasonable fine are provided to do the postprocessing in *CFD\_HighRiseExampleFine* (via a download link)
- Additional information and useful files saved from postprocessing in *CFD\_HighRiseExampleFine\additional\_information\_and\_data/*
- Auxiliaryfiles in *CFD\_HighRiseExampleFine\auxiliary\_files/*

- 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**



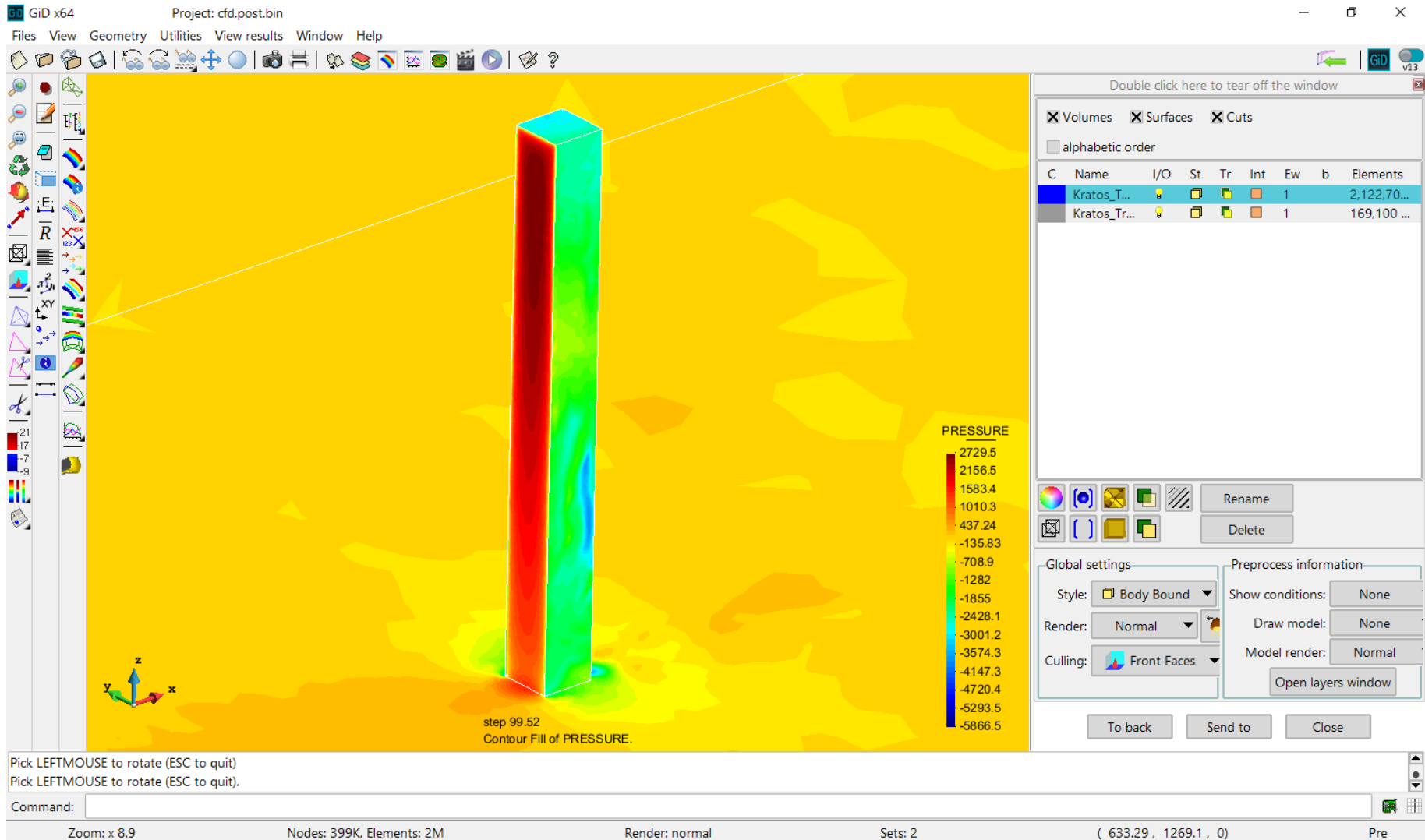
# Post processing

- Results for *pressure* in the last timestep:



# Post processing

- Close-up of the structure :





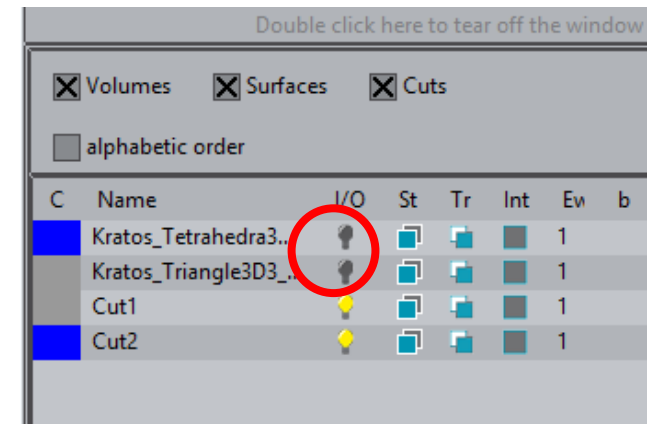
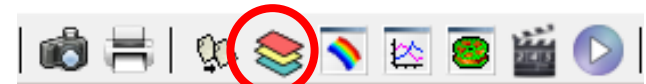
# Using cuts

- **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	2/3 H	2/3H	2/3H
X	Y	Z (alt.)	(Tut. 3)

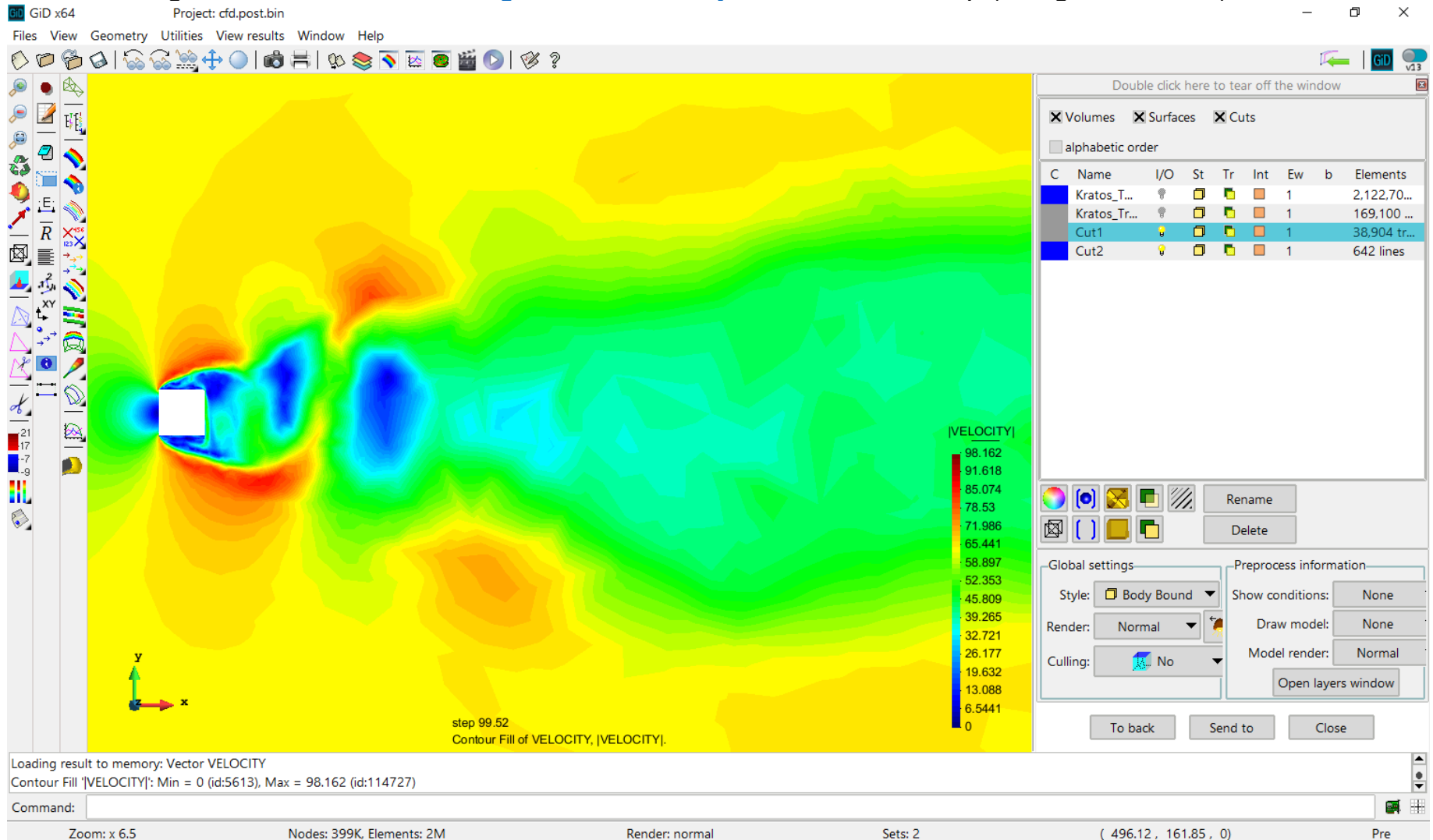
0.0	0.0	400.0	65.0
1.0	0.0	400.0	65.0
1.0	1.0	400.0	65.0

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



# Post processing - cuts

- Visualize the time evolution of the velocity using *Ctrl + m*
- The figure shows results for *magnitude of velocity* in the last timestep (using the first cut):



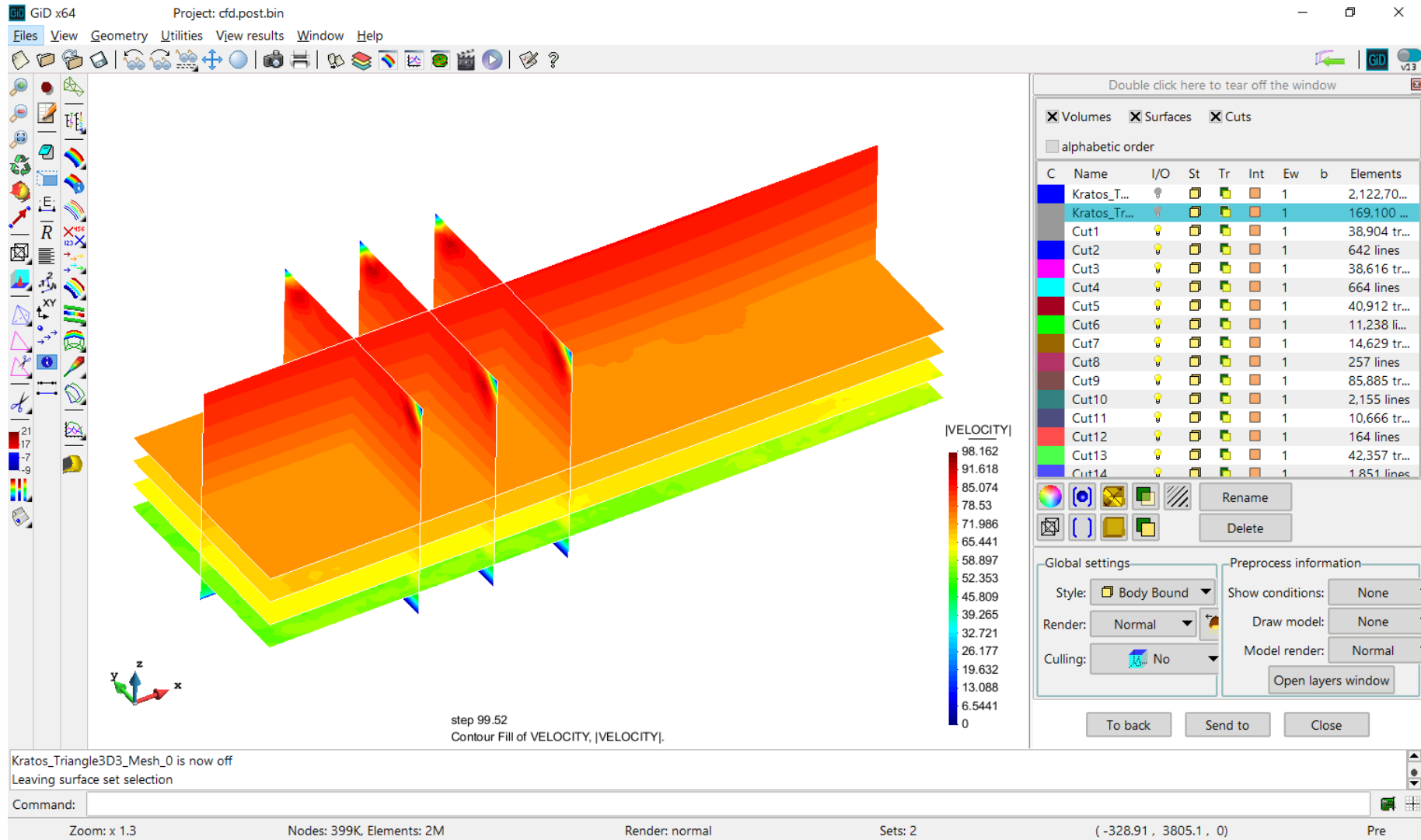
# Using cuts

- Create additional horizontal and vertical cuts
- Export the cuts  
*Files → Export → Cut → All cut*
- Cuts can be reused  
*Files → Import → Cut ...*
- *Play around with different plane coordinates*

Horizontal X	for example Y	Z
0	0	200
1	0	200
1	1	200
0	0	600
1	0	600
1	1	600
0	0	600
1	0	600
1	1	600

Vertical X	for example Y	Z
0	0	0
1	0	0
1	0	1
-600	0	0
-600	1	0
-600	1	1
0	0	0
0	1	0
0	1	1
600	0	0
600	1	0
600	1	1

# Using cuts

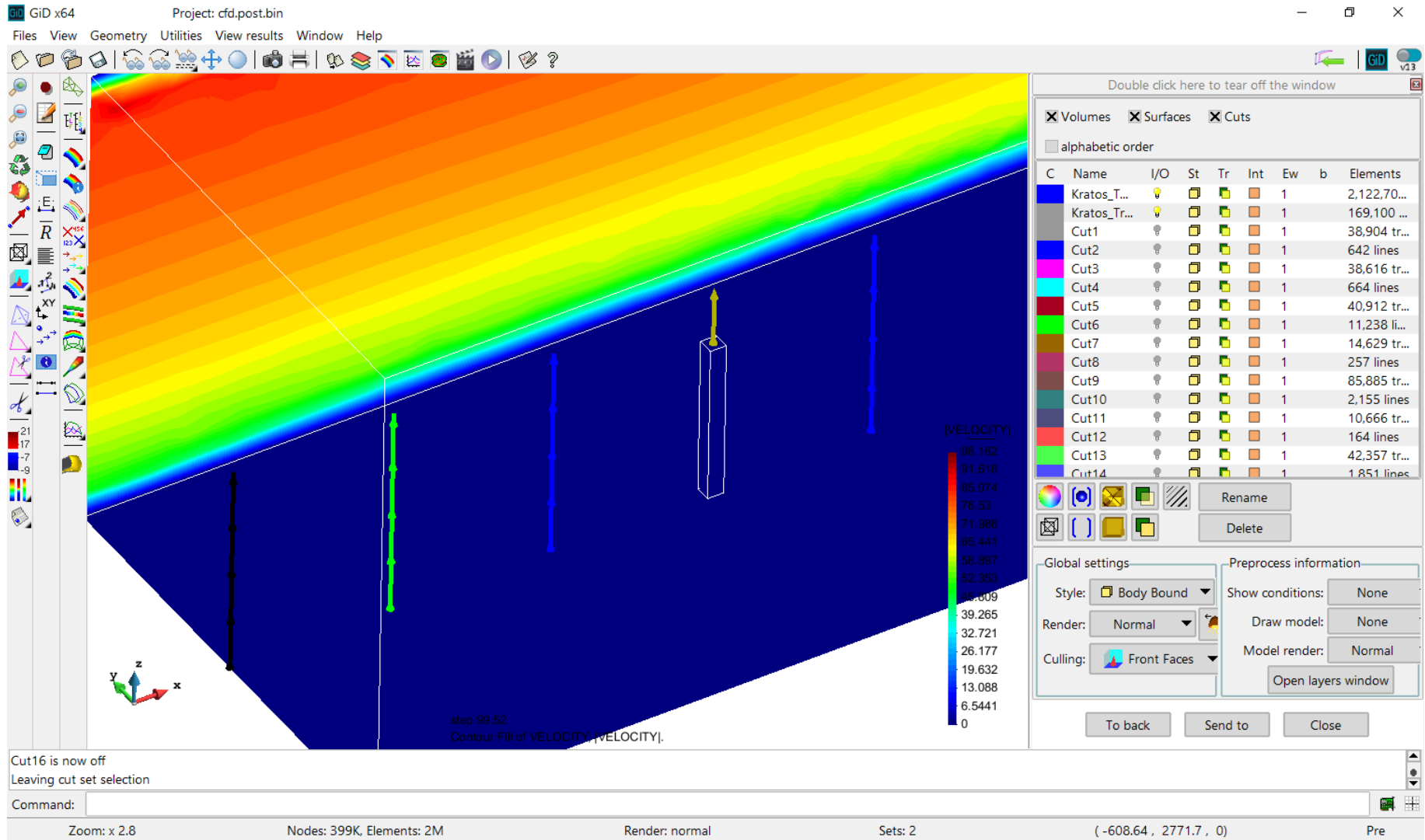


- Create a graph with several points at the inlet and the building  
*View Results → Graphs → Line graphs → VELOCITY → X\_VELOCITY*
- Exports the graph  
*Files → Export → Graph → All graphsets*
- Graph can be reused  
*Files → Import → Graph ...*

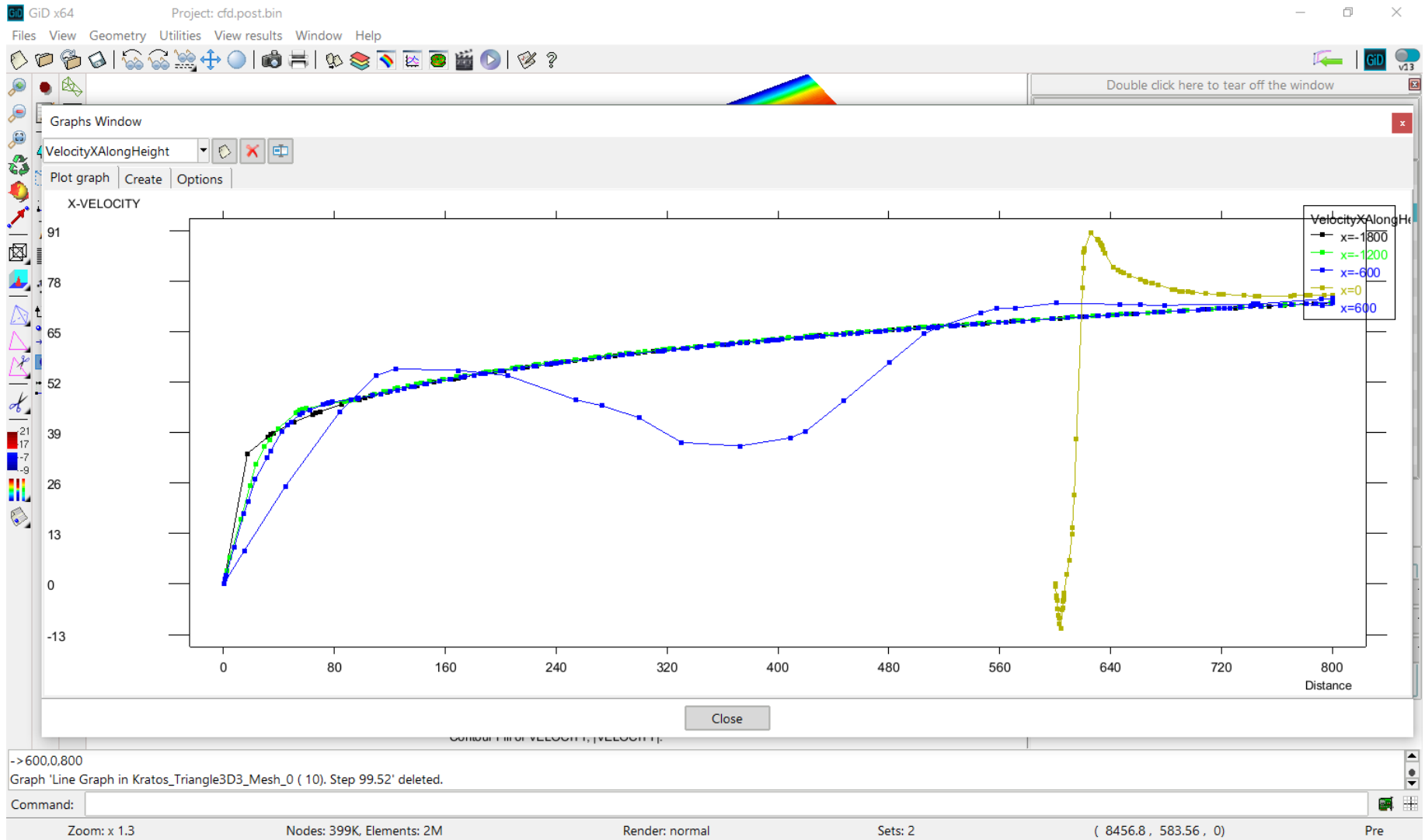
In the \*.grf file you have all the numerical data to create plots in Excel, Python etc. for your project report and presentation.

Graphs X	for example Y	Z	
-600	0	0	at the inlet
-600	0	500	
1200	0	0	
1200	0	500	
-300	0	0	
-300	0	500	
0	0	0	at the building
0	0	500	
600	0	0	
600	0	500	

# Using graphs



# Using graphs



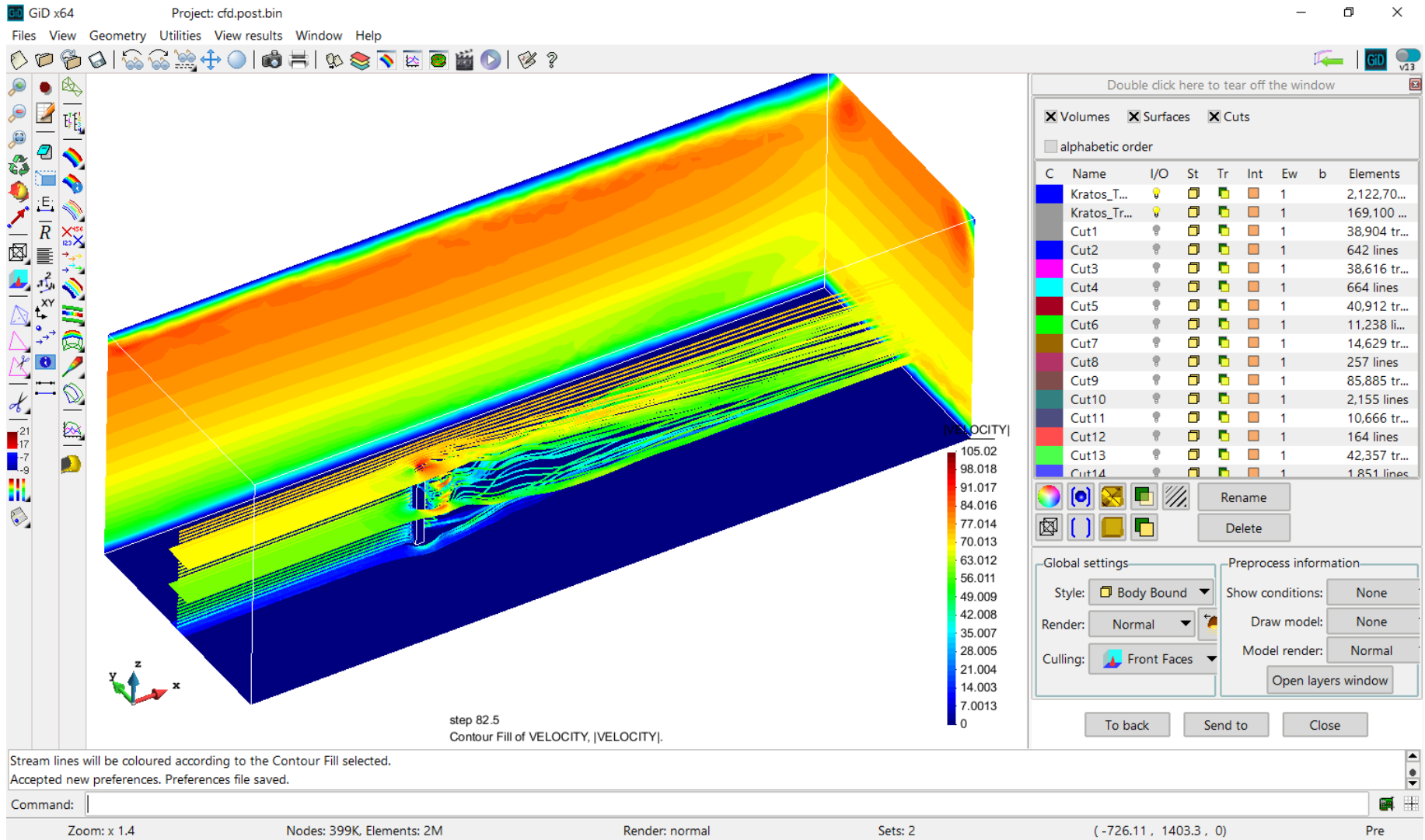
- Create [streamlines](#), for e.g. along lines (defined by X, Y, Z of the start and end points), like shown in the table (they are for a certain time step)  
[View Results](#) → [Streamlines](#) → [Along line](#) → [Velocity](#)
- Change the color property of the streamlines  
[Utilities](#) → [Preferences](#) → [Postprocess](#) → [Stream lines](#) → [Color mode](#) → [Result contour filled](#)
- Exports streamlines  
[Files](#) → [Export](#) → [Post information](#) → [Stream lines](#)

- Streamlines can be reused  
[Files](#) → [Import](#) → [Stream lines ...](#)

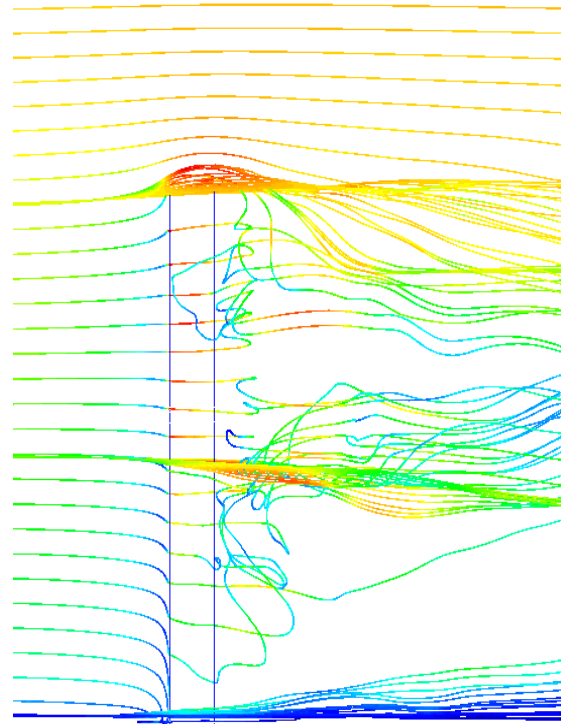
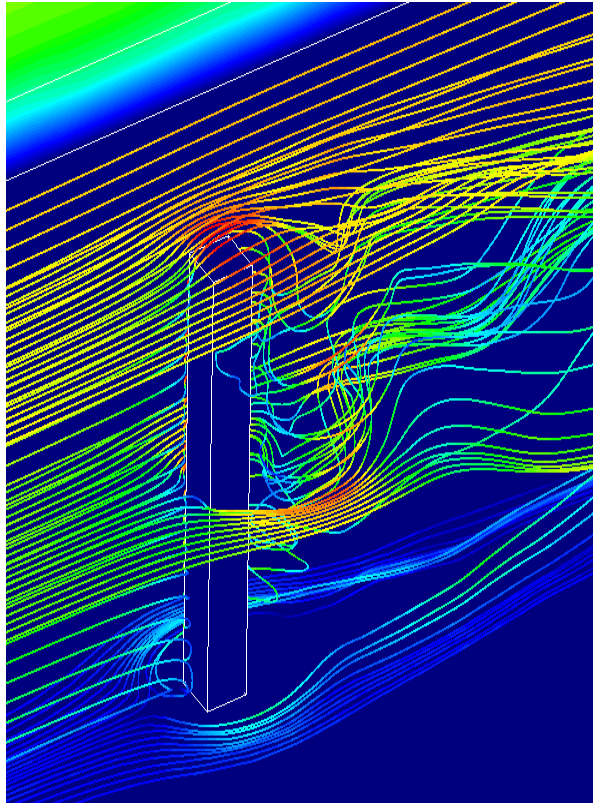
Steamline			
X	Y	Z	
-200	0	0	
-200	0	500	→ 30 points
-120	-120	10	
-120	120	10	→ 20 points
-120	-120	100	
-120	120	100	→ 20 points
-120	-120	190	
-120	120	190	→ 20 points



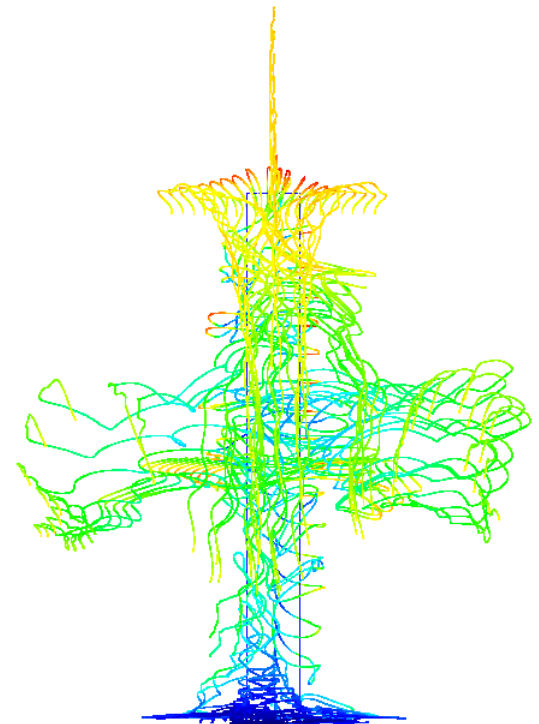
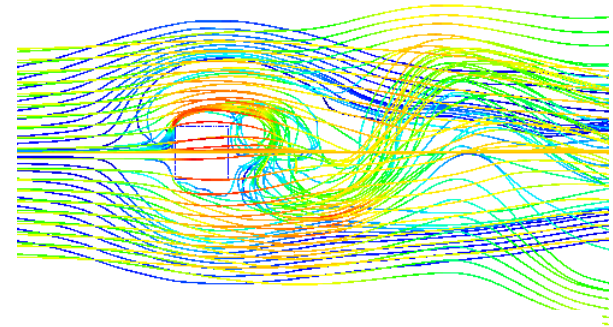
# Using streamlines



# Using streamlines



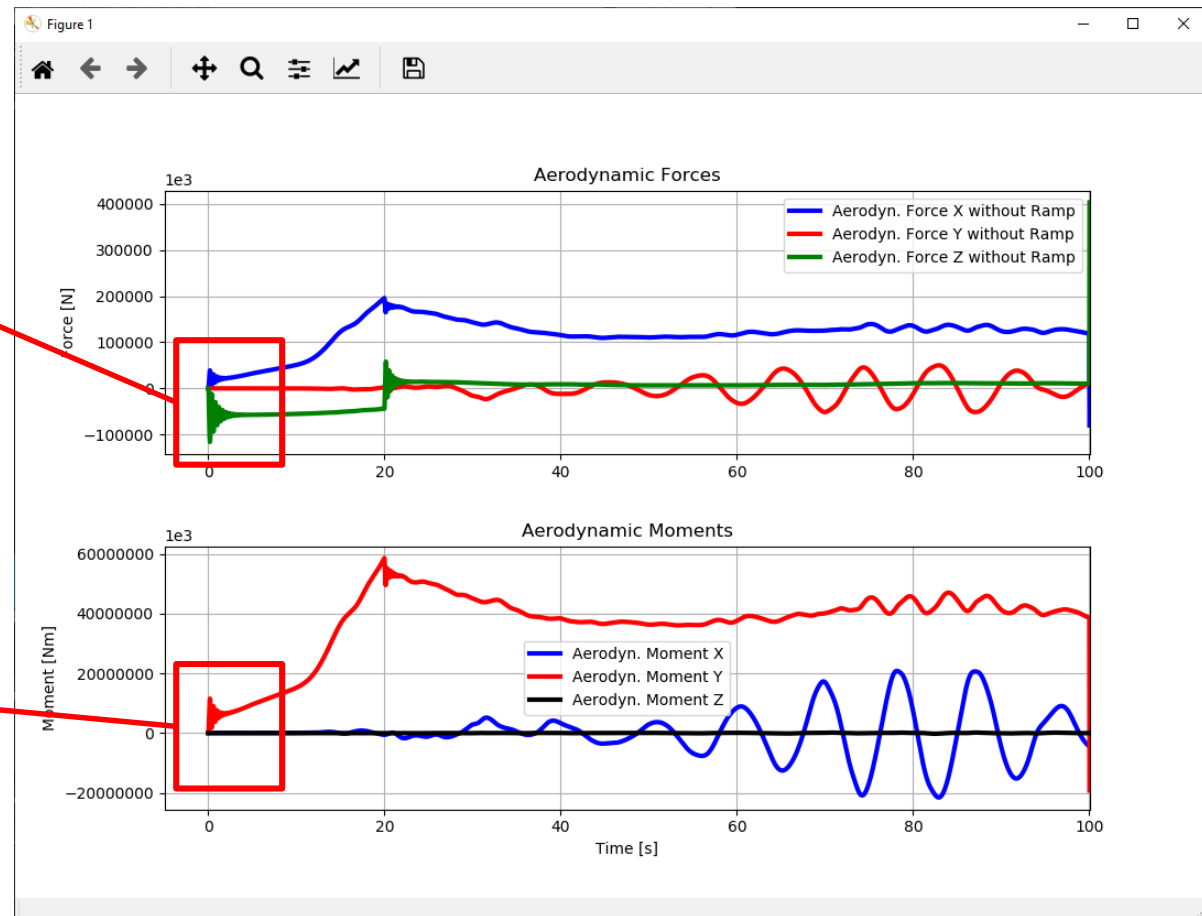
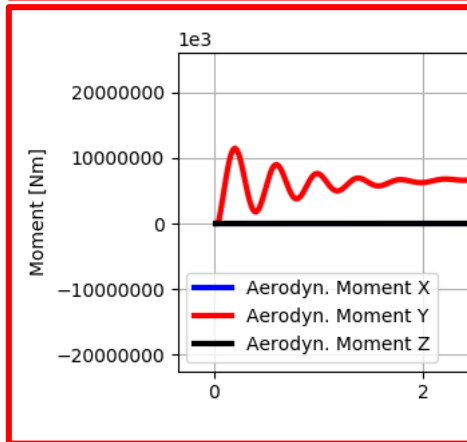
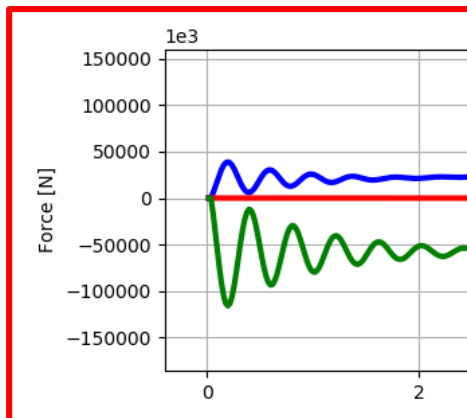
step 82.5  
Contour Fill of VELOCITY, |VELOCITY|.



step 82.5  
Contour Fill of VELOCITY, |VELOCITY|.

# Aerodynamic results

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



# Using a 3D geometry from a CAD software

→ suggested task for individual work, useful for projects

In this tutorial we will work with geometries of real buildings. The main challenge here is to clean up the geometry and therefore make it suitable for meshing/analysis.

## Covered topics:

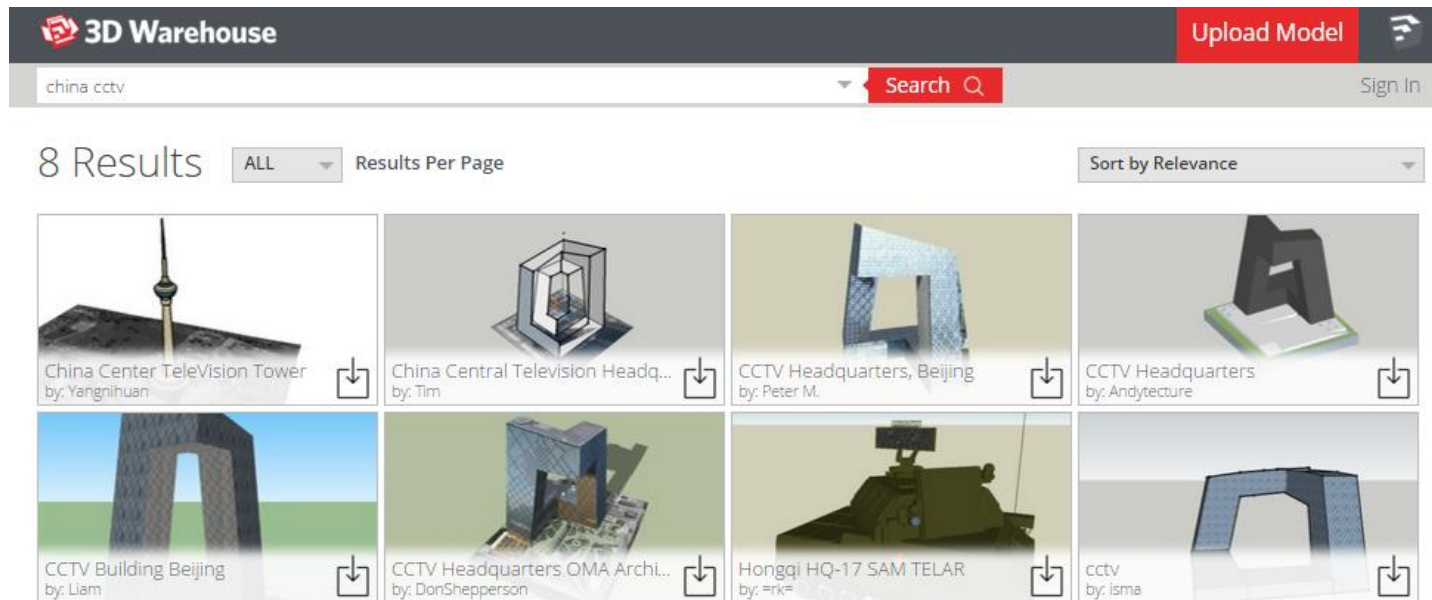
- Cleanup of CAD-geometries
- Creating and meshing fluid domains



[<http://blog.blockbrief.com/wp-content/uploads/2014/10/Dubai-Pearl.jpg>]

# Kratos 3D Fluid Tutorial – real geometry

You can create 3D models on your own or import some available (for e.g. from [3D Warehouse](https://www.3dwarehouse.sketchup.com/))



Necessary steps:

1. Import the SketchUp file format \*.skp to AutoCAD using this [plugin](#)
2. Export a common CAD format such as \*.dxf, \*.stl, etc.
3. Import the more common format into GiD
4. Clean and prepare the geometry for a CFD simulation → see steps on the following slides

Cleaning and preparing the geometry is necessary because for CFD you need to avoid overlapping geometric entities as in the end you will need a so-called „watertight“ geometry for meshing and simulation.

# Cleaning the Geometry

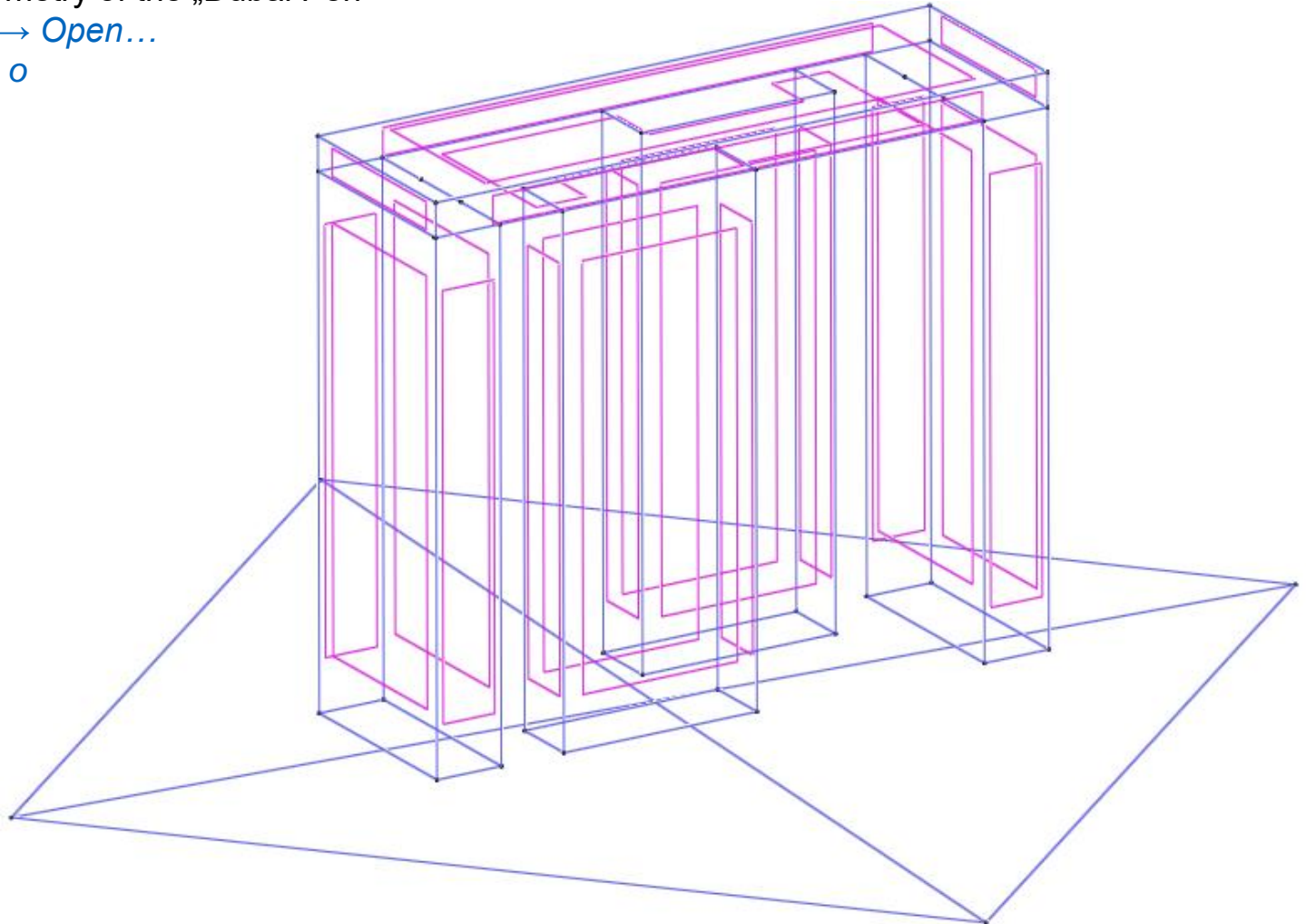
- Rotate the view to plane XY  
*View → Rotate → Plane XY*
- Create domain and base for building  
*Geometry → Create → Point*
  - Enter the following points in the command line
  - Press *Enter* after each point

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

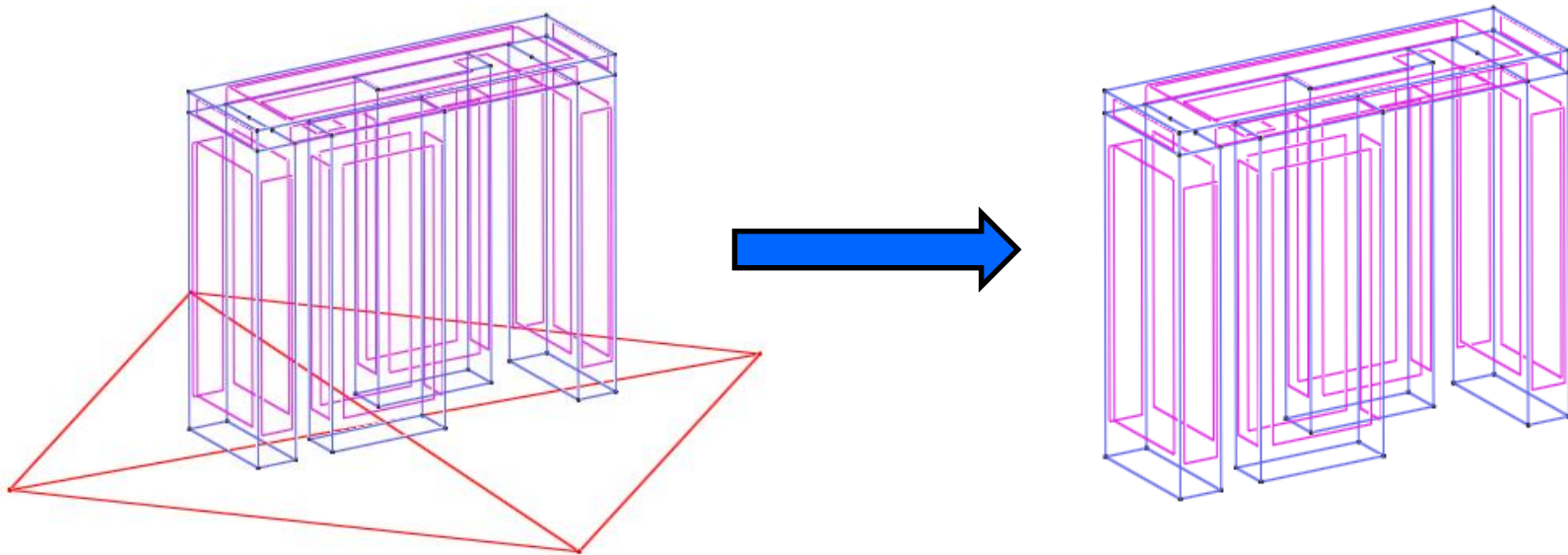
Boundary		
X	Y	Z
-600.0	300.0	0.0
-600.0	-300.0	0.0
1200.0	300.0	0.0
1200.0	-300.0	0.0



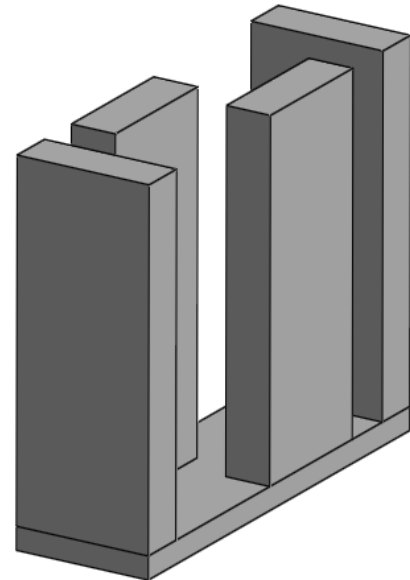
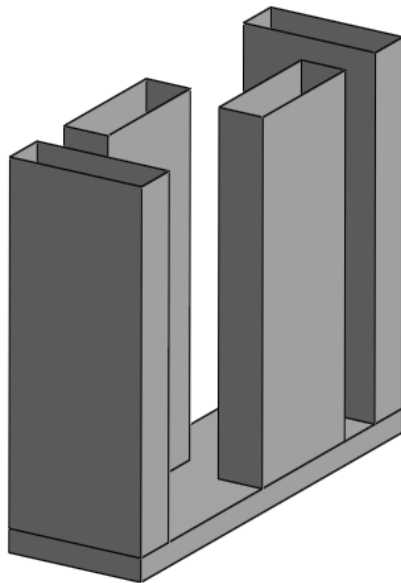
- Load the geometry of the „Dubai Perl“  
    *Files → Open...*  
or    *Ctrl + o*



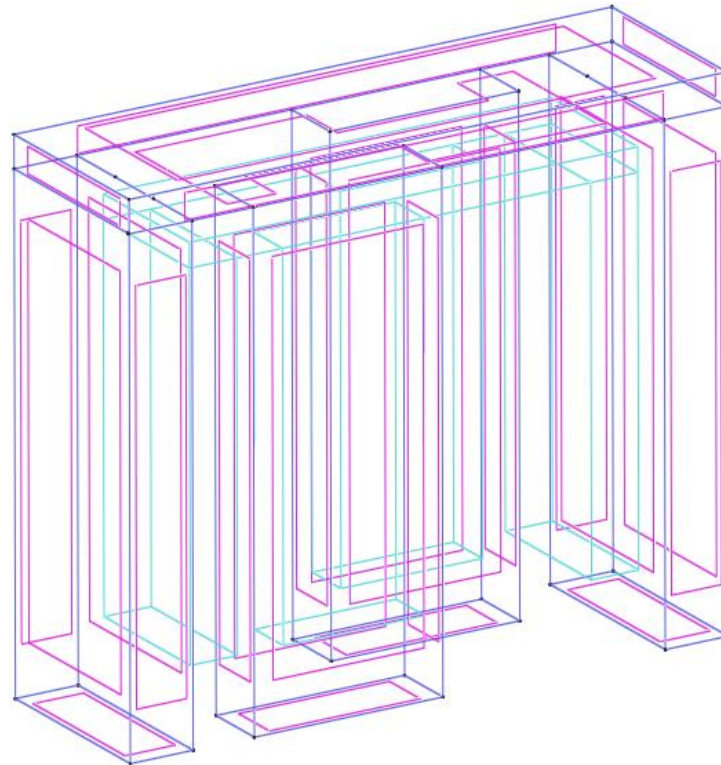
- To create a mesh, one first needs to create a **volume**  
=> This is not possible yet as the body is not fully closed by surfaces
- Delete the „artificial“ bottom lines & corner points  
*Geometry → Delete → All types*



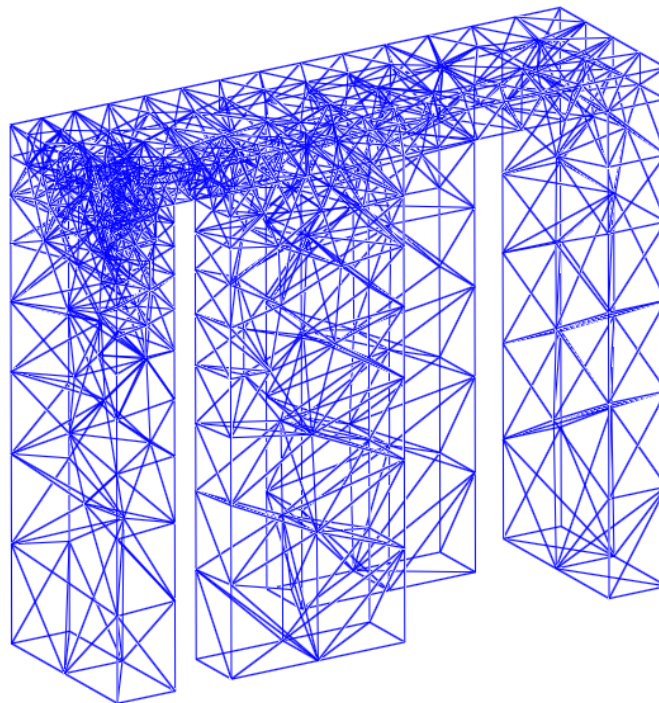
- The easiest way to check the model for missing surfaces is to change the rendering (switch from Normal to Flat)  
*View → Render → Normal / Flat*
- Rotate the model to find the missing surfaces
- Create the missing surfaces (4)



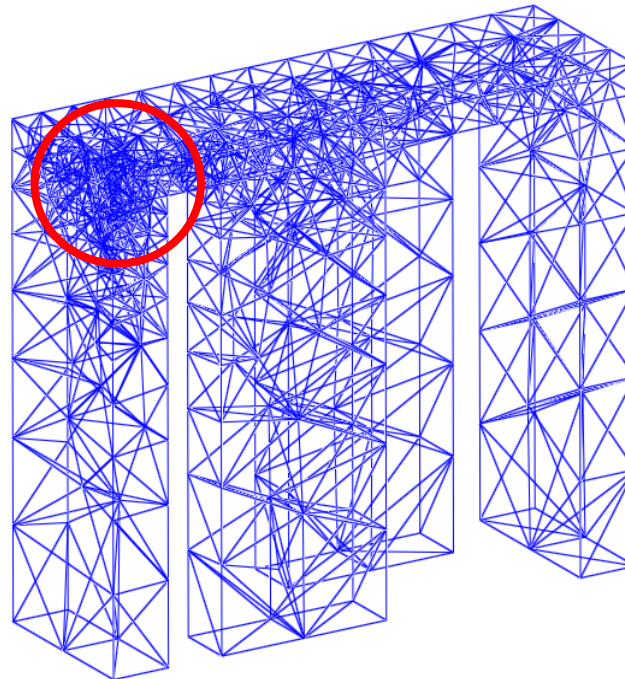
- Create a volume by selecting all surfaces
- If no volume can be created, it is an indicator, that there is something wrong with the geometry!



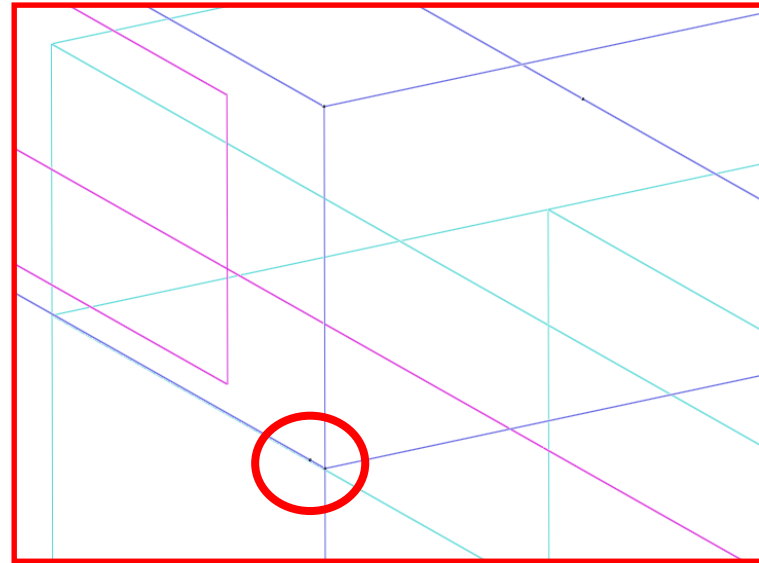
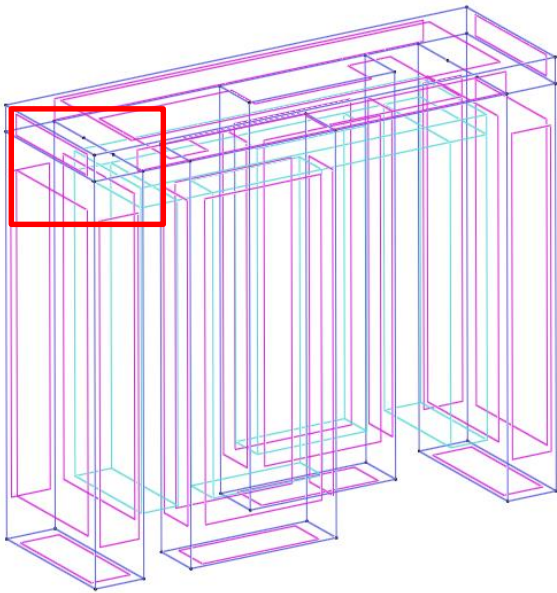
- To check the quality of the geometry, mesh the structure:
  - First reset all mesh data  
*Mesh → Reset mesh data*
  - Create the mesh using „10000“ as element size



- Check the quality of the mesh
- It is obvious that there is an unfortunate concentration of small elements in one corner
- This is an effect of an uncleaned geometry



- Switch back to geometry view and investigate the geometry in this area closely



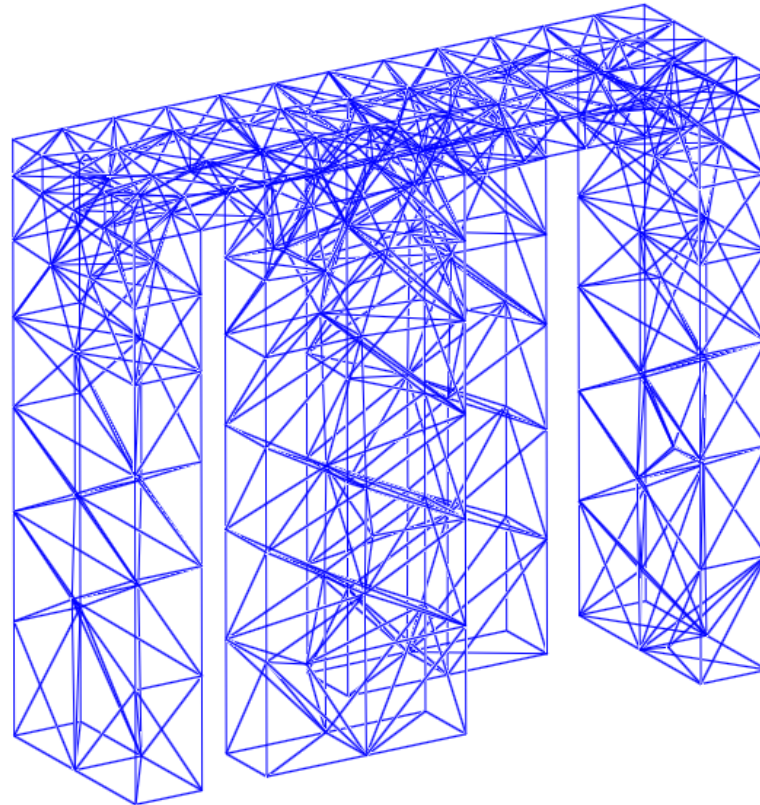
- Some tiny line segments exist, that cause the fine mesh in this area.
- Check the model for further very small primitives
- **This is a typical result form importing CAD-geometries!**

# Geometry – cleaning up

- One way to (automatically) clean the model is to collapse it  
*Geometry → Edit → Collapse → Model*
- This option joins small entities to larger one  
=> In our case 5 points / 5 lines were deleted (check command line), including the tiny line segments that caused problems
- Using the automatic model collapsing is not always a good idea, especially if small features want to be resolved.
- Therefore cleaning up the geometry can also be done manually. GiD provides many functionalities to so.  
*Geometry → Edit*



- Re-mesh the model
- The mesh is a lot more homogeneous than before



- As we want to compute the flow around the object, we have to create and mesh a fluid domain, just as in the first tutorials.
- Use the following domain size recommendations:
  - Domain height  $h_D$ :  $3h_B$
  - Domain width  $w_D$ :  $3h_B = 2 \times (1.5h_B - 0.5w_B)$ ; (symmetric)
  - Domain length  $l_D$ :  $9h_B$ ; ( $3h_B - 0.5l_B$  upstream,  $6h_B - 0.5l_B$  downstream)

