

8 POSTPROCESSING

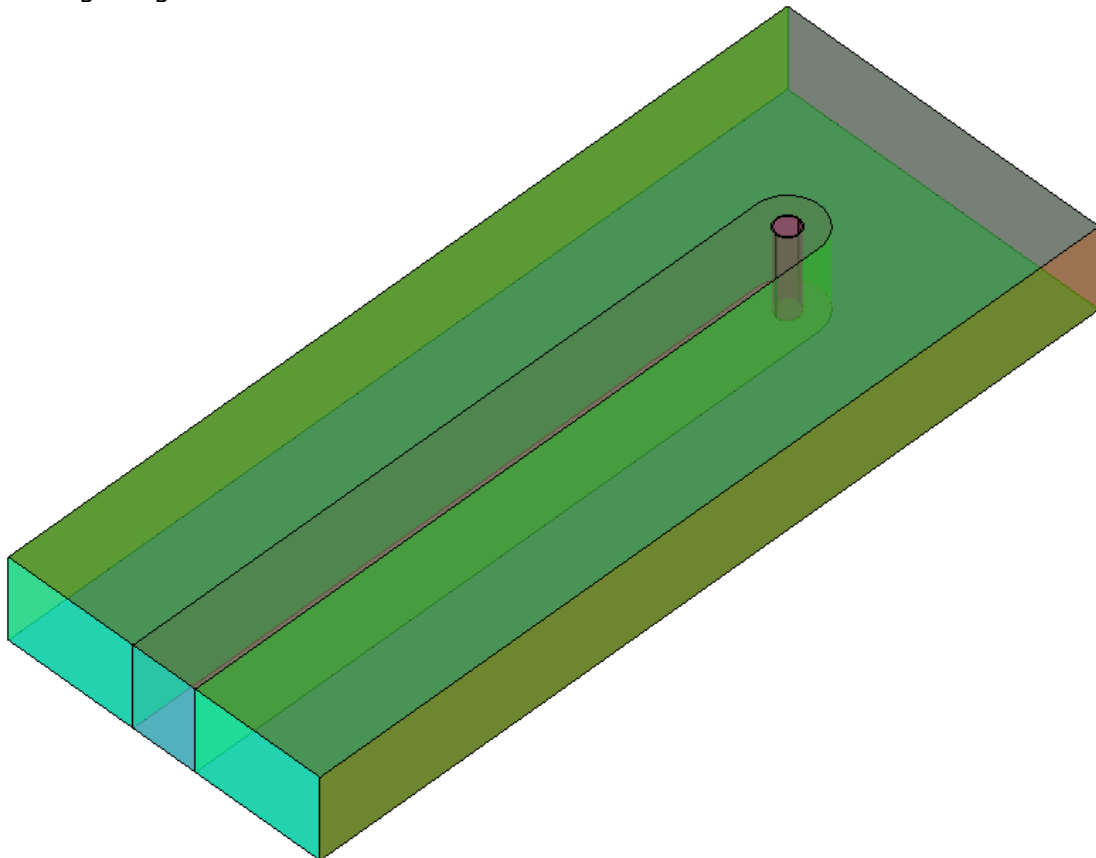
The objective of this tutorial is to do a postprocess analysis of an already calculated fluid simulations, no preprocess option is used.

Not only the model is already meshed and the constraints are assigned, but also the results have been calculated. For more information about the preprocess part of GiD, please check the preprocess tutorials.

In this tutorial, the model *Cylinder.bin* has been used. The problem type used to do this simulations is Tdyn, particularly the Ransol model. Tdyn is a fluid dynamic (CFD) simulation environment based on the stabilized Finite Element Method.

Steps followed in this tutorial:

- Loading the model
- Changing mesh styles
- Visualization of results
- Creating images




8.1 Loading the model

There are two ways to load the results simulation information into GiD:

- If the model has been calculated inside GiD, the results are also inside the GiD model, then just load the GiD project and change to postprocess mode. This can be achieved clicking on this icon:



, or selecting the **Files->Postprocess** menu entry.

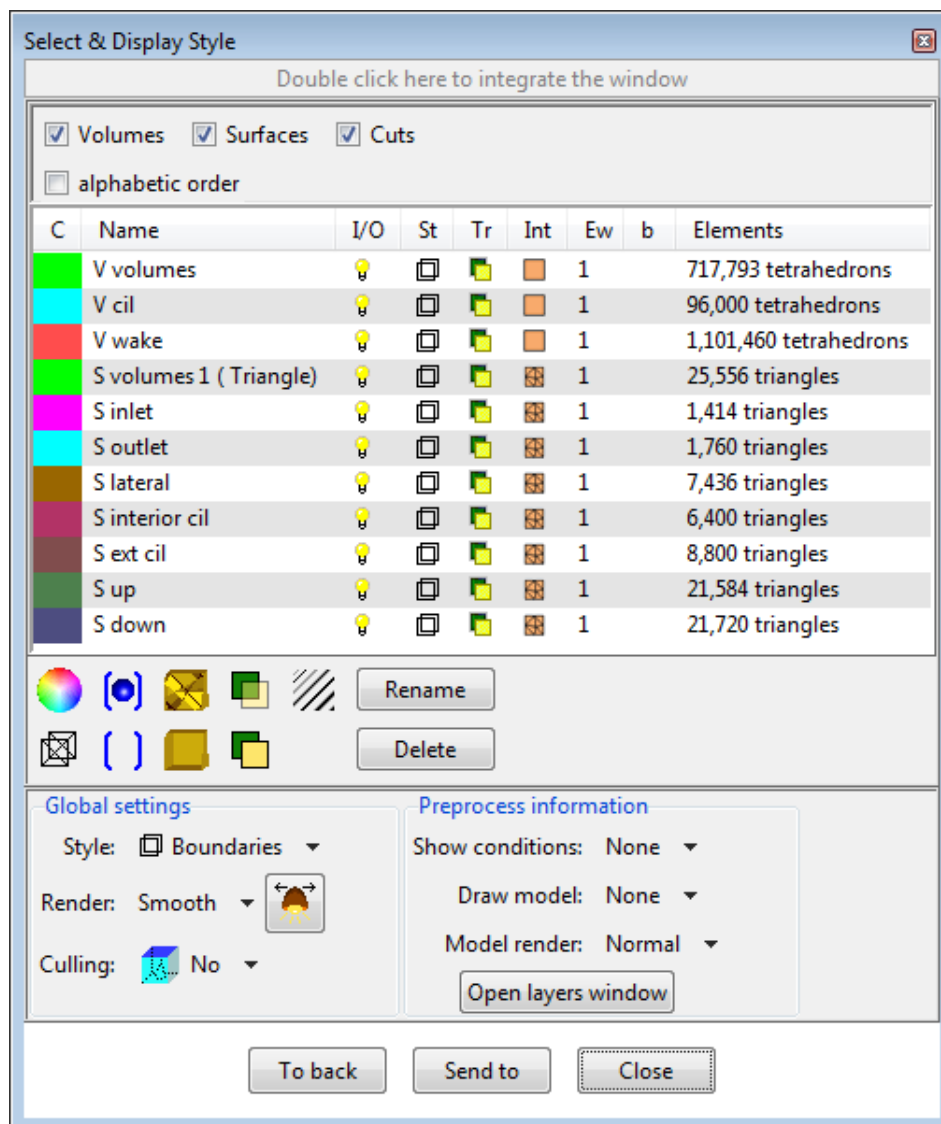
- If only a mesh and results file(s) are present then GiD should be started, and switched to postprocess mode () before loading the file(s).

For this tutorial we will use the file called "Cylinder.bin" that contains the postprocess information, so the steps to follow are:

- 1 . Start GiD
- 2 . Switch to postprocess mode:  or **Files->Postprocess**
- 3 . Open the model with: **Files->Open**, **Ctrl-o** or clicking on 

8.2 Changing mesh styles

- 1 . Select **Window->View style...**
- 2 . Select all the layers
- 3 . change the style to **Boundaries**
- 4 . Play a little with the options of these windows, but to continue the tutorial, let a **Boundaries** style selected for all meshes
- 5 . Change render mode to **Normal**

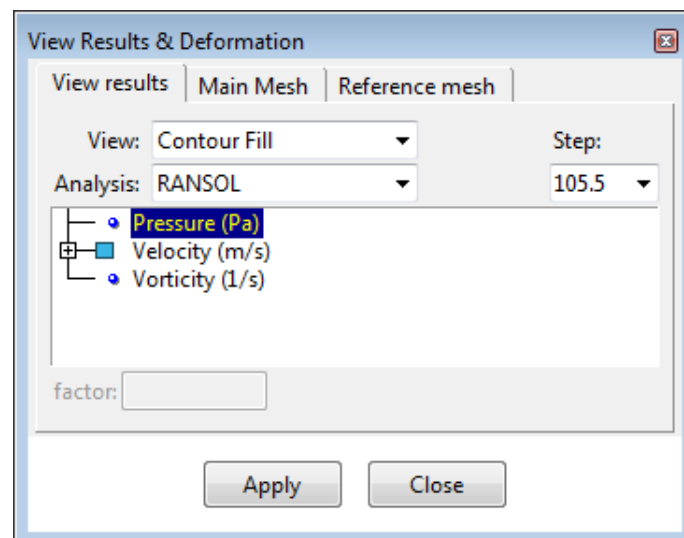


8.3 Viewing the results

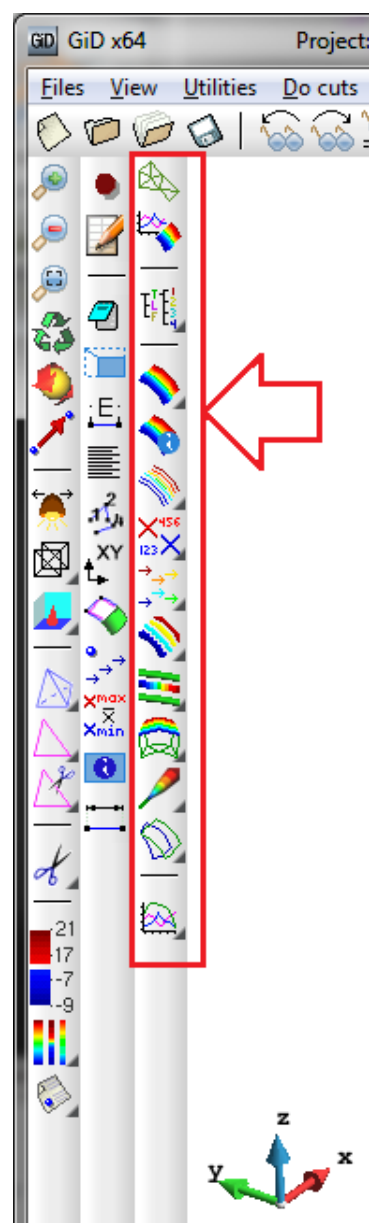
In the example, several results have been calculated for several time steps. You can check these results through the **Results** menu, opening the **View Results** window or through the results view icon bar.

Menu: View Results

Window->View Results...



Results view icon bar:




8.3.1 Iso surfaces

Menu: View results->Iso Surfaces

With this result visualization a surface, or line, is drawn passing through all the points which have the same result's value inside a volume mesh, or surface mesh. To create isosurfaces there are several options.

- 1 . Select **View results->Iso Surfaces->Automatic**

Width->Velocity(m/s)->|V|through the menu bar or clicking on  on the results view icon bar.

After choosing the result, you are asked for a width. This width is used to create as many isosurfaces as are needed between the Minimum and Maximum defined values (these are included).

- 2 . enter the value 0.25727 to get the picture below.
- 3 . Select **View->Render->Smooth** in order to get a better view.

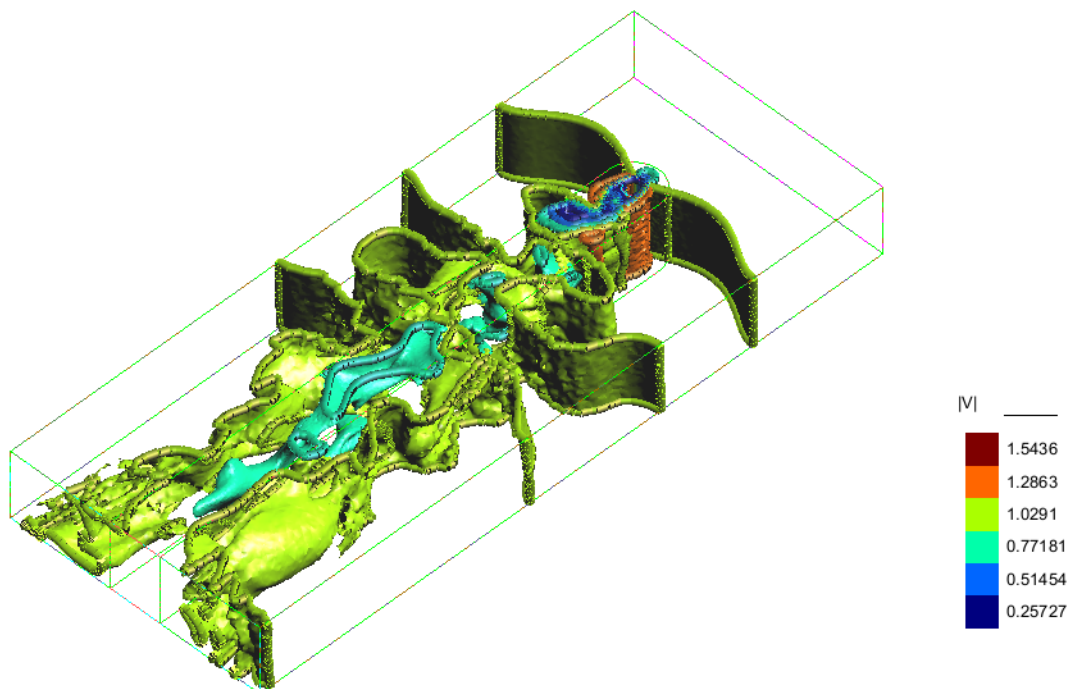
Several configuration options can be set via the Options menu.

Menu: Options->Iso surfaces

Using **Options -> Iso surfaces -> Display Style** the style of the iso-surface can also be changed as with the volume and surface meshes.

In order to see the inner zones we will set the transparency on the iso surfaces.

- 4 . Select **Options->Iso Surfaces->Transparency->Transparent**
- 5 . Move the model to see the inner zones
- 6 . Select **Options->Iso Surfaces->Transparency->Opaque**



Other interesting options are:

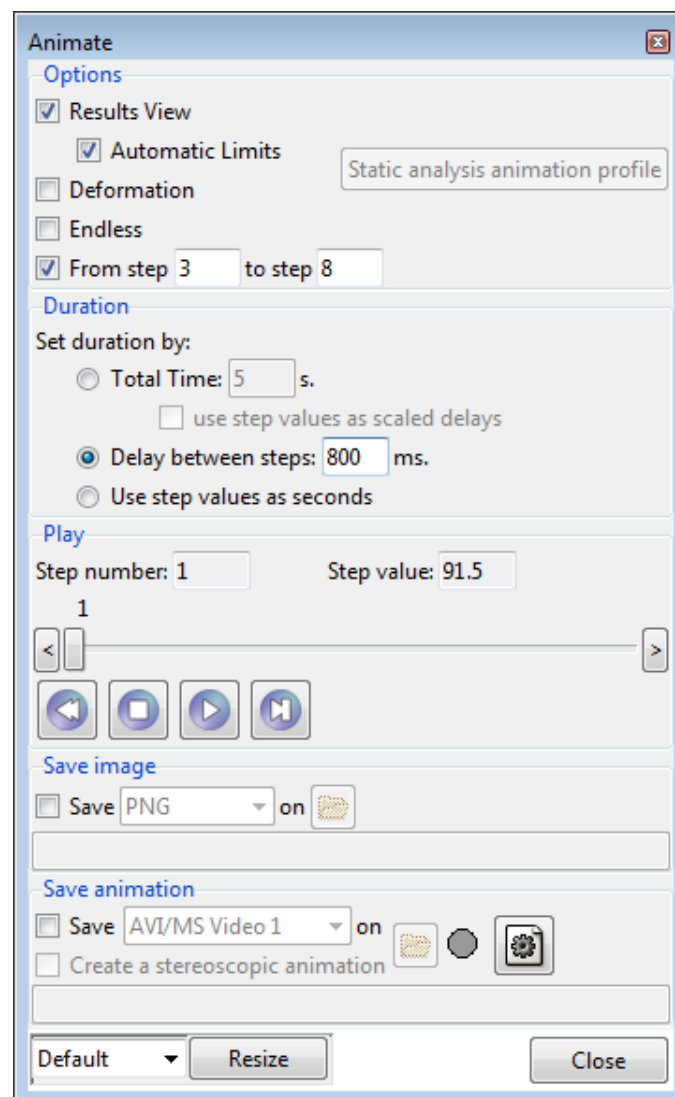
- **Options->Iso surfaces->Convert to cuts** which consolidates the isosurface as mesh which can be exported to a file.
- **Options->Iso surfaces->Color Mode** allows to draw the iso-surfaces with a single colour (**Monochrome**), according to the results used to create the iso-surface (**Result color**) or using the color map of the visualized contour fill result (**Contour fill color**).
- **Options->Iso surfaces->Show isolines** this option allows the user to switch isolines of surfaces on or off.
- **Options->Iso surfaces->Draw always** if this option is selected the iso-surfaces are always drawn even though all the meshes are switched off.

8.3.2 Animate

Menu:Window->Animate...

This window allows the user to animate the current visualized results.

If only one step is present, then the **Static analysis animation profile** button is enabled so that a custom animation profile can be step to animate that one step.



If one result has several steps you can visualize them in an animation. In this case we will

use the iso surfaces result.

- 1 . Select **View->Render->Smooth**
- 2 . Select **Window->Animate...** to open the animation window

Please notice that we have from step 1 to 13. We will do the animation only of some of these steps.

- 3 . Check the **From step** option and set 3 **to step** 8
- 4 . Select the **Delay between steps** option and set it to 800 ms. The animation should take 4 seconds
- 5 . Try it clicking on the **play** icon


We will record a video during the animation.

- 6 . Once the animation is finished check the **Save** option on the **Save animation** part

You can choose from several video formats.

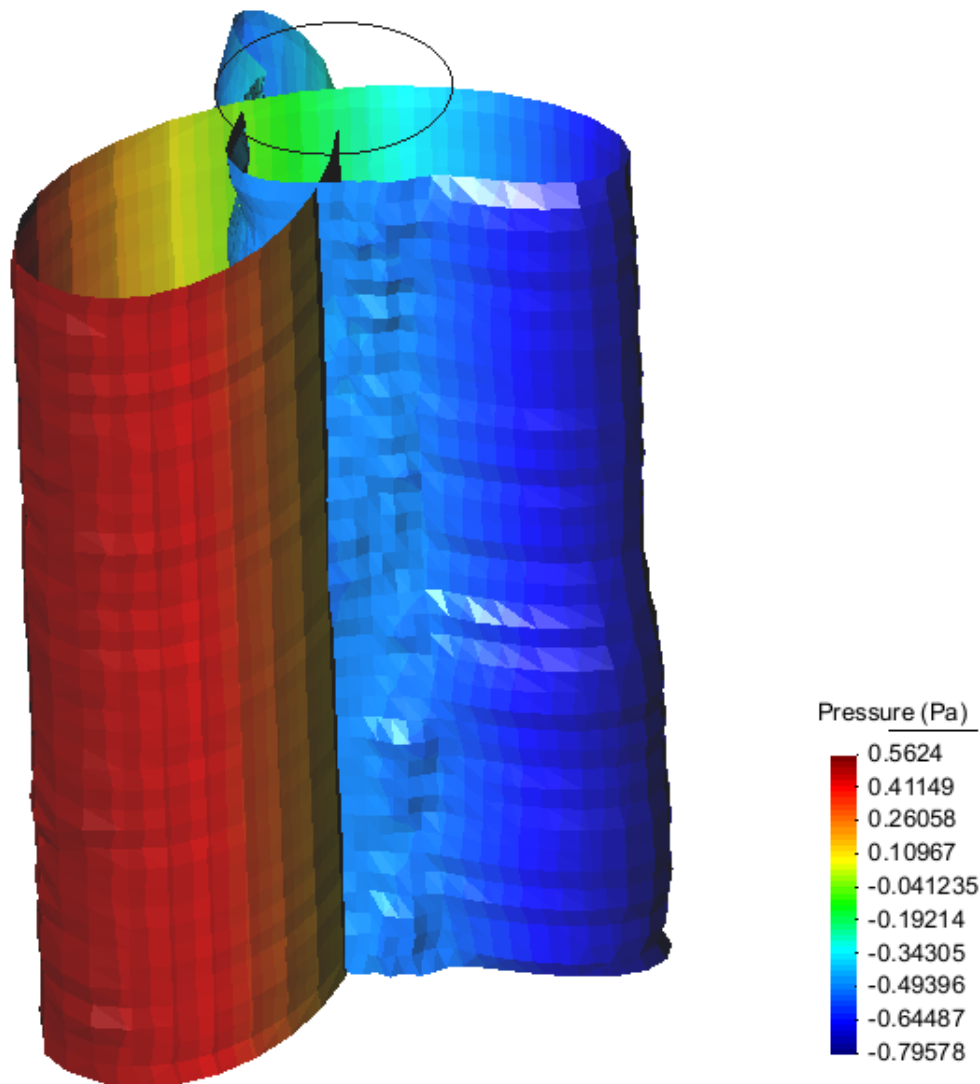
- 7 . Select AVI/mjpeg
(to include this animation in a MS PowerPoint an appropriate codec is needed like the one supplied with Combined Community Codec Pack, CCCP)
- 8 . Select a folder where the video will be saved clicking on the **folder** icon or writing the path in the text entry
- 9 . Click on the **play** button and the recording will begin. This step could take a little bit long. Wait until the red circle turns to green
- 10 . **Close** the Animate window

Now we will visualize another result but before we will clear all the results.



- 11 . Select **View results->No results** through the menu bar or using the icon 

8.3.3 Result surface



Another result visualization of interest is this one:



To get this visualization follow these steps:

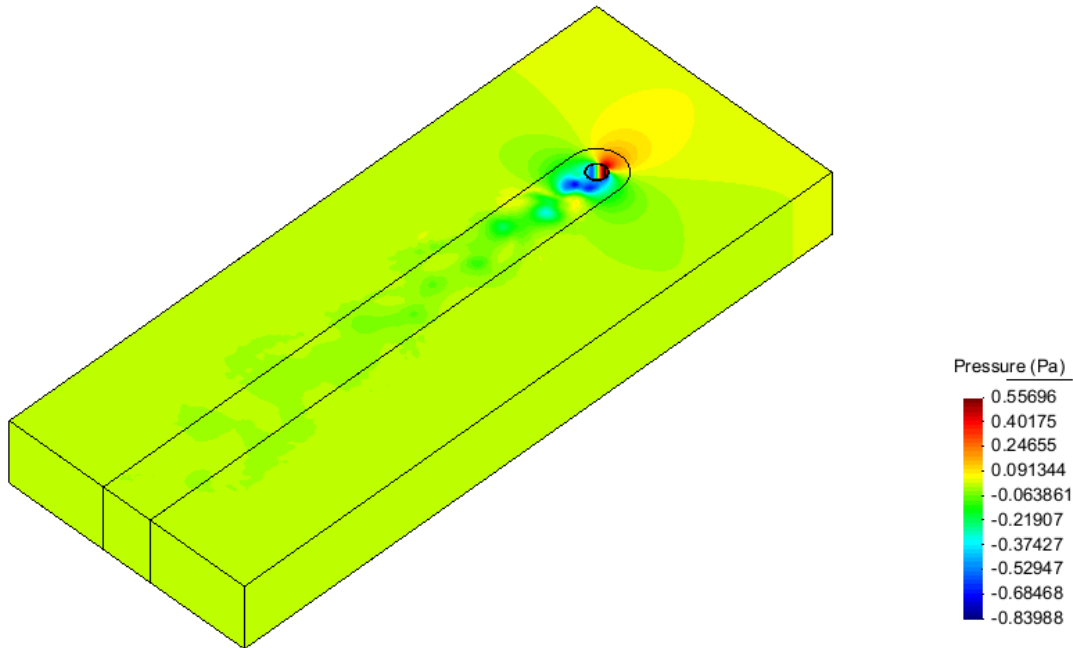
- 1 . Switch off all the sets except **S interior cil**. To do this:
 - Select **Window->View style...** in the menu bar.
 - Select all the sets except **S interior cil** pressing Ctrl while selecting with mouse.
 - Click on the **bulb light** icon  on the **I/O** column or click on the icon .
- 2 . Through the "View style" window change the Style to **Body Bound**.
- 3 . Select **View results->Result surface-> Pressure (Pa)**. A surface will be drawn which results from moving the nodes along its smoothed normal according to the results value for this node.
- 4 . Enter **5** as factor in the bottom command line.
- 5 . Select **Options->Result surface->Show elevations->None** .
- 6 . Select **Options->Result surface->Show elevations->Contour fill**. With this last option the surface is colored according to the pressure value.

Play with the other options as you will.


- 7 . Select **View results->No results** through the menu bar or using the icon 
- 8 . Switch on all the sets again through the "View style" window by selecting all sets and clicking on the  icon.

8.3.4 Contour fill, cuts and limits

Contour fill




Menu: View results->Contour Fill

- 1 . Please select **View results->Contour Fill->Pressure (Pa)** through the menu bar, or clicking on  or using the **Window->View results...** window.
- 2 If not all sets show the contour fill like the picture above, remember to select **BodyBoundary** mesh style for all the sets.

This option allows the visualization of coloured zones, in which a scalar variable or a component of a vector varies between two defined values. GiD can use as many colours as permitted by the graphical capabilities of the computer. The number of colours can be set through **Options->Contour->Number of colours**. A menu of the variables to be represented will be shown, and the one that is chosen will be displayed using the default analysis and step selected.

In the model the pressure has been calculated. We can visualize the result for each step in a contour fill.

You can choose the step that you want to view through the **View results** window or clicking on .

- 3 . Select the **step 103**

Several configuration options can be set via the Options menu.

Menu: Options->Contour

You can change the color scale in order to get a more comfortable view. You can select several predefined color scales. The default scale is *standard*, which is a rainbow colour map starting from blue (minimum) through green and yellow, to red (maximum).

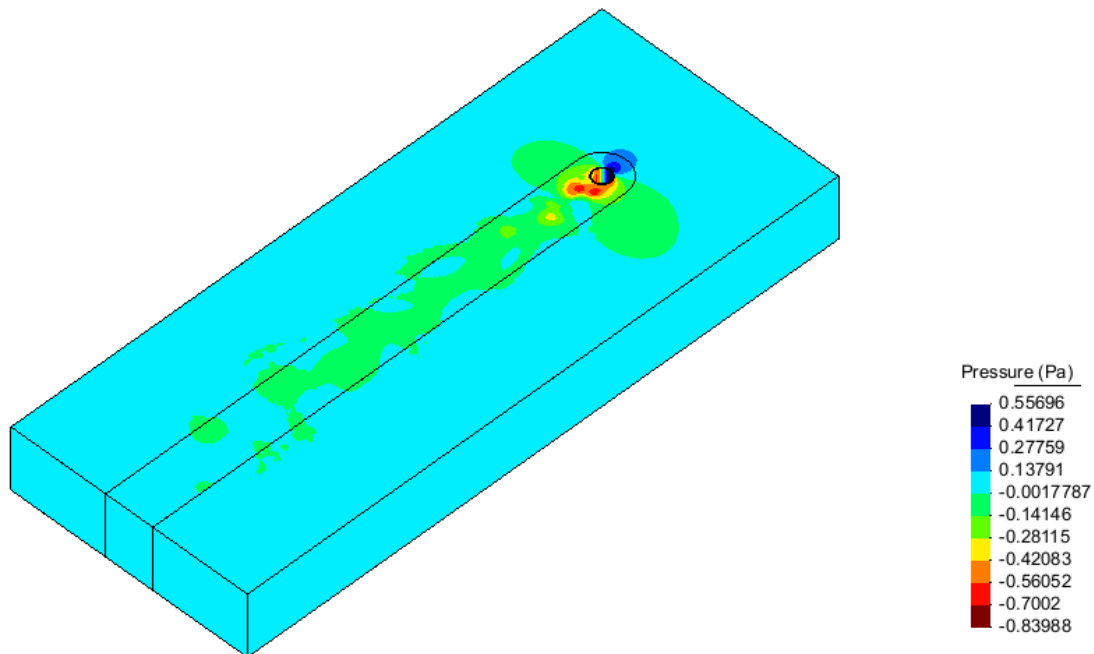
- 1 . Select **Options->Contour->Color Scale->Inverse Standard**

You can also define your own scale.

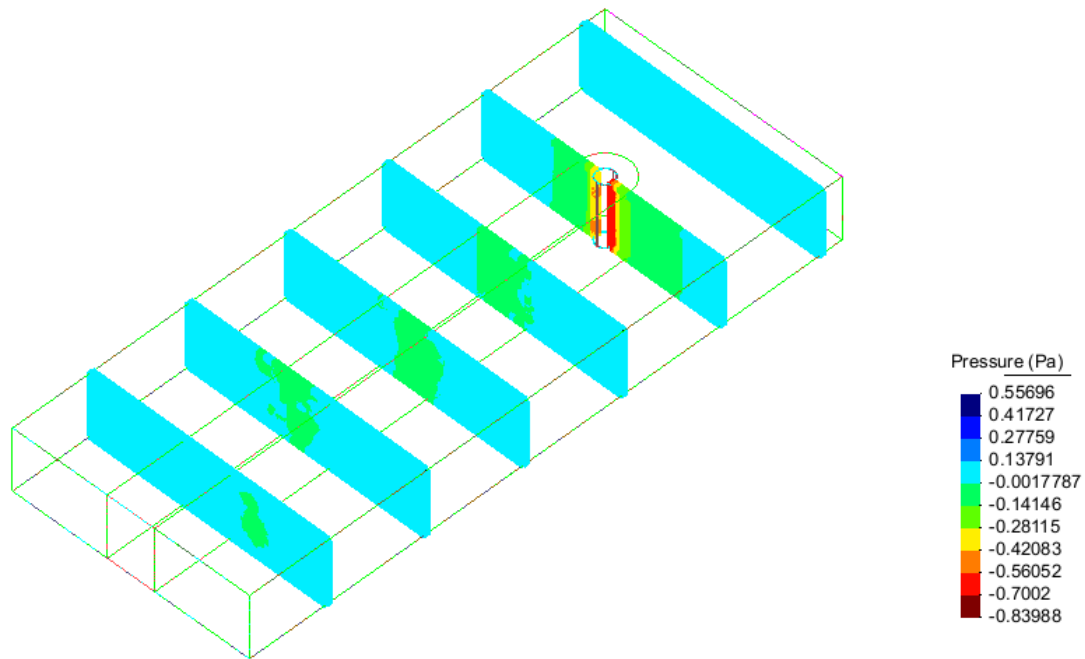
- 2 . Select **Options->Contour->Color scale->User defined...** or **Options->Contour->Color Window...**

In this window you can change the number of different colors used in the scale. If you need more accuracy you can increase this number, or decrease it for a higher contrast.

- 3 . Change the number of colors to **10**
- 4 . Click on **Apply** button
- 5 . Click on **Close** button





**Cuts**

In order to view the inner zone we will do several cuts along the model.



Menu: Do cuts

In order to make it easier first we will change the plane visualization.

- 1 . Please select **View->Rotate->Plane XY(Original)** through the menu bar, **Rotate->Plane XY(Original)** through the mouse menu or clicking on  and  .
Now you have a top view of the model.
- 2 . Select **Do cuts->Cut plane->Succession** through the menu bar or clicking on  and then .

With the **succession** option you specify an axis that will be used to create cut planes orthogonal to this axis. The number of planes is also asked for.


- 3 . Draw a line through the X axis in the middle of the model and ask for 7 cuts. You should obtain 7 parallel planes to Y axis.

Note: after clicking the first point, pressing the Alt key while moving the mouse the dynamic line will be axis aligned or at 45 degrees.

- 4 . Now change the display style (**Utilities->View style**) in order to see only the cuts. You can see that several layers have appeared a prefix like **CCutSetX** indicating which mesh or set has been cut. These names can always be changed through the **Window->View Style**. Select all the layers except the cuts and change their style to **Boundaries**. You can rotate the model in order to see the contour fill result on the cut planes.
- 5 . In the same window select all the **CCutSetX** and click on **Delete** button in order to delete all the cuts.
- 6 . Select **BodyBoundary** as mesh style to visualize the **contour fill** of **pressure** again.
- 7 . Select **Options->Contour->Reset all** in order to set all the defaults options.

Define limits

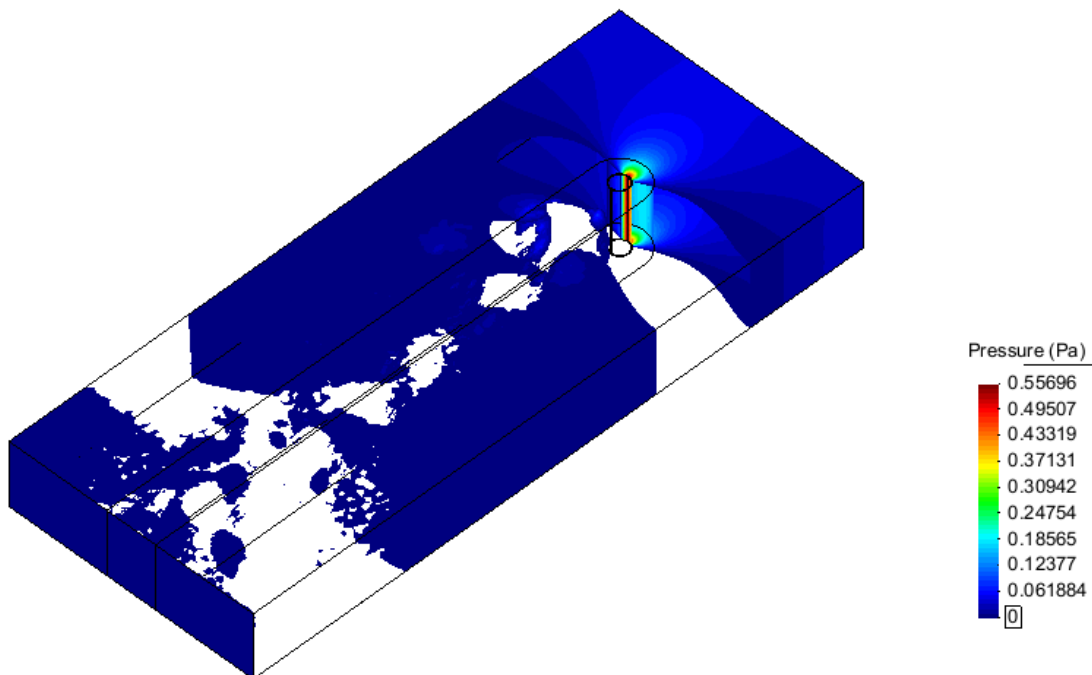
You can set the limit values for the contour fill. In our case we only want to see the positive values. In order to do this we will set the minimum value to 0.

1 . Select **Options->Contour->Define Limits...** through the menu bar or clicking on . Choosing the first option the Contour Limits window appears. With this window you can set the minimum/maximum value that Contour Fill should use.

- 2 . Check the Min checkbox
- 3 . Change the value to 0
- 4 . Click on the **Apply** button
- 5 . Click on the **Close** button

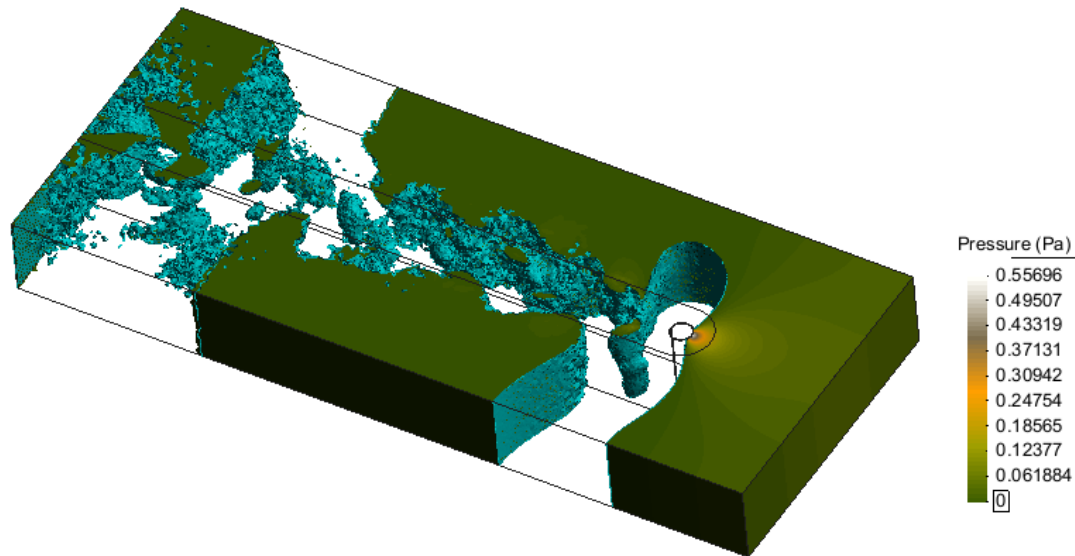
Outliers will be drawn in the colour defined in the Out Min Colour option. In order to view it better we will change this color to transparent.

- 6 . Select **Options->Contour->Min Options->Out Min Color->Transparent**




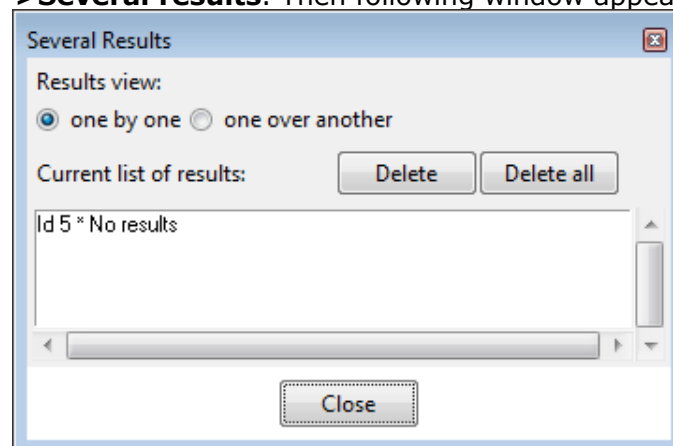
8.3.5 Combined results


An interesting postprocess options is to combine several result visualizations, like this one:



To get this view follow these steps:

- 1 . Clear all results visualizations with **View Results->No results** or the icon 
- 2 . Select **Window->Several results**. Then following window appears:

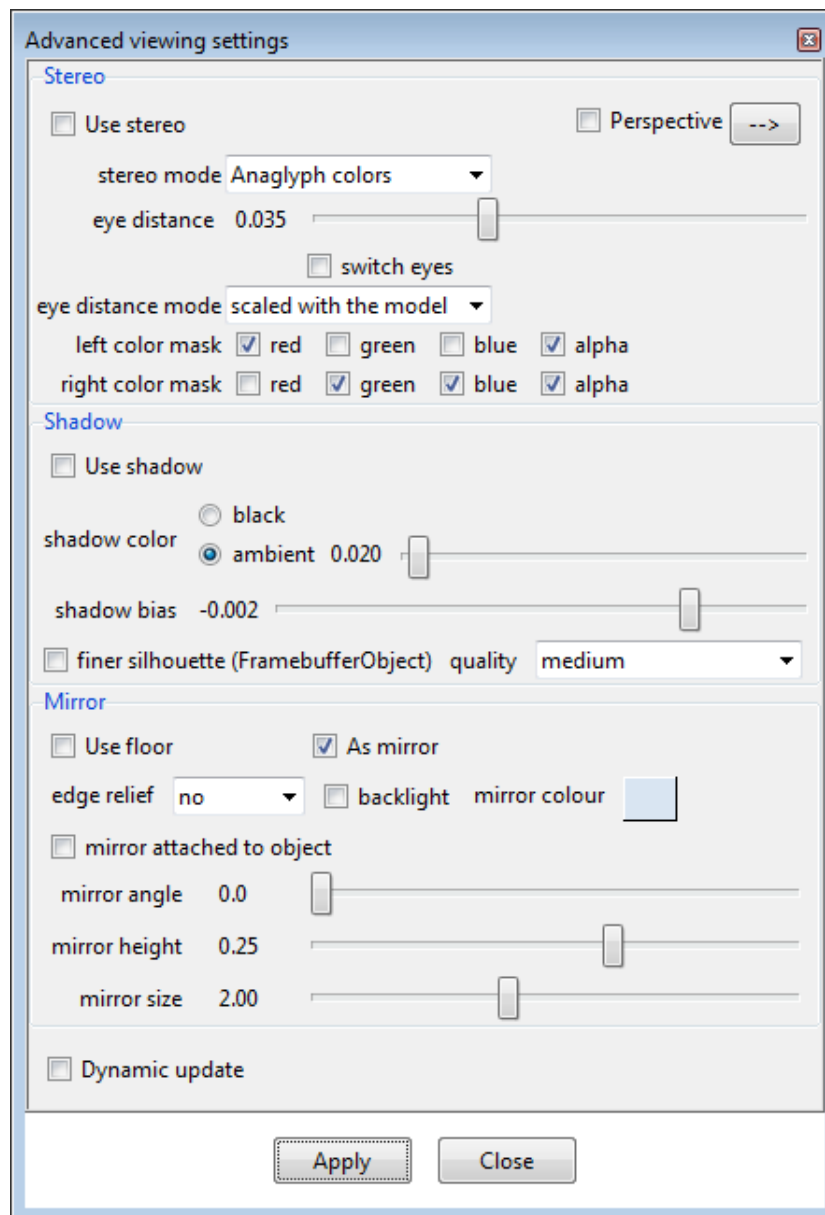


- 3 . In this window select **one over another**. With this option GiD is told to visualize one result over another
- 4 . Select **View Results->Default Analysis/Step->Ransol->103**
- 5 . Select **View Results->Iso surfaces->Exact->Pressure** through the menu bar or clicking on the 
- 6 . In the following questions: How many **isosurfaces**? Enter 1
- 7 . Enter the 1 value ...? Enter 0
- 8 . Select **View Results->Contour Fill->Pressure**
- 9 . Set the **minimum** value to 0
- 10 . Select **Options->Contour->Min options->Out min color->Transparent**
- 11 . Select **Options->Contour->Color scale->Terrain Map**
- 12 . Select **Options->Iso surface -> Color mode -> Monochrome**
- 13 . Select **Options->Iso surface -> Change color** to change the color of the iso

surface.

Note: On newer version of GiD, step 2 and step 3 is not needed.

8.3.6 Stereo mode (3D)



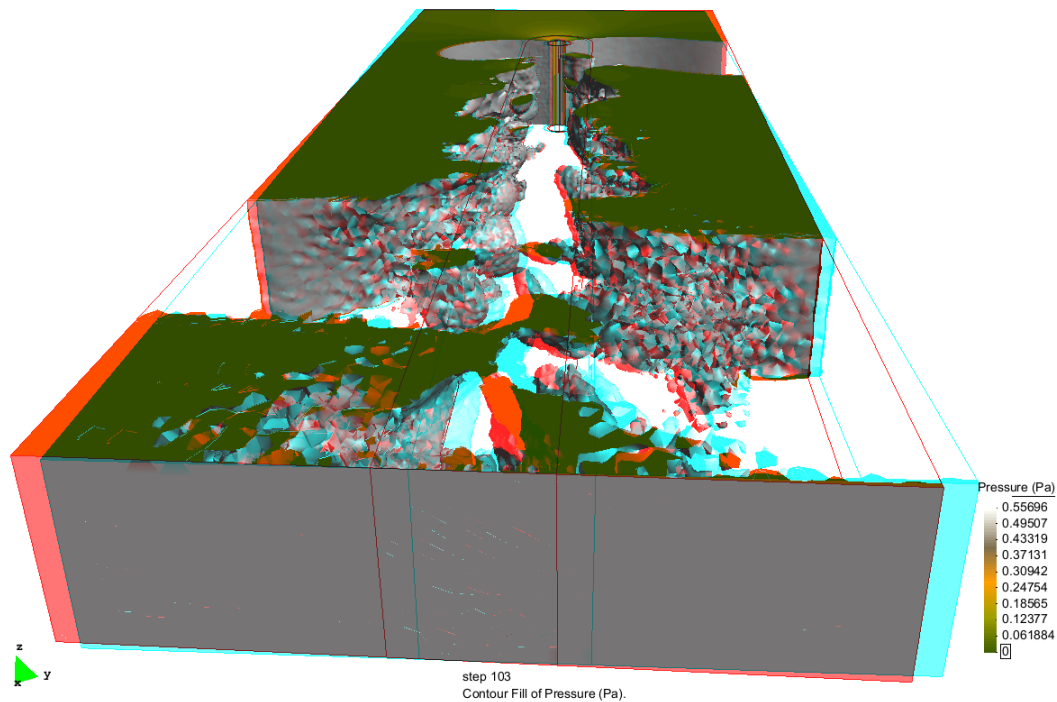
Menu: View->Advanced viewing settings...

If you have an **anaglyphic** glasses you can try this option. The model can be set as an anaglyphic image in order to provide a stereoscopic 3D effect, when viewed with 2 color glasses (each lens a chromatically opposite color, usually red and cyan).

Anaglyphic images are made up of two color layers, superimposed. Since the glasses act as red and cyan filters we should be careful with the model's colors. To avoid problems we will change the contour fill color scale.

- 1 . Select **Options->Contour->Color Scale->3D Anaglyphs**
- 2 . Select **View->Advanced viewing settings...**
- 3 . Check the **Use stereo** option

- 4 . Check the **Dynamic update** option in order to change the options without the need to click the Apply button
- 5 . Set the eye distance to the value where you can see the 3D effect
- 6 . Uncheck the **Use stereo** option
- 7 . **Close** the window
- 8 . Select **View results->No Results**
- 9 . Change the **view style** to boundaries for all the layers, like in **Changing style** chapter

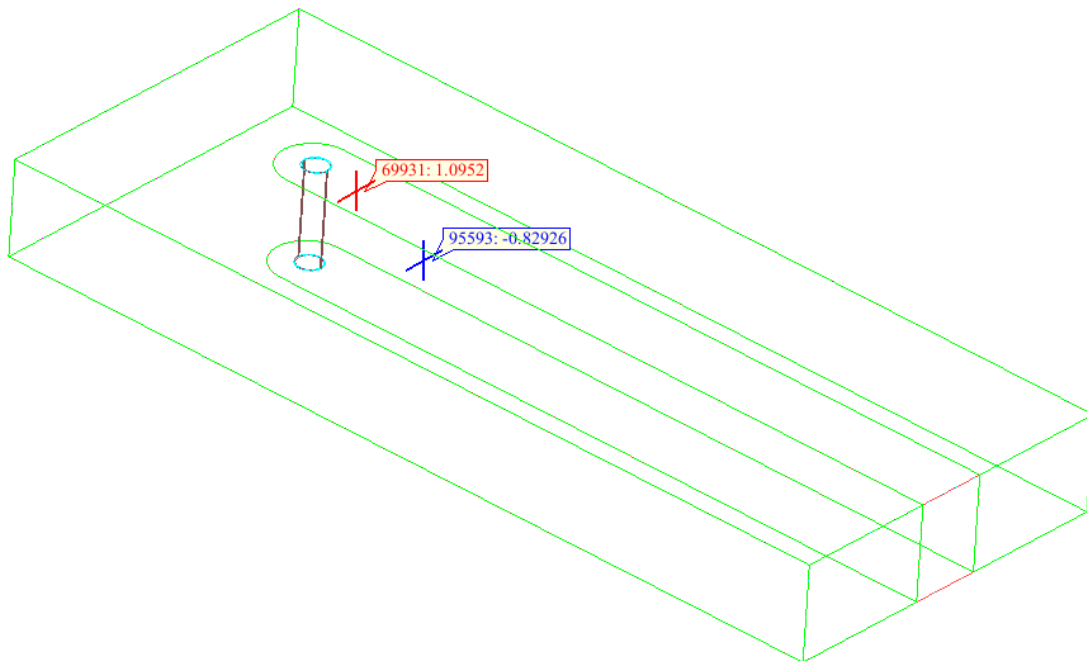


8.3.7 Show Min Max

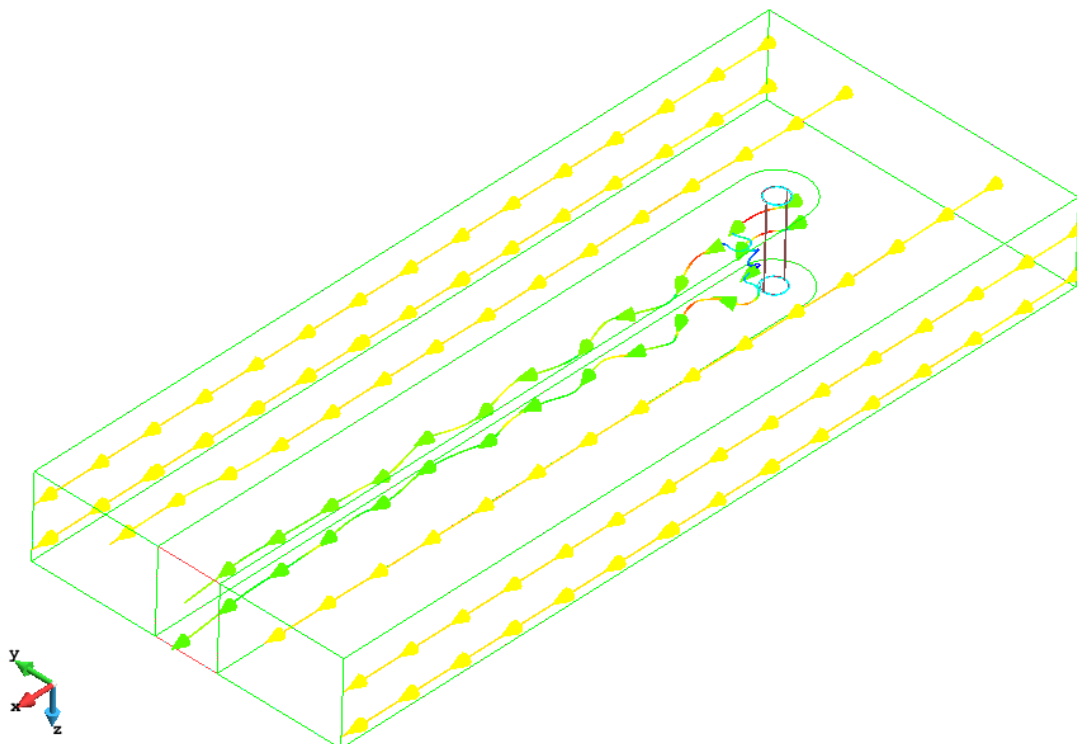
Menu: View results->Show Min Max

With this option you can see the minimum and maximum value of the chosen result in the chosen analysis step. In our case we will choose the V_y component of velocity result for the first analysis step.

- 1 . Select **View results->Default Analysis/Step->RANSOL->91.5** through the menu bar or clicking on
- 2 . Select **View results->Show both->Velocity (m/s)-> V_y** through the menu bar or clicking on . The label shows the node number and the value of the result.
- 3 . Select **View results->No Results**



8.3.8 Stream lines



Menu: View results->Stream Lines

With this option you can display a stream line, or in fluid dynamics, a particle tracing, in a vector field.


Note: stream lines are confined in a single volume mesh, i.e. they do not jump from one

volume mesh to the next volume mesh, even if they are close neighbours. In the provided example there are three volume meshes and stream lines will not cross the volume boundaries. You can join the volume meshes into a single volume mesh using **Utilities --> Join --> Volume sets**. Then you can delete the three separate volumes and switch the single joined volume mesh on.

The above image results from doing this tutorial with the three separated volumes.


The image at the end of this *stream lines* tutorial is achieved if following step is done before the enumerated *stream lines* tutorial steps.

- Select **Utilities --> Join --> Volume sets** to create a single volume mesh, and delete the three other volume meshes: *V volumes*, *V cil* and *V wake*. (The above image results from

- 1 . Select **View results->Default Analysis/Step->RANSOL->103.0** through the menu bar or clicking on 
- 2 . Select **View results->Stream Lines->Along line->Velocity (m/s)** through the menu bar

With this option you can define a segment along which several start points will be chosen. The number of points will also be asked for, including the ends of the segment. In the case of just one start point, this will be the center of the segment.



NOTE: This action could also be done clicking on  in the icon bar. In this case we have to select the way to define the start point through the mouse menu. In this case select **Contextual->Along line**.

We want to create several stream lines along the model doing 2 lines.

- 3 . Write the **initial** point in the command line 10 15 3
- 4 . Write the **final** point in the command line 10 -15 3
- 5 . You are asked for the **number of points** along the line. Enter **5** and click **Ok**.

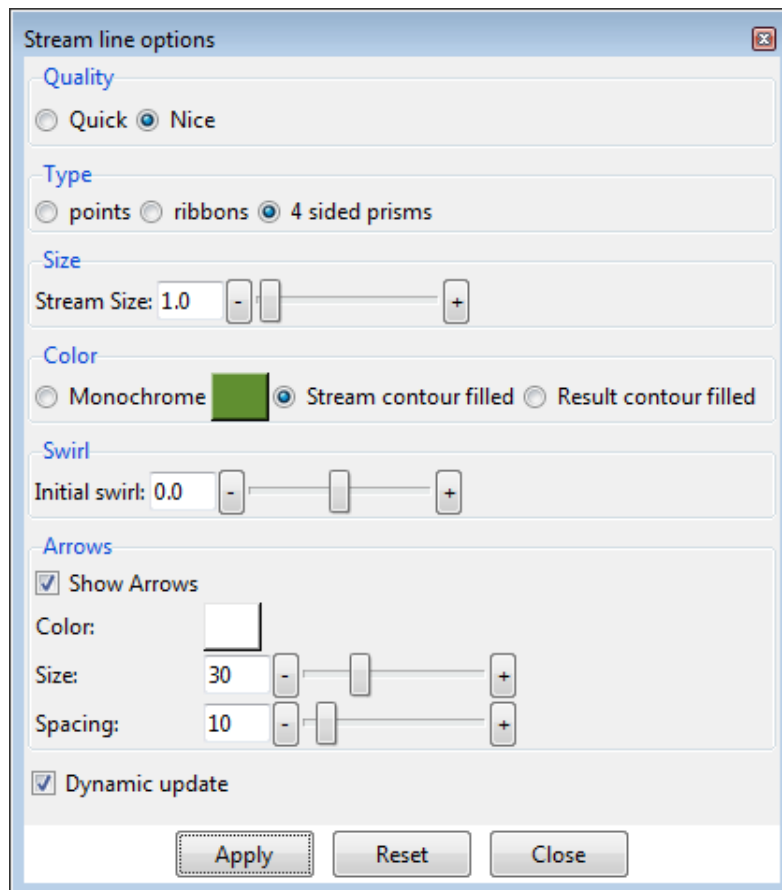
Note: (You can also press Ctrl-t to set the cursor in the command line)

The first line with 5 stream lines is created.

- 6 . Write the **initial** point in the command line 10 15 7
- 7 . Write the **final** point in the command line 10 -15 7
- 8 . You are asked for the **number of points** along the line. Choose 3.

The second line with 3 stream lines is created.

- 9 . Click the middle mouse button or press the **Esc** key in order to finish the operation.



Several configuration options can be set via the Options menu.

Menu: Options->Stream lines

The options can be also managed through the **Size & detail** window.

10 . Select **Options->Contour->Color Scale->Standard**

11 . Select **Options->Stream lines->Size & detail...**

12 . Check the **Dynamic update** option

13 . Select **Stream contour filled**

The stream lines will be drawn with the colors used in the velocity contour fill.

14 . In the **Arrows** options, set 30 for the **Size** option

15 . Set 10 for the **Spacing** option

16 . Check the **Show Arrows** option

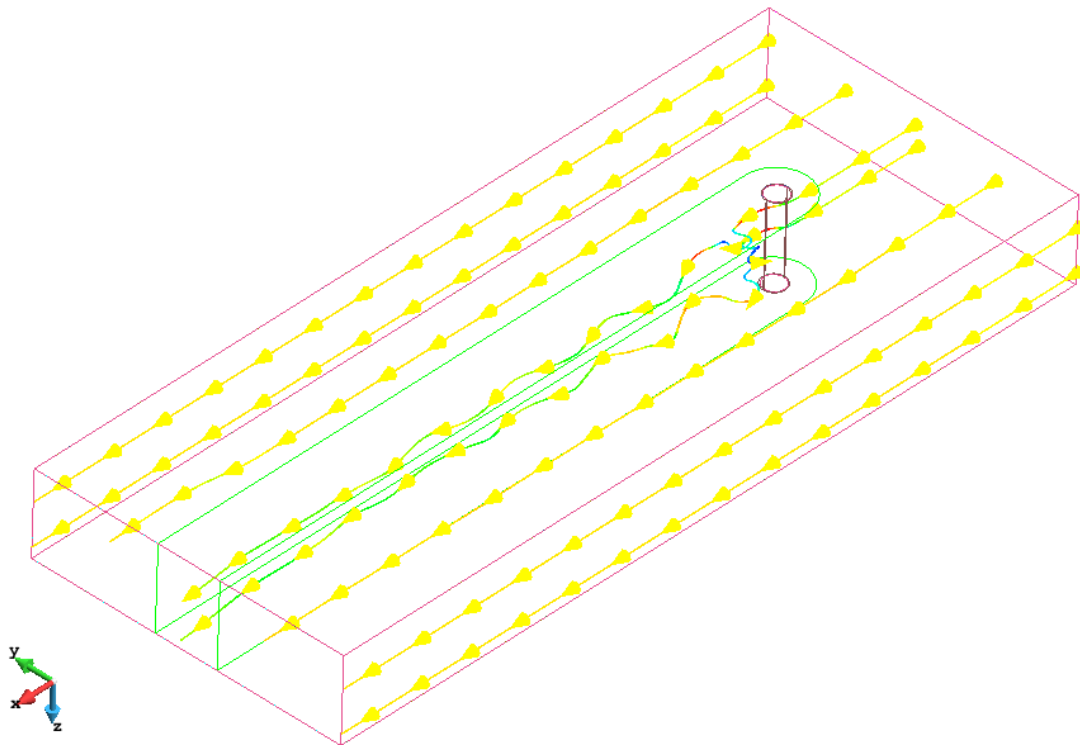
17 . You may play with the different stream types: points, ribbons or 4 sided prisms. If the ribbons type is selected you may adjust the initial swirl to rotate the ribbon.

18 . Close the window

19 . Select **Options->Stream lines->Delete all**



NOTE: A way to achieve the best results is to first create a cut of the volume mesh through the *region of interest* and then use these nodal information as support to create *stream lines* and its options: *along line*, *in a quad*, etc.



8.3.9 Graphs

Menu: View results->Graphs

From this menu several graphs types can be created, we will try some of them. Graphs are supported for results defined over nodes.

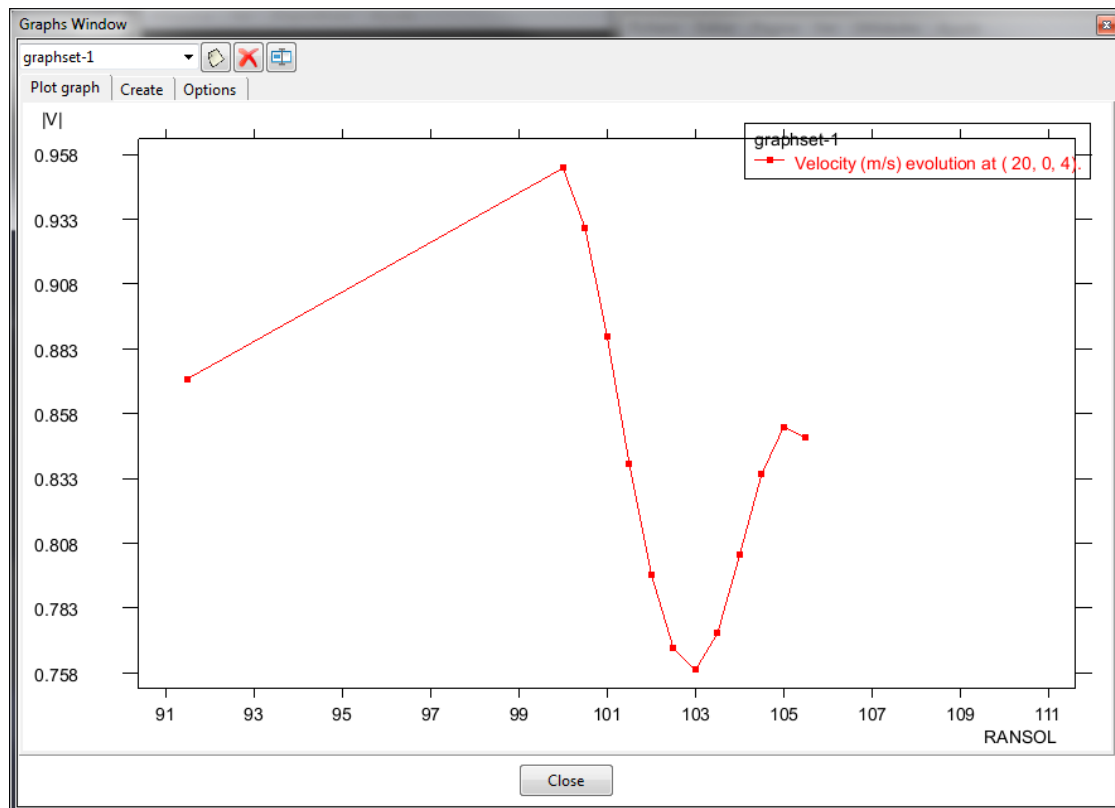
Graphs are organized into **graph sets** in order to ease the management. Each set shares the same units for each axis.

When a graph is created is placed in the current graphset if the units are the same, otherwise a new graphset is created.

In order to work with graphs we will use the 'graphs window'.

The **Point evolution** graph displays a graph of the evolution of the selected result along all the steps, of the default analysis, for the selected nodes.

- 1 . Select **View results->Graphs->Point evolution->Velocity(m/s)->|V|**
- 2 . Write 20 0 4 in the command line in order to specify the point.
- 3 . After pressing the **Escape** key, or the middle mouse button, the graph will be shown in a separate window:

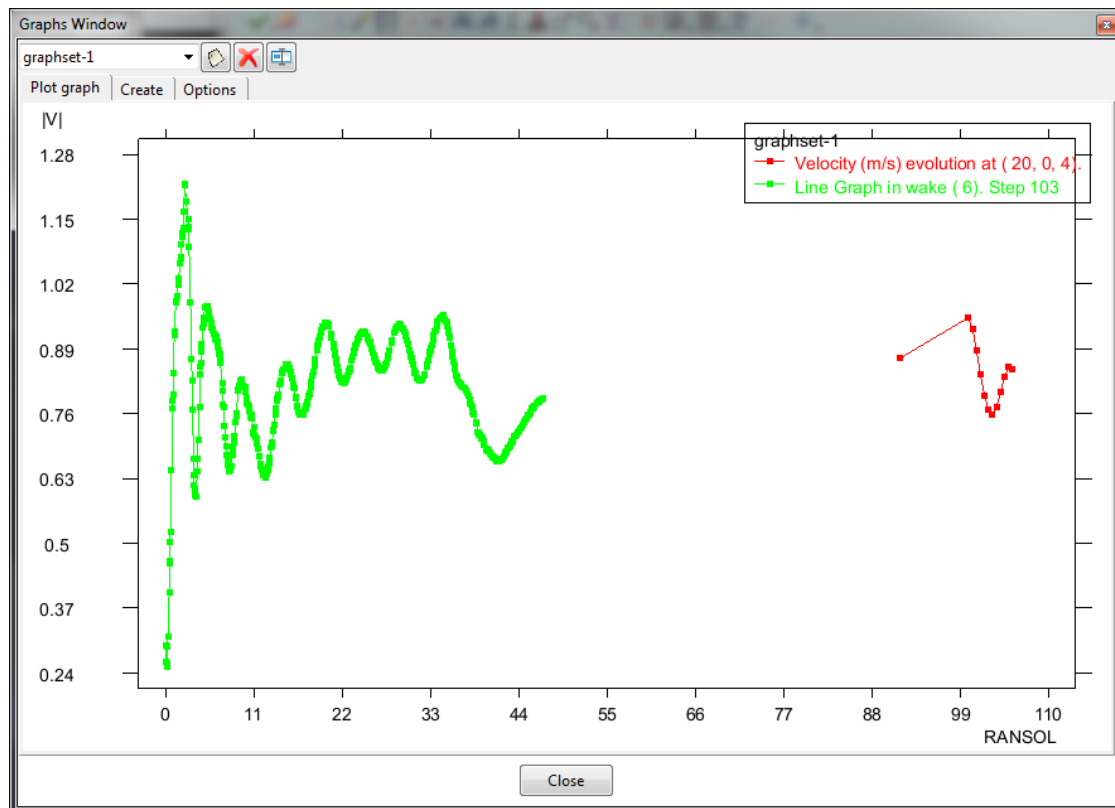


The graph is created in the graphset-1. We will create another graph in the same graph set.


The **Line graph** displays a graph defined by the line connecting two selected nodes of surfaces or volumes, or any arbitrary points on any projectable surface and in any position.

- 4 . Switch all surface meshes off, and let only the three volume meshes on: V volumes, V cil, V wake.
- 5 . Select **View results->Graphs->Line graph->Velocity(m/s)->|V|**
- 6 . Write 3 0 4 in the command line in order to specify the initial point.
- 7 . Write 50 0 4 in the command line in order to specify the final point.

Now both graphs are showed in the same graph set:




We will rename the graph set.

8 . In the top part of the window click the  icon.

9 . A window will appear asking for a new name. Enter 'Velocity', for example.

We will create a new graph set.

10 . In the top part of the window click the  icon.

A new graph set is created with default name 'graphset-1'. When a new graph set is created becomes the current one. We can see that there are no graphs on this new graph set.

It's also possible to create graphs from the graph window.

11 . Go to **Create** tab and select **Point evolution** int **View** option.

12 . In **Y Axis** list double click **Pressure (Pa)**.

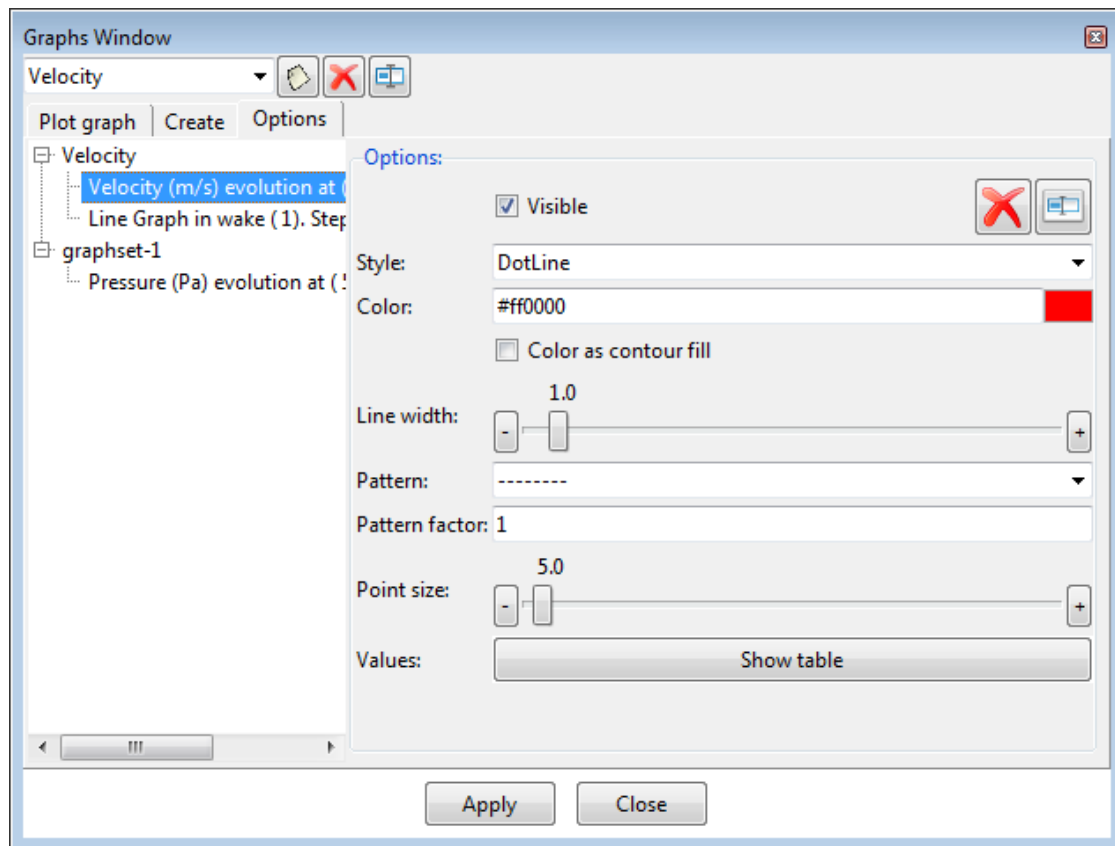
13 . Write 50 0 0 in the command line in order to specify the point.

14 . Press **Escape** to finish the graph.

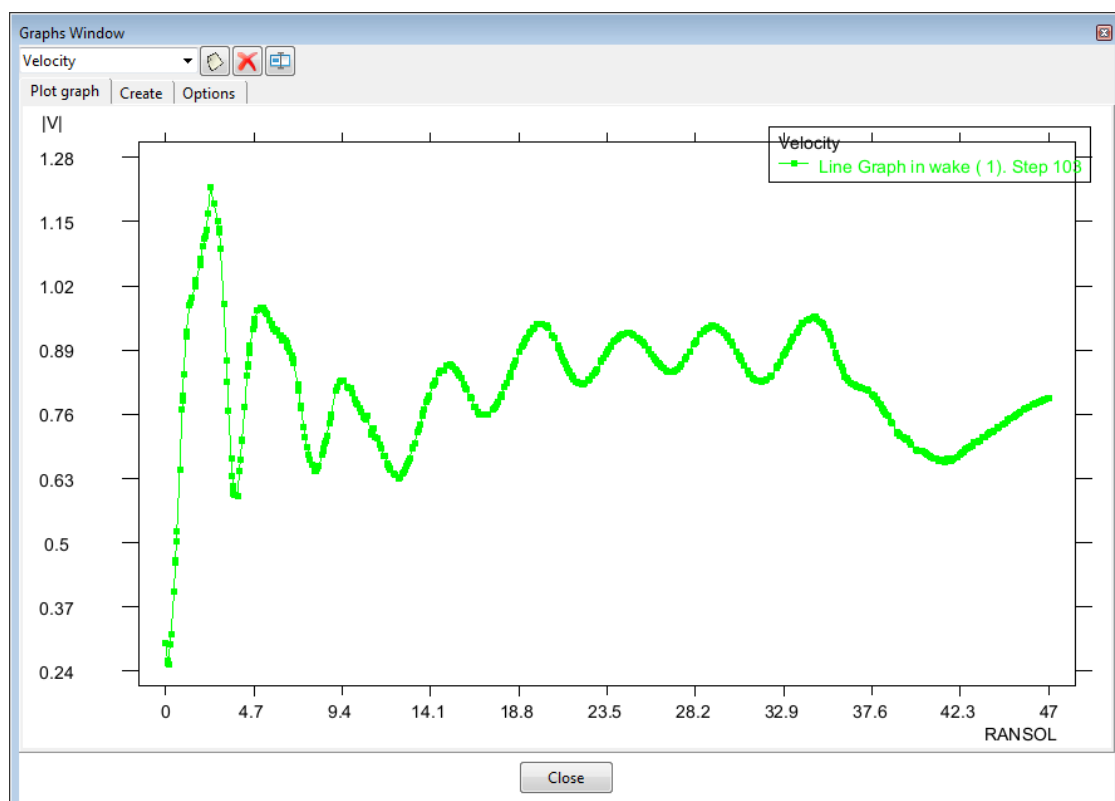
We can manage graphs and graphs sets in the Options panel. Depending if we are selecting a graph set or a graph in the tree we will see different options in the tab.

15 . Go to the **Options** panel, select the 'Velocity (m/s) evolution at (20, 0, 4)' graph and delete it pressing the button with the red cross.

16 . A confirmation window appears. Click **Yes**.



17 . Please notice that the current graph set have been changed to 'Velocity'. Now the Plot graph panel will show only one graph:

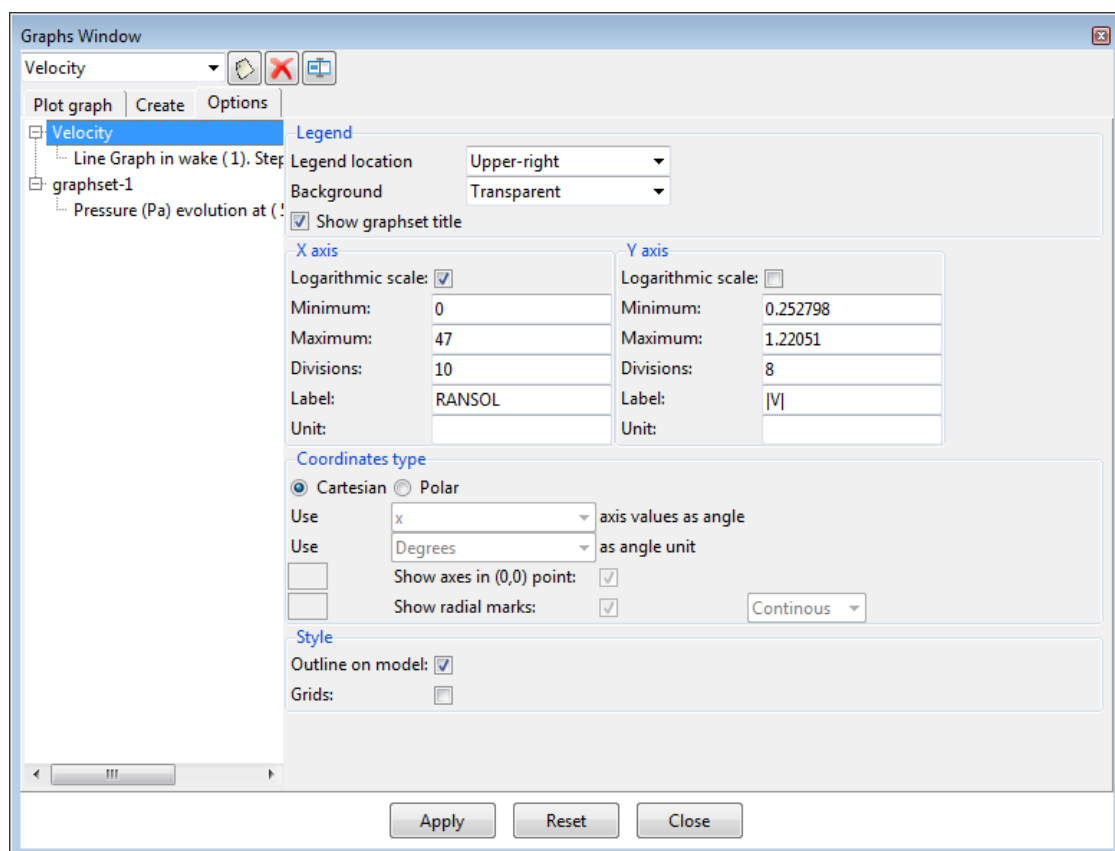


The graph size is readapted. We can will change several style options of a graph.

- 18 . **Double click** in any point of the graph and we will access to the **Options** tab.
- 19 . Choose **Line** in the **Style** option
- 20 . Set to red the **Color** option. You can do it writing #ff0000 or selecting the red clicking on the right color window
- 21 . Set to 4.0 the **Line width**
- 22 . Click on **Apply** button

Graph sets options can be managed selecting the set in the tree.

- 23 . Select 'Velocity' branch. The options will change.
- 24 . For instance mark 'Logarithmic scale' option in X axis.
- 25 . Click on **Apply** button



We can export the graph information in order to open it later with GiD.

- 26 . Select **Files->Export->Graph->All**. You are asked for the location where to save the .grf file.
 - 27 . Choose the location
- Now you can import the selecting **Files->Import->Graph**

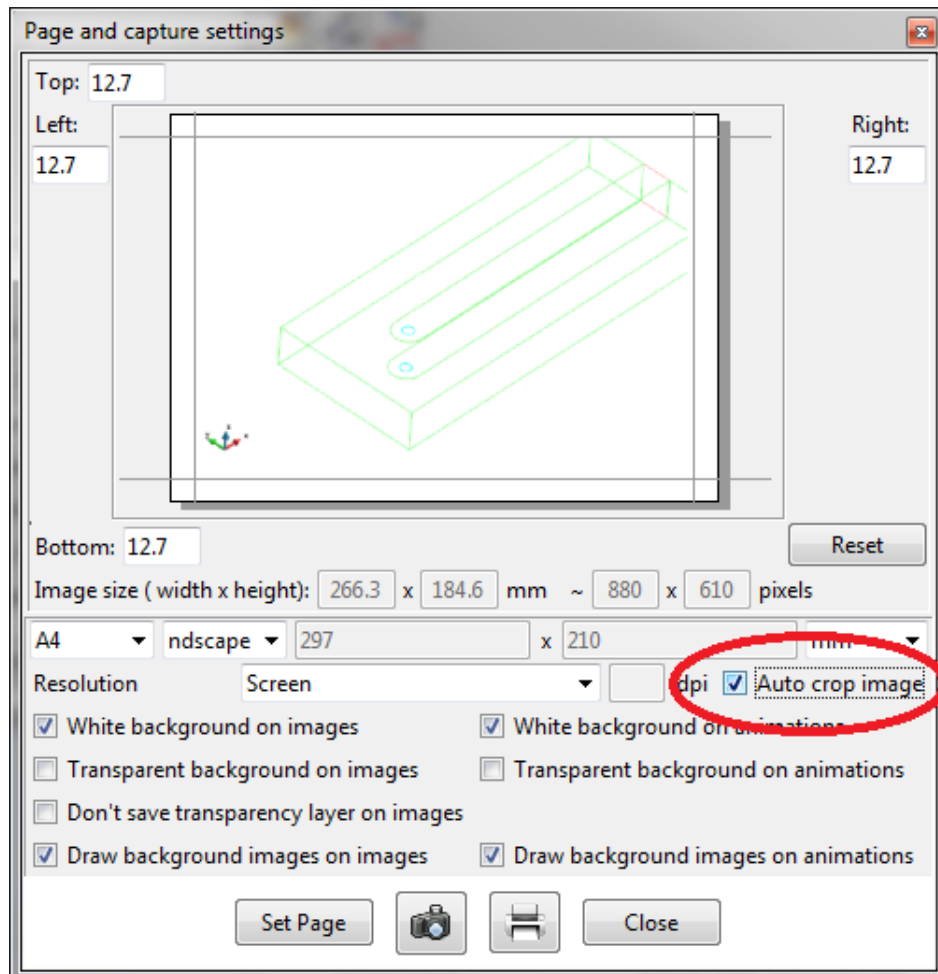
- 28 . Select **Options->Graphs->Clear graphs** in order to delete all the graphs

8.4 Creating images

Menu:Files->Page and capture settings...

Finally we will take some snapshots of our model. You can save images in several formats. The properties of the image (resolution, size, etc.) can be assigned in Page and capture settings option.

- 1 . Select **Files->Page and capture settings...**
- 2 . Check the **Auto crop image** option in order to cut the image in the model limits
- 3 . Click on **Set Page** button
- 4 . Click on **Close** button




Menu:Files->Print to file

This option asks you for a file name and saves an image in the required format with the defined properties in **Page and capture settings**.

- 1 . Select **Files->Print to file->PNG...** through the menu bar
- 2 . Choose the location where you want to save the image
- 3 . Choose a name for the file
- 4 . Click on **Save** button



NOTE: This action could also be done by clicking on  through the icon bar. This icon can also be found in the "Page and capture settings" window. In this case the image format is chosen while saving the file in the "Files of type" combobox.