GiD

The universal, adaptative and user
friendly pre and post processing
system for computer analysis
in science and engineering

GiD basic course 2014

1 Introduction

This courses are prepared to be followed using the version 12.0 of GiD.
They are divided in GiD basic courses and GiD advanced courses. We strongly recommend to make the GiD basic courses before the advanced ones.

1.1 Installing GiD

To install GiD go to the GiD Convention USB unit, from now on we will assume it is 'D:', if you have auto-run function active, the file index.html will be opened automatically, if not, please double click on 'D:\index.html'.

Choose from the installer link **Windows 32** or **Linux 32**.

Click on the option corresponding to your OS (Windows or Linux), and then follow the instructions of GiD installer to install GiD into your computer.

Even if your OS is x64 bits, we will use for this course the **x32 bits version of GiD**, because some calculation module we will use later on is only available for x32 bits (dynamic libraries loaded in GiD must be compiled for the same platform of GiD).
Note: A x64 bits OS can run applications of x64 and x32 bits.

On web page www.gidhome.com you can find two types of GiD versions:

- **Official versions of GiD** are stable versions of the program. They don't include the newest capabilities but they have passed all the validation tests.
- **Developer versions of GiD** are more modern versions of the program that have new capabilities. They have not passed the validation tests that a official version does. So, use them with care. Only download and install one of these versions if you know of a new capability very important for your needs or if you want to try the new improvements in the program.

1.2 Registering GiD

GiD can work with no license (unregistered), but in this way user can only manage a models with a few number of nodes (about 1000)

In case you want to try using a mesh with a higher number of nodes, a free password for one month can be downloaded from the web site (or a permanent one buying a licence)

- Password for GiD: http://www.gidhome.com/purchase/password
- Password for Compass problemtypes: http://www.compassis.com/compass/en/Passwords

NOTE: The USB memory sticks of the course are already registed with a one year password of GiD 12 (USB passwords is valid for both Windows and Linux platforms)

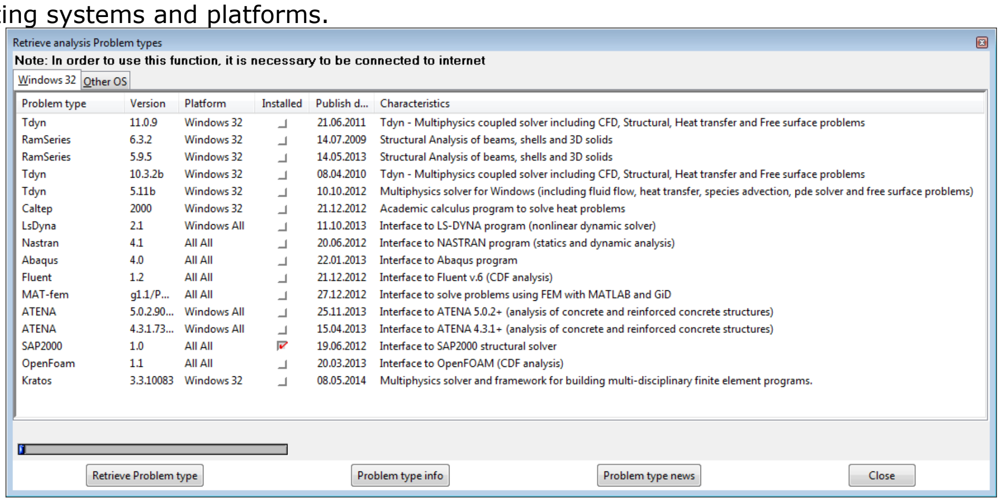
1.3 Installing Problemtypes

Following GiD terminology, a 'problem type' is a calculation module able to perform a simulation and which customizes GiD so as the user can apply to the model the material properties, boundary conditions and other information needed for the simulation process.

After the GiD installation process, it is needed to add the required 'problem types', to do is usually is only needed to copy the problemtype folder inside the \problemtypes gid folder.

It is possible to manually copy these problemtypes (e.g. from the USB GiD\problemtypes folder to our current GiD) or dowload them from Internet

- If you don't have internet connection, you can simply copy the files from other location.
- If you have internet connection, start GiD and go to menu: **Data->Problem type->Internet retrieve**. A window with the available problem type modules will appear, splitted in the different operating systems and platforms.



There is a mark on the currently installed problemtypes.
To get some new ones, select them and click "Retrieve Problem type" to install them.

Once the problemtype has been downloaded, close the window and you can check it is present in the list of problem types installed in the **Data->Problem types** menu.

NOTE: The basic course requires for the 'Run a CFD simulation' chapter to install the CompassFEM (Tdyn) problemtype, that is copied inside the GiD\problemtypes folder of the memory stick (only the Windows x32 version). The example uses a coarse mesh that doesn't require any CompassFEM password.

1.4 Material location

The PDF documents and most of the models of the basic and advanced courses can be found in the GiD USB memory stick.
The same PDF documents and all models used in both courses can also be found at ftp://www.gidhome.com/pub/GiD_Convention/2014, inside the corresponding folders.

Table of Contents

1 Basic GiD management

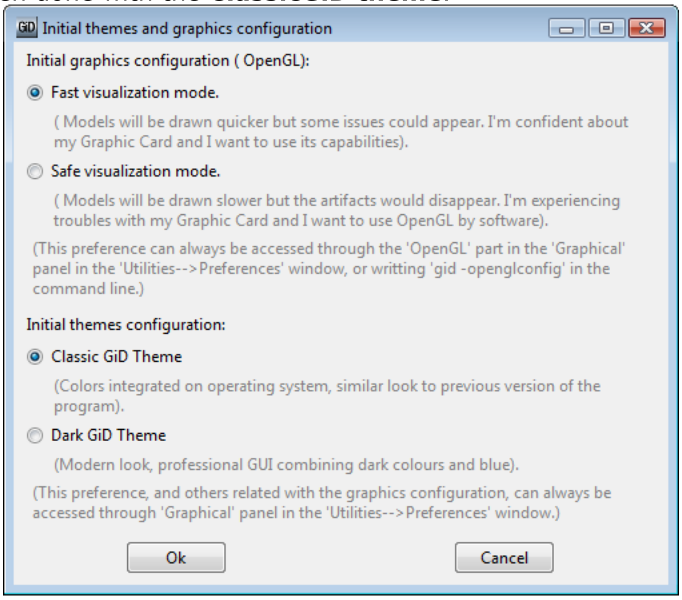
The philosophy of this course is to get familiarized with GiD: how to change the views of the model, how to manage the Layers, and other basic features. Some of this features are both in the preprocessing and the postprocessing parts of GiD, although the examples shown are from the preprocessing one.

Many times the text will make reference to *'entities'.* Almost all the options explained in this course are valid both for geometrical and mesh entities, although the examples used are often geometrical ones.

1.1 User interface

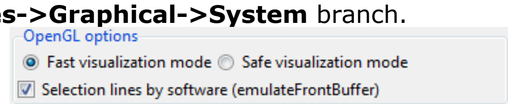
1.1.1 Starting GiD

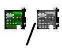
When GiD is started for the first time a pop-up appears where you can choose OpenGL working mode and GiD theme.
If you are confident with your graphic card please choose **Fast visualization mode**. With this mode the model will be drawn quicker but some issues could appear.
These courses have been done with the **ClassicGiD theme**.



These two options can be changed later in the preferences window (select **Utilities->Preferences**).

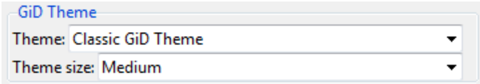
Change OpenGL option
This option can be changed in GiD Theme option
inside **Utilities->Preferences->Graphical->System** branch.



Also it's possible to change it by clicking on  in the bottom right corner of GiD window.

Change theme
User can choose between Classic and Dark themes, which change drastically the GUI appearance. User can also choose between some icon sizes in each theme.

These options can be changed in **GiD Theme** option inside **Utilities->Preferences->Graphical->Appearance** branch.



1.1.2 Preprocess and Postprocess modes

GiD basically works in two modes: preprocessing and postprocessing.
To change between both modes please select **Files->Postprocess** or **Files->Preprocess** (or clicking in the upper toolbar).

1.1.3 Warnline

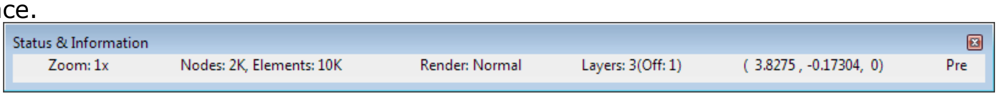
In some of the operations made in GiD by the user, GiD gives information about what is expected to do by the user. This information is very useful the first times GiD is used as a guideline for the user. The place were GiD shows this kind of information is the lower part of its main window.

1.1.4 Command line

Using GiD, sometimes the user is asked to introduce data with the keyboard. The 'Command line' must be used for this purpose. It is placed in the lower part of GiD window.

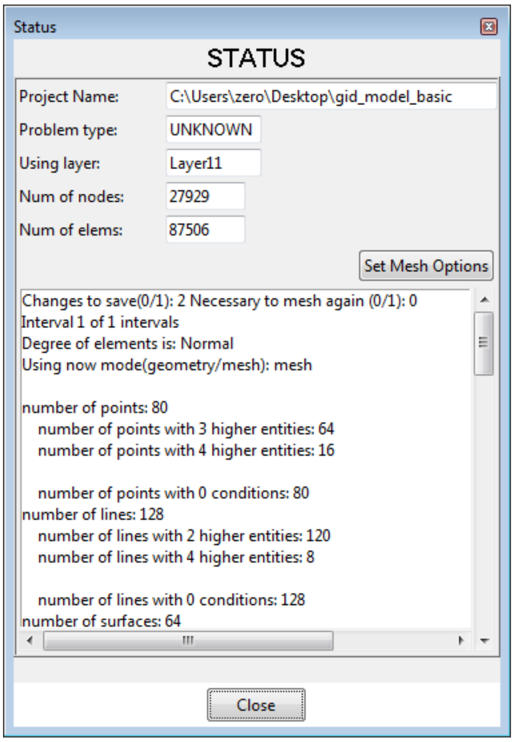
1.1.5 Status bar

The Status & Information bar located at lower part of GiD's Window, provides basic information at a quick glance.



From left to right you can find:

- Zoom factor
- Current number of nodes and elements (Click to acces to Status Window)

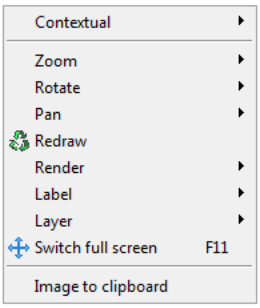


- Current renter mode (Click to change render)
- Number of layers in Pre, number of sets in Post
- Mouse coordinates (Click to open "Coordinate window" in Pre and "Change result units" in Post)
- Current Mode: Pre or Post

1.1.6 Mouse menu

Clicking the right mouse button on the main graphical window opens an on-screen menu with some visualization options. To select one of them, use the left or right mouse button; to quit, left-click anywhere outside the menu.

The first option in this menu is called **Contextual**. You can select from different options relevant to the function currently being used, e.g. when asking for a point they appear options like "Point in line", to select a point over a line, or "Arc center" to select the coordinates of the center of an arc.



- **Zoom, Rotate, Pan, Redraw** and **Render** are the same options as in pre and postprocess mode.
- **Label** in preprocess shows a label with the current entity number, depending if we are in preprocess, geometry and mesh, or in postprocess.
- **Layer/Mesh** (depending if we are in pre or postprocess mode) menu lets user to switch on/off layers.
- **Switch full screen** changes GiD main window to full screen mode.

- **Image to clipboard** creates an image fo the current visualization and placed in the clipboard.

1.1.7 Escape function

An important thing a GiD user should know as a general philosophy of use of the program is the **Escape** key functionality: In almost all the actions performed by the user, to declare the action as done the user should press **Escape** key (or press the center mouse button).

1.2 Change views of the model

In the **View** menu user can find the options to change the point of view in which the model is shown. Many of these options are also accessible by the right mouse button menu, or the icons toolbar.

1.2.1 Zoom

To zoom in or out the model user can choose the corresponding options in the **Zoom** section of the **View** menu or the right mouse button menu.
A user friendly way of zooming the model is to use the wheel of the mouse, or clicking the center button of the mouse while the **Shift** key is pressed.
To get a view which includes the whole model the **Frame** option must be selected.

The icons corresponding the zoom operations are the following ones:
Zoom in:
Zoom out:
Zoom frame:

1.2.2 Pan

To move the view of the model user must select the option **Pan**. This option is accessible from the **View** menu, the right mouse button menu, or moving the mouse while the **Shift** key and the right mouse button are pressed.
The corresponding icon for the pan option is the following one:

1.2.3 Rotate

In **View->Rotate** menu (also present in the right mouse button menu) there are the options to rotate the view of the model.
A user friendly way of rotating is to move the mouse while its **left button** and the **Shift** key are pressed.
The corresponding icon for rotating the model is the following one:

1.2.3.1 Set center of rotation

An interesting option for rotating the view of the model is to set the center of rotation. To change it:

1 . Select **View->Rotate->Center** from top menu or **Rotate->Center** from right button mouse menu. Then, the cursor changes into the selection mode.
2 . Select an existing point of the model.
3 . Now rotate the model and check that the center of the rotation is the one selected.

1.2.4 View management

There are several preloaded views in order to align the object axes with the screen axes. These views can also be selected clicking on icon.
When we are working it's possible to change to previous views using  and icons.
It's also possible to save in a file an specific view to use it later in order to take some snapshots of our model. In order to do that select **View->View->Save...** and to read it just select **View->View->Read...**

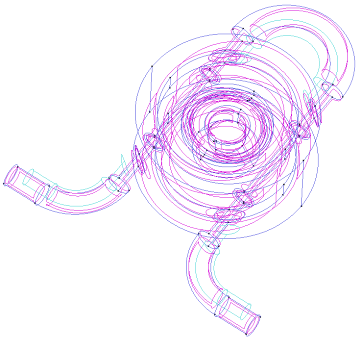
This option is useful to get the same perspective of the same model when we work on different sessions.

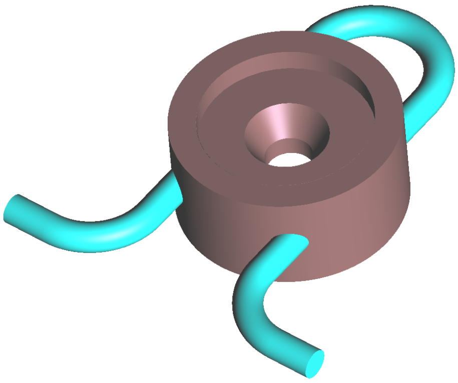
1.3 Render modes

In the **View** menu user can find the **Render** options. They are also accessible from the right mouse button and the status bar.

1 . Select **View->Render->Normal**
2 . Select **View->Render->Flat**
3 . Select **View->Render->Smooth**

In **Normal** render mode, user can see the entities drawn in different colors, depending on the kind of entity: volumes in light blue, surfaces in pink, lines in blue, and points in black.
Flat render mode draws each geometrical entity using the colour of the layer it belongs to, and **Smooth** mode uses also this criterion, but lines are not drawn to represent the geometry in a smoother way. The following figure shows the visualization of the model changing render modes:

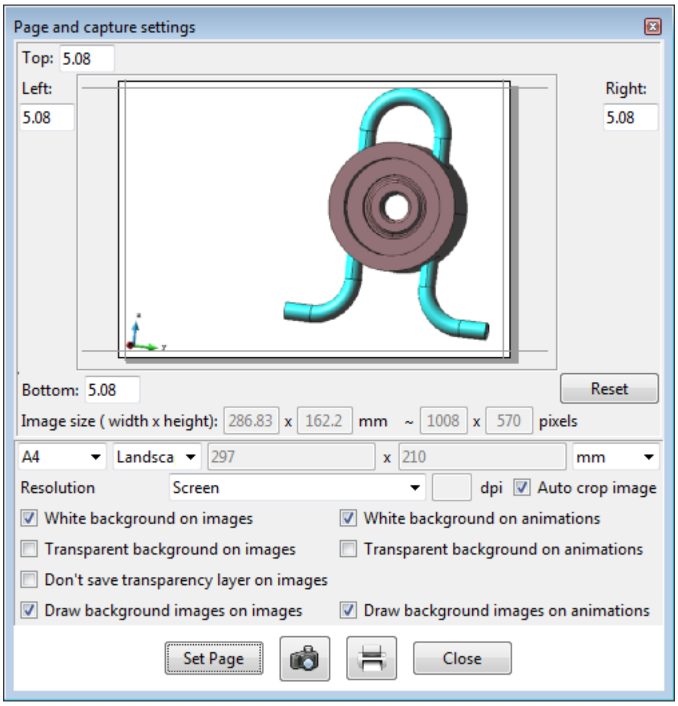

Render normal


Render smooth

1.4 Creating images

We can take 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



In order to save the image with the defined properties in **Page and capture settings:**

1 . Select **Files->Print to file** and the desired image formatthrought 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

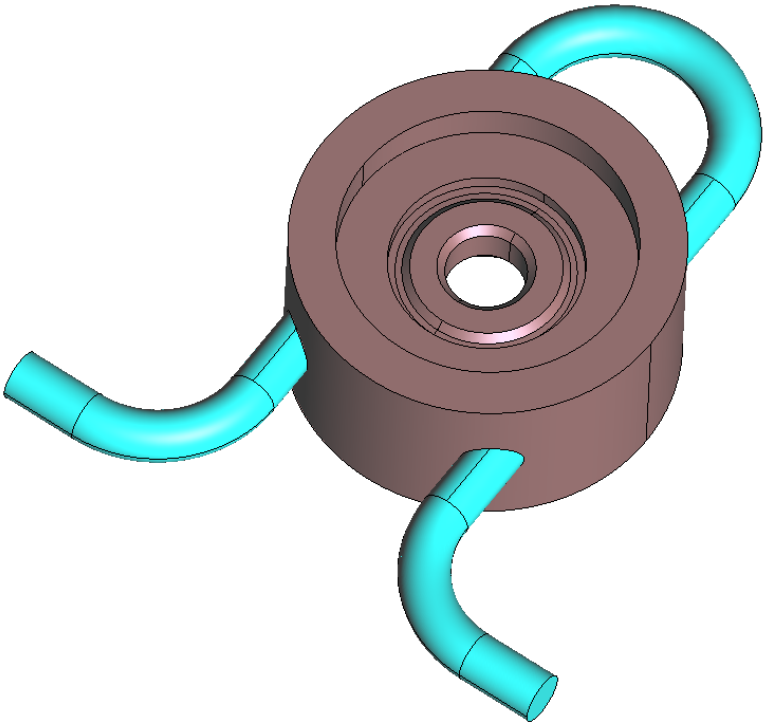
NOTE: This action could also be done by clicking on through the icon bar. This icon can also be found in the "Page and capture settings" window. In this case the image format is choosen while saving the file in the "Files of type" combobox.

1.5 Preprocess

1.5.1 Load a model

In the **Files** menu user can find the typical operations for managing the GiD projects like save a project, open an existing project, import and export files, print or quit the program. Most of this options are also accessible from the icons toolbar. The corresponding icon is shown in the menu, next to the option.

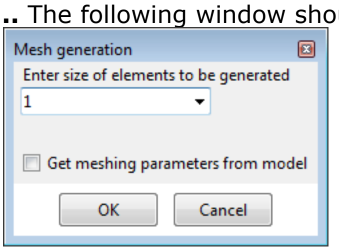
1 . Click on **Files->Open...** and select the GiD model **gid_model_basic.gid**. GiD also can load a model just with **drag & drop**.The following model should be loaded:



1.5.2 Geometry and Mesh modes

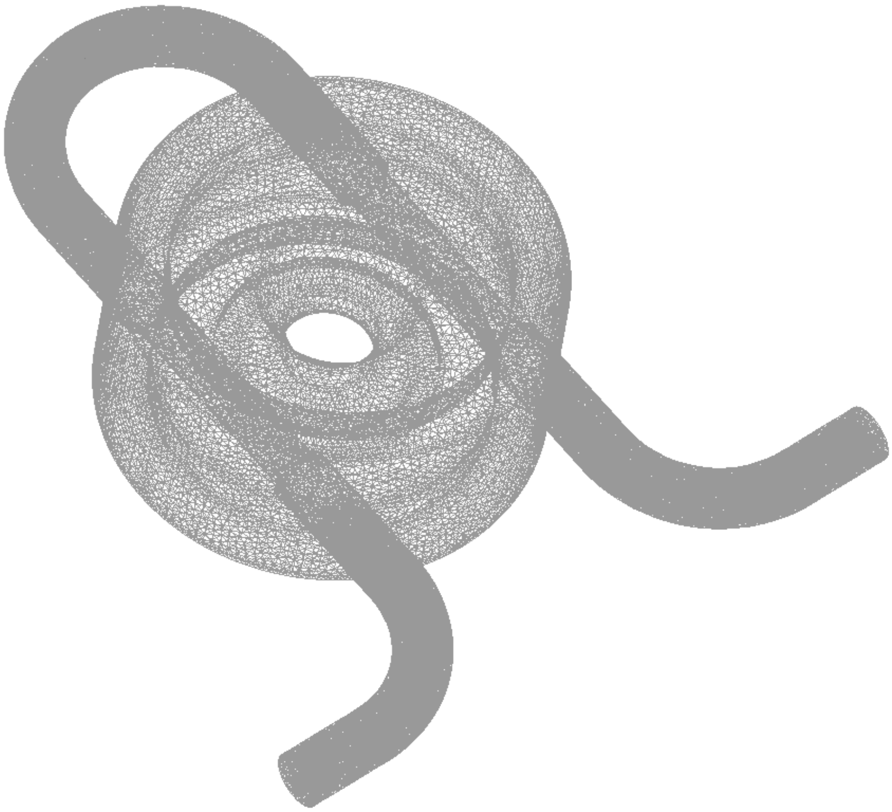
In the preprocessing part of GiD there are two basic modes the user can work with: geometry and mesh. Just in order to see how the mode can be changed we are going to generate a mesh with all the default parameters.

1 . Select **Mesh->Generate mesh...** The following window should appear.



2 . Click OK and wait for the mesh generation. Once the mesh is generated, a window pops up and show the user the result from the mesh generation.

3 . Click on 'View mesh' option, and the following visualization of the model should appear:



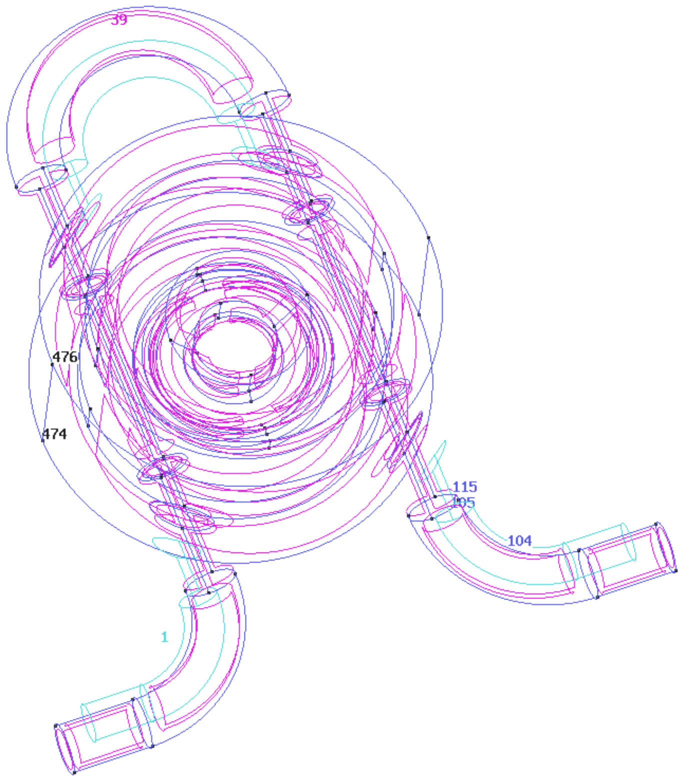
Now we are in 'mesh' mode. Changing the render mode user can see that the color of the mesh entities also follows the Layer colors.
Selecting **View->Mode->Geometry** user can change to the geometry mode again. The icon in the toolbar switch between both modes.

1.5.3 Entities information

1.5.3.1 Labels

Using the option **Labels** present in the **View** menu (and also in the right mouse button menu), user can see the number of the entities of the model. Either for points, lines, surfaces or volumes user can choose between viewing the numbers of all the entities, or just the selected ones.

In the following figure the model can be seen with the number of some entities:



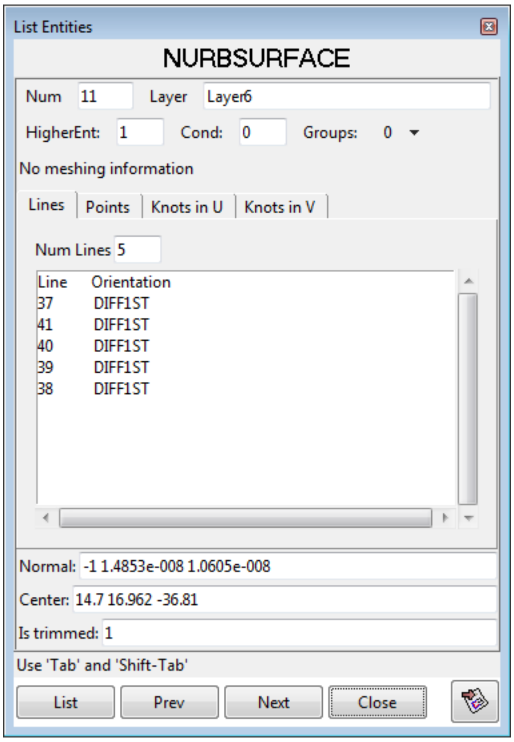
1.5.3.2 List entities

User have also the option of viewing all the characteristics of a specific entity by selecting **List** in the **Utilities** menu (or clicking in the icons toolbar).

For example:

1 . Select **Utilities->List->Surfaces** in the top menu
2 . Select some surfaces of the model
3 . Press **Escape**

An example of the information got using this option is the following figure:



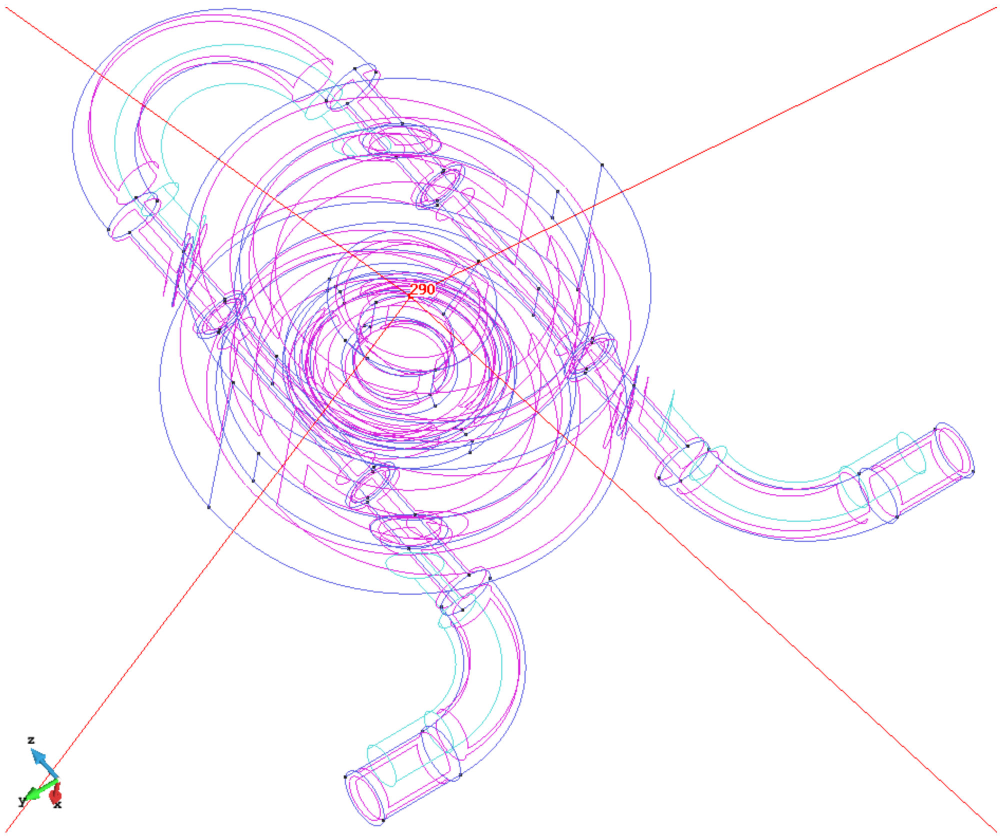
1.5.3.3 Signal

In complex geometrical models sometimes it is hard to localize an specific entity. Using the **Signal** option in the **Utilities** menu user can know graphically where the entity is, as GiD shows with a red lines cross its potition.

As an example we will signal the line number 290:

1 . Selec **Utilities->Signal->Lines**

2 . Write in the Command bar the number 290 and click ENTER. The result is shown in the next figure:

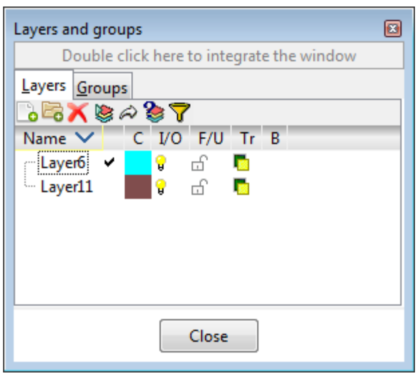


The red lines are centered always onto the specific entity independently on the rotations or view movements.

1.5.4 Layers and groups

A really useful way for organizing the different parts of the model is using 'Layers'.

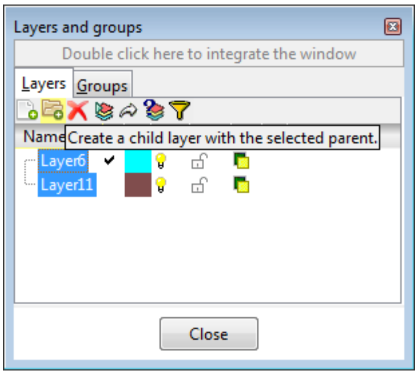
1 . Open the Layers window by selecting the **Utilities->Layer and groups** option or clicking in the upper icons toolbar. The following window should raise up:



As it can be read in the upper part of the window, if user double click on that part, the Layers window is integrated in GiD window. User can choose to work with the Layers and groups window integrated or not.

All the actions related with layers and groups can be accessed by clicking the right mouse button onto the Layers and groups window. Most of them can be also used by the corresponding icon in the upper part of the Layers window.

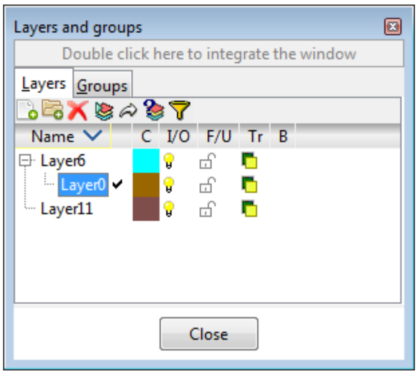
By moving the mouse over the icons of the upper part of the window and staying 2 seconds onto an icon, a help message is shown in order to give the user information about the action associated with the icon.



1.5.4.1 Create a layer

GiD allows to create a hierarchical structure of Layers, so as a Layer can contain sub-layers. Let's create a Layer into another one as an example:

1 . Select (using the left button of the mouse) the 'Layer6'.
2 . Select the **New child** option in the right mouse button menu, or click in the upper part of the Layers and groups window. Automatically, a layer named 'Layer0' should appear, as shown in the following figure:

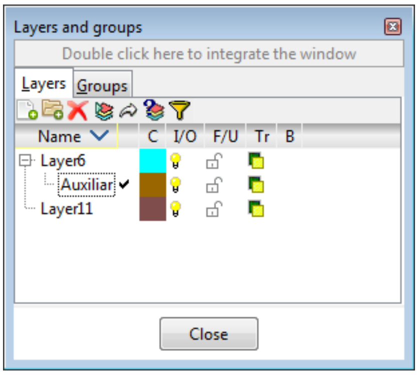


1.5.4.2 Rename a layer

To rename a Layer user should select the layer in the Layers and groups window and press **F2** key, or select the **Rename** option in the right mouse button menu.

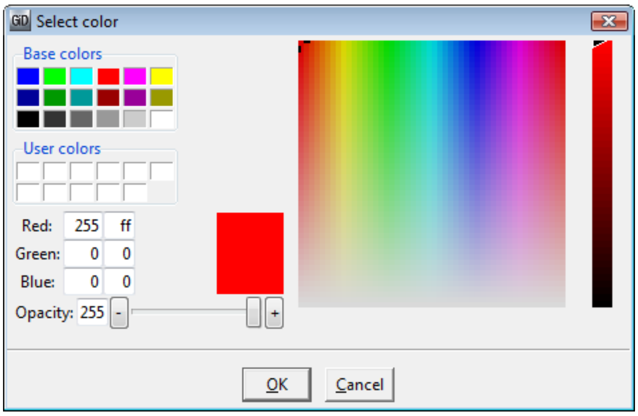
1 . Select the Layer0
2 . Rename it to 'Auxiliar'

Now the Layers window should look like the following picture:



1.5.4.3 Change the color of a layer

By clicking on the colored square next to each layer name, the following window pops-up, allowing the user to change the color of the layer:

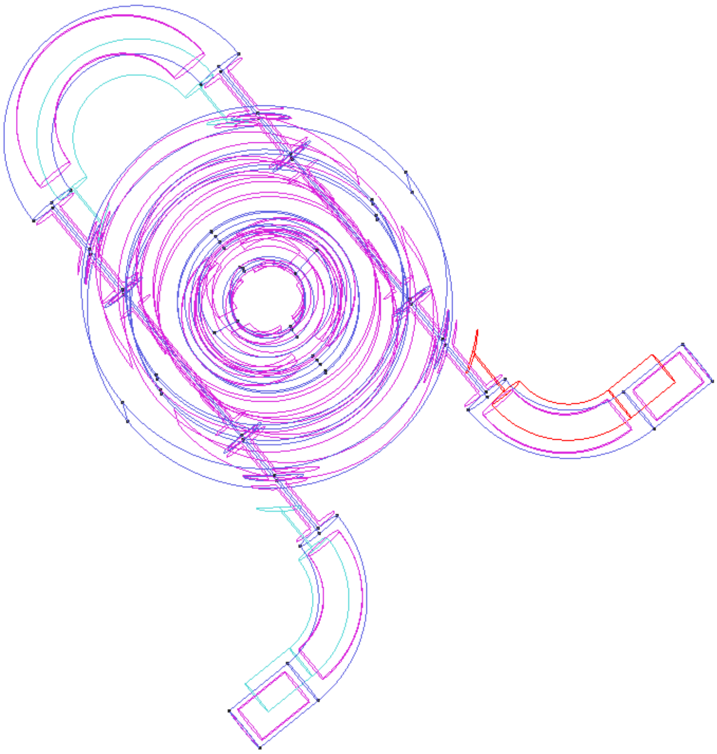


Window to change the color of the layer.

1.5.4.4 Send entities to a layer

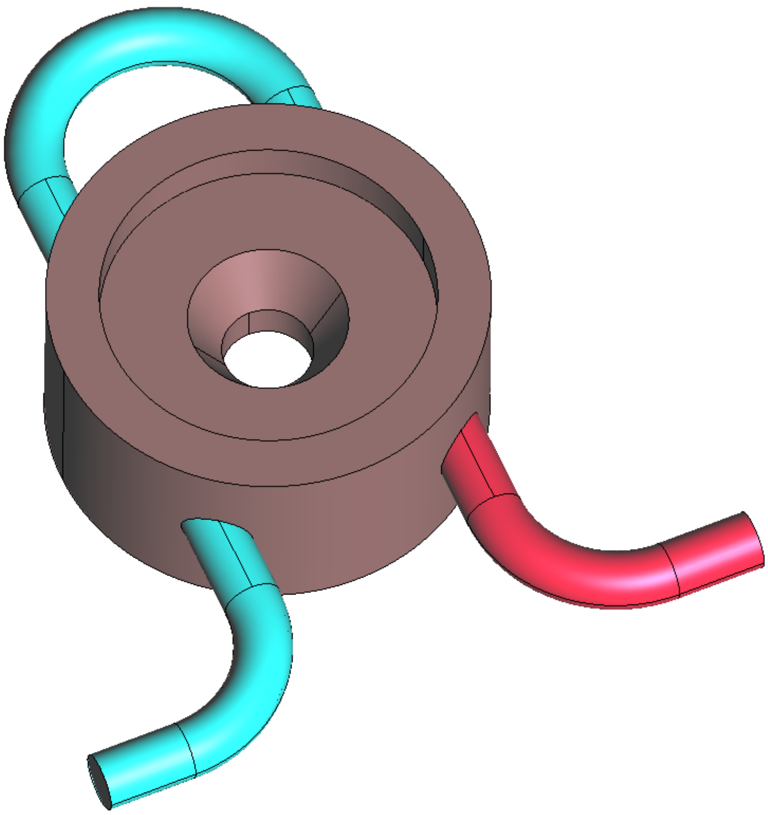
User can send entities to a specific layer. As an example we are going to send to the layer 'Auxiliar' a part of the model:

1 . Select the layer 'Auxiliar' in the Layers window
2 . Select the option **Send to** from the right mouse button (or click icon)
3 . Select **Volumes** and select the volume shown in red in the following figure:



4 . Then press **Escape** to exit the selection mode.

5 . Set the render mode to **Flat**. The color of selected volume has changed to the one of the layer 'Auxiliar', as shown in the following figure:



1.5.4.5 Switch on/off

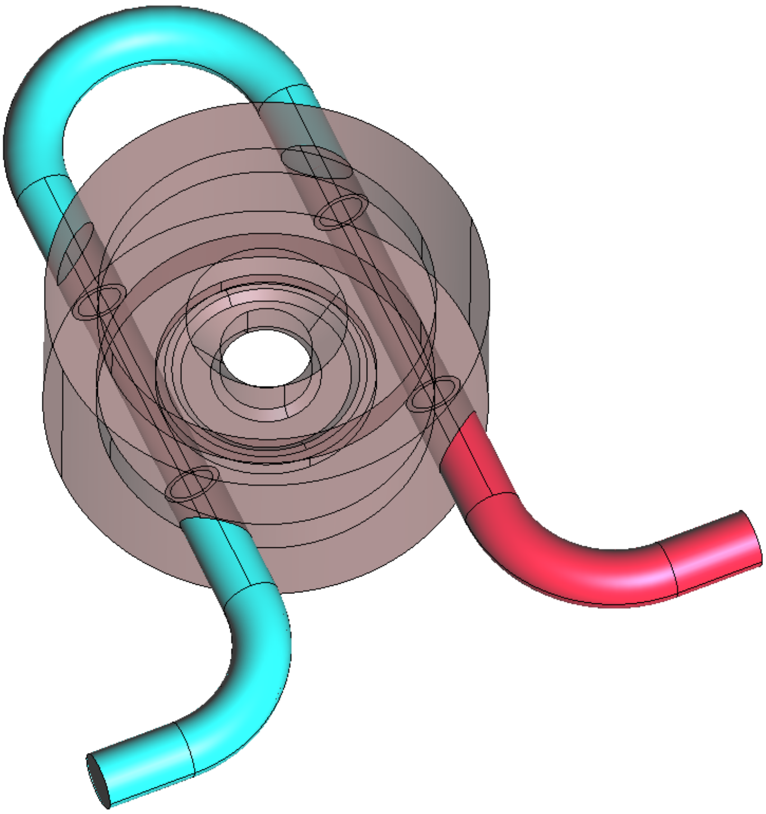
By clicking on the icon which is next to each Layer inside the Layers and groups window, user can switch on and off the corresponding layer. This is very useful in order to visualize just some specific parts of the model.

1.5.4.6 Freeze a layer

At the right side of the bulb, user can set an icon which is a lock. If the lock is closed, the layer is frozen. If a layer is frozen, GiD won't apply anything to the entities of that layer. For instance, if user select some entities to be deleted, if they are into a frozen layer they won't be erased.

1.5.4.7 Transparency

Next to the 'lock' icon of each layer is the transparency icon. By clicking there, the user can set a layer to be transparent or not. The following figure shows the model with the Layer11 set as transparent:



1.6 Postprocess

1.6.1 Load results

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

- If the model has been calculated inside GiD, and so the results are inside a GiD project, then just loading the GiD project and the changing to postprocess mode is enough. This can be achieved clicking on this icon: , or selecting the **Files->Postprocess** menu entry.
- If only a mesh and results file(s) is present then GiD should be started, and switched to postprocess mode () before loading the file(s).

In this case we will use the first option.

1 . Switch to postprocess mode: or **Files->Postprocess**

1.6.2 Entities information

1.6.2.1 Labels

Using the option **Label** present in the **View** menu (and also in the right mouse button menu), user can see the number of the entities of the model. Either for nodes, elements or results user can choose between viewing the numbers of all the entities, or just the selected ones.

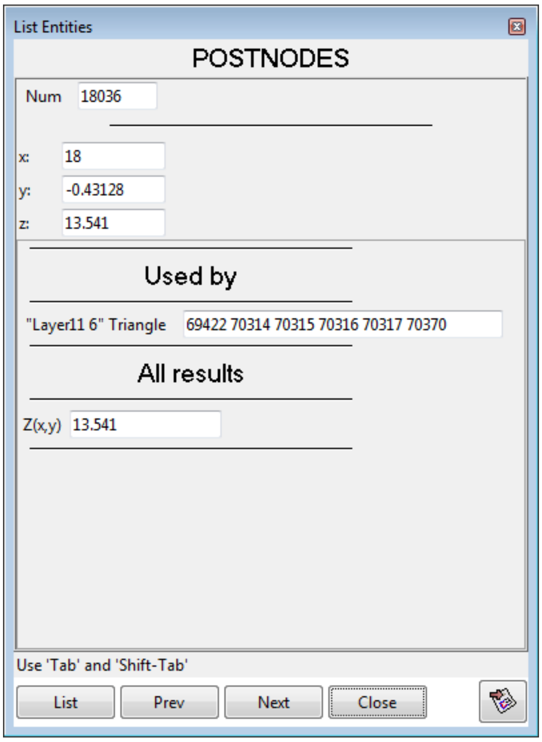
1.6.2.2 List entities

User have also the option of viewing all the characteristics of a specific entity by selecting **List** in the **Utilities** menu (or clicking in the icons toolbar).

For example:

1 . Select **Utilities->List->Nodes** in the top menu
2 . Select some nodes of the model
3 . Press **Escape**

An example of the information got using this option is the following figure:

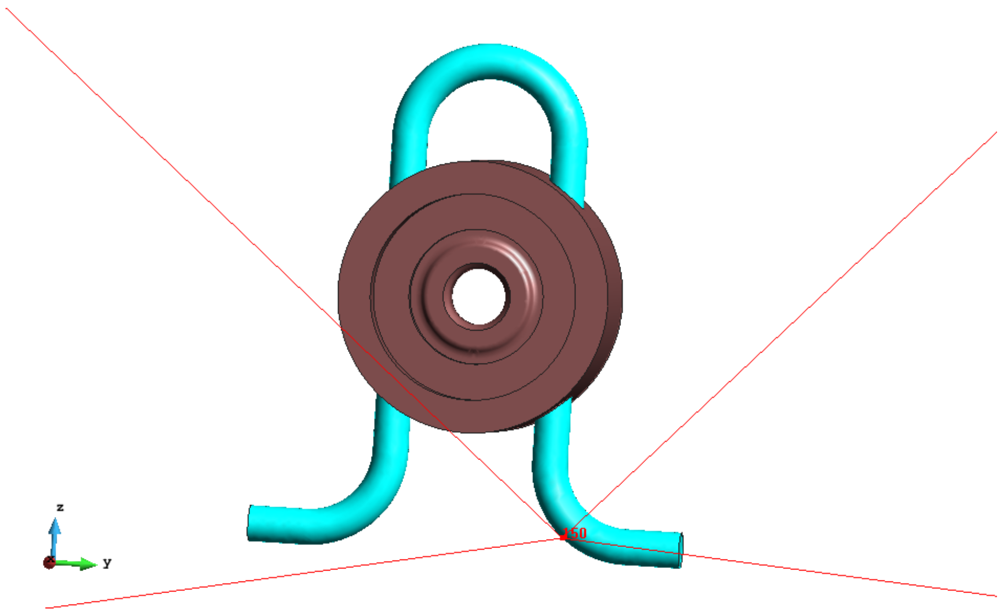


1.6.2.3 Signal

Using the **Signal** option in the **Utilities** menu user can know graphically where the entity is, as GiD shows with a red lines cross its potition.

As an example we will signal the element number 150.

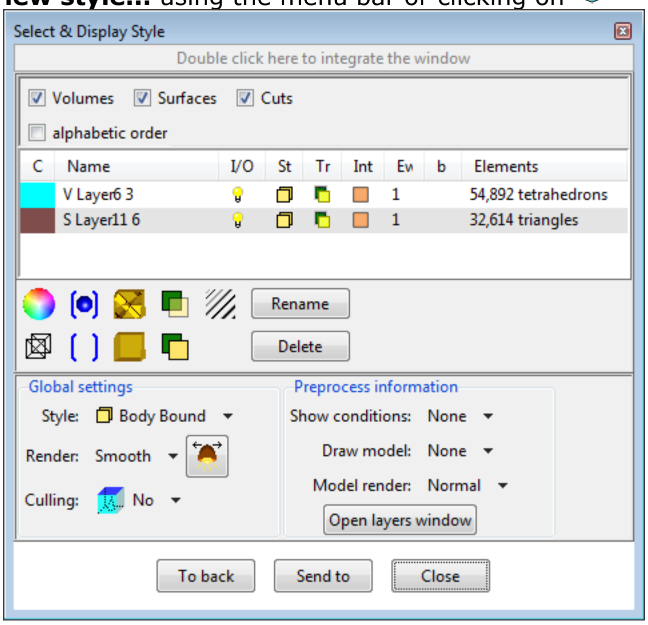
1 . Selec **Utilities->Signal->Element**
2 . Write in the Command bar the number 150 and press ENTER. The result is shown in the next figure:



1.6.3 Select and display style

Through the **Select & Display Style** window several options can be specified for volumes, surfaces and cuts. Among these options volumes, surfaces and cuts can be switched on and off, their colour properties can be changed, and their transparency too.
Other interesting options which can be changed are the style of the set and the width of the elements' edges.
From this window, volumes, surfaces or cuts can be deleted or their names can be modified.
The Select & Display Style window can be integrated inside GiD interface, just double click on the upper bar of the window. To tear it off again, double click the upper bar again.

1 . Select **Window->View style...** using the menu bar or clicking on



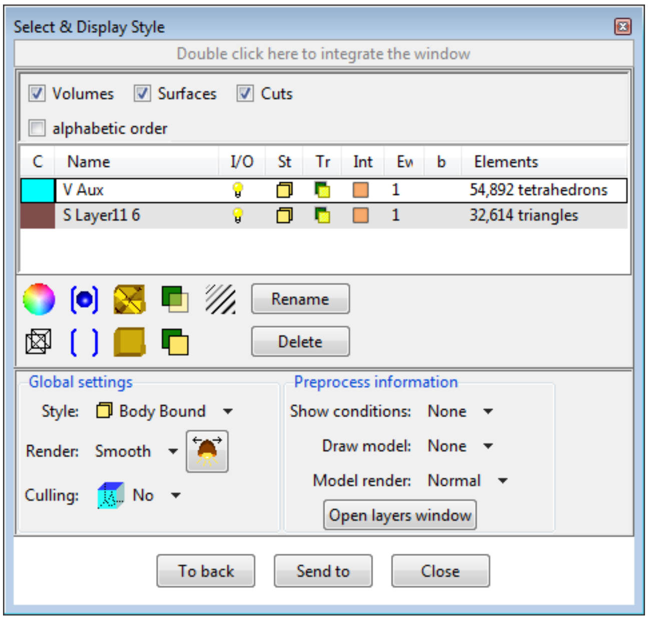
1.6.3.1 Rename a layer

To rename a layer user should select the layer in the Select and display window and press **F2** key, or press the **Rename** button.

1 . Select the **V Layer6 3**

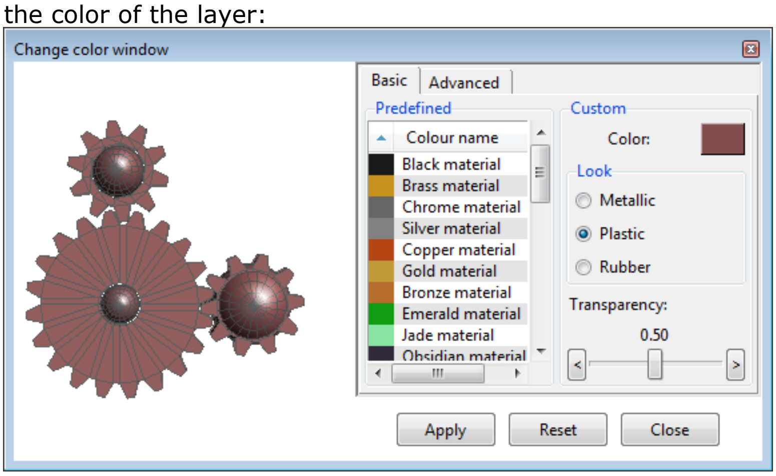
2 . Rename it to **Aux**

Now the Layers window should look like the following picture:



1.6.3.2 Change the color of a layer

By clicking on the colored square next to each layer name, the following window pops-up, allowing the user to change the color of the layer:



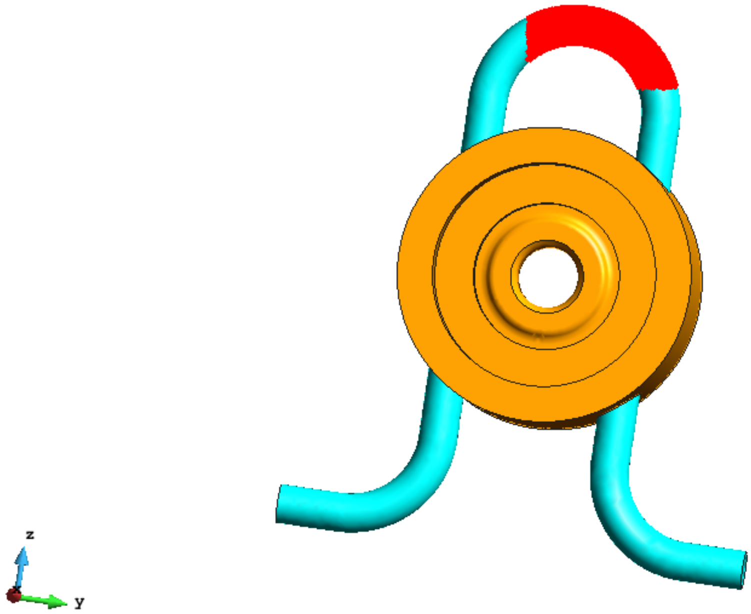
1.6.3.3 Send entities to a layer

User can send entities to a specific layer. As an example we are going to send to the layer 'Test' a part of the model:

Our model only has 2 layer. We will create a new layer with some elements.

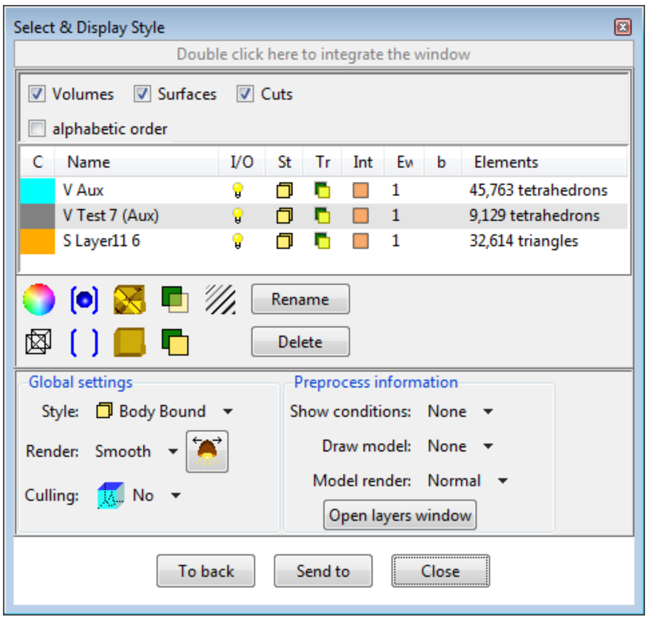
1 . Press button **Send to->New set long name**.

2 . Select some elements.



3 . Press **Escape**.
4 . A window appears asking for a name. Enter 'Test'.
5 . Press **Accept**.

A new layer is created with the selected elements. Now we can change the color of the new layer.



1.6.3.4 Switch on/off

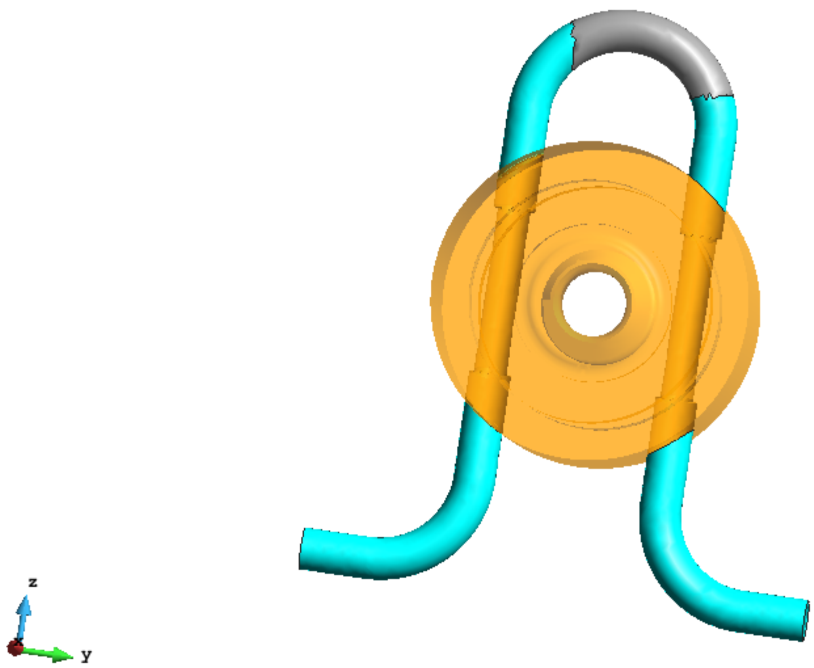
By clicking on the icon which is next to each layer inside the Select and display window, user can switch on and off the corresponding layer. This is very useful in order to visualize just some specific parts of the model.
It's also possible to switch on/off the layers using the and icon

1.6.3.5 Style and transparency

By selecting one set and clicking on one corresponding style icon at the **St** column or clicking on , the style for the selected mesh is changed. Bear in mind that a boundary visualization of the surface of a sphere nothing will be visualized, as a sphere surface has no borders.
Mesh styles can also be changed clicking on the icon , placed in the left icon bar. This style affects all sets of the model.
Next to the style icon of each layer is the transparency icon . By clicking there, the user can set a layer to be transparent or not.

1 . Select SLayer11 6 and change the style to **Body**
2 . Set it to transparent

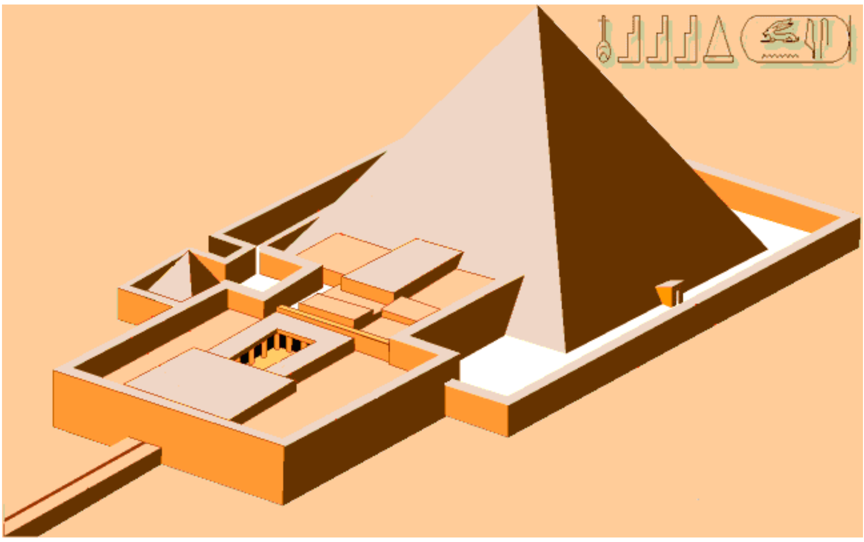
The model shoul look like this:



2 Geometry creation and edition

2.1 Description

This tutorial is focused in the use of the basic geometric tools of GiD.
We will explain how to create a CAD model of the Egyptian construction shown in the figure below.



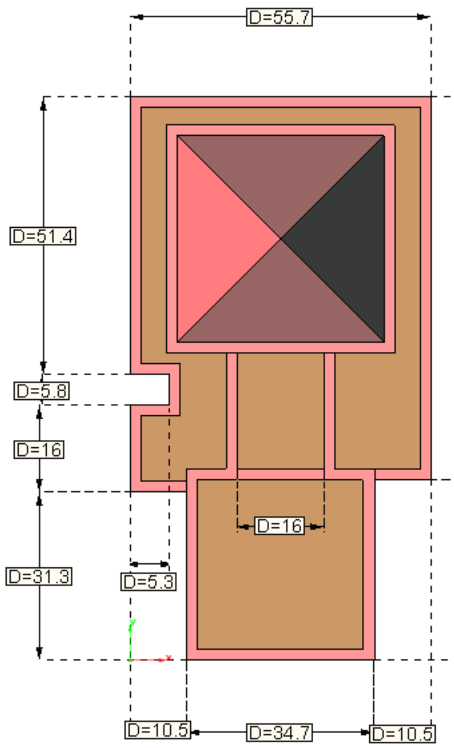
Egyptian funerary complex perspective

Some simplifications of the geometry are allowed. In the following parts of the course some schemes describe the dimensions of the simplified shape of the model

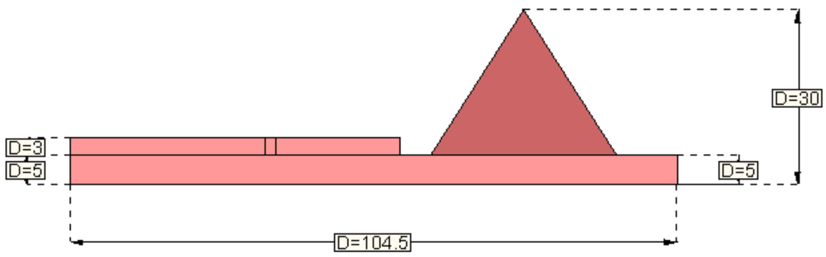
We want to create a volume of a portion of the air around a control volume. We use a 300x200x100 m box so that the faces of the box are relatively distant from the temple (e.g. imagine that we want to run a CFD simulation analysis of the airflow around the building).

2.2 Simplified model

In this section we will use the GiD facilities (creation of lines, surfaces, copy tool, use of layers, etc.) to create the geometry of the Egyptian construction shown following scheme (dimensions are in meters).



Plant scheme



Profile scheme

The width of all the walls is 2 meters.

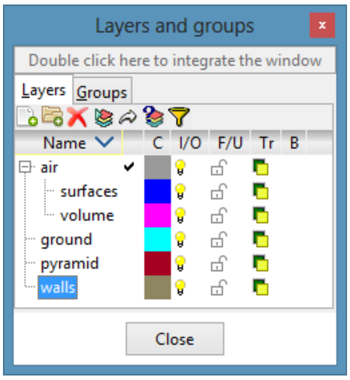
Of course, there is not a single way to build this shape. We will use several tools in this course which may not be the optimal ones, but the objective is to show as more tools as possible for the geometry creation using GiD.

NOTE: Not all dimensions are specified in the scheme, the undetermined ones should be approximated based on the images.

2.3 Points and lines creation

We will start creating the wall lines located the ground (plane z=0)

To facilitate visualization and selection tasks we will separate entities in several layers with this tree structure:



Final layers structure.

We will start creating the lines of the plant, in the z=0 plane.

Be sure the layer in use is the 'walls' one. The layer in use is marked with a 'check' between the name and the color of the layer. To set the 'walls' layer as the layer in use just double click onto the name of the layer.
New entities will be created in the current 'layer to use'.

2.3.1 Line creation

1 Choose the **Line** option, by going to **Geometry->Create->Straight line** or by going to the GiD Toolbox (the GiD Toolbox is a window containing the icons for the most frequently executed operations. For information on a particular tool, click on the corresponding icon with the right mouse button).

2 . Enter the coordinates of the beginning and end points of the auxiliary line. The coordinates of a point may be entered on the **command line** either with a space or a comma between them (but not both). If the Z coordinate is not entered, it is considered 0 by default. After entering the coordinates of one point, press **Return**. Another option for entering a point is using the **Coordinates Window**, found in **Utilities->Tools->Coordinates window**...

For our example our coordinated origin will be set at the lower-left of the bounding box of the model, and we will create the lower line that starts at x=10.5 and ends at x=10.5+34.7=45.2 units, the coordinates are then (10.5,0) and (45.2,0) respectively. Besides creating a straight line, this operation implies creating the end points of the line.
Write then:

- 10.5,0**<Return>**
- 45.2,0**<Return>**

3 . Press **<ESC>** twice to indicate that the process of creating the line is finished. (Pressing the **<ESC>** key is equivalent to pressing the center mouse button.).

4 . If the entire line does not appear on the screen, use the **Zoom Frame** option, which is located in the GiD Toolbox and in **Zoom** option in the mouse menu.

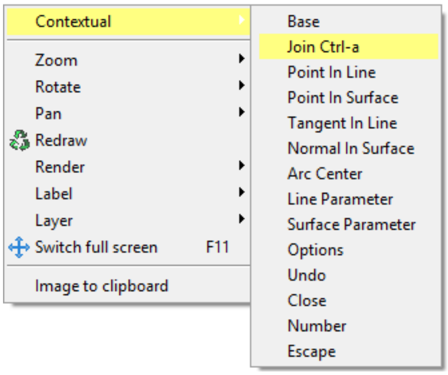
Figure 6. Creating a straight line

NOTE: When creating a line, after clicking the first point, you can press the **Alt** key to snap the dynamic line to the screen horizontal, vertical or 45º diagonals.

NOTE: The **Undo** option, located in **Utilities->Undo**, enables you to undo the most recent operations. When this option is selected, a window appears in which all the operations to be undone can be selected.

2.3.2 Join NoJoin option

When generating geometry, very often it is needed to select some point: a new one (like the ones that have been defined in the previous steps for defining the lines), or an existing one. When an already existing point is wanted to be selected, user must go to **Contextual->Join Ctrl-a** in the mouse menu (right-click)

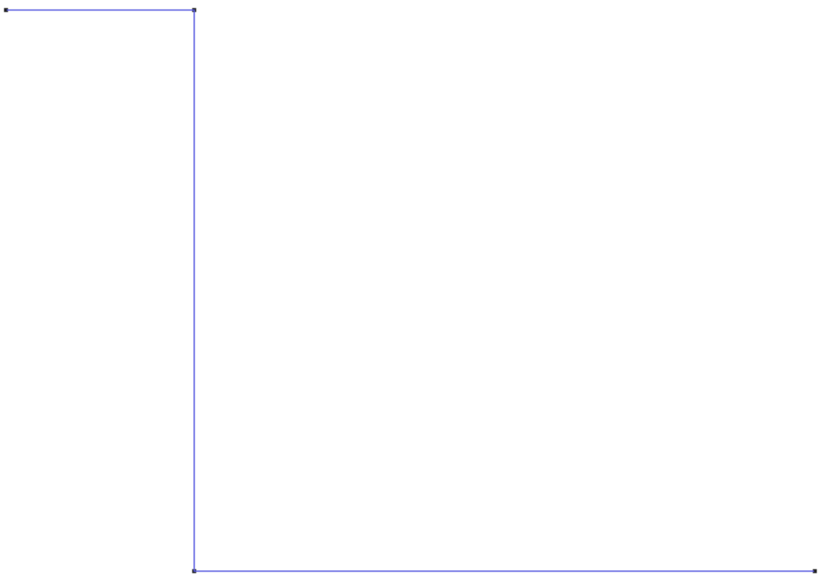


Contextual menu when expecting a point

(or pressing **Ctrl+a**). The pointer will become a square, which means that you may click an existing point. User can change alternatively between both modes: (clicking an existing point or a new one) in the same selection process.

1 Choose the **Line** option, by going to **Geometry->Create->Straight line** or by going to the GiD Toolbox.
2 Press **Ctrl+a** to select an existing point (ensure the mouse pointer is square-like)
3 Click on the point on the left.
4 Press **Ctrl+a** again to select now new points (ensure the mouse pointer is now a cross)
5 Then enter the following points by coordinates (like in the previous steps):
    - 10.5,31.3
    - 0,31.3
6 Press **ESC** twice to indicate that the process of creating lines is finished.

Now the model should look like the figure.

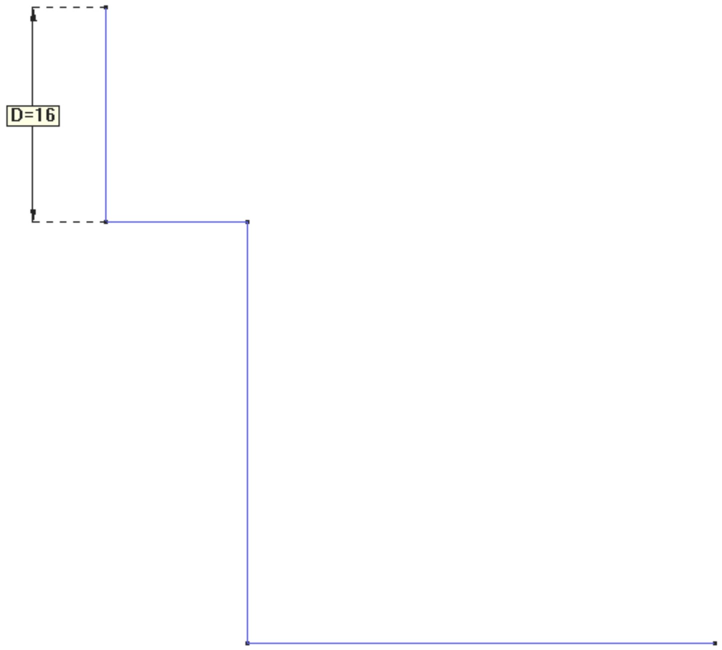


State of the model geometry at this point.

2.3.3 Relative coordinates

In GiD there are two ways of defining the coordinates of a point: absolutely (like all the coordinates we have entered since now in this course) or relatively to the last point entered (using @ symbol). We will now enter a point using this second option, which is very useful when points wants to be entered knowing some distance.

1 Choose the **Line** option, by going to **Geometry->Create->Straight line** or by going to the GiD Toolbox.
2 Press **Ctrl+a** to select an existing point (ensure the mouse pointer is square-like), and select the point which is more at the left in the model.
3 Press **Ctrl+a** to select a new point (ensure the mouse pointer is a cross).
4 Enter the following relative coordinates: (@0,16)
5 Press **ESC** two times to leave the line creation mode.



Line created with a length of 16 meters, entered as a relative coordinate to its second point.

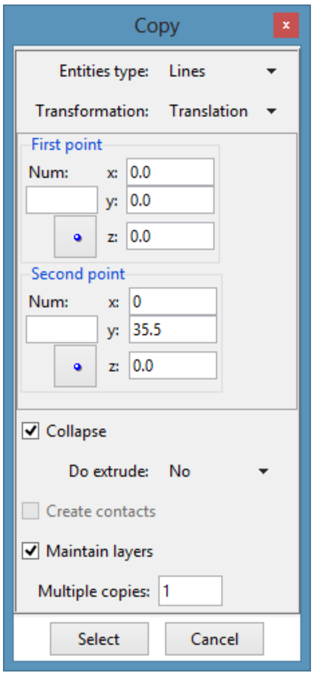
As it can be seen in the figure, the new point has been created at 16 meters (as the relative coordinates indicated) of distance from the first selected.

2.3.4 Copy tool

Another useful option for creation of geometry is using the different geometrical transformations GiD offer in the **Copy** and **Move** windows.

Now we are going to copy the lower line translated 35.3 m in y direction:

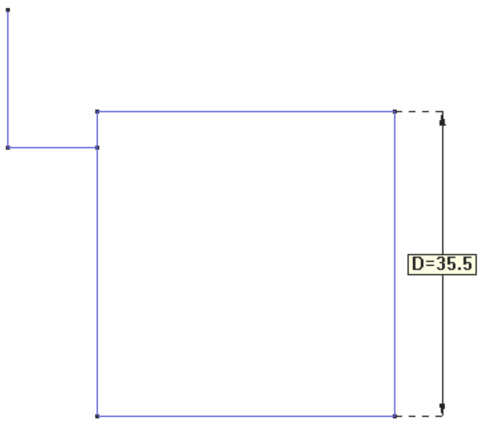
1 Use the **Copy** window, which is located in **Utilities->Copy** (you can access that window also using the icon from the Toolbox, or the keys <Ctrl-Shift-C>).
2 Within the **Copy** menu and from among the **Transformation** possibilities, select **Translation** . The type of entity to receive the translation is a line, so from the **EntitiesType** menu, choose **Lines**.
3 We want to make a translation of 35.5 units in the y direction, so we should leave the coordinates of the **First Point** at (0,0,0), and set the coordinates of the **Second point** at (0,35.5,0) (the translation is defined with the relative vector from the first to the second point)
4 The following options of the window should be the same as the ones in the figure.



Copy options to be used.

5 Click on Select and select the line to be copied (the lower line)
6 Press **ESC** to exit the selection mode.

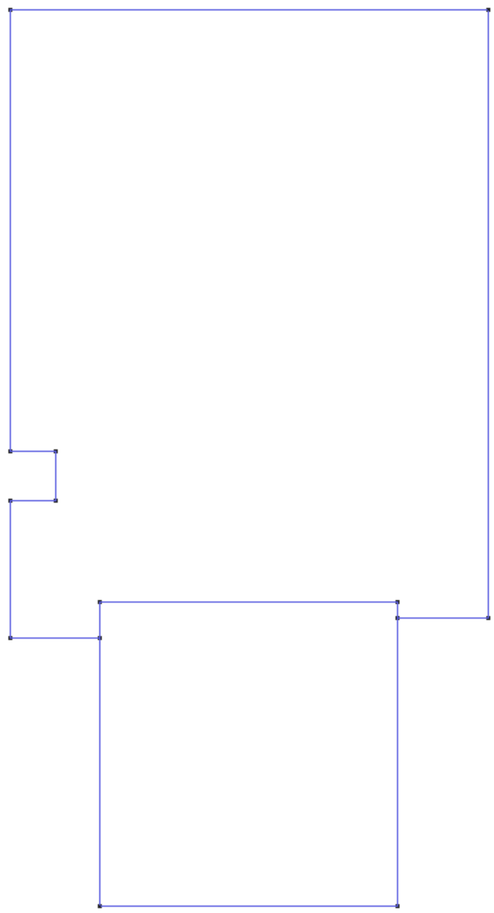
Then we create new lines between existing points to complete the square.Finally, you would get the model like the one showed in next figure.



State of the model at this point.

The copy tool can create entities by extrusion. We can create the new lines of the external perimeter of the wall doing a copy of point extruding a line along the translation. Now the 'entities type' must be set to points, and 'do extrude' to lines.

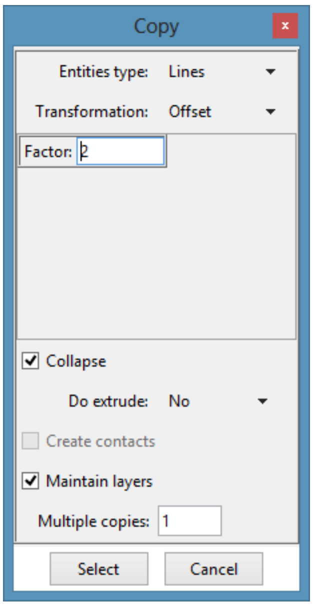
Set relative translation x=5.3 and select the upper-left point.
Repeat it changing the relative translation based on the scheme dimensions and selecting the last point until complete the external perimeter

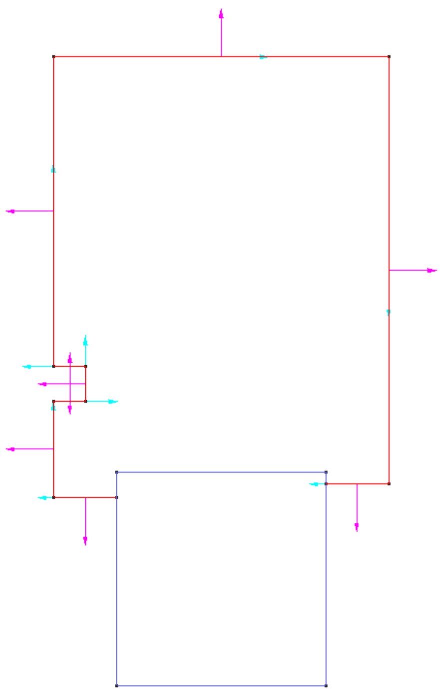


Closing the outer perimeter.

2.3.5 Offset

The walls have a width of 2 meters, now we will build the inner perimeter using the **Copy** tool with **'Offset'**

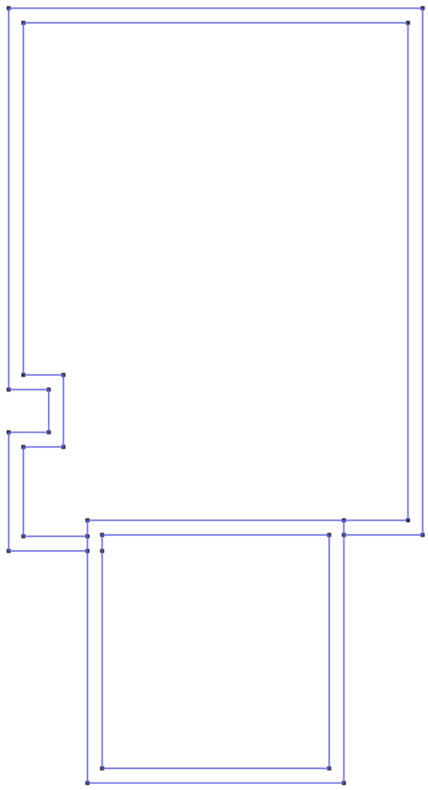




The offset factor is positive pointing to the line normals direction, or negative in the opposite one. When selecting the lines to be copied we see that all lines are pointing outside, we can first change its orientation. We cancel the copy deselecting all before finish (can do it with "Clear selection" of the contextual menu)

To swap the line orientation use the menu **Utilities->Swap normals->Lines** and pick the lines then could repeat the copy of lines by offset.

Repeat the operation for the lower square, obtaining this result

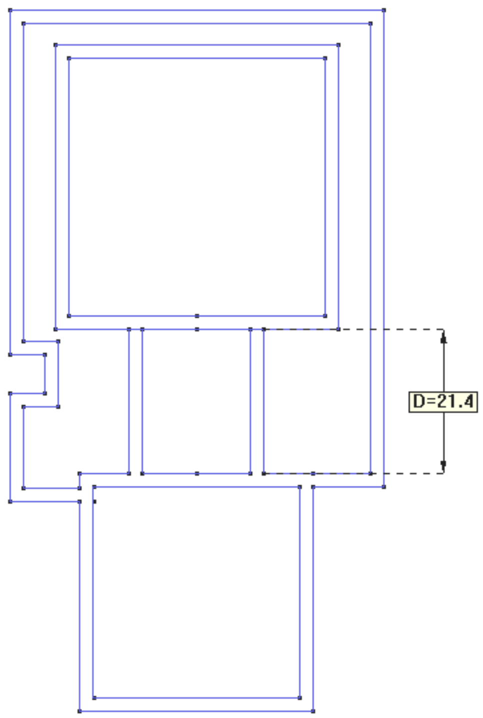


Inner perimeter created by two offsets.

2.3.6 Finalize the lines creation

Now you should complete the remaining geometry until obtaining a figure similar to the one shown below. Use the explained geometrical operations to complete by you own the geometrical definition of it.

It may be useful for you to use some other geometrical operations, like the ones present in Geometry menu. They are easy to use, and its names are intended to be self-explicative, we encourage you to try some of them for the creation of the geometry (e.g. division, intersection of lines, deletion, collapse of points, etc.)



Current state of the model.

2.4 Surface creation

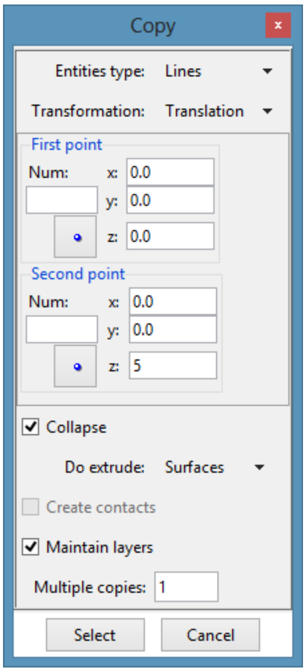
In the following points we are going to complete the geometrical model definition using different surface creation techniques.

2.4.1 Extrusion of walls

For creating the walls we will use the translation tool inside **Copy** window for the extrusion of the lateral surfaces, and then create the top surfaces.
First of all let's use the **Rotate** function (**Rotate->Trackball** from the mouse right button menu, or clicking the corresponding icon) to see the lines of the model from a different point of view, which will help us to understand the model.

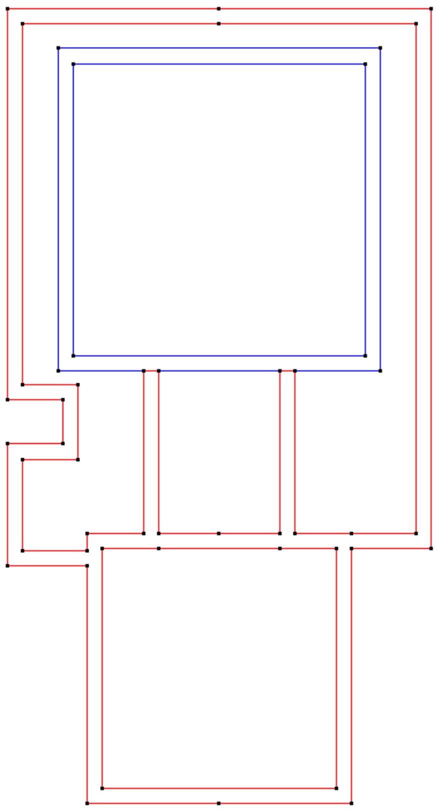
Ensure the active layer is the one named 'walls' and follow the next steps:

1. Open the **Copy** window, which is located in **Utilities->Copy** (you can access that window also using the icon from the Toolbox).
2. We are going to apply a translation of 5 units in z direction to all the lines. For this purpose ensure all the options of the window are like the ones showed in the figure.



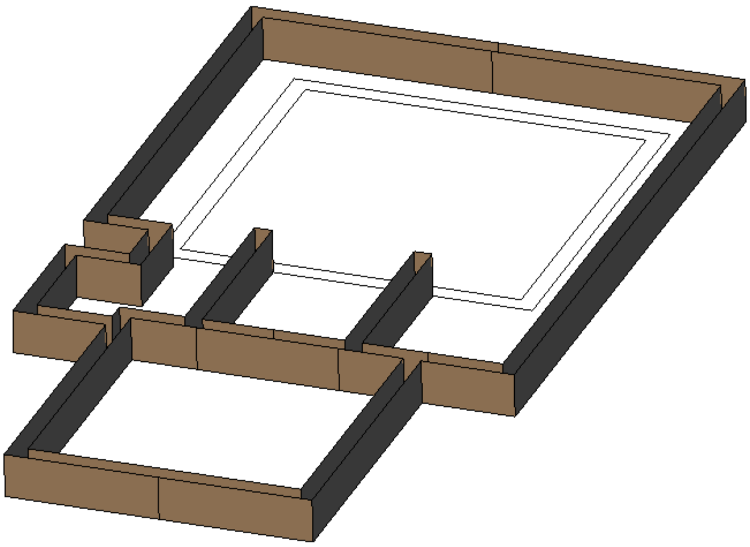
Options to be applied

3 Select the lines of shown in next figure. (some lines should be divided before to show the same points that the image)



Lines to be selected for the walls creation.

4 Click on finish button. The result should be like showed in the figure

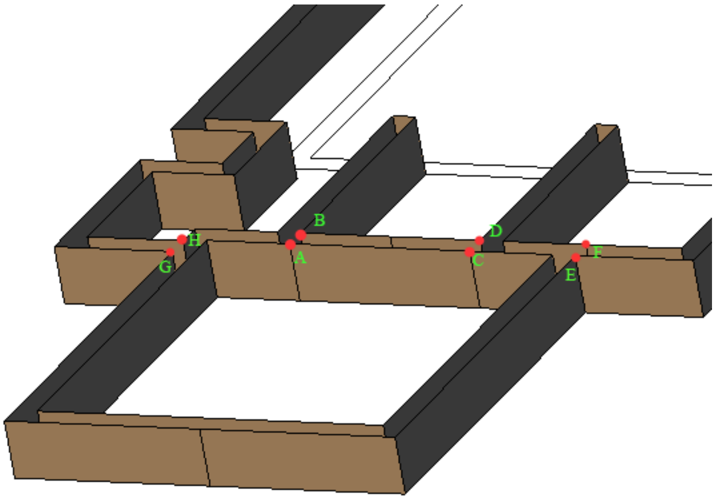


State of the model after the lines extrusion.

Take care on the different visualization modes that can be used for visualizing the geometry. This options are **Normal**, **Flat** and **Smooth**, and can be selected from the **Render** option in the mouse right button menu.

Note: in Flat render mode surfaces are painted with the layer color. Don't worry if you are seeing different colors, layer colors could be changed.

Now we must create two straight lines (using the way of creating lines explained in previous steps) between points A-B, C-D, E-F and G-H. This points are showed in the figure.

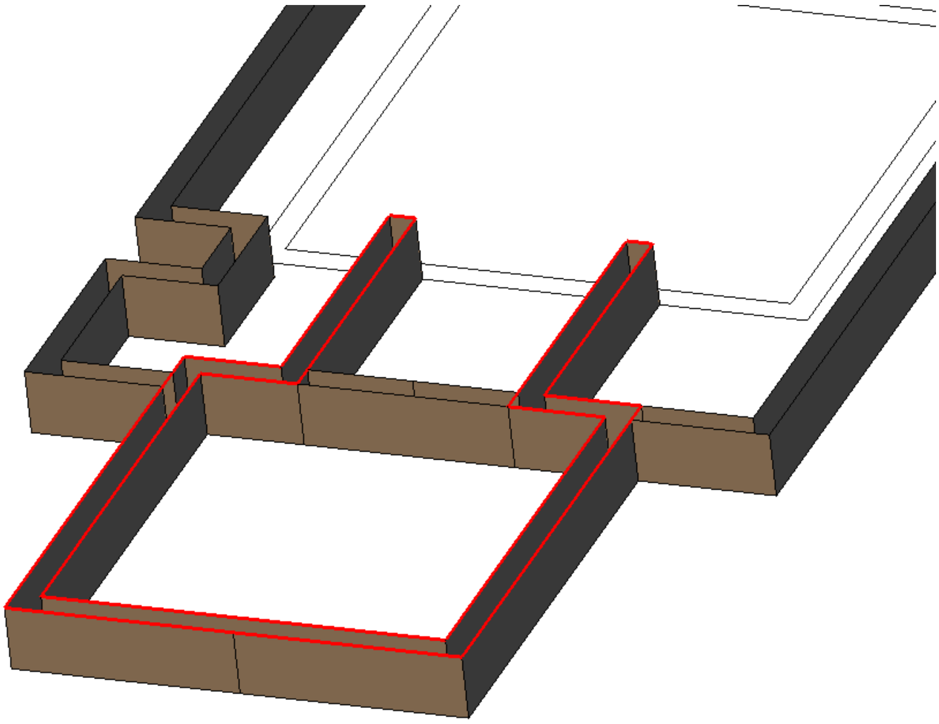


Points to be conected by straight lines.

NOTE: If some point doesn't exists it could be created dividing the line, for example with

Geometry->Edit->Divide->Line->Near point
selecting an existing poing with Join and then selecting the line to be divided projecting this close point.

Now it is needed to make higher certain walls (not all of them). For this purpose, we are going to use the **Translation** operation (analogously as the previous steps, but only 3 units in z direction, and selecting only the lines showed in next figure).



Lines to be selected for the second extrusion.

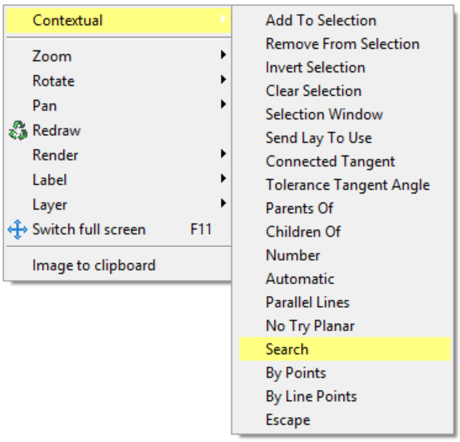
2.4.2 Creation top of walls

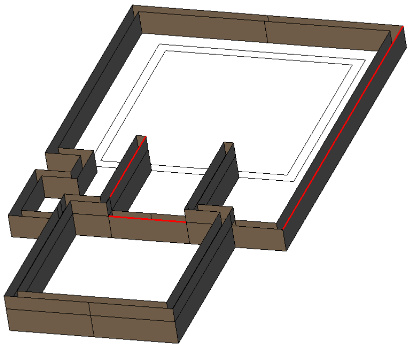
Now we have created the lateral surfaces of the walls, and we must create the top surface.

We have all the contour lines of the surfaces to be created, so we have several options of creating them. The simplest one is to select **Geometry->Create->NURBS surface->By contour** (or clicking the corresponding icon), and select all the contour lines which define the contour of the surface.

For simple geometry configurations, GiD can detect the contour lines of a possible surface automatically. We will use this option.

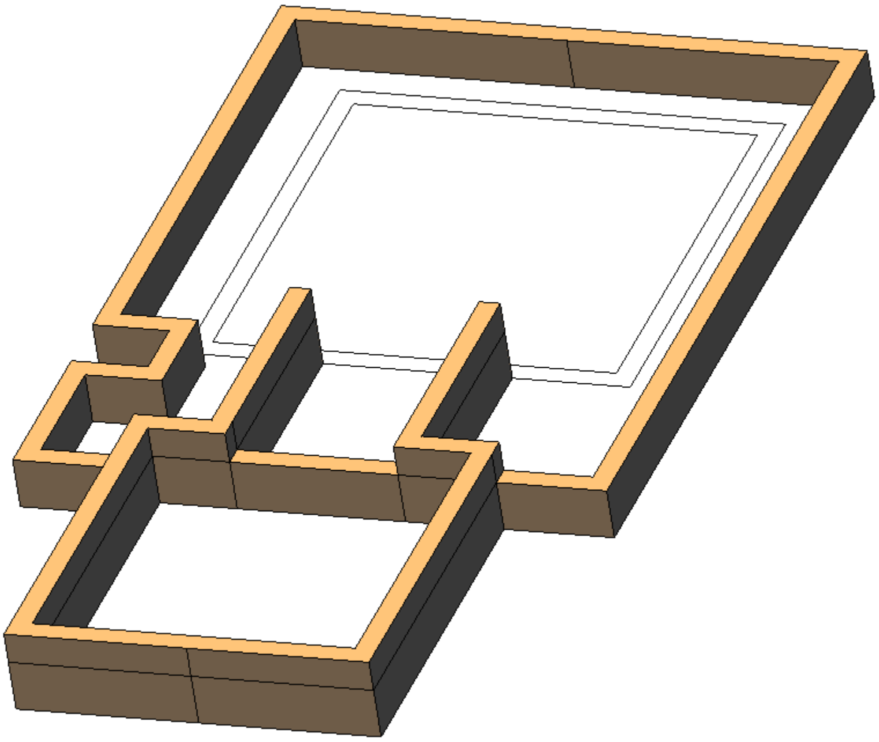
1. Select **Geometry->Create->NURBS surface->By contour.**
2. Then click on the right mouse button and select **Search** in the Contextual menu.
3. Then click once the lines showed in the figure.





As you can see, each time you click a line, GiD find automatically the other lines which close a surface parting from the selected line.

Now all the walls are created. The result of this operation should be the one showed in next figure.



View of the model once the walls are created.

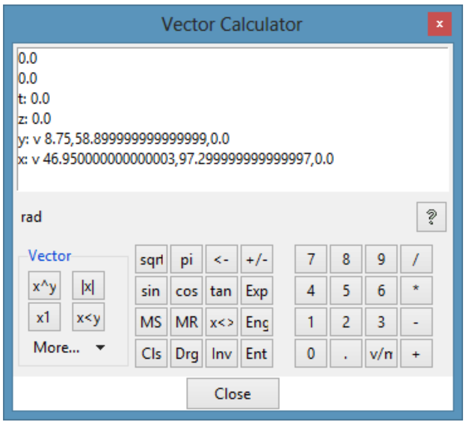
If do you have obtained different surfaces must delete the wrong ones and try the automatic search mode starting from another line, or without this tool specifying all boundary lines of the wanted surface.

2.4.3 Creation of the pyramid

We are going to create the pyramid in the layer named 'pyramid', so you have to set this layer as the one in use.

For creating the pyramid, first of all we need to create the vertex of it, and need to calculate its coordinates.
You know that the total height of the pyramid is 30 meters, but you don't know the x,y coordinates of

the projection on the ground.
You can calculate it as the midpoint bethween two diagonal points.
Open the calculator with
Utilities->Tools->Calculator...



Calculator window after select two diagonal corners.

Select the **More...->Get point** and click a corner of a diagonal
their coordinates will appear in the calculator
8.75,58.8999,0.0

Repeat it and now get the opposite diagonal corner.
46.95,97.2999,0.0

now to calculate the sum pressing button
+
now to divide by 2 enter this number pressing the buttons
2 Ent
and pres the button
/

the vector result is shown
27.85,78.1,0.0

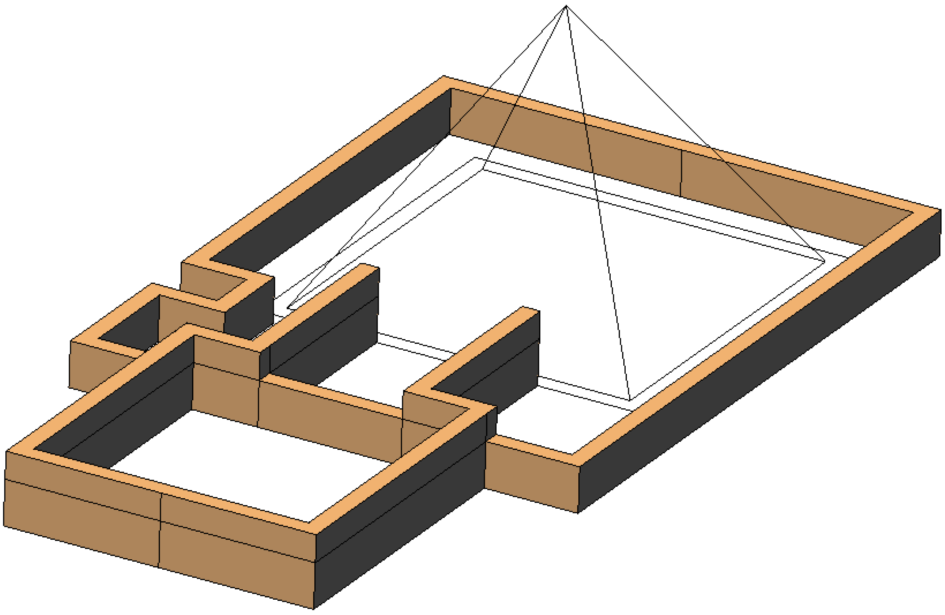
to send this value to the GiD comand line press
More...>Write

1 Select **Geometry->Create->Point** and modify in the command line the coordinates changing z=0 to z=30

27.850000000000001,78.099999999999994,30
and press enter to creat the point with this coordinates

Note that the points can only be seen in **Render Normal** (not in **Flat** or **Smooth** mode).

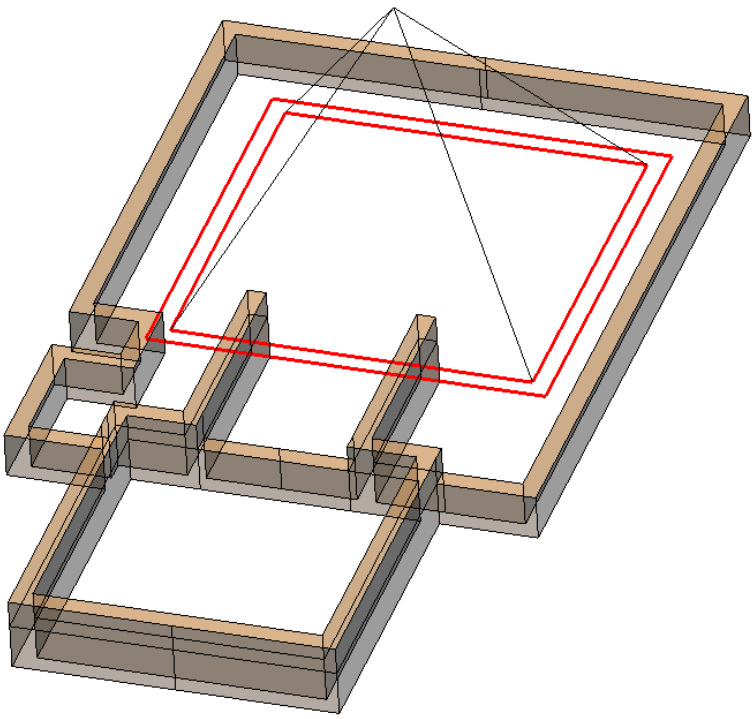
2 Then create one straight line from each vertex of the quadrilateral base of the pyramid to the vertex of the pyramid. The result should be like the model showed in the figure.



Lines of the pyramid created.

We are going to create the surface of the squared frame on the base of the pyramid. For this purpose:

3 Select **Geometry->Create->NURBS surface->By contour** (or click the corresponding icon), select the lines showed in the figure and press **ESC**.



Lines to be selected for creating the base frame of the pyramid.

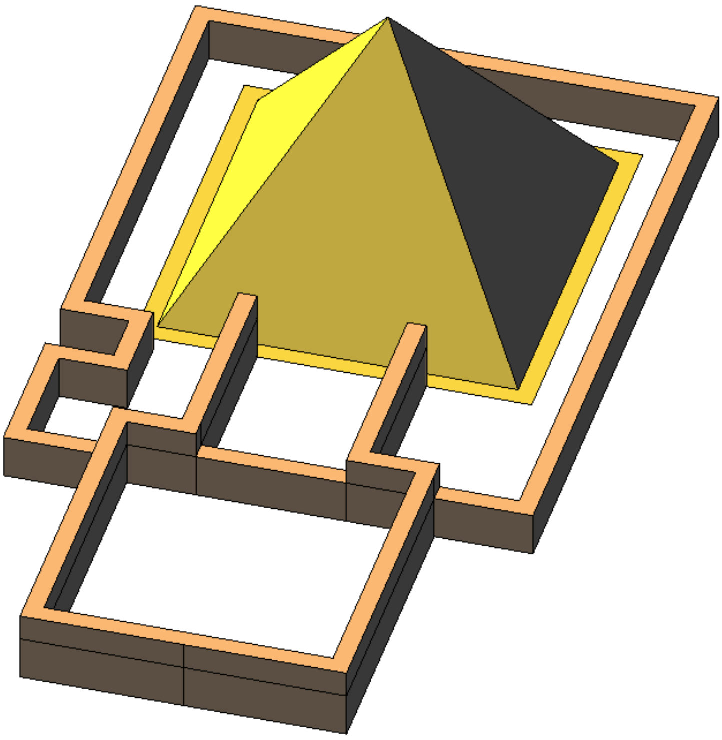
For creating the lateral surfaces of the pyramid:

4 Select **Geometry->Create->NURBS surface->By contour** (or click the corresponding icon), select the three lines of one face of the pyramid and press **ESC**.

5 Then select the three lines of the other face and press **ESC**, and the same for the other faces.

(you can use also the **Search** method to define easily these surfaces)

The geometrical model at this moment should be like the one showed in the figure.



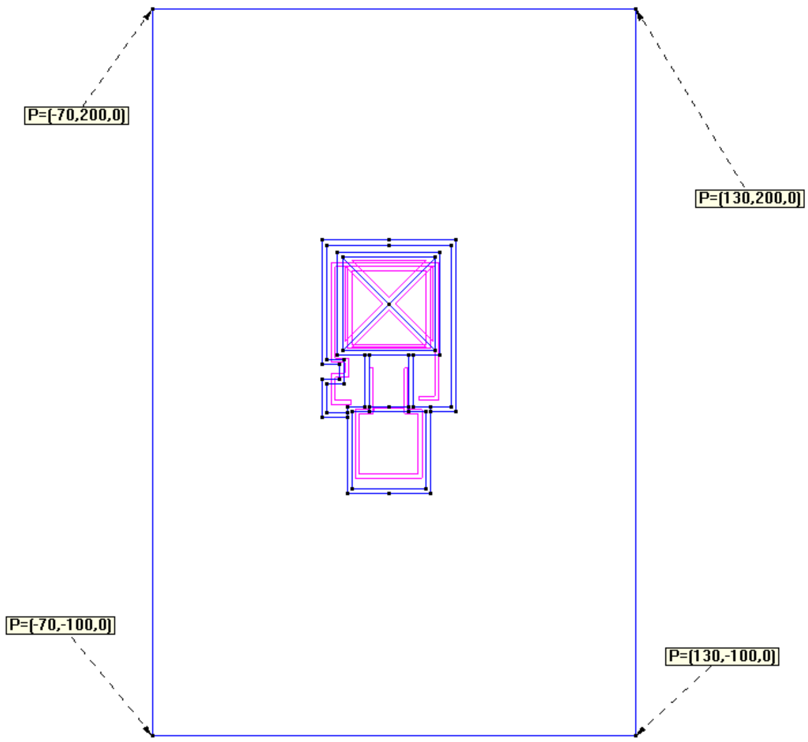
State of the geometrical model at this point.

2.4.4 Creation of ground surface

We are going to create now the surface of the ground. We should define the limits of the domain into where the simulation will be done. This limits should be far enough from the model to avoid the effect of the boundary conditions in the results.

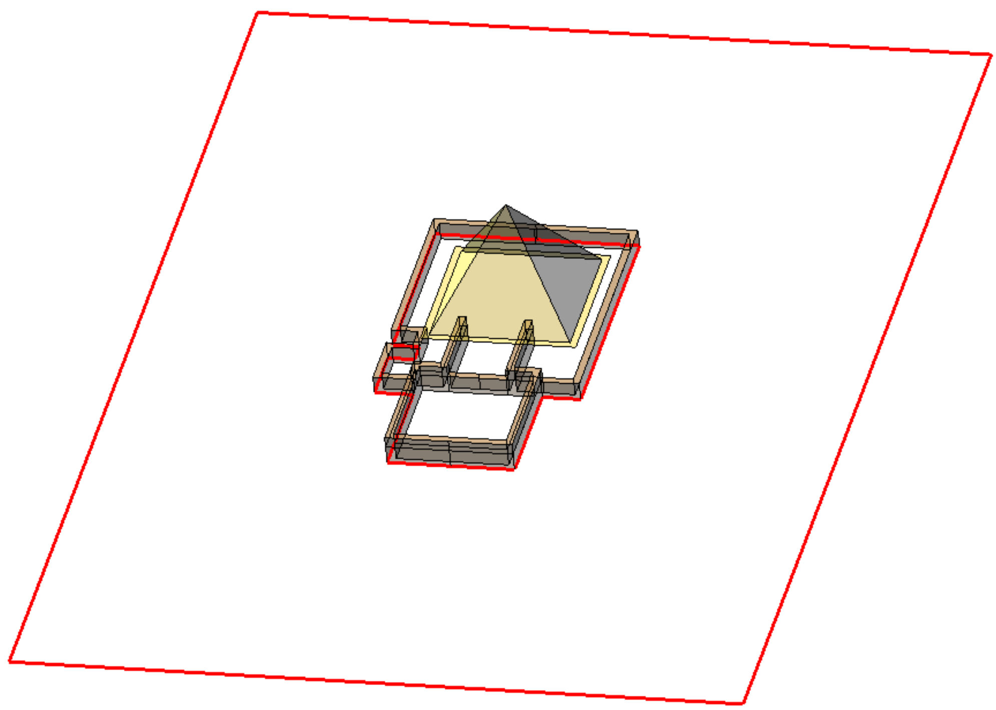
1 Set the 'ground' layer as the one in use.

2 Create the four lines showed in the figure, taking into account the coordinates of the vertex points.

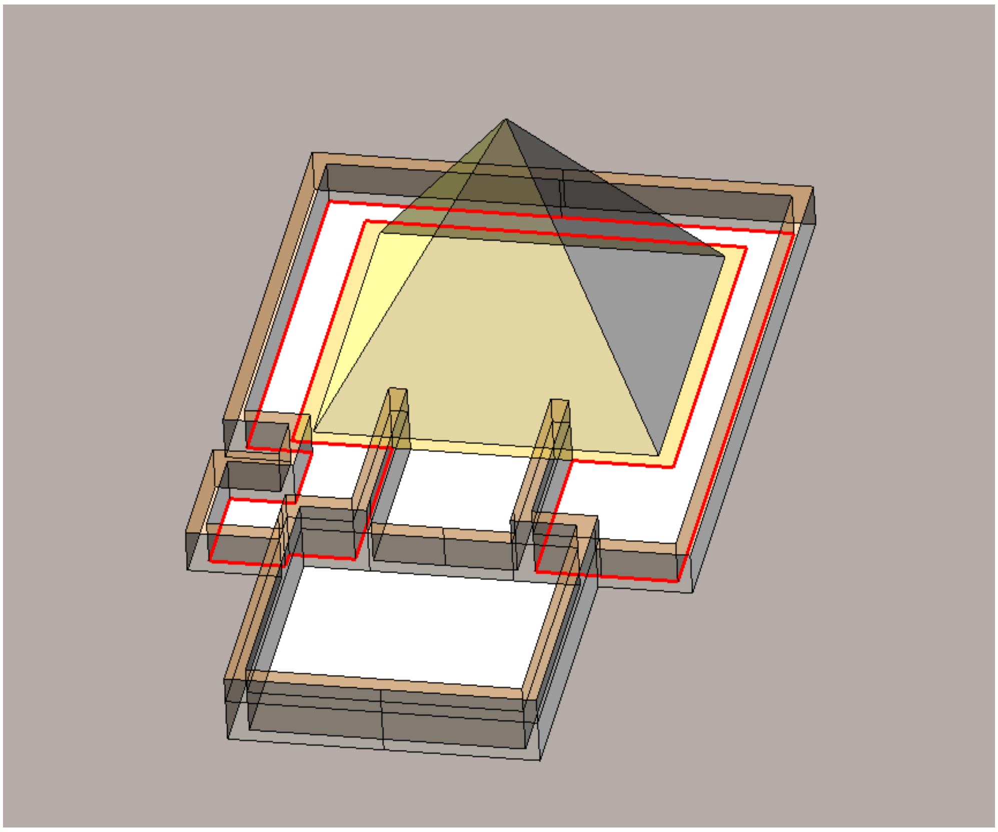


Lines defining the limits of the ground for the simulation.

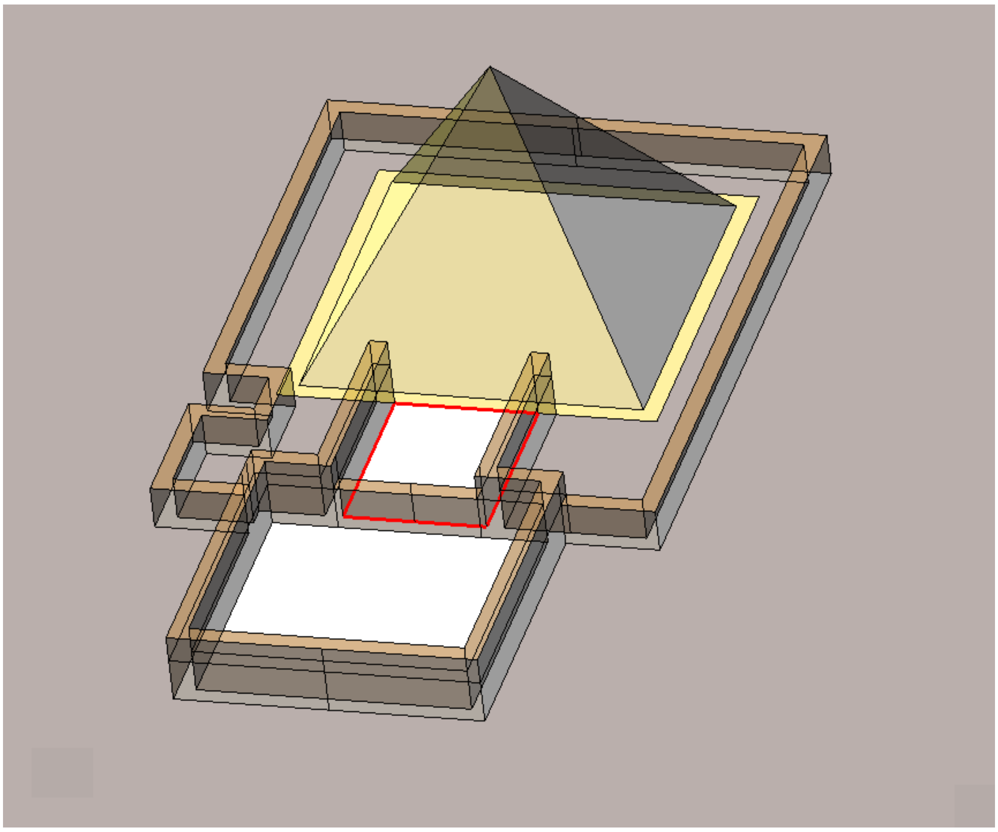
We have to create now 4 surfaces (using the methods explained before) defined by the lines showed in the next figures.



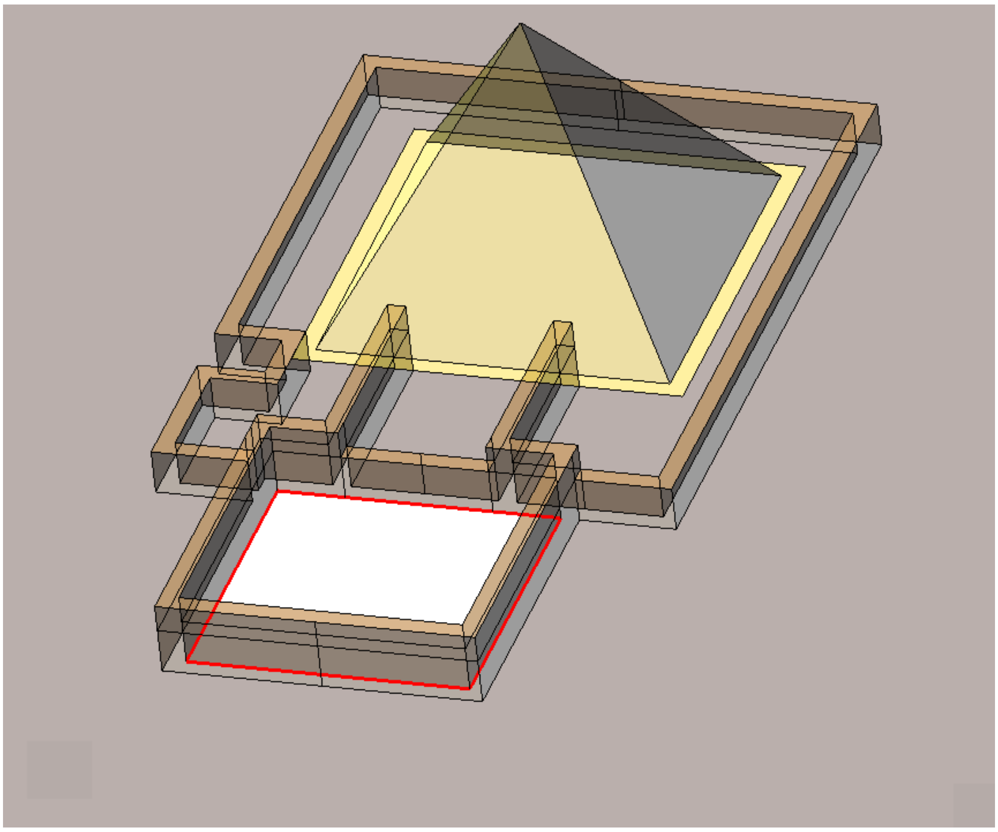
Lines which define the first ground surface to be created.



Lines which define the second ground surface to be created.



Lines which define the third ground surface to be created.

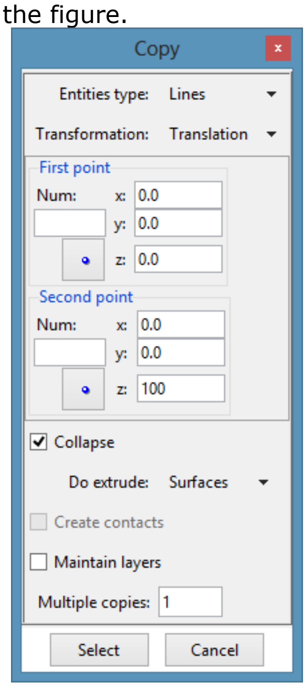


Lines which define the fourth surface to be created.

2.4.5 Creation of control volume surfaces

For creating the outer surfaces of the control volume we will use again the **Translation** tool of the **Copy** window.

1 Set the 'air//surfaces' layer as the one in use.
2 Open the **Copy** window, which is located in **Utilities->Copy**, and set all the values of the parameters as the ones showed in the figure.

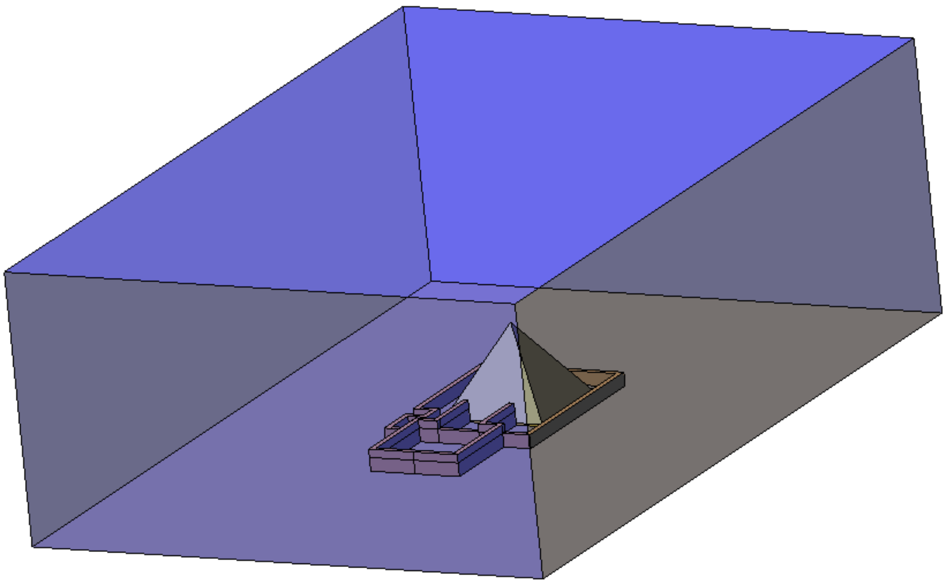


Parameters to be used for the extrusion of the outer lines of ground surface.

3 Select the 4 outer lines of the ground surface and press **Finish.**

Now the lateral surfaces of the control volume have been created.

4 Create the top surface of the control volume, and the model should be like the one showed in the figure 29.



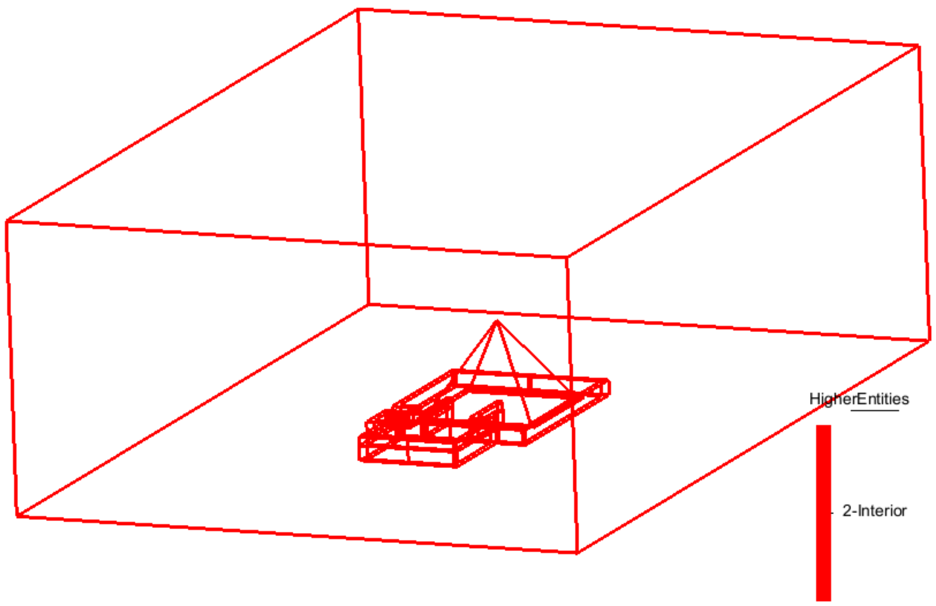
State of the model creation at this point.

Note that you can apply transparency to one Layer by clicking in the icon placed on the right part of the corresponding layer in the Layer Window.

2.5 Volume creation

Due to the hierarchical definition of the geometry inside GiD, a volume is needed to have a closed path of surfaces closing it. For ensuring that a path of surfaces is water-tight (closed), GiD offers one graphical tool (**Higher entities**) which is very useful.

1 Click on **View->Higher entities->Lines**. You should obtain the information of how many surfaces own each line. In this case all the values should be 2, like is shown in next figure.



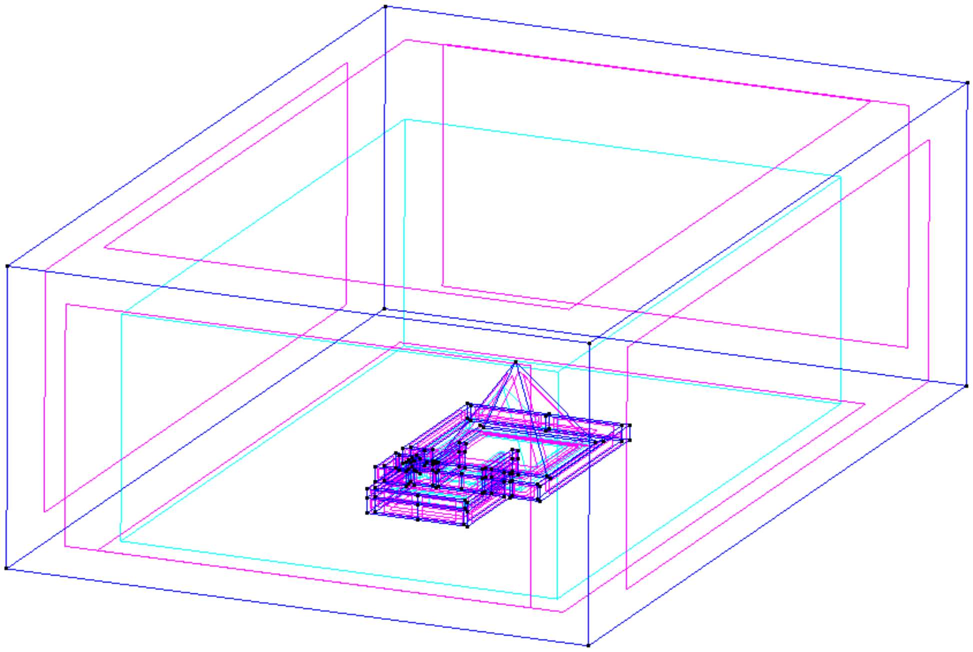
Higher entities of the lines of the model.

Press **ESC** to return to the normal render mode.

The last step for the geometry creation of this model is create the volume:

2. Select **Geometry->Create->Volume->By contour** and select all the surfaces of the model.
3. Then press **ESC**, and the volume should be created. Note that the volume (using GiD Normal render mode) is represented by a light blue line following the contour lines.

Now the geometrical model is finished, and should look like the one presented in the figure.



Geometrical model finished.

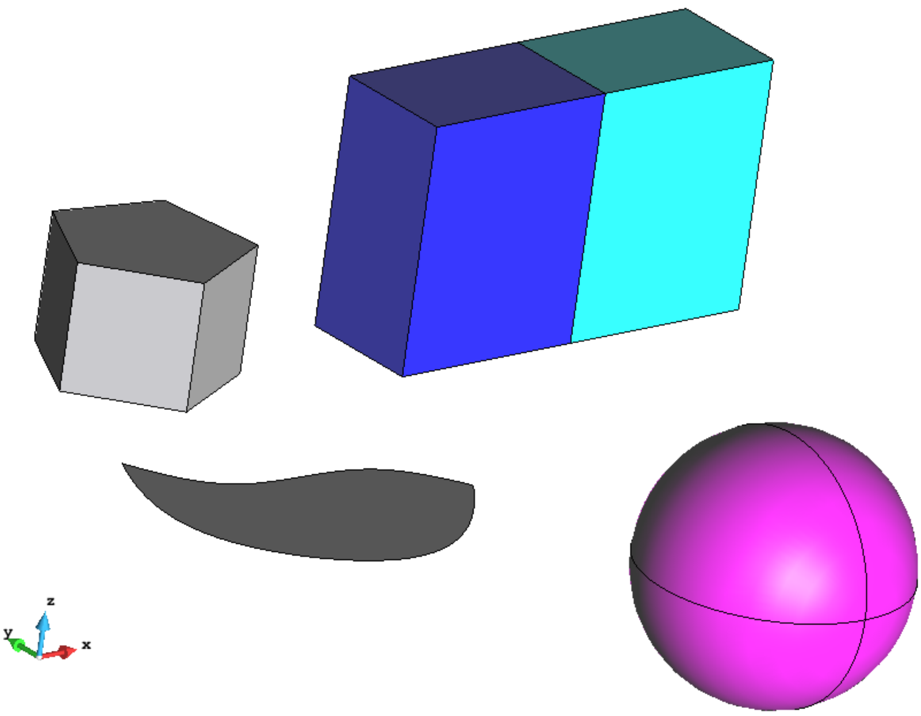
Save this gid project with **Files->Save as..** with the name 'Pyramid_geometry'

It is recommended to try to generate a coarse mesh to be sure that all is ok and they are not geometric problems for a future simulation.
E.g. use **Mesh->Generate mesh...** with a general meshing size=20 m, you must obtain a mesh with only tetrahedra elements.

3 Meshing basic features

This course is focused in the basic meshing features of GiD.

To follow this course, the model **gid_model_basic_meshing_course.gid** must be loaded from the folder where the courses material is.



View of the model used for this course.

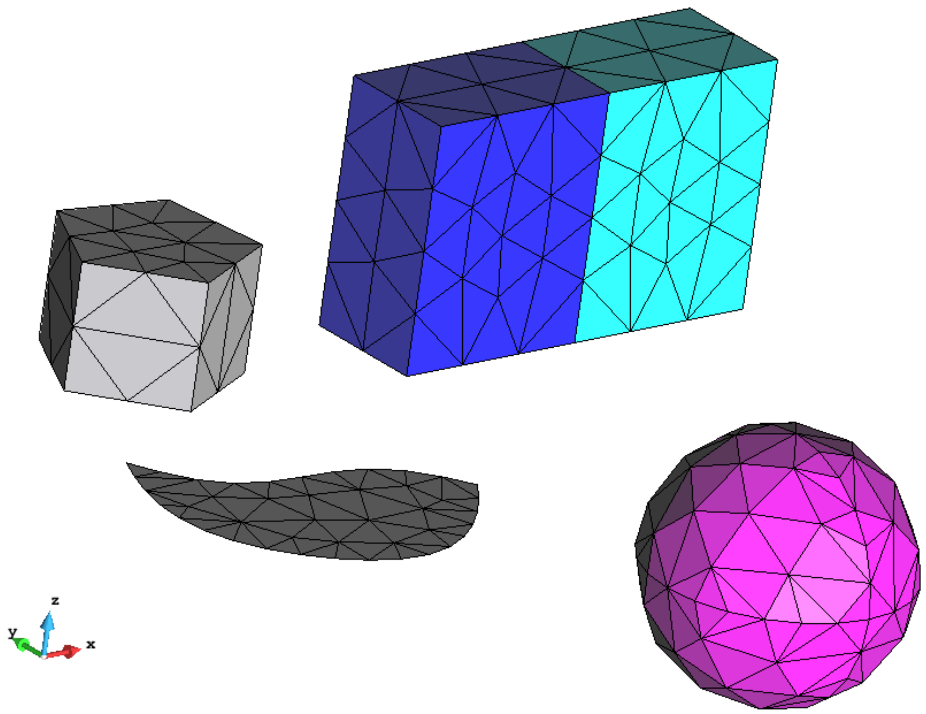
3.1 Default settings

First of all we should reset all the meshing preferences in order to ensure the meshing options got from GiD are the same as the ones needed to reproduce exactly this course. For this purpose:

- Open the preferences window (Utilities->Preferences)
- Set the 'Meshing' branch.
- Press the 'Default values' button.
- Press 'Accept' and close the preferences window.

If user does not assign any meshing property to the model, GiD assigns an automatic mesh size based on the size of the model. Elements by default are triangles (for surfaces) and tetrahedra (for volumes) and the mesh is unstructured. Let's have a look at this first automatic mesh.

- Select the option 'Generate mesh' from the 'Mesh' menu.
- A window will appear asking for the mesh size to be used. For this model, the value proposed by GiD by default is 1. In this window also appears the option 'Get meshing parameters from model'. If this option is set, GiD load the meshing parameters used the last time the model was meshed. Now unset this option, and click OK.
- A window will pop up giving the basic information of the result mesh (number of nodes and elements, memory consumed, etc.). Click on 'View mesh' and the following mesh should be shown:



Mesh got using the default options of GiD.

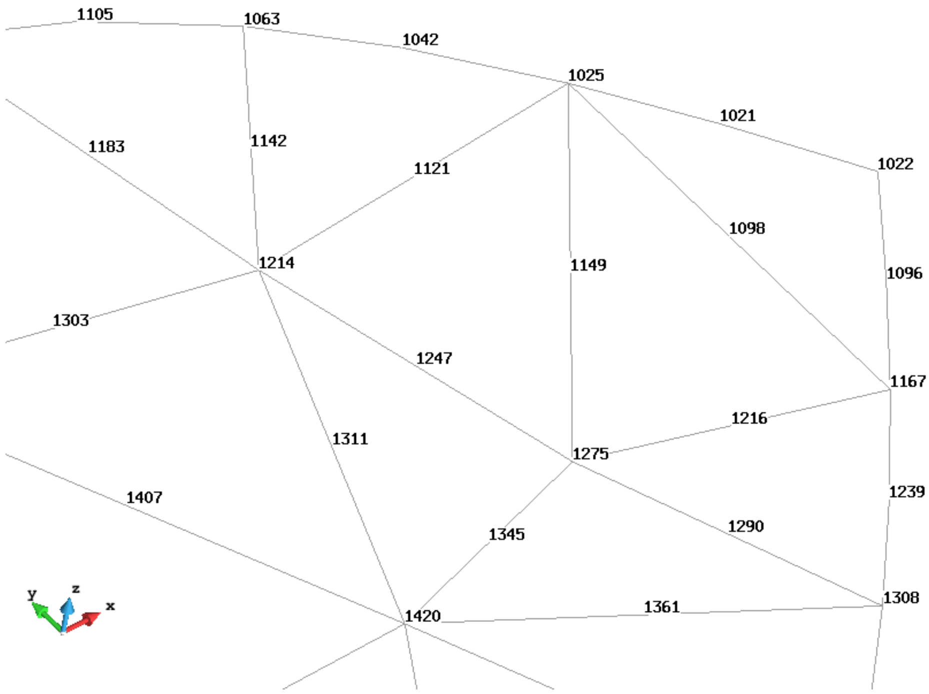
3.2 Quadratic type

User can choose between different quadratic element types by selecting the 'Quadratic type' option in the 'Mesh' menu.

- 'Normal' option corresponds to the linear elements.
- 'Quadratic' option corresponds to the linear element with quadratic nodes in the middle of its edges.
- 'Quadratic9' option is like the Quadratic one, but with quadratic nodes also in the center of the faces of the element, and the center of the element. This quadratic type is only applied to quadrilateral or hexahedra.

Let's make a quadratic mesh of the model.

- Select 'Quadratic' as the quadratic type in the mesh menu.
- Then generate the mesh again. The resultant mesh should look like the previous one, but if we zoom in at some area and we label all the nodes we should see that the elements are quadratic:



Detail of the mesh of a surface of the model, where all the nodes are labeled, so the quadratic nodes can be seen.

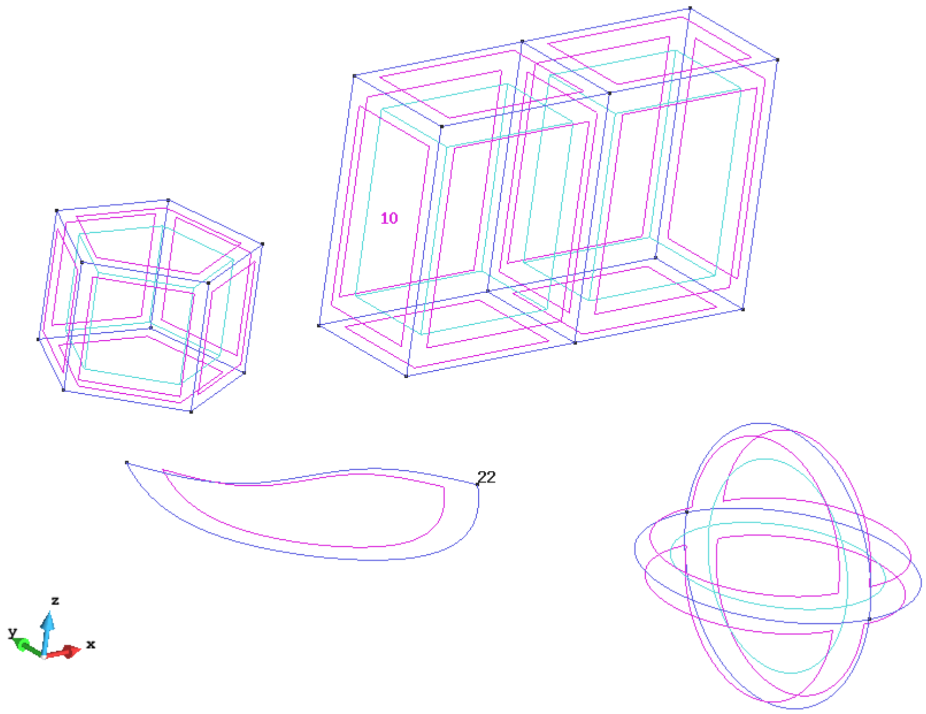
3.3 Types of meshes

Different kinds of meshes can be generated using GiD, depending on the topology required: Cartesian, structured, semi-strucrtured and unstructured.
Set as 'Normal' the Quadratic type of the mesh (from Mesh mesnu) in order to reach the same meshes as the shown in this chapter.

3.3.1 Unstructured mesh

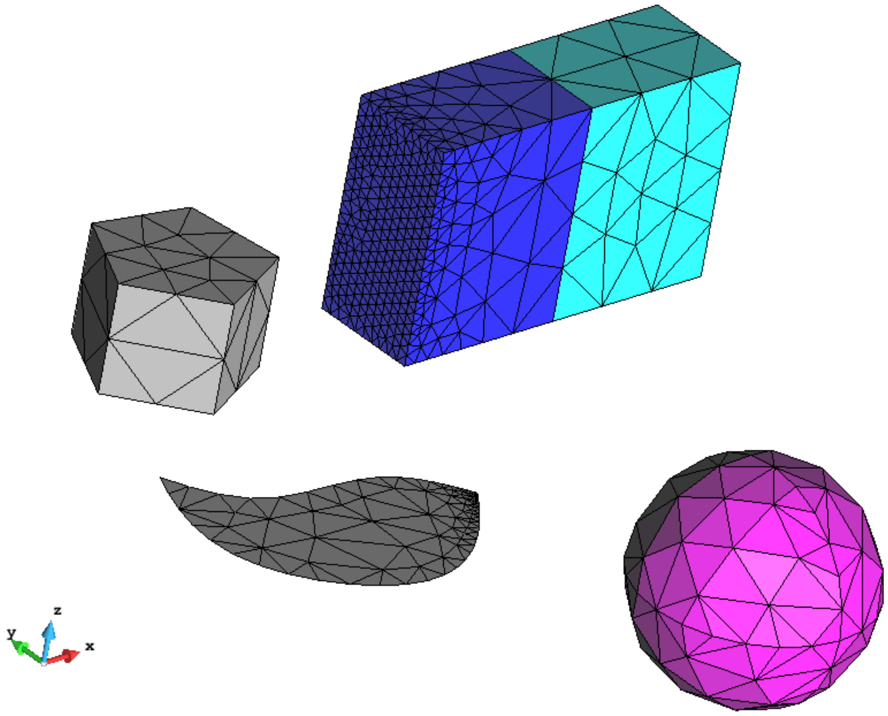
3.3.1.1 Assign sizes

As said in previous steps, the default mesh generated by GiD is a unstructured mesh, and a general size is set to the whole model. However, it is possible to refine specific regions of the model by assigning different sizes to the geometrical entities. Let's see it in an example where the entities labeled in the following figure will have different size assigned.



Entities to be selected in the following steps to assign mesh size.

- Select the 'Unstructured->Assign sizes on surfaces' in the 'Mesh' menu, and set 0.2 in the appearing window.
- Click on Assign and assign the surface number 10. Then press ESCAPE to finish the selection, and close the size window.
- Select the 'Unstructured->Assign sizes on points' in the 'Mesh' menu and set 0.05 to the point number 22.
- Generate the mesh again (Mesh->Generate mesh), and the following mesh should appear.

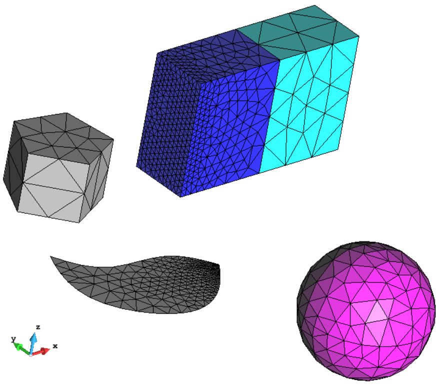


Result mesh of the model once some specific sizes have been assigned.

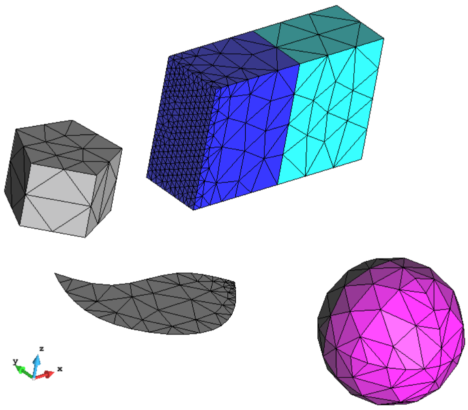
3.3.1.2 Size transition factor

When different unstructured mesh sizes are present in the model, a transition between them is applied in the unstructured mesh. User can control whether this transition is smoother or sharper by the 'Unstructured size transitions factor' present in the 'Meshing' branch of 'Preferences' window (Utilities->Preferences).

- Open the preferences window (in the 'Utilities' menu) and set the 'Unstructured size transitions factor' to 0.2. Click OK in the Meshing branch. Then generate the mesh again (Mesh->Generate mesh).
- Repeat the operation setting the 'Unstructured size transitions factor' to 0.9 to see the difference between the resultant meshes. Those meshes are shown in the next figure.



Mesh using 0.2 as unstructured size transition factor.



Mesh using 0.9 as unstructured size transition factor.

3.3.1.3 Surface meshers

This is a very simple geometrical model, but user may see the difference between the two main unstructured surface meshers available in GiD: RFast and RSurf.

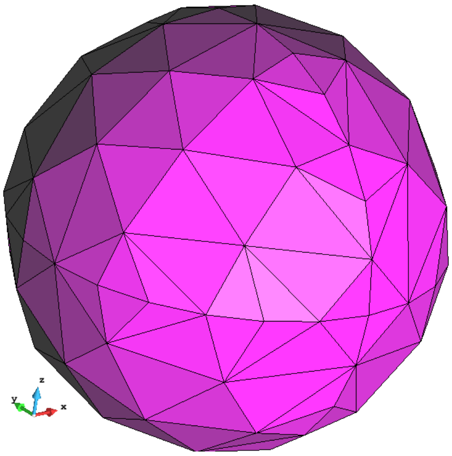
RFast mesher generates the mesh of the surface in the 2D iso-parametric space of the surface, and RSurf generates it directly in the 3D space. This characteristics gives to that meshers two basic properties:

- RFast mesher is faster, but in some cases it may give worse quality elements.
- RSurf mesher is slower, but in general gives a better quality elements, in special in case of curved surfaces.

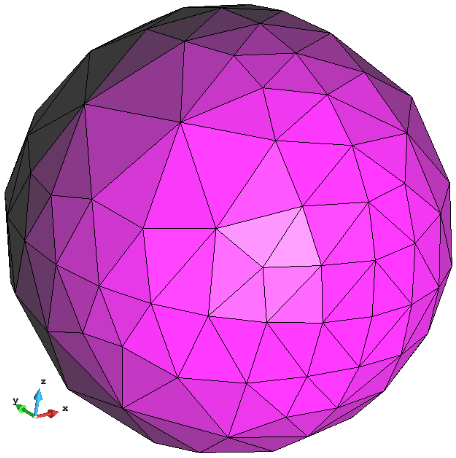
In this model the only curved surfaces are the ones of the sphere, and it is hard to see the difference of quality between both meshers. However we can see a slight difference between them.

- Open the 'Preferences' window (from the 'Utilities' menu) and set the RFast mesher in the Meshing branch. Set the 'Unstructured size transitions' factor to 0.6.
- Click Accept and generate the mesh.
- Repeat this process setting RSurf as surface unstructured mesher.

In the following figure the results meshes of the sphere are shown using RFast an RSurf meshers:



Mesh of the sphere of the model using RFast mesher.



Mesh of the sphere of the model using RSurf mesher.

In complex geometrical models in which the surfaces curvatures are important it is strongly recommended to use RSurf mesher in order to get a high-quality final mesh.

3.3.1.4 Volume meshers

GiD has three unstructured volume meshers (all of them generating tetrahedra): Advancing front, Tetgen and Octree.
The main characteristics of them are the following ones:

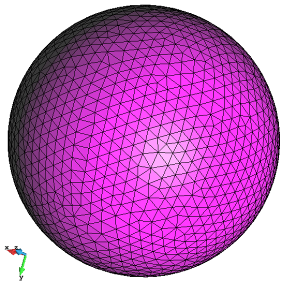
- Advancing front: it is based on advancing front technique. It may be slow, but it typically generates better quality of elements.
- Tetgen: it is based on Delaunay method, and it is faster than the previous one. In some cases it may lead to poor quality elements near the boundaries of the volumes.
- Octree: it is an octree-based mesher, which is very fast and robust. This mesher can skip lines which are part of the contours of the volume, if they have no specific property assigned.

First of all, reset the meshing data (Mesh->Reset mesh data).

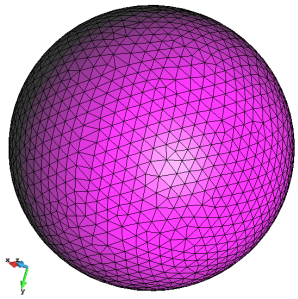
We will focus on the sphere volume (volume number 3). For this purpose we will not mesh the rest of the entities. For this purpose, set 'Mesh->Mesh criteria->No mesh' and select the other volumes of the model, as well as the isolated surface (surface number 18).

In order to appreciate better the difference between the difference volumes meshers, let's unset the 'Automatic correct sizes' in the Mesh branch of the Preferences window (set it to None). Set also as unstructured surface mesher the RSurf one.

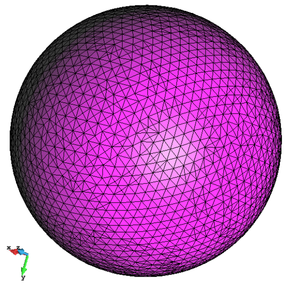
- Generate the mesh of the model with a size of 0.2 setting the three types of unstructured volume mesher (one mesh for each mesher). The results of the mesh are shown in the following figure:



Mesh generated with Advancing front mesher



Mesh generated with Tetgen mesher



Mesh generated with Octree mesher

One can appreciate that the different meshers generate different number of elements, and they are not equally fast.

Looking at these three meshes, the external view of the ones generated by Advancing front and Tetgen is the same, as they are constrained meshers and they use the same surface mesh as an input. This is not the case of the Octree mesh, where the regular grid of the octree structure can be appreciated in the nodes distribution.

It can be also seen that the lines of the volume surfaces are not represented in the mesh comming from the octree mesher.

3.3.2 Structured mesh

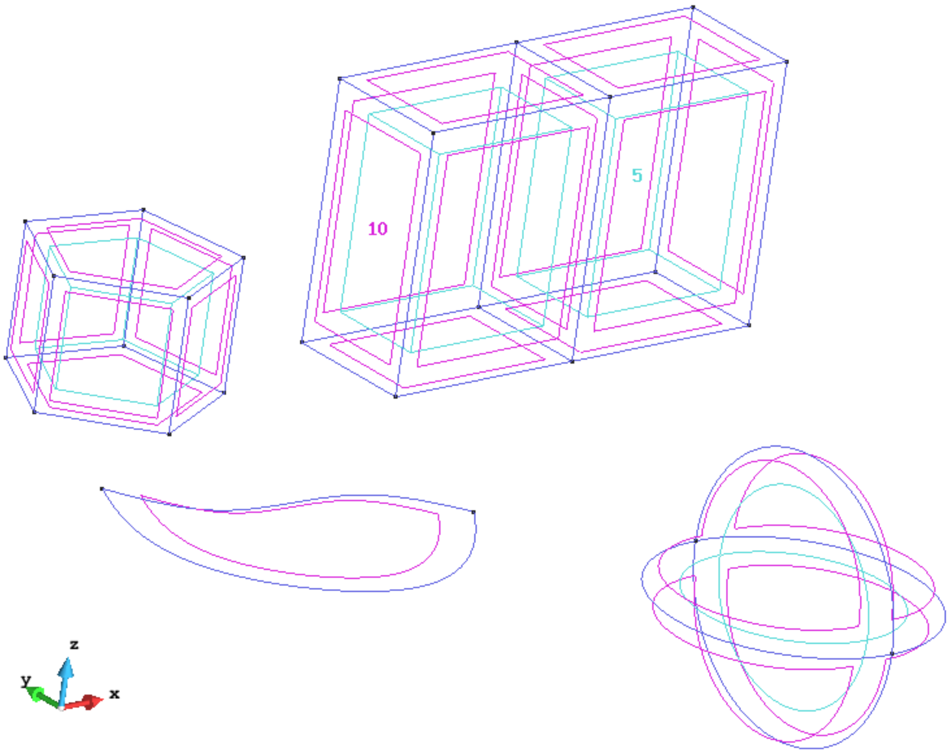
Structured meshes can also be generated in GiD. Structured meshes are the ones got from a predefined pattern, and they are restrictive in the topology of the geometrical entities they are applied in.

- Structured surfaces must be '4-sided' surfaces; this means that they must have four lines, or four clear angles close to 90 degrees, so as the mesher could make a topological correspondance between the contour of the surface and a square.
- Structured volumes must have 6 contour surfaces (the mesher will make a topological correspondance between the volume and a cube).

Let's see an example of how to assign this meshing property, but first of all it is better to unassign the previously assigned meshing properties to focus just in this structured part.

- In order to unassign the mesh properties assigned to the model click on the 'Reset mesh data' option in the 'Mesh' menu.

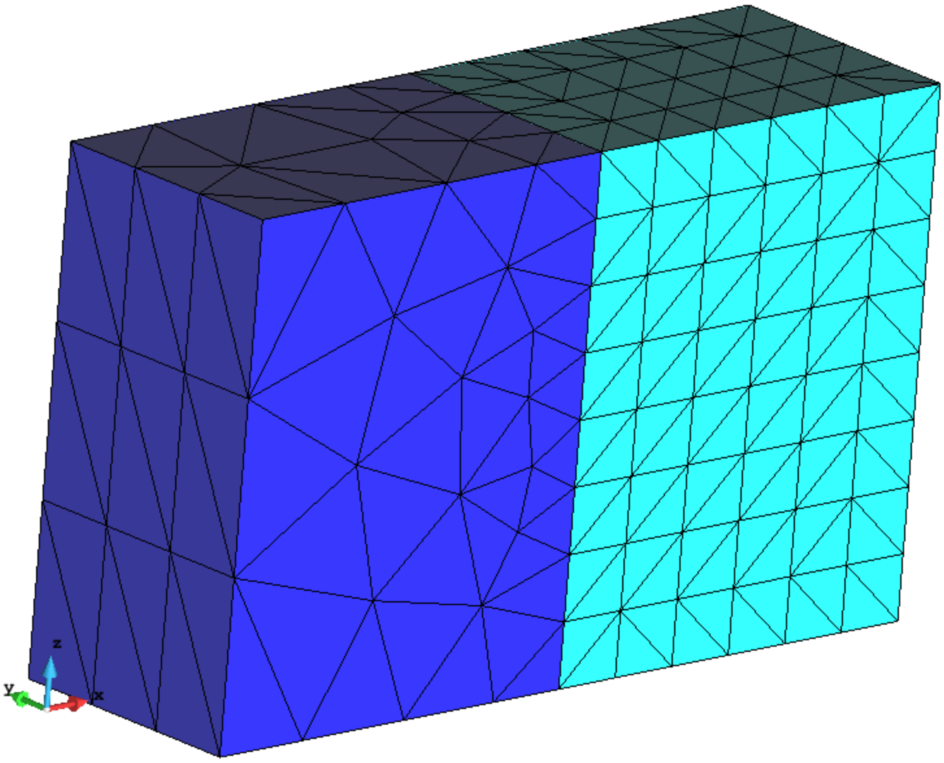
Lets assign structured meshing type to the volume number 5 and surface number 10 (shown in the following figure).



Entities to be set as structured.

In order to set a structured mesh onto a geometrical entity, the number of divisions (or the size) in its contour lines must be set.

- Select 'Mesh->Structured->Surfaces->Assign number of cells' and select the surface number 10. Then press ESCAPE to leave the selection mode.
- Set the number of divisions to 3 and select all its contour lines.
- Select 'Mesh->Structured->Volumes->Assign size' and select the volume number 5. Then press ESCAPE to leave the selection mode.
- Set a size of 0.5 and select the contour lines of the volume.
- Now generate the mesh again, and the result mesh should be like the one showed in the following figure:

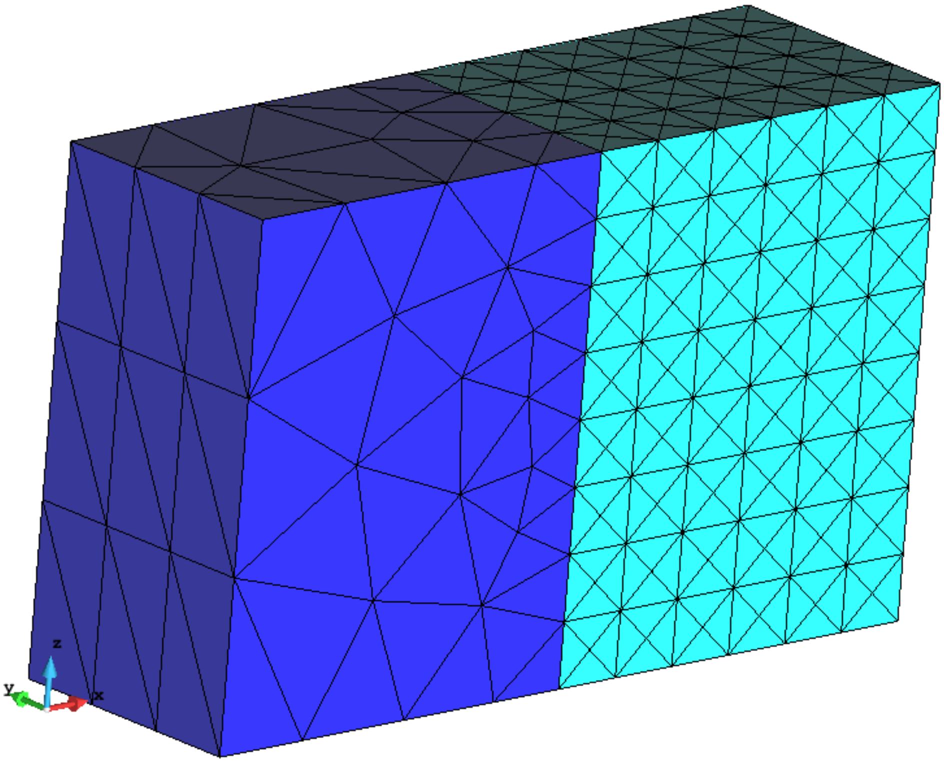


View of the structured mesh of some entities of the model

The structured pattern of elements is clearly recognized in the entities we have selected.

Different number of divisions can be set to the different structured directions of the entities. You can try to set different numbers and see how the final mesh changes.

In case of triangles and tetrahedra, the structured mesh comes from an internal splitting of an intermediate mesh made of quadrilateral or hexahedra. The way these elements are splitted can be symmetrical or not, depending on the 'Symmetrical structured' variable of the Meshing branch of Preferences window.
Setting the tetrahedra to be symmetric, for example, would give a mesh like the shown in hereafter:



View of the structured mesh of some entities of the model, seting as 'Symmetric structured' the tetrahedra.

3.3.3 Semi-structured mesh

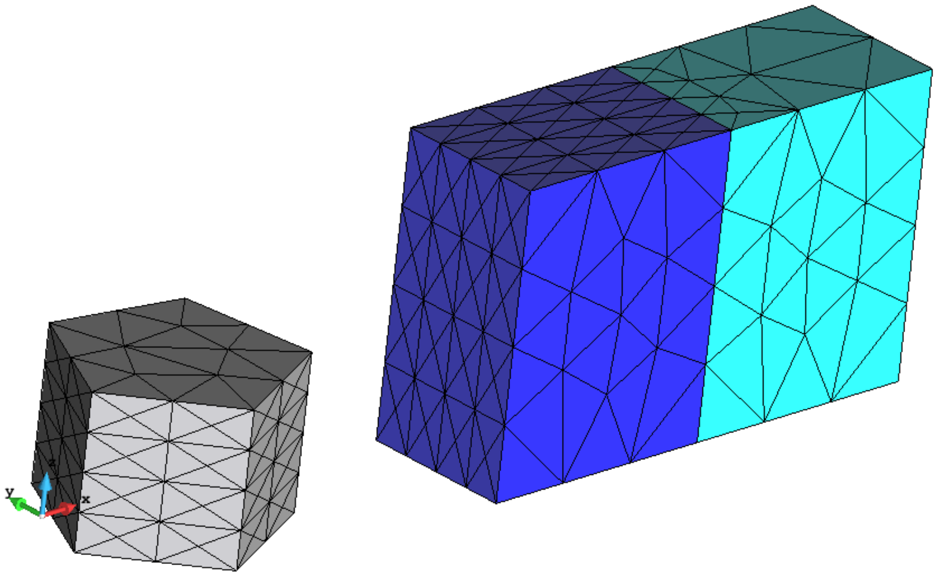
Semi-structured meshes can be applied to volumes. This kind of meshes follows a prismatic topology, so these volumes must be prismatic: two tops can be identified, as well as a number of lateral surfaces (which must be able to be meshed as structured).

In the model of this course, the volumes number 1, 4 and 5 are prismatic, so they can be meshed with semi-structured volume.

- Again, reset the mesh data previously assigned (Mesh->Reset mesh data).

Let's make semi-structured the volumes 1 and 4.

- Select Mesh->SemiStructured volumes and set as 4 the number of divisions to be assigned.
- Then select volumes number 1 and 4 and press ESCAPE to leave the selection mode.
- Generate the mesh. The result should be as the one showed in the following figure:

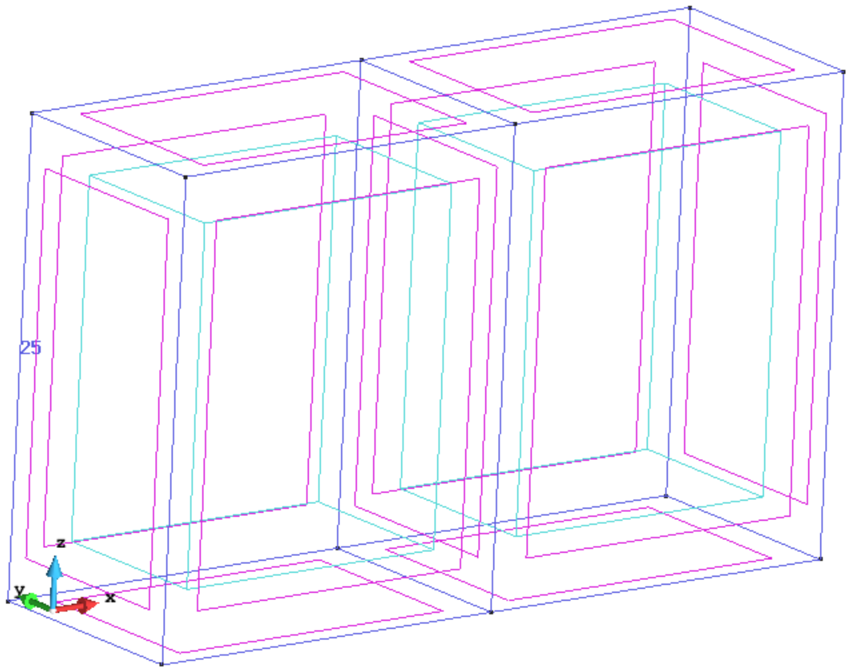


Detail of the semi-structured mesh of the volumes 1 and 4.

3.3.3.1 Change structured direction

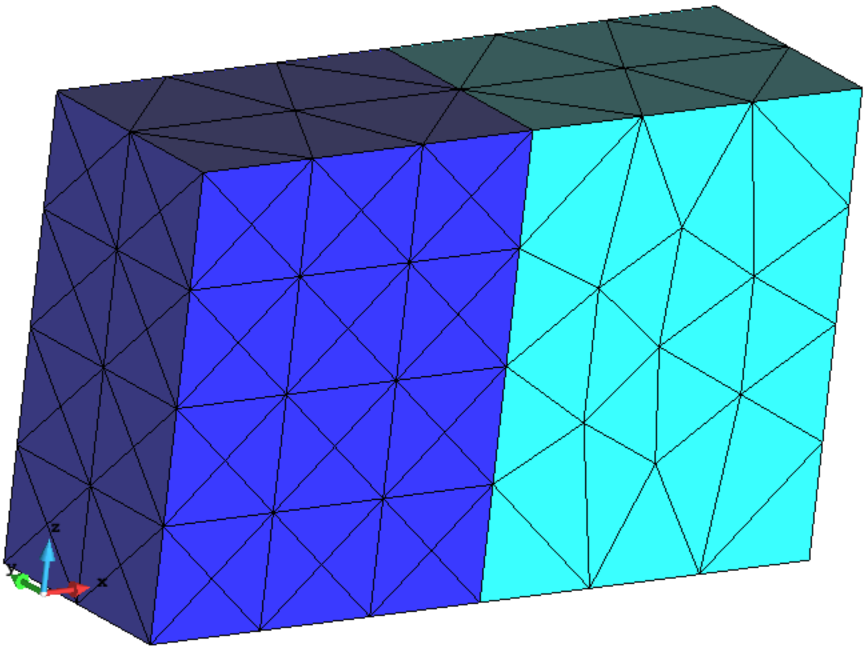
Volume number 1 is topologically prismatic only in one direction, but volume number 4 is prismatic in its 3 directions (as is has 6 boundary surfaces). The structured direction of volume 4 has been set automatically by GiD, but user may want to set another one.

- Select Mesh->SemiStructured->Set->Structured direction. Then select line 25 (the line labeled in the next figure).



Line 25, to be set as structured direction.

- Generate the mesh again. The resulting mesh should be as this one:



Semi-structured mesh when structured direction of volume 4 is the one defined by line 25.

GiD offers two options to force the structured direction of a semi-structured volume: set the structured direction via a line following that direction (the option just done before), or set one of the tops of the prism ('Mesh->SemiStructured->Set->Master surface').

3.4 Element types

The different element types GiD can generate are the following ones:

- Surface: triangle, quadrilateral, circle
- Volume: tetrahedra, hexahedra (only structured or semi-structured mesh), prism (only structured or semi-structured mesh), spheres and points.

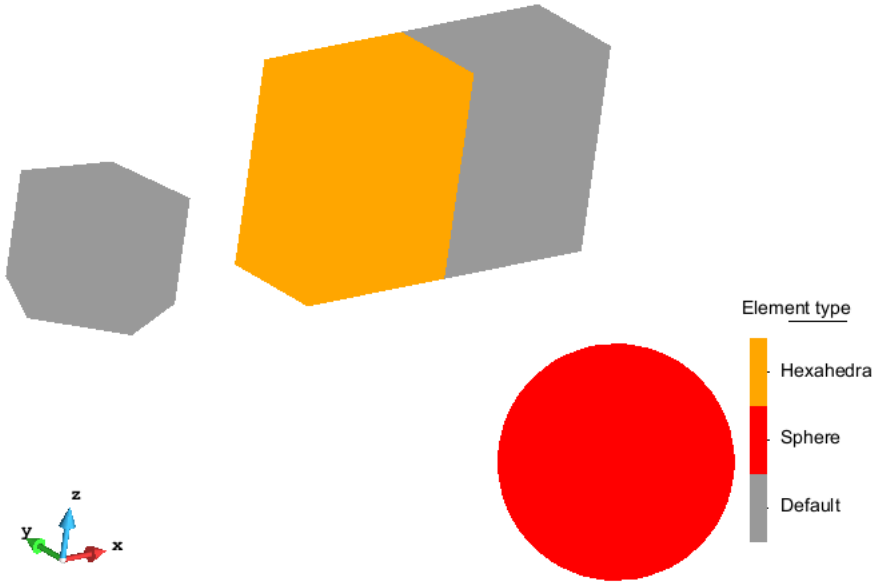
Apart from the specific requirements in terms of kind of mesh for the hexahedra and prism elements for volumes, selecting an element type is as easy as going to the 'Mesh' menu, setting the 'Element type' and select the desired entities.

In order to see different types of elements on this model follow this steps:

- Reset previous mesh data (Mesh->Reset mesh data).
- Set as structured volume number 4, with 5 divisions in its contour lines.
- Set as semi-structured volume number 5 with 5 structured divisions.
- Set Hexahedra as element type for volume number 4.
- Set 'Sphere' as element type of the spherical volume (volume number 3).
- Set 'Quadrilateral' as element type of surface number 18.

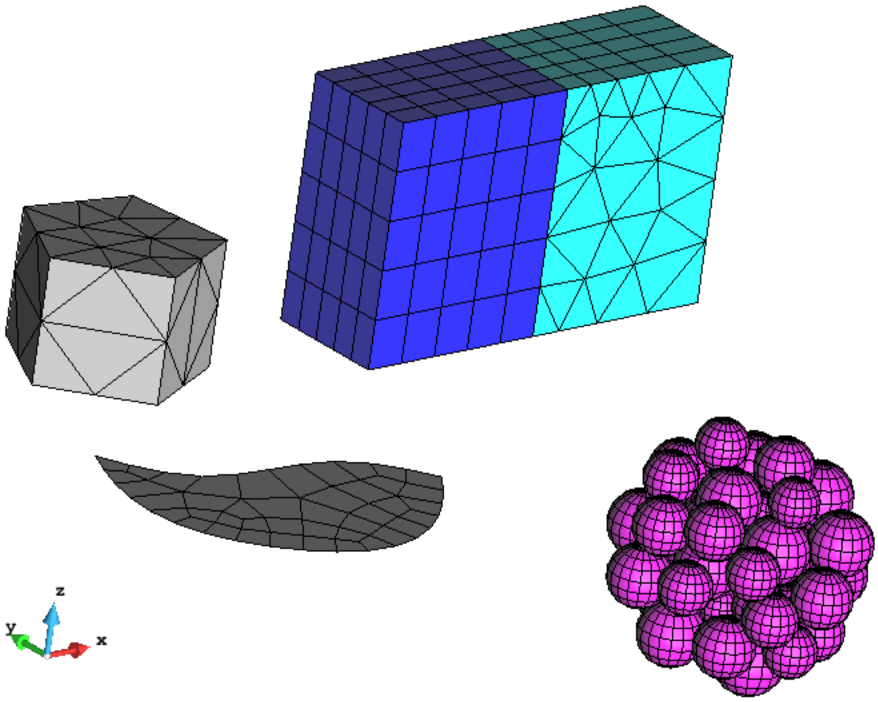
To check the element types (or other meshing properties) assigned to the model graphically, the 'Draw' option in the 'Mesh' menu is really useful.

- Select Draw->Element type->Volumes in order to check the element types assigned. The result image should be like the following one:



Different element types assigned to the model volumes shown using the Mesh->Draw option.

- Generate the mesh. The resulting mesh should be like the one showed in the following figure:



View of the mesh generated with the different element types assigned.

Note that the element type of volume 5 is prism, but the user has not set it. Just before mesh generation, GiD makes compatible the element types of the geometrical entities in order to fit the user requirements.

4 Run a CFD simulation

The model to be calculated is the flow of the air around a pyramid building.
Files->Open...
and select the pyramid_geometry model.
Then it is time to load the CFD calculation problem type and assign the material properties and the conditions to the geometry, so as the simulation could be run.

We won't focus this course in the simulation itself or the parameters needed for it, but only in the way of assigning properties and how to run a simulation within GiD.

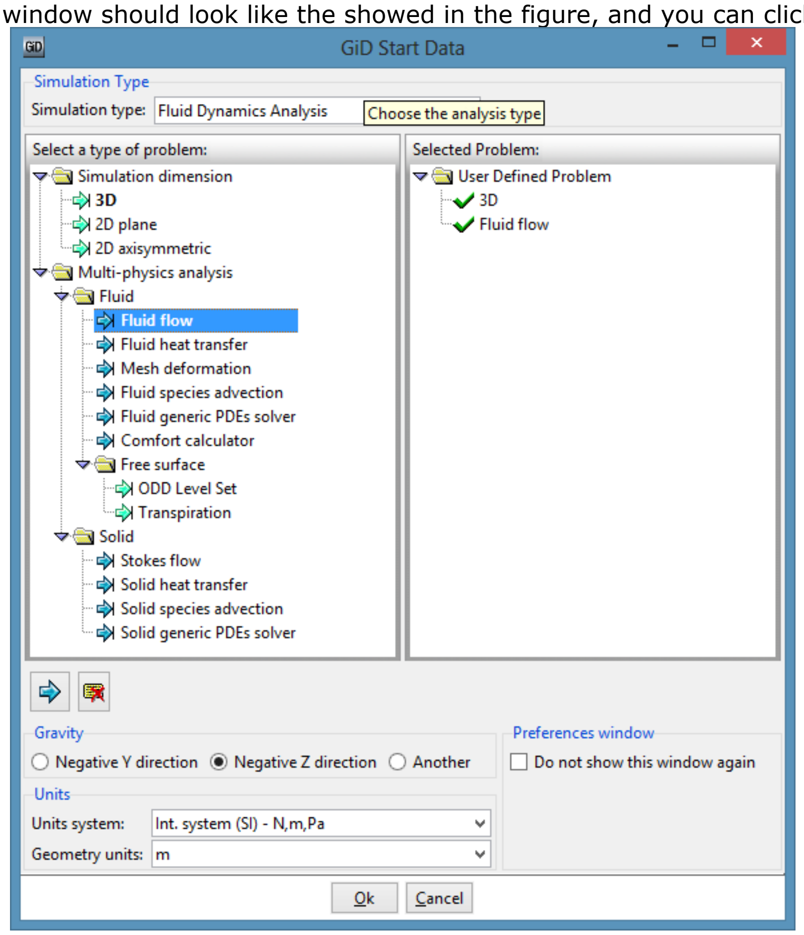
The boundary conditions and physical properties to use in this simulation are:

- Fluid physical properties: air at 25ºC.
- Density: 1.17 Kg/m^3
- Dynamic viscosity = 1.8e-5 Kg/m·s
- Fluid velocity=1 m/s in the Y axis direction. There are no flux through lateral walls of the control

4.1 Load problem type

For loading the problem type you should go to the **Data->Problem types** menu and select **CompassFEM12.4/compassfem**

The start data window of the problem type appears. We are going to run a simulation of a 3D fluid flow, so we must uncheck **Flow in Solids** option, which is in the Selected Problem part (right side of the window). The window should look like the showed in the figure, and you can click **OK**.



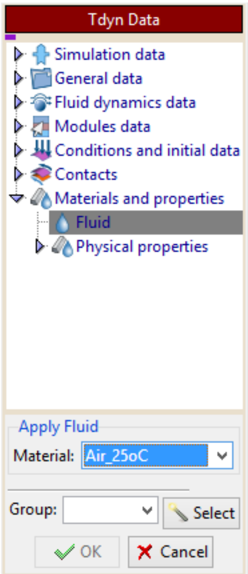
Options to be set when loading the CompassFEM module.

You can see that a new icons bar appear in the left side of the window. This is the problemtype toolbar.

4.2 Fluid material

For applying the fluid material properties click on the icon of the problem type toolbar.

You can see that a window (like the one showed in the figure) is opened with a data tree containing all the information required for the simulation.



Tdyn data window.

Click twice onto **Fluid** material, and you will see in the lower part of the window a field where to put the material. Select **Air_25oC** and select the volume of the model (click on **Select** and select the volume and press Finish to end the selection). Then press **OK** to accept the changes.

The properties of the fluid corresponding to Air_25oC seen and edited (if needed) by clicking on the Physical Properties branch of the data tree.

4.3 Fluid boundary

- Click on icon for applying the Fluid Boundary properties.

First of all we are going to force the fluid to have null velocity in the 'solid' surfaces of the model, which are the ones of the building and the ground.

- Click on **Wall/Bodies** and a window will appear in the lower part with some data.
- Select **V FixWall** in the **Bound.Type** field.
- Click on **Select** and select all the building surfaces as well as the ground surfaces.
- Click **OK**.

Now we are going to set the conditions for the outlet surface (the face of the control volume with maximum Y coordinate).

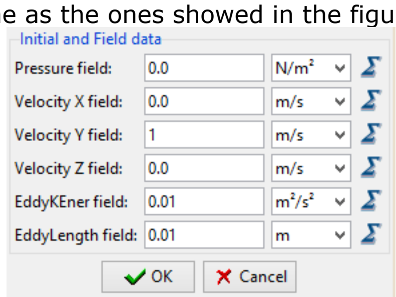
- Select the **Outlet** option (click twice) in **Fluid flow** branch.
- Ensure the options set are
    - Outlet of: **Fluid**
    - Bound. Type: **OutletPres**

  - Press Field: **0.0 Pa**
- Click on **Select** and select the outlet surface (the one with maximum y coordinate).
- Click **OK**.

4.4 Initial data

An initial uniform velocity with Y axis direction must be set as initial data (time=0).

- Click twice onto the **Initial and Field data** option of **Initial and Conditional data** branch of the data tree.
- Ensure the options are the same as the ones showed in the figure and click **OK**.



Initial data to be set for the simulation.

4.5 Boundary conditions

The velocity field must be fixed on lateral and top surfaces of the control volume to have a flux parallel to these surfaces. For this purpose:

- Click on **Velocity field** option (inside **Fluid flow** options in the data tree).
- Set the **Fix field X** option
- Click on **surfaces** type (this indicates the type of geometrical entities to be selected).
- Click on **Select**.
- Select the two lateral surfaces of the control volume.
- Click **OK**.

Repeat the same but now setting **Fix field Z** for the top surface

We should now fix the velocity field in the inlet surface (the one with minimum Y coordinate). We want to maintain its three components equal to the initial one (1m/s in +Y direction). For this purpose:

- Click on **Velocity field** and set as fixed the **Field** in **X**, **Y** and **Z** direction.
- Click on **surfaces** and select the inlet surface (the one with minimum Y coordinate).
- Click **OK**.

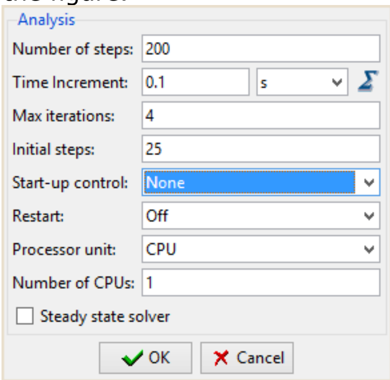
Now the velocity field is fixed, but as default it is fixed to 0. We must set the Y component equal to 1 m/s.

- Click on **Fix velocity component** and set the **Y Axis** to **1 m/s**.
- Click on **surfaces** and select the inlet surface.
- Click **OK**.

4.6 General data

Now it is time to set the general data for the solver, like solver parameters, etc.

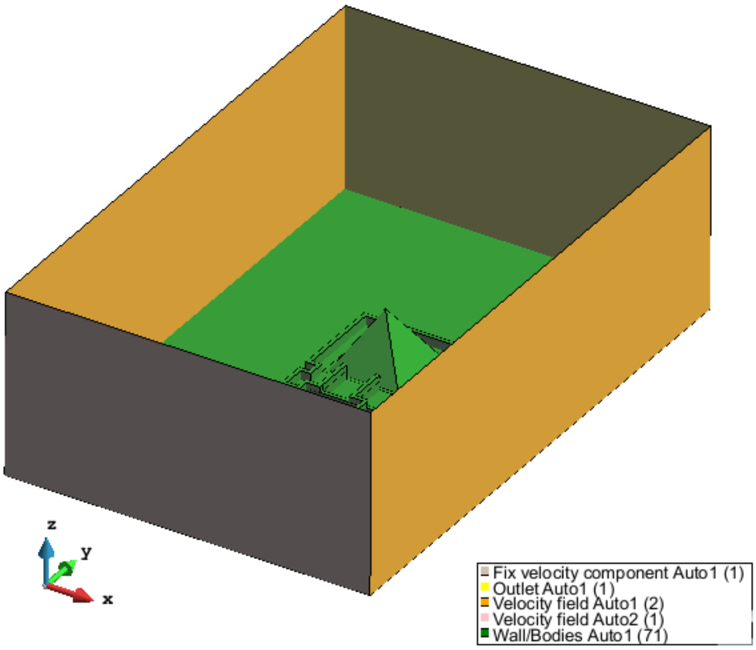
The default values in **General data** section of the data tree should be correct. We are going to modify some value inside the **Fluid dynamics data**. For this purpose click on **Analysis** part and fill the values like the ones showed in the figure.



Values for the Analysis data.

4.7 Check properties assigned

Clicking on the tree with the right-mouse button, user can check the information assigned to the model, drawing with colors the different conditions, materials, etc... (contextual menu: Draw->Draw groups...)



Groups with applied properties.

Tree items with data applied to groups are highlighted in bold characters

Note that Tdyn includes its own help menu, where there can be found the meaning of the fields of the conditions, materials, theory, etc., as well as some tutorials that can be interesting to learn to use it.

4.8 Calculation

It is necessary to generate a FEM mesh for the analysis, to do it press the icon and accept the default values.

Before beginning the calculation it is needed to save the model (**Files->Save as...**).
and save it as 'pyramid_tdyn'

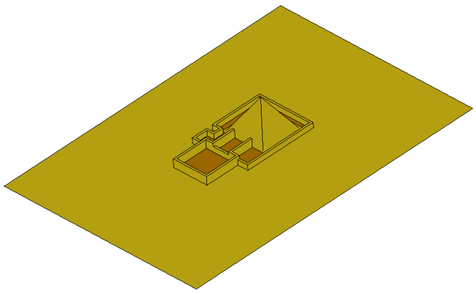
Then go to the menu **Calculate->Calculate**, and the calculation process will begin. User can check the status of the calculation by selecting **Calculate->View process info**.

Once the calculation is finished, a window appears. User can go directly to postprocess the results by clicking **Postprocess** in that window.

Tdyn has its own postprocess windows or it can use the 'Traditional post' (GiD standard postprocess) that will be explained in next tutorials.
To use the traditional post, it is necessary to set the **Fluid dynamics data**->**Results**->**Results file** to Binary1 or ASCII instead of Default, and recalculate.

5 Results visualization

5.1 Description



The objective of this course is to do a postprocess analysis of an already calculated fluid simulation. The simulation is the one run at the basic preprocess course, but we are going to open another model, which is already meshed (and the constraints are assigned) and also the results have been calculated. The reason for useing this other model is to ensure all the participants have the same results, which will make more easier to follow the course.

In this course, the project *pyramid.gid* will be used. The solver used to do this simulations is Tdyn3D, particularly the Ransol model. Tdyn3D is a fluid dynamic (CFD) simulation environment based on the stabilized Finite Element Method.

Steps followed in this course:

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

5.2 Loading the model

In this course we will use the pyramid project, so the steps to follow are:

1 . Start GiD
2 . Open the **pyramid.gid** project with: **Files->Open**, *Ctrl-o* or clicking on .
3 . Switch to postprocess mode:  or **Files->Postprocess**
4 . Select **Utilities->Preferences**
5 . In **General** branch choose **Normal** in the **Popup messages** option to avoid some popups messages

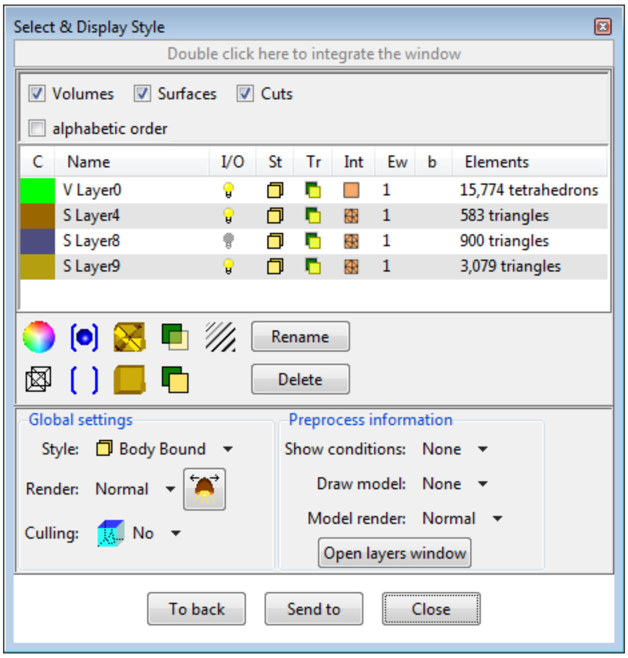
5.3 Changing mesh styles

Menu

Window->View style...

Description

1 . Select **Window->View style...** using the menu bar or clicking on 



Our model is composed by 4 layers. In order to get a better visualization we will disable the volume layer.

2 . Select the **VLayer0** layer and switch it **off**

3 . Select the **SLayer8** layer and switch it **on**

4 . Click on the icon in the **Tr** column or click on the icon

5 . Click on **Close** button

Play a little with the options of these windows, but to continue the tutorial, let a **Body Bound** style selected for all meshes

5.4 Viewing the results

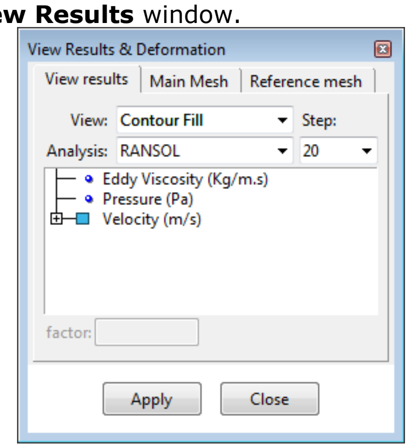
Menu

View Results

Window->View Results...

Description

Several results had been calculated for several time steps. You can check these results through the **Results** menu or opening the **View Results** window.



5.4.1 Iso surfaces and animations

Model used

The model `pyramid.gid` used in this example can be found at Material location.

Menu
View results->Iso Surfaces

Description
With this result's 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 . From preprocess mode open the model
2 . Switch to postprocess mode
3 . Turn off the **VLayer0** and **SLayer8** in order to get a better view
4 . Set as **Body Bound** the layers style
5 . Select the 12.5 step through **View results->Default Analysis/Step->RANSOL->12.5** or clicking on
6 . Select **View results->Iso Surfaces->Automatic Width->Pressure(Pa)** through the menu bar or clicking on

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).

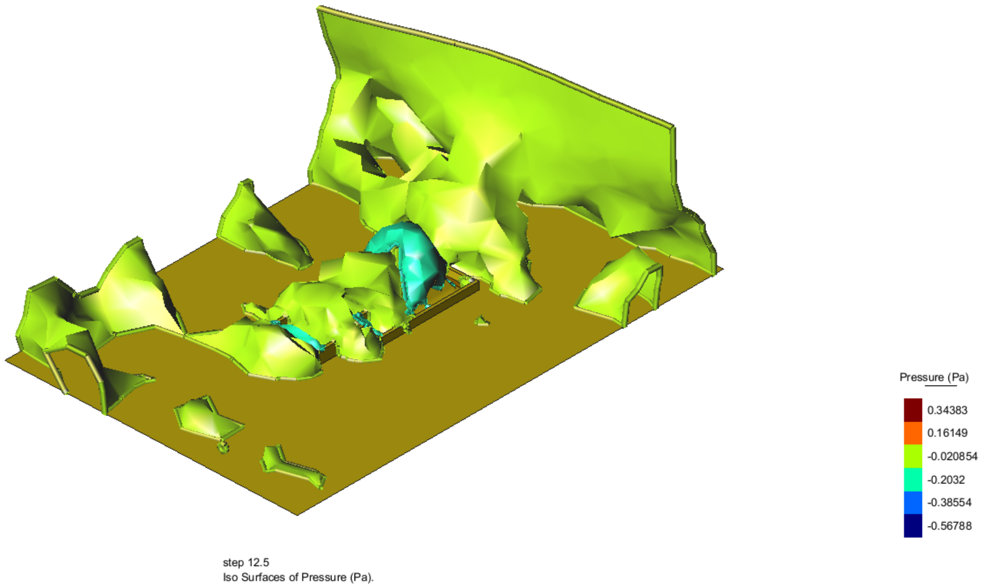
7 . Select the default value 0.182341

Several configuration options can be set via the Options menu.

Menu
Options->Iso surfaces

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

8 . Select **Options->Iso surfaces->Transparency->Transparent**
9 . Move the model to see the inner zones
10 . Select **Options->Iso surfaces->Transparency->Opaque**
11 . In order to improve the visualization and to get a more realistic view select **View->Render->Smooth**



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->Contour fill color** allows to draw the contour fill of any result over the isosurface. Select this option and then do a contour fill of any result.
- **Options->Iso surfaces->Show isolines** this options allows the user to switch isolines of surafaces on or off.

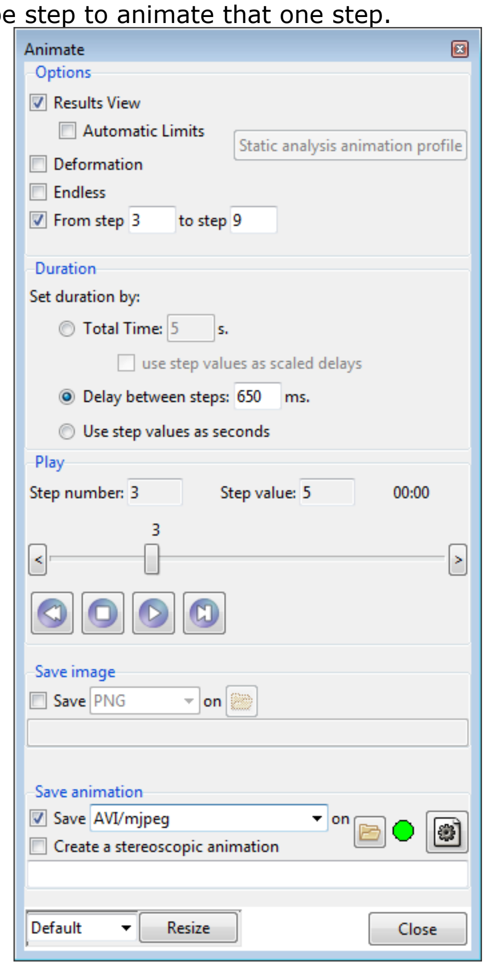
Menu

Window->Animate

Description

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.

12 . Select **Window->Animate...** to open the animation window

Please notice that we have from step 1 to 9. We will do the animation only of some of these steps.

13 . Check the **From step** option and set 3 **to step** 9

14 . Select the **Delay between steps** option and set it to 650 ms. The animation should take around 4 seconds

15 . Try it clicking on the **play** icon

We will record a video during the animation.

16 . Once the animation is finished check the **Save** option

You can choose from several video formats.

17 . Select **AVI/mjpeg**

18 . Please select a folder where the video will be saved clicking on the **folder** icon or writing the path

19 . Please 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

20 . **Close** the Animate window

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

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

5.4.2 Result surface

Model used

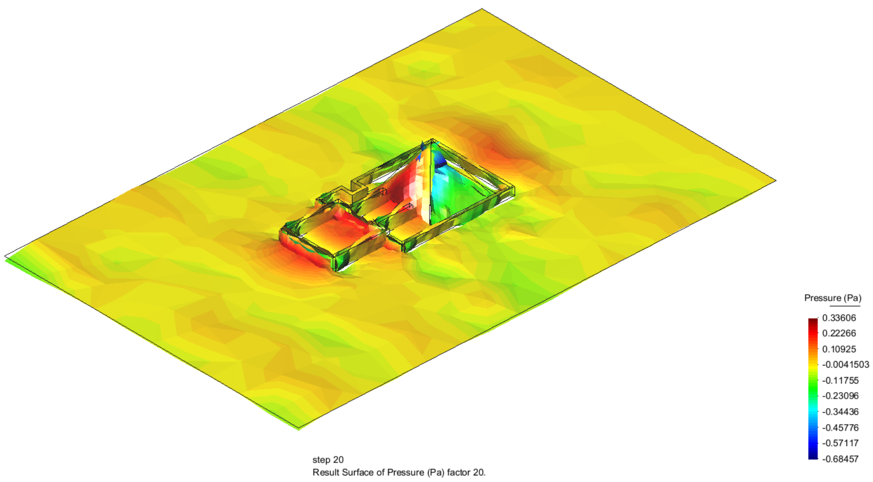
The model `pyramid.gid` used in this example can be found at Material location.

Menu

View results->Result Surface

Description

This option uses a result component, or a scalar value, and draws a 3D surface above the mesh following the normals of this mesh.



1 . From preprocess mode open the model

2 . Switch to postprocess mode

3 . In order to get a better view turn off the **VLayer0** and **SLayer8** layers

4 . Select **View->Render->Smooth**

5 . 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

6 . Enter **20** as factor in the command line

7 . Due we have positive and negative values please set as **Boundaries** the layers style. Now all the result surface can be seen easily

8 . Select **Options->Result surface->Show elevations->None**

9 . 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.

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

5.4.3 Contour fill, cuts and limits

Model used

The model `pyramid.gid` used in this example can be found at Material location.

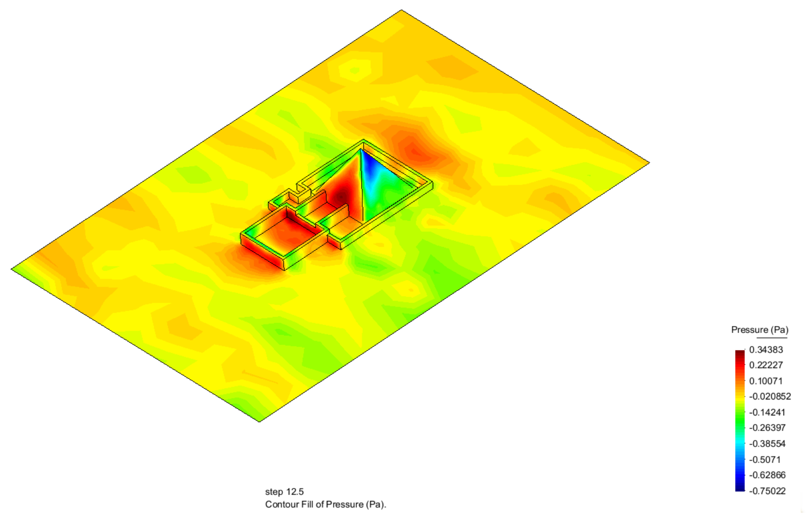
Contour fill

Menu

View results->Contour Fill

Description

This option allows the visualization of coloured zones, in which a scalar variable or a component of a vector varies between two defined values.



1 . From preprocess mode open the model
2 . Switch to postprocess mode
3 . In order to get a better view turn off the **VLayer0** and **SLayer8** layers
4 . Set as **Body Bound** the layers style
5 . Select **View->Render->Normal**
6 . Please select **View results->Contour Fill->Pressure (Pa)** through the menu bar, or clicking on or using the **Window->View results...** window

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 Colors**. 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.

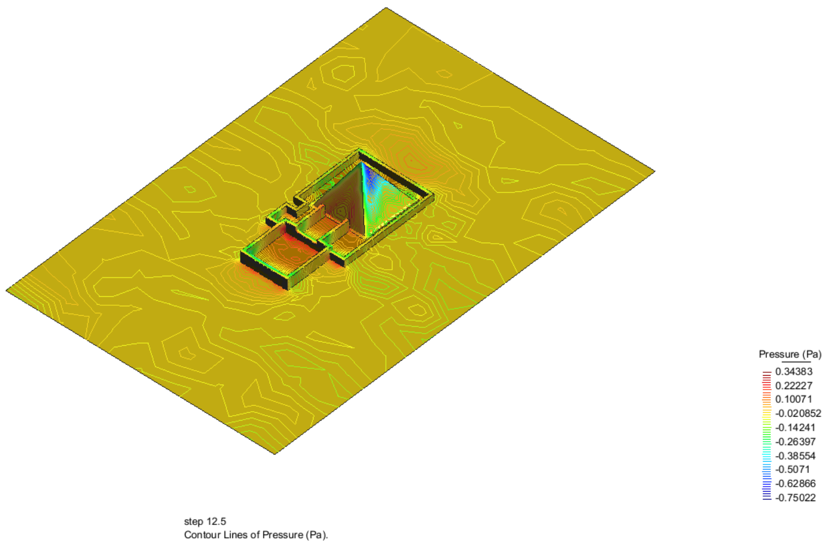
In the model the pressure has been calculated. We can visualize the result for each step in a contour fill.

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

7 . Select the step 12.5

Several configuration options can be set via the Options menu.

NOTE: Another similar result visualization is **Contour Lines** but in this case the isolines of a certain nodal variable are drawn. In this case, each color ties several points with the same value of the variable chosen.



Menu

Options->Contour

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

1 . Select **Options->Contour->Color Scale->Terrain map**

You can also set your own scale.

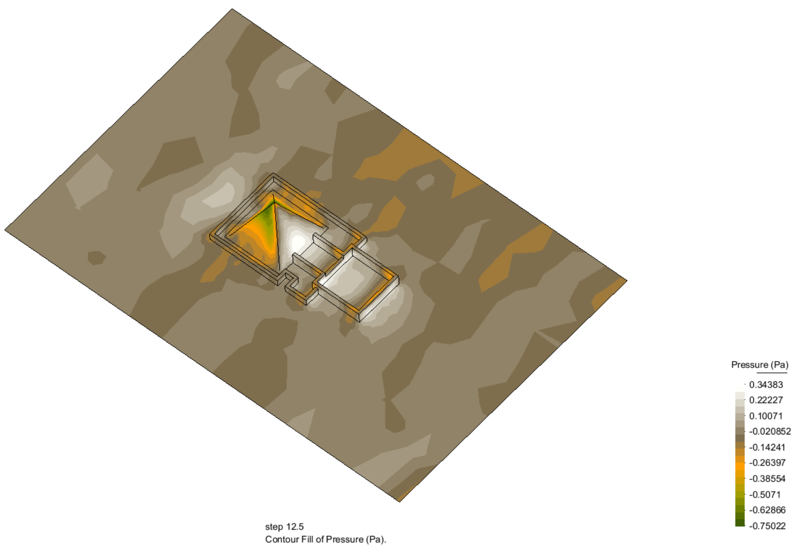
2 . Select **Options->Contour->Color scale->User defined...**

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 **20**

4 . Click on **Apply** button

5 . Click on **Close** button

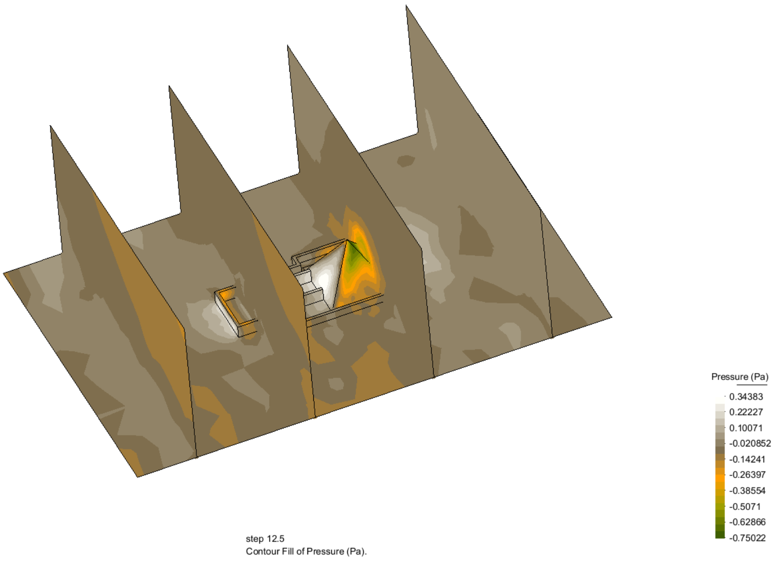


Cuts

Menu

Do cuts

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



We want to cut the inner volume in order to see the pressure in the air.

First of all we have to make visible the volume.

1 . Select **Window->View style...**
2 . Select **VLayer0** and make it **visible** clicking on the icon
3 . Select **Do cuts->Cut plane->Succession** through the menu bar or clicking on and then

With the **succession** option you specify a line which will be used as axis to create cut planes orthogonal to this axis.

NOTE: after clicking the first point, you can press the *Alt* key to snap the dynamic line to the screen horizontal, vertical or 45º diagonals.

The number of planes is also asked for.

4 . The axis is defined by two points, please write the first one in the command line **30 200 0**
5 . You are asked for the second point, introduce **30 -100 0**
6 . Choose **4** cuts. You should obtain 4 parallel planes to Y axis.
7 . Select **Window->View style...** You can see that several layers had appeared a prefix like **CCutSetX** indicating which mesh or set has been cut. These names can always be changed through this window
8 . Set off the **VLayer0** again through the **View style** window. You can rotate the model in order to see the contour fill result on the cut planes.
9 . In the same window select all the **CCutSetX** and click on **Delete** button in order to delete all the cuts.
10 . Select **Yes**
11 . 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.

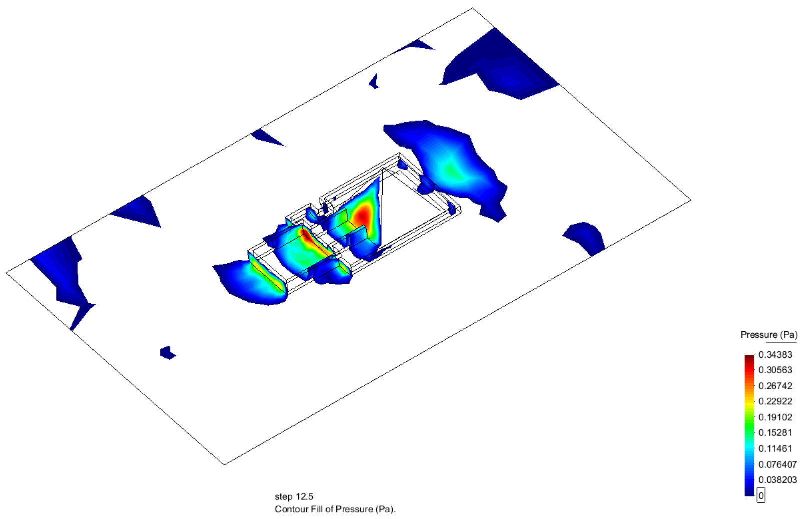
1 . Select **Options->Contour->Define Limits...** through the menu bar or clicking on

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**



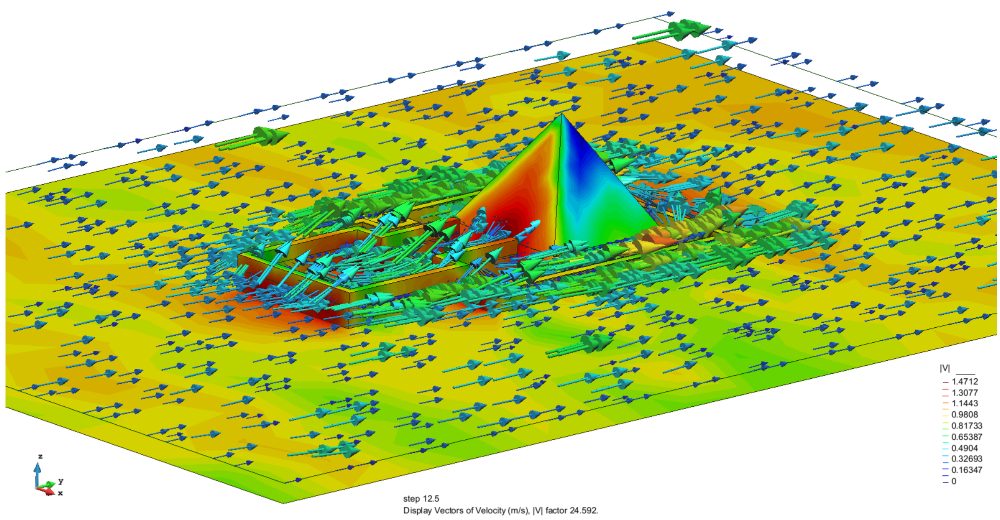
7 . Select **Options->Contour->Reset Limit Values**
8 . Select **View results->No Results**

5.4.4 Combined results

Model used
The model `pyramid.gid` used in this example can be found at Material location.

Menu
Window->Several results...

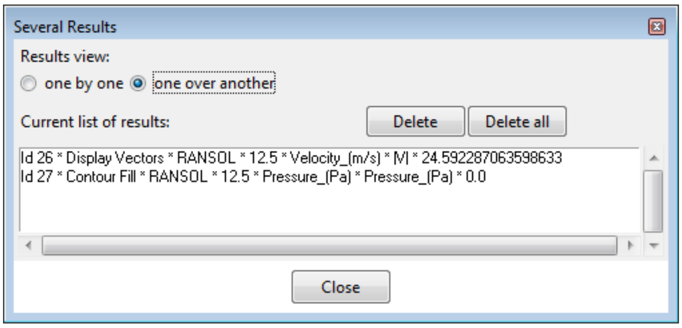
Description
Through this window you can select several results in order to visualize them at the same time. From this window you can also delete the undesired results visualizations.



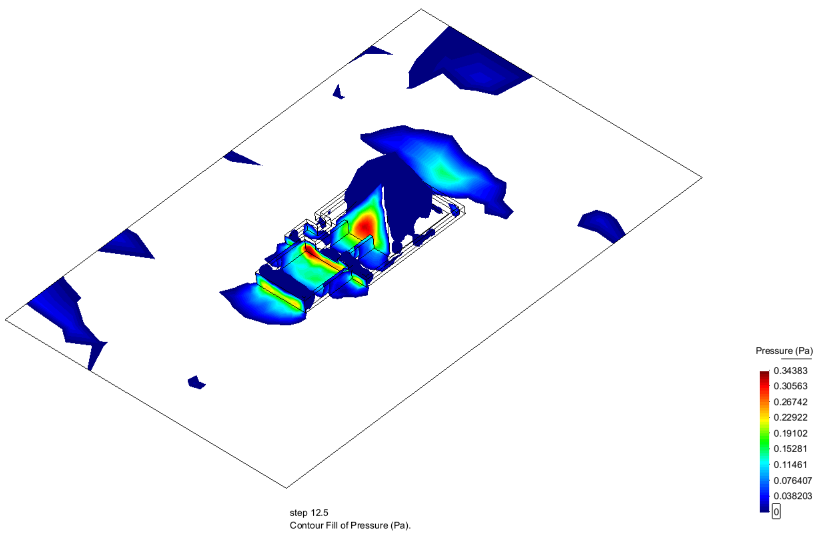
1 . From preprocess mode open the model
2 . Switch to postprocess mode
3 . In order to get a better view turn on all the layers except **SLayer8**
4 . Set as **Body Bound** the layers style

In order to see the inner vectors first we will do a cut through the volume. We want a cut parallel to the XY plane and near to the pyramid base.

5 . Select **View->Rotate->Plane YZ** to get a proper view to make de cut
6 . Select **Do cuts->Cut Plane->2 points**
7 . Select 2 points by your own near the pyramid base
8 . Press **ESC** to leave the cut function
9 . Turn off the **VLayer0**
10 . Change the cut layers style to **Boundaries**
11 . Select **Window->Several results...**
12 . In this window select **one over another**. With this option GiD is told to visualize one result over another
13 . Select **View Results->Default Analysis/Step->Ransol->12.5**
14 . Select **View Results->Contour Fill->Pressure**
15 . Select **View results->Display Vectors->Velocity (m/s)->|V|**
16 . Select **Options->Vectors->Color Mode** and click **Colour Module**

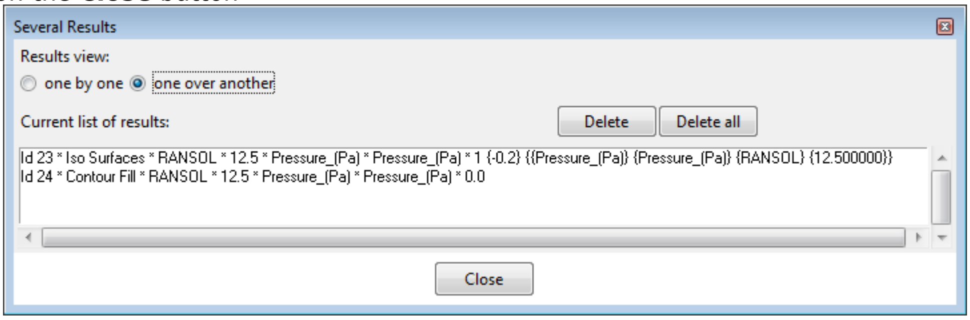


17 . Select **View results->No Results**



18 . From preprocess mode open the model
19 . Switch to postprocess mode
20 . In order to get a better view turn off the **VLayer0** and **SLayer8** layers
21 . Set as **Body Bound** the layers style

22 . Select **Window->Several results...**
23 . In this window select **one over another**. With this option GiD is told to visualize one result over another
24 . Select **View Results->Default Analysis/Step->Ransol->12.5**
25 . Select **View Results->Iso surfaces->Exact->Pressure** through the menu bar or clicking on the
26 . In the following questions: How many **isosurfaces**? Enter **1** and click **Ok**
27 . Enter the 1 value ...? Enter **-0.2** and click **Ok**
28 . Select **View Results->Contour Fill->Pressure**
29 . 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.
30 . Check the **Min** checkbox
31 . Change the value to **0**
32 . Click on the **Apply** button
33 . Click on the **Close** button



34 . Select **Options->Contour->Min options->Out min color->Transparent**
35 . **Close** the **Several Results** window
36 . Select **Options->Contour->Reset Limit Values**
37 . Select **View results->No Results**

5.4.5 Show min max

Model used

The model `pyramid.gid` used in this example can be found at Material location.

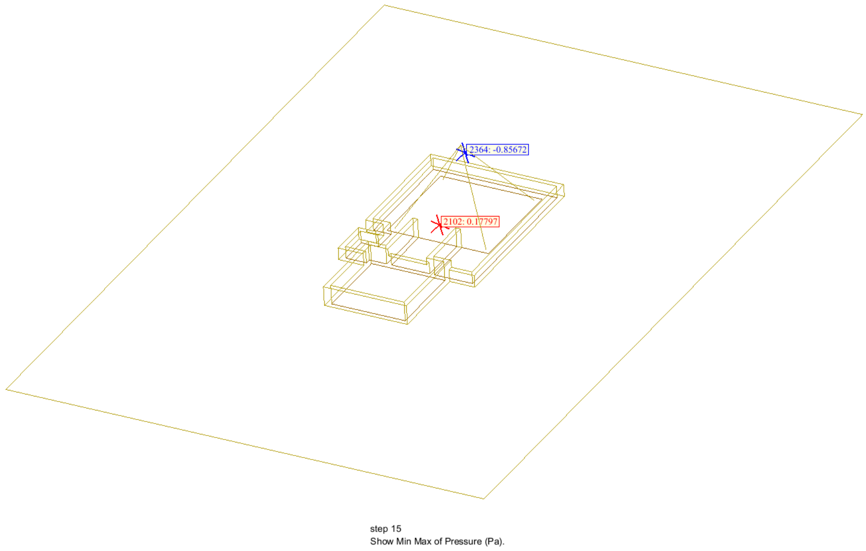
Menu

View results->Show Min Max

Description

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 Vy component of velocity result for the first analysis step.

1 . From preprocess mode open the model
2 . Switch to postprocess mode
3 . Turn off the **VLayer0** and **SLayer8** in order to get a better view
4 . Change the style to **Boundaries** for all the layers
5 . Select **View results->Default Analysis/Step->RANSOL->15** throught the menu bar or clicking on
6 . Select **View results->Show Min Max->Show both->Pressure (Pa)** throught the menu bar or clicking on . The label shows the node number and the value of the result
7 . Select **View results->No Results**



5.4.6 Display Vectors

Model used

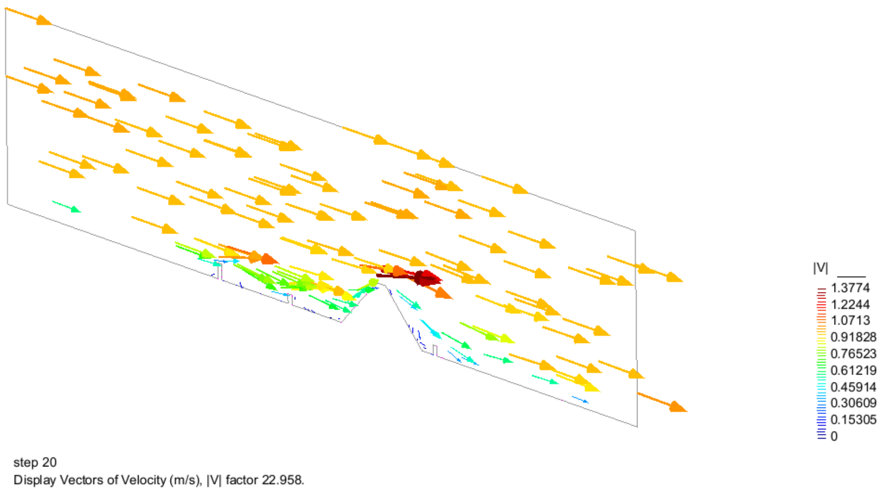
The model `pyramid.gid` used in this example can be found at Material location.

Menu

View results->Display Vectors

Description

With this display option the nodal vectors of the chosen result are shown.



We want to display the vectors of velocity in a cut.

1 . From preprocess mode open the model
2 . Switch to postprocess mode
3 . Turn on all the layers except **SLayer8**
4 . Select **View results->Default Analysis/Step->RANSOL->20** throught the menu bar or clicking on 

In order to see the inner vectors first we will do a cut through the volume. We want a cut parallel to the YZ plane and near to the pyramid center

5 . Select **View->Rotate->Plane XY (Original)** to get a proper view to make de cut
6 . Select **Do cuts->Cut Plane->2 points**

7 . Select 2 points by your own or enter the following coordinates in the command line. 30 200 and 30 -100
8 . Press **ESC** to leave the cut function
9 . Turn off all the layers except the ones with name **CCutSetX** in order to only see the cut
10 . For all the cut layers change the style to **Boundaries**
11 . Select **View results->Display Vectors->Velocity (m/s)->|V|**

We can set some options
12 . Select **Options->Vectors->Color Mode** and click **Colour Modules** in order to see the vectors by colors depending on their value
13 . Select **Options->Vectors->Number of Colors**, enter 50 and click **Ok** to get more accuracy
14 . Select **Options->Vectors->Filter factor**, enter 5 and click **Ok**. This option changes the number of displayed vectors

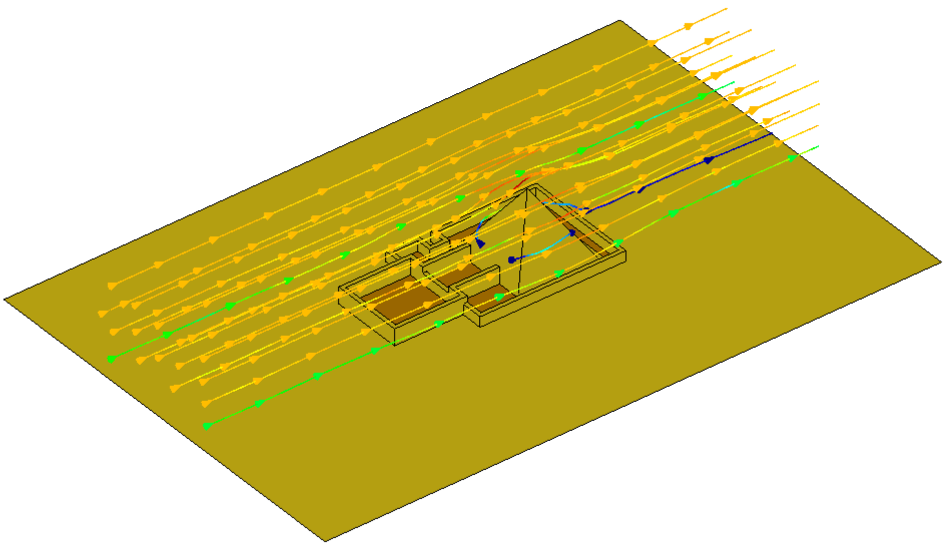
15 . Delete the layers with name **CCutSetX**
16 . Select **View results->No Results**

5.4.7 Stream lines

Model used
The model `pyramid.gid` used in this example can be found at Material location.

Menu
View results->Stream Lines

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



1 . From preprocess mode open the model
2 . Switch to postprocess mode
3 . Turn on all the layers except **SLayer8**
4 . Set as **Body Bound** the layers style

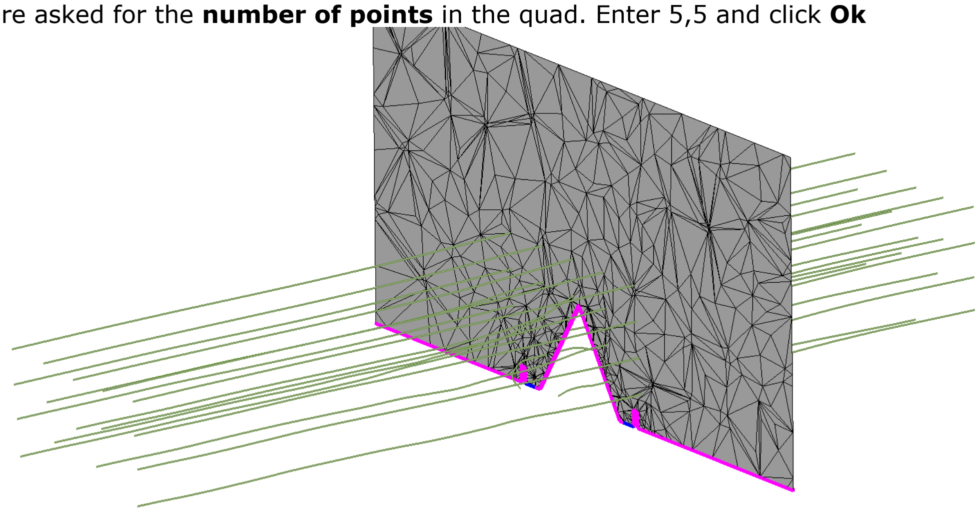
We want to create some stream lines near the pyramid center in order to plot the velocity result near it. So we will make a cut near the center and we will select there the points where we want to plot the stream lines. We want a cut parallel to the XZ plane and near to the pyramid center.

5 . Select **View->Rotate->Plane XY (Original)** to get a proper view to make de cut
6 . Select **Do cuts->Cut Plane->2 points**
7 . Select 2 points by your own or enter the following coordinates in the command line. -70 78 and 130 78
8 . Press **ESC** to leave the cut function
9 . For each cut created change the style to **Body Lines**
10 . To get a better view set off all the layers except the ones with name **CCutSetX**
11 .Select **View->Rotate->Plane XZ**

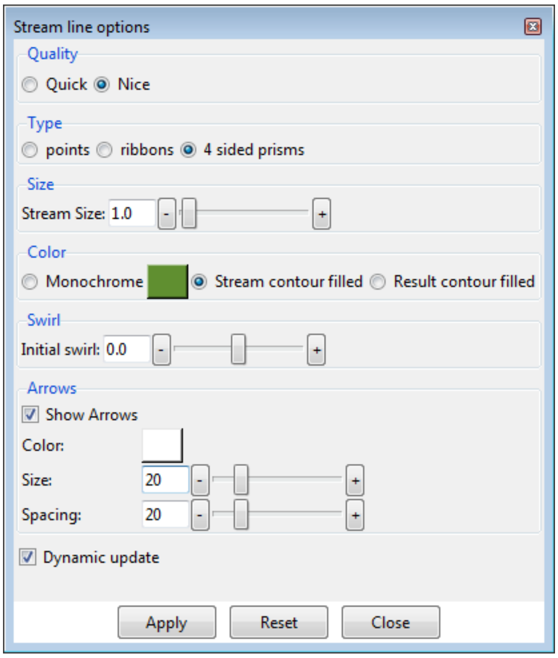
12 . Select **View results->Stream Lines->In a quad->Velocity (m/s)** throught the menu bar
With this option you can define a quadrilateral area which will be used to create a N x M matrix of points. These points will be the start for the stream lines.
We want to create several stream lines around the pyramid.

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->In A Quad.**

13 . From the mouse menu (right button click) select **Contextual->Join Ctrl-a**
14 . Define a quadrilateral area near the pyramid shape clicking on 4 mesh points
15 . You are asked for the **number of points** in the quad. Enter 5,5 and click **Ok**



The stream lines are created.
16 . Click the middle mouse button or press the **ESC** key in order to finish the operation
17 . Turn off the cuts layers and set on the layers **SLayer4** and **SLayer9**



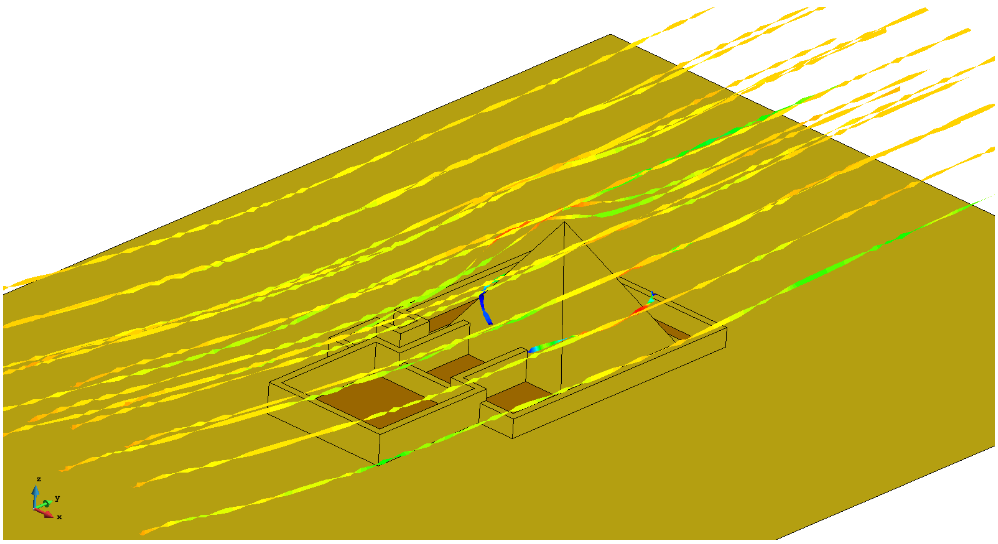
Several configuration options can be set via the Options menu.

Menu
Options->Stream lines

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

18 . Select **Options->Stream lines->Size & detail...**
19 . Choose **Nice** option in order to activate the other visualization options
20 . Check the **Dynamic update** option
21 . Select **Stream contour filled** option
The stream lines will be drawn with the colors used in the velocity contour fill.

22 . In type otions select **ribbons**. With this option the streams show the swirl of the velocity field



23 . Change the type to **4 sided prism** again
24 . In the **Arrows** options, set **20** for the **Size** option

25 . Set **20** for the **Spacing** option
26 . Check the **Show Arrows** option
27 . **Close** the window
28 . Select **Options->Stream lines->Delete all**

5.4.8 Graphs

Model used

The model `pyramid.gid` used in this example can be found at Material location.

Menu

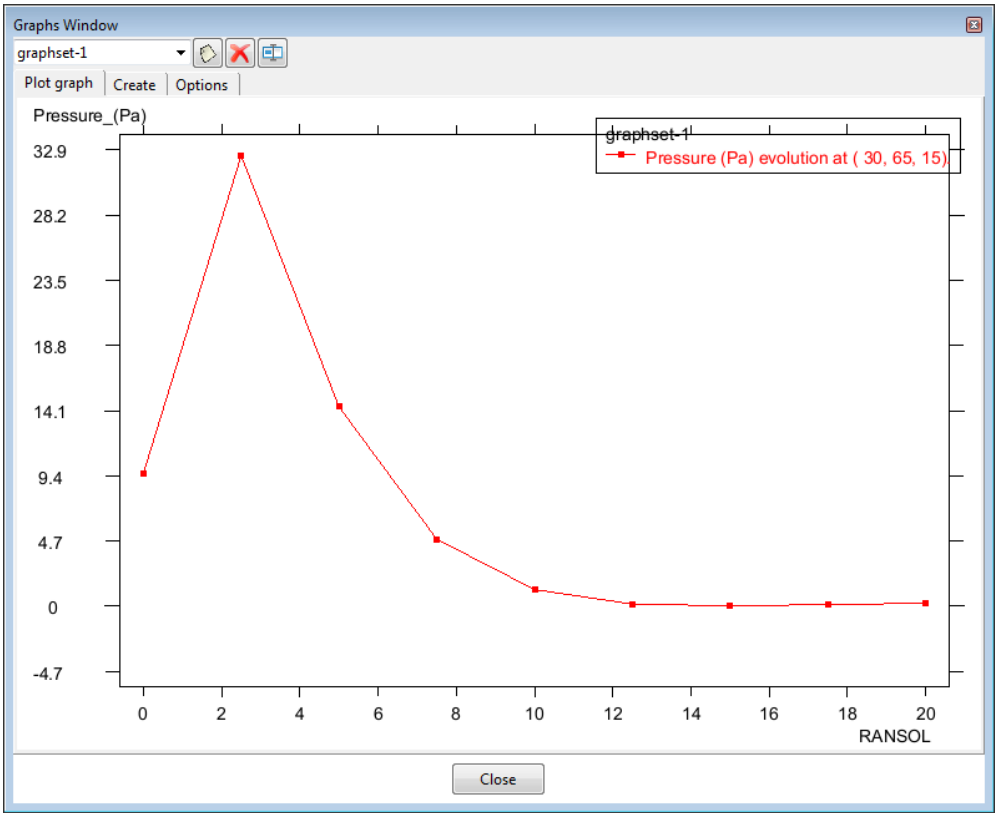
View results->Graphs

Description

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'.

1 . From preprocess mode open the model
2 . Switch to postprocess mode
3 . Turn on **all** the layers except the **SLayer8** through the **View style** window
4 . Change the mesh style to **boundaries** for all the layers

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.
5 . Select **View results->Graphs->Point evolution->Pressure (Pa)**
6 . Write 30 65 15 in the command line in order to specify the point
7 . Affter 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.

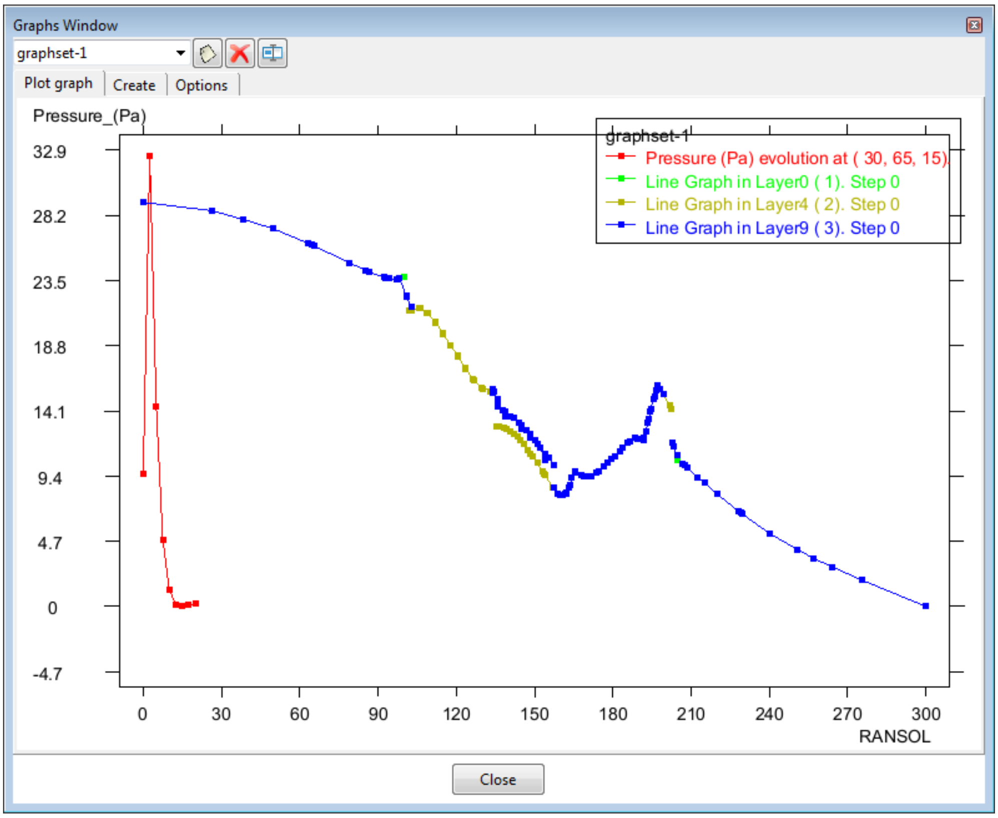
8 . Select **View results->Default Analysis/Step->RANSOL->0**

9 . Select **View results->Graphs->Line graph->Pressure (Pa)**

10 . Write 30 -100 0 in the command line in order to specify the initial point

11 . Write 30 200 0 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.

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

13 . A window will appear asking for a new name. Enter 'Pressure', for example

We will create a new graph set.

14 . 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.

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

16 . In **Y Axis** list double click **Velocity (m/s)->|V|**

17 . Write 30 65 15 in the command line in order to specify the point

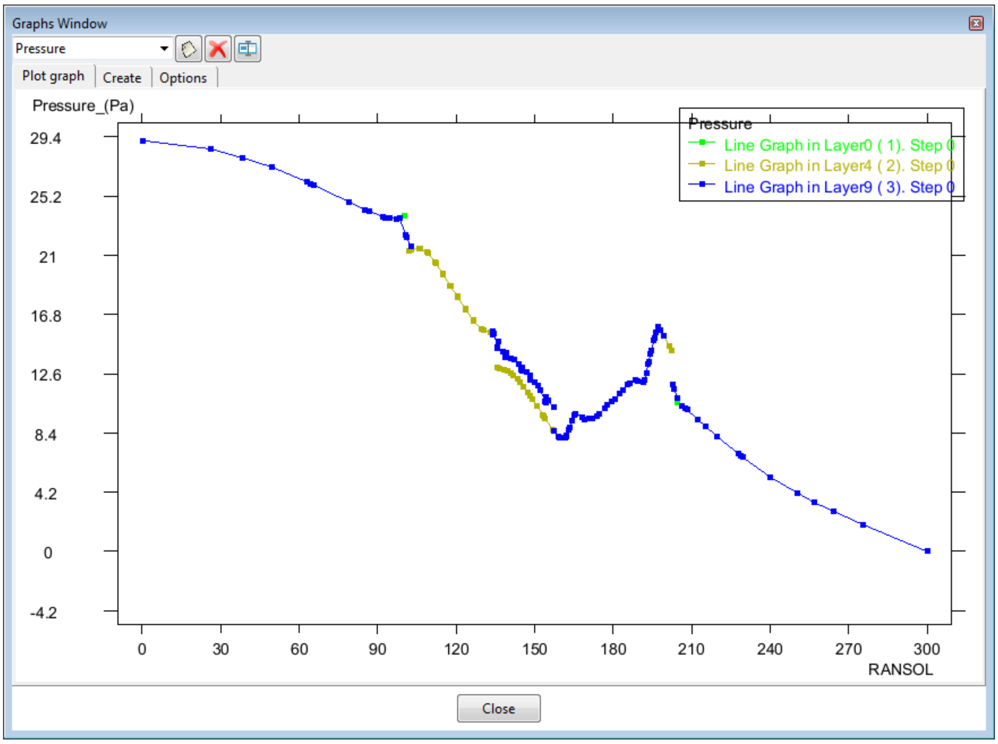
18 . Press **Escape** to finish the graph.

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.

19 . Go to the **Options** panel, select the 'Pressure (Pa) evolution at (30, 65, 15)' graph and delete it pressing the button with the red cross.

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

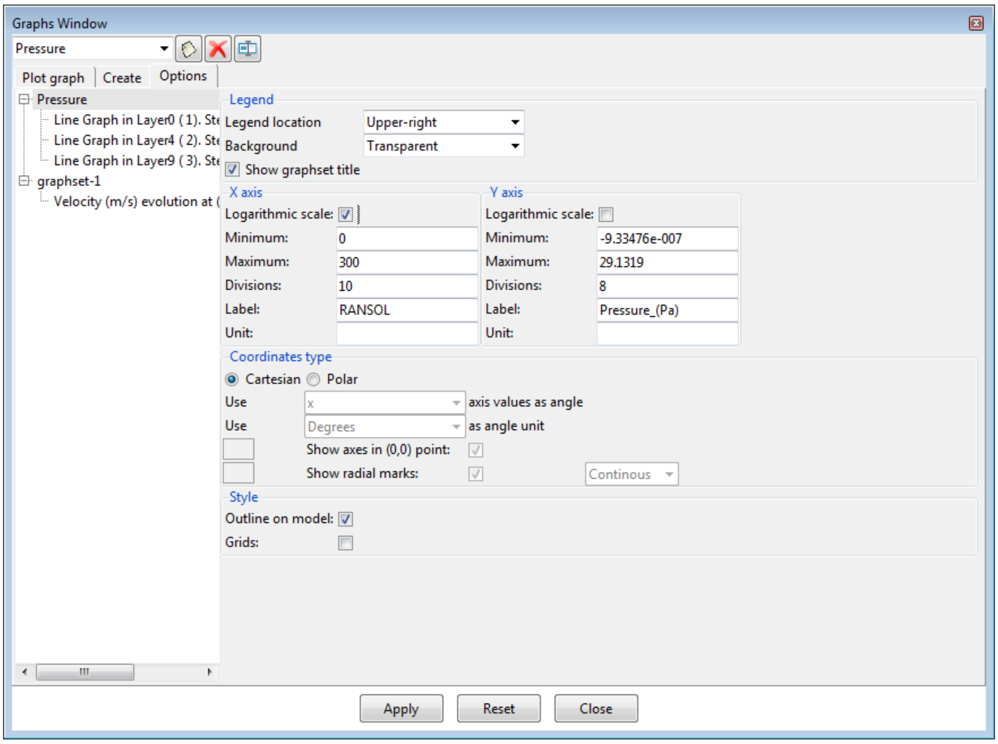
Please notice that the current graph set have been changed to 'Pressure'. Now the Plot graph panel will show only two graphs:



The graph size is re-adapted. We will change several style options of a graph.

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

Graph sets options can be managed selecting the set in the tree.
26 . Select 'Pressure' branch. The options will change.
27 . For instance mark 'Logarithmic scale' option in X axis.
28 . Click on **Apply** button



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

29 . Select **Files->Export->Graph->All graphsets**. You are asked for the location where to save the .grf file

30 . Choose the location

Now you can import the graphs selecting **Files->Import->Graph**

If we select **Options->Graphs->Clear graphs** we will delete all graphs in current graph set. We can also delete them through the graphs window selecting the graph set in the tree and clicking the red cross button.

5.4.9 Creating results

Model used

pyramid.gid

In this tutorial we will learn how to create several types of new results. To do this we can operate between a previous result and a scalar or vector, or between 2 previous results.

5.4.9.1 Create result

Model used

The model `pyramid.gid` used in this example can be found at Material location.

Menu

Window->Create result...

Description

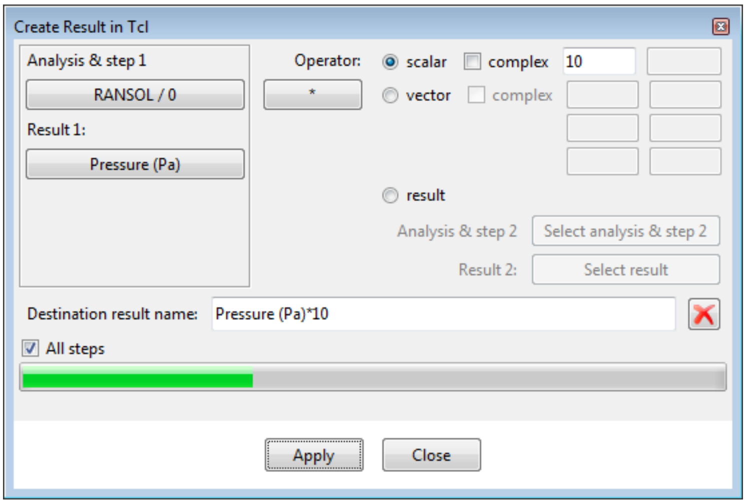
With this option we can create new results based in another ones.

We will create a new result based in the Pressure result for all steps with a multiplier factor of 10.

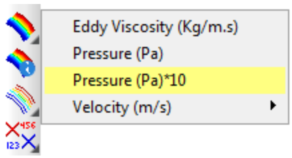
1 . From preprocess mode open the model

2 . Switch to postprocess mode

3 . Select **Window->Create result...** and the following window appears



4 . Select **Ransol->0** as Analysis & step 1
5 . Select * as Operator
6 . Select **scalar** and write **10** in the entry
7 . Select the **All steps** option
8 . Write a name for the new result and click **Apply** to create it
9 . Now the result can be selected



5.4.9.2 Create statistical result

Model used

The model `pyramid.gid` used in this example can be found at Material location.

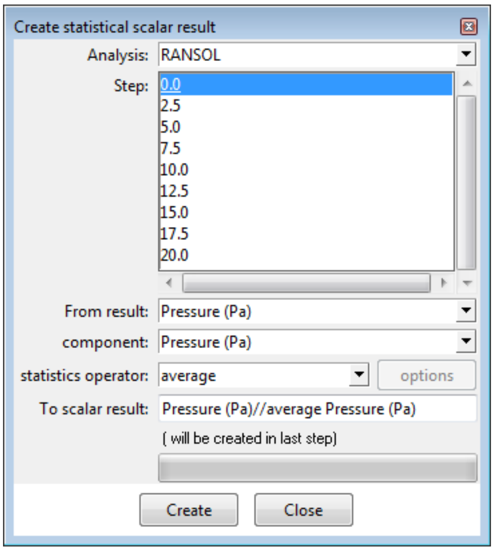
Menu

Window->Create statistical result...

Description

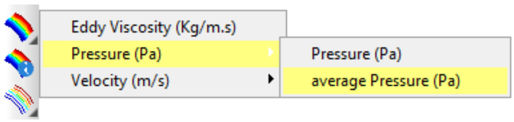
This option allows the user to create statistics from their results.

We will create the average of all steps of Pressure result.
1 . From preprocess mode open the model
2 . Switch to postprocess mode
3 . Select **Window->Create statistical result...** and the following window appears



4 . Select **Pressure (Pa)** as From result
5 . Select **average** as statistics operator
6 . We can change the name for the new statistical result
7 . Select all the steps and click **Create**

The new results are created in the last step, in this case step 20.
8 . Select **View results->Default Analysis/Step->RANSOL->20** throught the menu bar or clicking on 
9 . Now the result can be selected



5.4.9.3 Create graphs

Model used
The model `pyramid.gid` used in this example can be found at Material location.

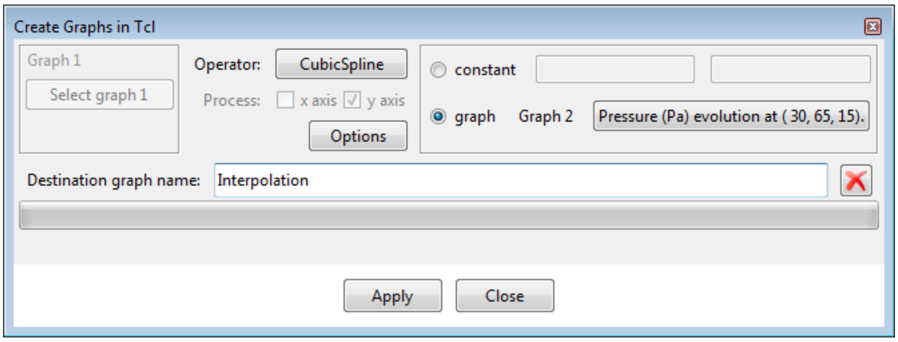
Menu
Window->Create graphs...

Description
With this window the user can create graphs from other graphs.

1 . From preprocess mode open the model
2 . Switch to postprocess mode

First we will create a graph that will be used in the operations.
3 . Select **View results->Graphs->Point evolution->Pressure (Pa)**
4 . Write 30 65 15 in the command line in order to specify the point and press Enter
5 . Press **Escape**

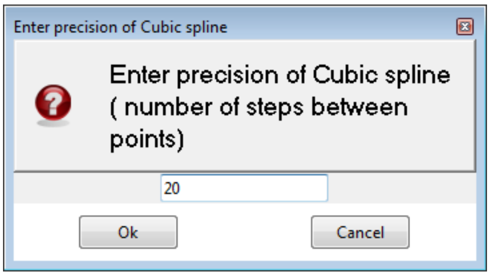
The created graph appears. Now we will create a interpolation of this graph using cubic splines
6 . Select **Window->Create graphs...** and the following window appears



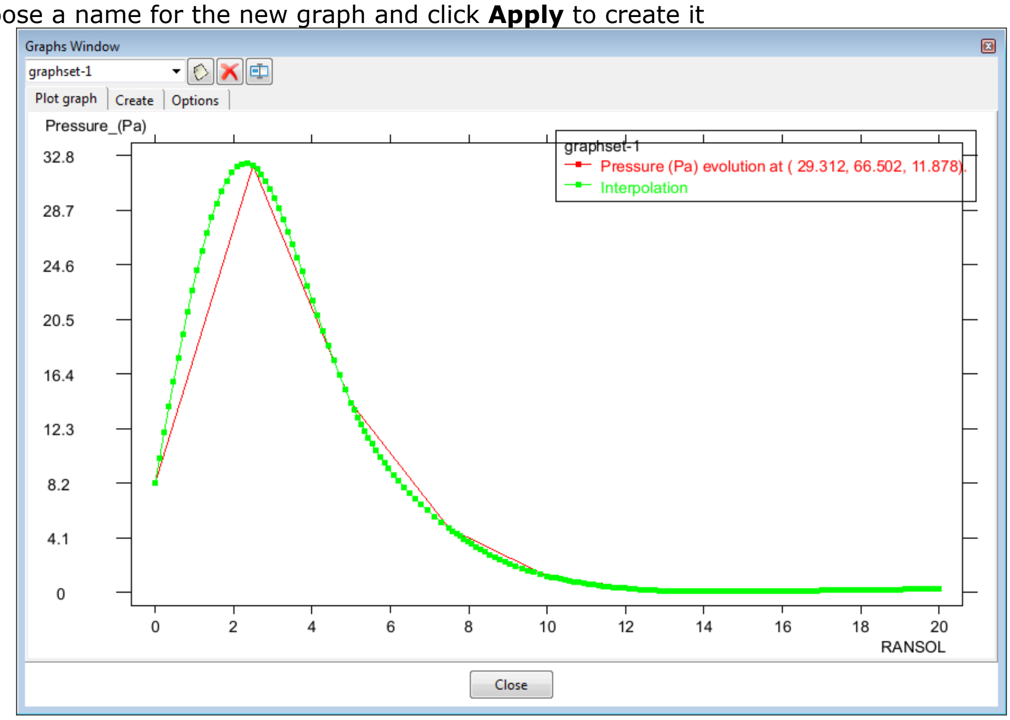
7 . Select **CubicSpline** as Operator

8 . Select the created graph clicking on the **Select graph 2** button

9 . In this case we can set the precision of cubic spline clicking on the **Options** button. Leave the default value and click **Ok**



10 . Choose a name for the new graph and click **Apply** to create it



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