

RamSeries tutorials manual

<http://www.compassis.com>

info@compassis.com

November 2018

Version: 15.1.0

1. Introduction

RamSeries can be used to perform the structural analysis of either beams, shells, solids, cables and membranes, or any combination of all them, using the finite element method. In general, it is considered that linear elastic material properties and small displacements assumptions apply to the entire structure under analysis. Nevertheless, material non-linearities (plasticity), and geometrical non-linearities (large displacements) can also be taken into account under some circumstances. Non-linear elastic boundary conditions can be used as well. Finally, RamSeries can also be used to dimensionalize the concrete beams and shells using the necessary steel and based on the EHE Spanish regulation. The present document intends to include a comprehensive set of tutorials trying to cover most of the basic program capabilities. If using the interactive Help system, the following link can be used to access a comprehensive list of tutorials and to automatically load the corresponding models in Tdyn.

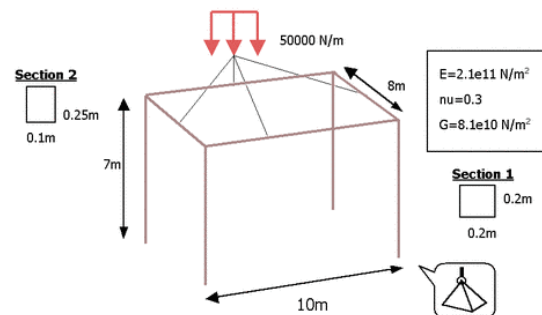
Tutorials list

Some theoretical aspects concerning the mechanical behavior of beams, shells and solids, and the most important hypotheses assumed in the implementation of the RamSeries code are briefly described in the appendices of the RamSeries User's Manual. Some important issues concerning the definition of local axes for beams and shells are also addressed in the User's manual. The definition of local axes is going to be used systematically in many of the tutorials and is necessary during the preparation of many actual models in RamSeries. Hence, it is strongly recommended to fully read the RamSeries User's manual before starting an intensive use of RamSeries.

All the analysis modules in RamSeries are embedded within GiD, a finite element pre and post-processor. Some degree of knowledge and some confidence on the use of GiD is assumed to ensure a correct development of the case studies presented within this manual. In particular, it is assumed that the user has some practice on using GiD to create the geometry, generate the mesh and visualize the results in the post-processing stage. Please, refer to the GiD manual for more details about its usage.

2. Tutorial 1 - Analysis of a beam structure

This tutorial is aimed to perform a simple structural analysis of a spatial frame subjected to the action of an homogeneous distributed load applied to the horizontal beams located at the top of the frame structure. A schematic representation of the problem under analysis is shown in the figure below. Beam's section and material properties are also included.



Schematic representation of the spatial frame under analysis.

The geometry can be constructed in GiD using the following useful pre-processing commands which are described in detail in the GiD User's Manual.

Geometry ► Create ► Line ► 0,0,0 ► 10,0,0

Utilities ► Copy ► Translation

View ► Rotate ► Trackball

The final model will be composed of 8 lines and all the corresponding connection points.

2.1. Problem specification

To load the RamSeries type of analysis, choose the following option in the Start Data window:

Simulation type ► Structural analysis

The same option can be alternatively activated through the Tdyn data tree by using the following option:

Simulation data ► Simulation type ► Structural analysis

The kind of problem under consideration in this tutorial, requires the following basic options to be selected:

- Simulation dimension: 3D

- Structural analysis element types: Beams

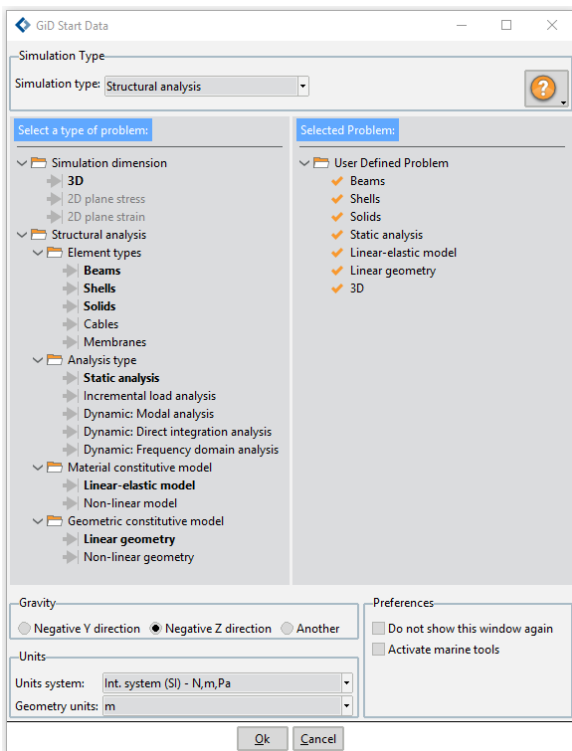
- Analysis type: Static analysis

- Material constitutive model: Linear-elastic model

- Geometrical constitutive model: Linear geometry

This setup can be configured from the Start Data window that can be accessed from the Data menu as follows:

Data ► Start Data



Start Data window as it appears after configuration of the type of problem required to perform structural analysis of the beams in a spatial frame structure

2.2. Boundary conditions and properties

Constraints

First, constraint conditions must be applied. In this case, the four inferior points of the structure have a prescription in their movements in the three global directions X, Y and Z. Rotations in the points are not prescribed.

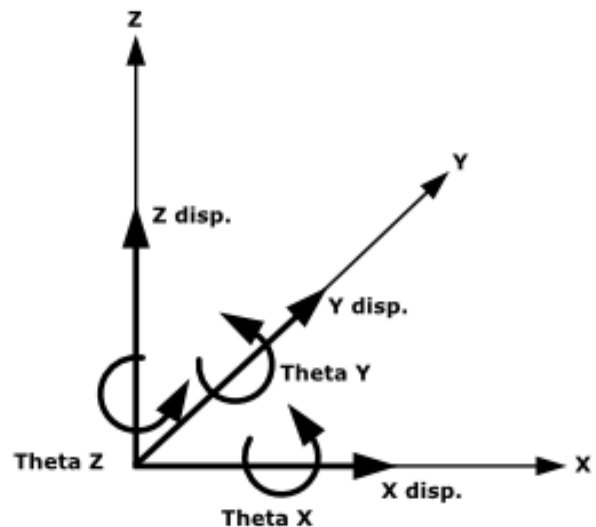
Constraints ► Fixed constraints

Activation		Values	
X constraint	1	X value	0.0
Y constraint	1	Y value	0.0
Z constraint	1	Z value	0.0
θ_x constraint	0	θ_x value	-
θ_y constraint	0	θ_y value	-
θ_z constraint	0	θ_z value	-

In this condition, the local axes are no related to the beam local axes defined in the properties section. The GLOBAL option means that the prescriptions assigned are related to the global axes of the problem. This is used in this case. Local axes are used to prescribe the displacement or rotation in a direction not

coincident with any of the global axes. The *values part* of the condition is used to prescribe a fixed amount of displacement or rotation. Default units are meters for the X, Y and Z displacements and radians for the prescribed rotations. In this case, the prescribed displacements and rotations are all zero.

X Constraint, Y Constraint and Z Constraint mean the displacements along the axes. θ_x Constraint, θ_y constraints and θ_z constraints mean the rotations around the axes. Signs are as follows (right hand rule):



This condition must be assigned to the four points in the bottom part of the structure (minimum Z).

Material properties

Next, section and material properties must be defined. To assign the section and material properties to the beams choose:

Materials and properties ► Beams ► Rectangular section

The beams will have the following rectangular section properties:

-Width Y= 100 mm (width of the rectangular section along the Y' direction)

-Width Z= 250 mm (width of the rectangular section along the Z' direction)

-E= 2.1e11 N/m² (Young modulus)

-G= 8.1e10 N/m² (torsion modulus)

-Specific Weight= 0.0 (the self-weight of the beam is not considered)

-Maximum stress= 0.0 (this parameter does not have influence in the calculation; it is only useful as a reference for the user when verifying stress results in postprocess).

Note: Remember that for an isotropic material: $G = \frac{E}{2 \cdot (1 + \nu)}$
In this case $\nu=0.3$

To assign properties to the beams, it is first necessary to define local axes. To this aim, in the present case **Automatic** local axes are used. Hence, following the standard criteria, X' axes (prime refers to local axes) will remain aligned along the horizontal beams longitudinal axis. On the other hand, Y' will be horizontal and perpendicular to the beam longitudinal axis, while Z' will have the same direction and sense than the global Z axis.

Once the values of all rectangular section parameters have been provided, the condition must be assigned to all the horizontal beams.

When assigning the properties on the horizontal beams, the Local axes system **Automatic** is chosen, following the criteria given before.

The same must be made with the vertical beams. In this case the Y' axe points to the direction of the X axe, for local axes system **Automatic**.

The conditions that have been assigned can be viewed right-clicking on the created group:

Draw ► Draw values / Draw groups / Draw symbols

Load conditions

Finally the loading conditions of the problem must be specified. In the present case, all the horizontal beams have a distributed load of 50000 N/m along the negative global Z-axis. So the condition **Global Beam Load** will be used:

Loadcases ► Loadcase 1 ► Beams ► Pressure load

Factor	1.0
Loadtype	Global
X pressure	0.0 [N/m]
Y pressure	0.0 [N/m]
Z pressure	-5.0e4 [N/m]

This condition shall be assigned to all the horizontal beams.

2.3. Mesh generation

For most of the geometrical models, especially if all the beams are straight, it is only necessary one beam element per line. To obtain it, generate the mesh with a big default element size. It should be chosen longer than any of the lines length. For this example 100 units can be chosen.

If there are curved lines, it can be necessary to mesh several beam elements for every line. To do so, use GiD options to control the mesh size like:

Meshing ► Assign unstructured sizes ► Points

Meshing ► Assign unstructured sizes ► Lines

Meshing ► Assign unstructured sizes ► By chordal error

To generate the mesh use:

Meshing ► Generate ► 100

RamSeries only accepts the default 2-noded beam elements. So *quadratic* option is not accepted for beams. Check the GiD manual for details.

2.4. Calculate

If the model has not been saved yet, use:

Files ► Save

and give a name to the model. To begin the analysis choose:

Calculate ► Calculate

The analysis runs as a separate process. Then, it is possible to continue working with GiD or exit the program. When the analysis is finished, a window will appear notifying it. If it has not run successfully, one window showing some error info will be supplied. Correct the error and run again.

It is possible to visualize, when the process is running, some information about its evolution. Press:

Calculate ► View process info

2.5. Postprocess and results analysis

The part of the program dedicated to visualize the results of the analysis is called Post-process. Once the calculation is finished, to enter in the post-process part and load automatically the results select:

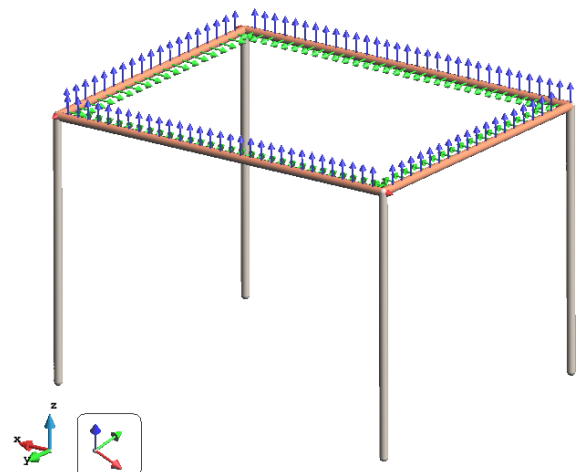
Postprocess ► Start

To perform the operation correctly, it is necessary to have the model loaded inside the pre-processing part before going to post-process.

• Visualization of local axes

The first important checking that must be done is to see if the local axes for every beam are equal to the ones supposed when assigning the properties. Choose:

Postprocess panel ► Results ► Automatic local axes ► Local axes mesh - beams ► Local axes



If they are different than supposed, go to the pre-processing part, change the properties accordingly and calculate again.

If the drawing of the local axes appear too big or too small, the scale drawing factor may be changed by entering a different factor in:

Postprocess panel ► Preferences ► Vectors & Local axes ► Factor

Deformation results

To visualize the deformed (with displacements contour fill on it) of the structure, chose the following options:

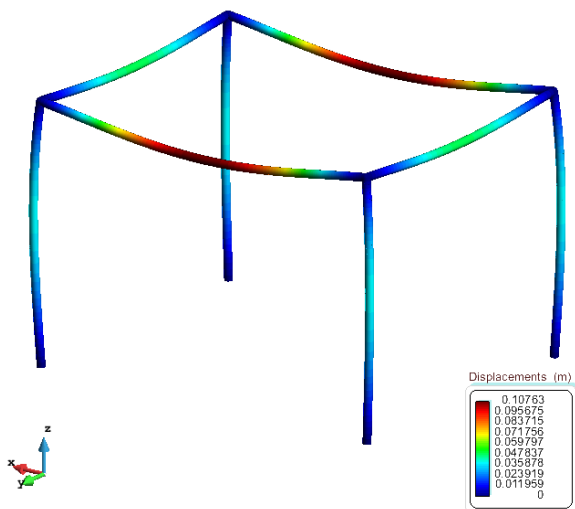
Postprocess panel ► Meshes ► Line style ► 2nd option

Postprocess panel ► Results ► Static ► Displacements ► Contour fill

Postprocess panel ► Preferences ► Deformed ► Draw:[Yes]

Postprocess panel ► Preferences ► Deformed ► Factor:[5]

Postprocess panel ► Preferences ► Deformed ► Result:["Static Displacements"]



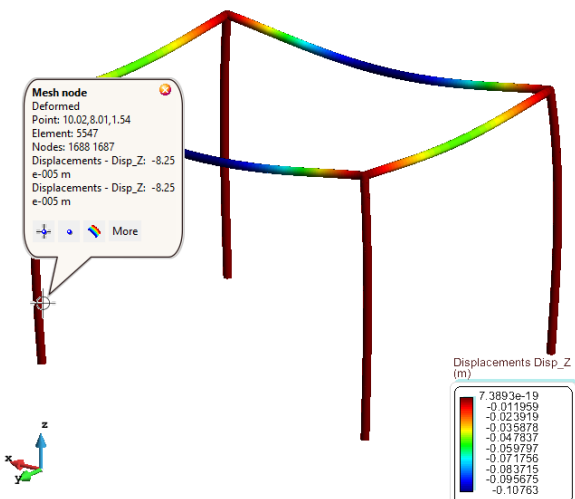
Deformed of the structure with contour fill of displacements [m]

The deformed of the structure is drawn magnified in the screen (factor 5 chosen). The magnifying factor can be automatically calculated if "Adimensional factor" is checked instead.

To see the numerical values of the deformation choose at any point, node or element:

Postprocess ► Mesh information

Try also pressing mouse left button over any structure element in the screen.



Make the appropriate zooms to visualize properly the values.

The values are the maximums and lateral values of every beam and represent the vector, expressed in global coordinates, of the displacement. The default units are meters.

To switch off the labels press:

Mouse right button ► Label ► Off

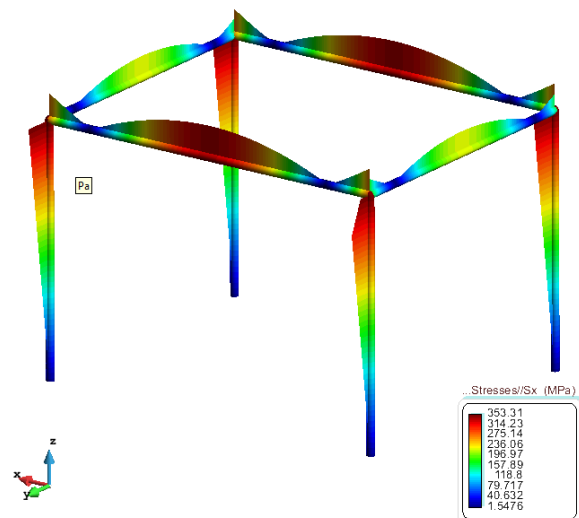
In order to choose the visualization style for the diagrams, try:

Postprocess panel ► Meshes ► Line style

Strength results

All of the strengths are represented related to the local axes systems of every beam. The visualization is made with a diagram that will be more separated from the beam as the magnitude of the strength increases. It will be drawn in either side of the beam depending on the sign. To draw labels with the numerical value of the strength that is being visualized, press the left mouse button over the screen.

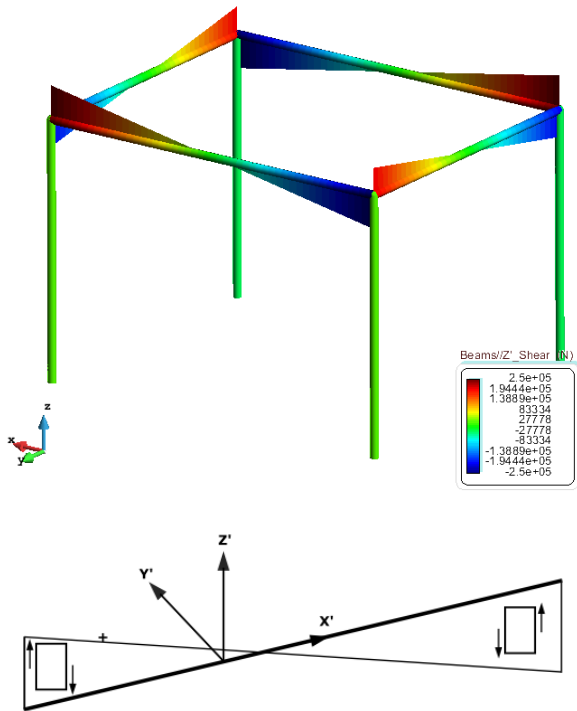
Postprocess panel ► Results ► Static ► Beams ► Max stresses ► Sx ► Line diagram



Shear, momentum and axial forces

This is the shear in the X'Z' plane. Default units are N.

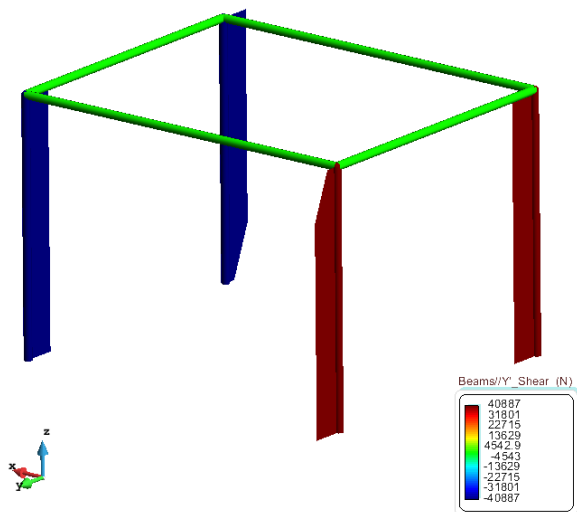
Postprocess panel ► Results ► Static ► Beams ► Z' shear ► Line diagram



In the figure the sign criteria for the shear is shown.

This is the shear in the $X'Y'$ plane. Default units are N.

Postprocess panel ► Results ► Static ► Beams ► Y' shear ► Line diagram

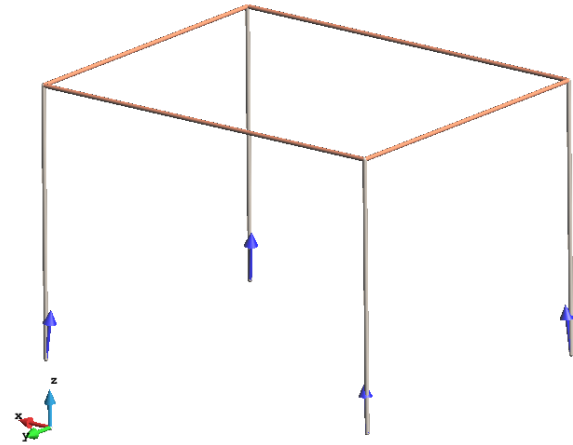


In the figure the sign criteria for the shear is shown.

These are the reactions that appear in the constraints. All nodes

that have no constraint will have a null reaction. Default units are N.

Postprocess panel ► Results ► Static ► Reactions ► Vectors



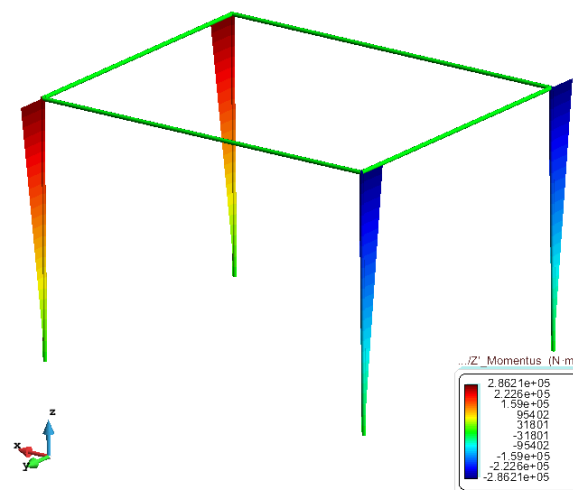
For obtaining the numerical result of all the reactions go to:

Postprocess ► Compose reactions

If labels are displayed, four numbers appears. The first three are the reaction vector and the last is the module of this vector.

This is the momentum that rotates around the Z' axe. Default units are N·m.

Postprocess panel ► Results ► Static ► Beams ► Z' momentum ► Line diagram



Sign criteria for Z' momentum is:

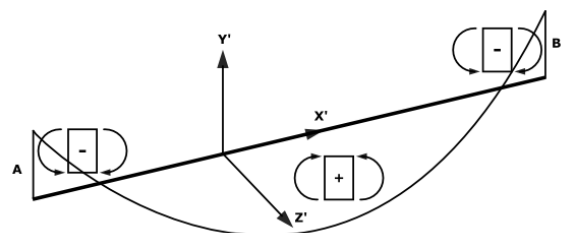
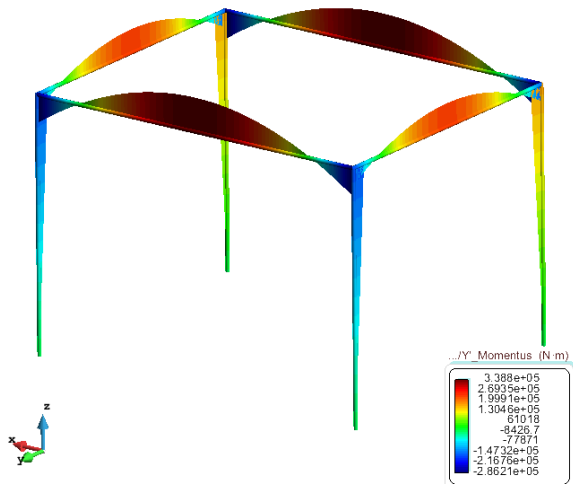


Diagram is drawn in the plane $X'Y'$ and in the side of the beam

where the traction is. Positive values of the momentum mean that traction is in the $-Y'$ side (in the negative side of Y').

This is the momentum that rotates around the Y' axe. Default units are N·m.

Postprocess panel ► Results ► Static ► Beams ► Y' momentum ► Line diagram



Sign criteria for Y' momentum is:

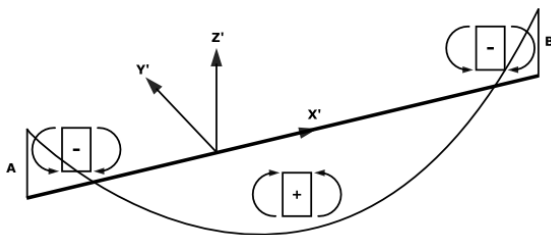
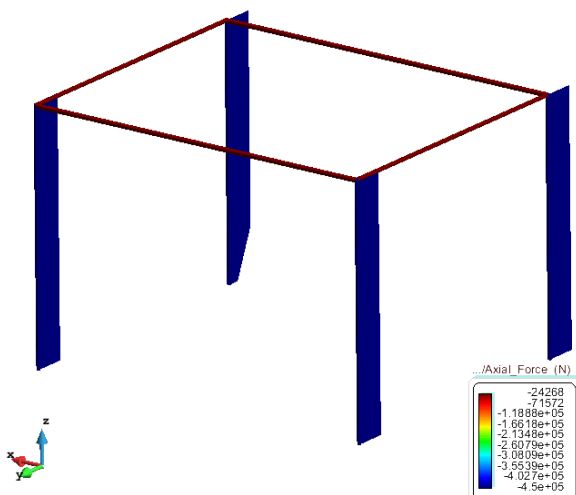


Diagram is drawn in the plane $X'Z'$ and in the side of the beam where the traction is. Positive values of the momentum mean that traction is in the $-Z'$ side (in the negative side of Z').

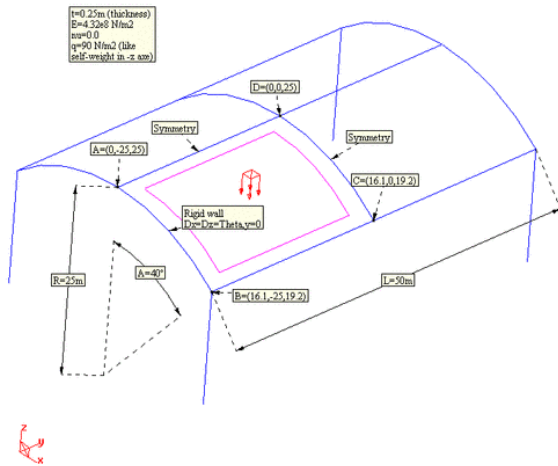
This is the axial force in the beam axe. Default units are N.

Postprocess panel ► Results ► Static ► Beams ► Axial force ► Line diagram



3. Tutorial 2 - Analysis of a shell structure

This tutorial aims to demonstrate the use of RamSeries capabilities for modelling shell structures. In particular, this model describes the analysis of a curved roof taking advantage of the double symmetry of the actual problem configuration. A sketch of the whole problem is shown in the figure below. One fourth of the total shell structure and the corresponding symmetry conditions are indicated to illustrate the actual problem being solved.



Schematic representation of the problem under analysis. Only one fourth of the entire structure is actually modelled taking advantage of the double symmetry resulting from the actual configuration of the problem.

The geometry can be constructed in GiD using the following usefull pre-processing commands which are described in detail in the GiD User's Manual.

Geometry ► Create ► Line

Geometry ► Create ► Arc

Utilities ► Copy ► Rotate ► Do extrude lines

Geometry ► Create ► NURBS surface ► By contour

The final model will be composed of 1 surface, 4 lines and all of its connecting points. Be careful to position the structure related to the global axes as seen in the figure to make it easier to follow the set up of the example as explained in the remaining part of the tutorial. The coordinates of the four vertices of the shell under analysis are reported in the following table:

	X coordinate (m)	Y coordinate (m)	Z coordinate (m)
Point A	0.0	-25.0	25.0
Point B	16.07	-25.0	19.15
Point C	16.07	0.0	19.15
Point D	0.0	0.0	25.0

3.1. Problem specification

To load the RamSeries type of analysis, choose the following option in the Start Data window:

Simulation type ► Structural analysis

The same option can be alternatively activated through the Tdyn data tree by using the following option:

Simulation data ► Simulation type ► Structural analysis

The kind of problem under consideration in this tutorial, requires the following basic options to be selected:

Data ► Start data

- Simulation dimension: 3D
- Structural analysis: Shells
- Analysis type: Static analysis
- Material constitutive model: Linear-elastic model
- Geometrical constitutive model: Linear geometry

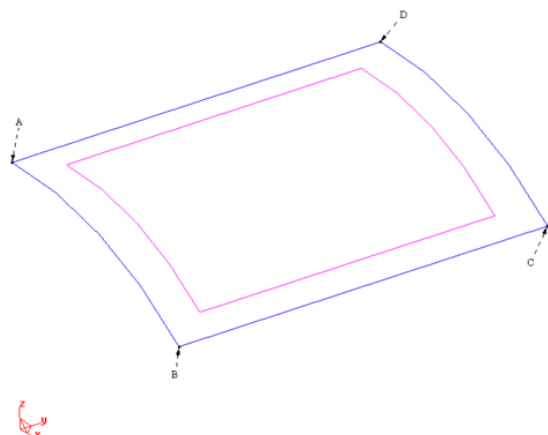
This setup can be configured from the Start Data window as explained in detail in the [Tutorial 1 - Analysis of a beam structure](#) tutorial. Alternatively, all these options can be modified in the data tree if necessary.

3.2. Boundary conditions and properties

Constraints

Constraints for shells can be either applied to points or lines. In this example, fixed constraints are applied to 3 boundary edges of the shell by using the following option of the data tree:

Constraints ► Fixed constraints



In particular, a condition resembling a rigid wall is applied to the edge AB. To this aim, X and Z displacements must be restrained as well as the rotation about Y global axis. Hence, the degrees of freedom are activated as indicated in the following table (non-activated degrees of freedom remain free) and the corresponding displacements are fixed to zero:

	Activation	Values
X constraint	1	X value 0.0
Y constraint	0	Y value -
Z constraint	1	Z value 0.0

θ_x constraint	0	θ_x value	-
θ_y constraint	1	θ_y value	0.0
θ_z constraint	0	θ_z value	-

On the other hand, lines CD and DA must have symmetry conditions. To this aim, degrees of freedom must be prescribed as shown in the following tables for CD and DA edges respectively:

	Activation		Values
X constraint	0	X value	-
Y constraint	1	Y value	0.0
Z constraint	0	Z value	-
θ_x constraint	1	θ_x value	0.0
θ_y constraint	0	θ_y value	-
θ_z constraint	1	θ_z value	0.0

	Activation		Values
X constraint	1	X value	0.0
Y constraint	0	Y value	-
Z constraint	0	Z value	-
θ_x constraint	0	θ_x value	-
θ_y constraint	1	θ_y value	0.0
θ_z constraint	1	θ_z value	0.0

Finally, edge BC remains completely free. In all these cases, the degrees of freedom refer to the global system axes.

Shell material properties

The shell under analysis in this tutorial is considered to be isotropic. To assign the section and material properties of an isotropic shell, use the following option of the data tree and setup the corresponding properties as shown below:

Materials and properties ► Shells ► Isotropic shell

- Thickness = 250 mm
- $E = 4.32e8 \text{ N/m}^2$ (Young modulus)
- $\nu = 0.0$ (Poisson coefficient)
- Specific Weight = 0.0 (self-weight of the shell is not considered here)

Remember that for an isotropic material: $G = \frac{E}{2 \cdot (1 + \nu)}$ which in this case results to be $G = \frac{E}{2}$

Note also that the default local axes system is used. Following the axes criteria explained in detail in the RamSeries' users manual, X' local axis will point towards the positive global X axis. Z' axis will be normal to the elements and Y' will be contained in the plane XZ.

Once the shell material properties are filled in, the condition must be assigned to the surface that defines the shell. Assigned conditions can be viewed at any moment by right-clicking on the created group and using the following option from the contextual menu:

Draw ► Draw values / Draw groups / Draw symbols

Load conditions

The load condition of the structure under analysis in this problem consists on a distributed load of 90 N/m^2 directed to the negative direction of the global Z axis. Hence, a pressure load condition can be applied by using the following option of the data tree:

Loadcases ► Loadcase 1 ► Shells ► Pressure load

Factor	1.0
Loadtype	Global
X pressure	0.0 [N/m]
Y pressure	0.0 [N/m]
Z pressure	-90.0 [N/m ²]

The resulting condition must be assigned to the surface that defines the shell under analysis.

3.3. Mesh generation

RamSeries can generate both, 3-noded and 6-noded triangular meshes. hence, it is possible to mesh the surface of the shell under analysis with either linear or quadratic triangles. The use of linear or quadratic elements can be controlled by using the following main menu option:

Mesh ► Quadratic type ► Normal | Quadratic | Quadratic 9

Note that the Quadratic9 option only applies to quadrilateral elements for which an extra node is located at the center of the element.

Note also, that in the structural analysis data tree an option exists to control the type of triangular element being used internally by the structural solver:

General data ► Analysis ► Element types ► Internal triangular element

This option is set by default to DKT. Nevertheless, a 6-noded option is also available in which case 3-noded triangular meshes are internally converted to 6-noded meshes. Finally, a third option exists that allows to use an improved Drill-rotation triangular element.

As usual, the more elements are used to mesh the geometry, better accuracy in the results will be obtained. Conversely, larger computer times and RAM memory resources will be needed. The following option can be used at the end of the calculation to get information on the memory and computer time requirements:

Calculate ► View process info

It is usually convenient to mesh using smaller elements in those zones of the shell where larger result's gradients are anticipated to exist. Mesh element sizes can be controlled using the following options to assign mesh sizes to the various geometrical entities (points lines surfaces and volumes) of the model.

Meshing ► Assign unstructured sizes ► Points

Meshing ► Assign unstructured sizes ► Lines

Meshing ► Assign unstructured sizes ► By chordal error

In this model, a default size of 1 is chosen to generate triangular meshes. To actually generate the mesh using such a default mesh size, use the following main menu option indicating the maximum element size allowed:

Meshing ► Generate ► 1

The resulting mesh contains 642 nodes and 1146 linear triangular elements.

To get an insight on the precision of the results related to the number of nodes in the mesh, take a look at the benchmark graphics provided in the appendix .

3.4. Calculate

If the model has not been saved yet, use:

File ► Save

and give a name to the model.

To begin the analysis choose:

Calculate ► Calculate

The analysis runs as a separate process. Then, it is possible to continue working with GiD or exit the program. When the analysis is finished, a window will appear notifying it. If it has not run successfully, a window showing some error info will be supplied. Correct the error and run again.

It is possible to visualize, when the process is running, some information about its evolution. To get it, press:

Calculate ► View process info

One of the most important information that can be obtained in this window is the amount of RAM memory, expressed in Megabytes, needed by the solver. The total amount of memory required by the analysis code is somewhat bigger. For skyline solver, the total amount is about 10% more. For Sparse solver, it is about the double of memory. This data gives an idea of the maximum problem that can be solved in a given computer.

This specific analysis needs about 40Mb of memory to execute (it should be necessary a RAM of 64Mb or 96Mb to run it correctly). Using an Sparse solver, it would need much less memory.

3.5. Postprocess and results analysis

The part of the program dedicated to visualize the results of the analysis is called Post-process. Once the calculation is finished, to enter in the post-process part and load automatically the results select:

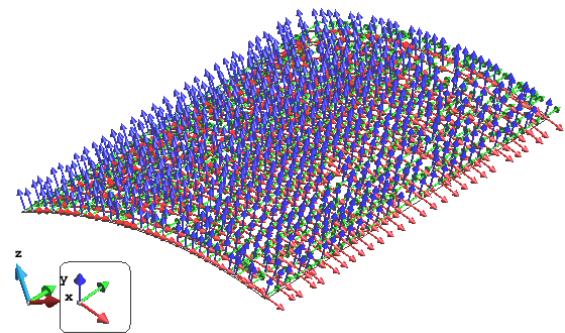
Postprocess ► Start

To perform the operation correctly, it is necessary to have the model loaded inside the pre-processing part before going to post-process.

• **Visualization of local axes**

The first important checking that must be done is to see if the local axes for every shell element are equal to the ones supposed when assigning the properties. Choose:

Postprocess panel ► Results ► Static ► Shells ► Local axes ► Local axes



Local X' axis is drawn in color red, Y' axis in green and Z' axis in blue.

If they are different than supposed, go to the pre-processing part, change the properties accordingly and calculate again.

Note: It is possible to enter a different factor to change the display size of the axes. Just enter the new factor in:

Postprocess panel ► Preferences ► Vectors & Local axes ► Factor

• **Deformation results**

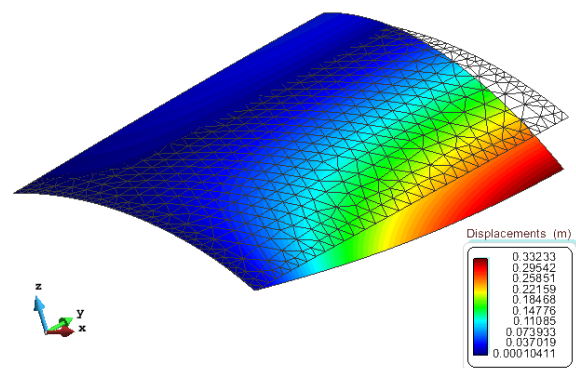
To visualize the deformation of the structure select the following option and visualization preferences in the postprocess panel:

Postprocess panel ► Results ► Static ► Displacements ► Contour fill

Postprocess panel ► Preferences ► Deformed ► Draw:[Yes]

Postprocess panel ► Preferences ► Deformed ► Factor:[10]

Postprocess panel ► Preferences ► Deformed ► Result:["Static Displacements"]

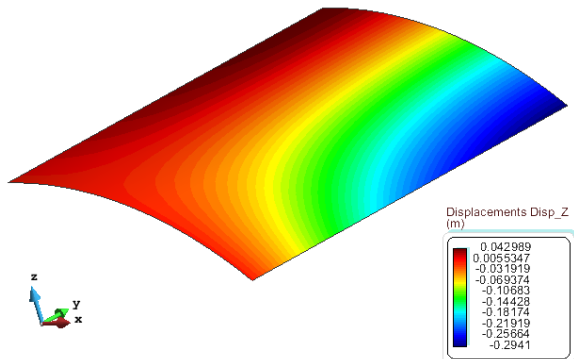


The deformation of the structure is drawn magnified in the screen in accordance to the deformation factor provided in the postproces preferences panel. The magnification factor is automatically calculated if the "Adimensional factor" option is selected instead of providing a fixed factor. The deformation and the original configuration of the structure can be drawn together for the sake of reference. To this aim, select the following option if the preferences section of the postprocess panel:

Postprocess panel ► Preferences ► Deformed ► Draw original

Another way to see the magnitude of the deformation of the shell is to use:

Postprocess panel ► Results ► Static ► Displacements ► Disp z ► Contour fill

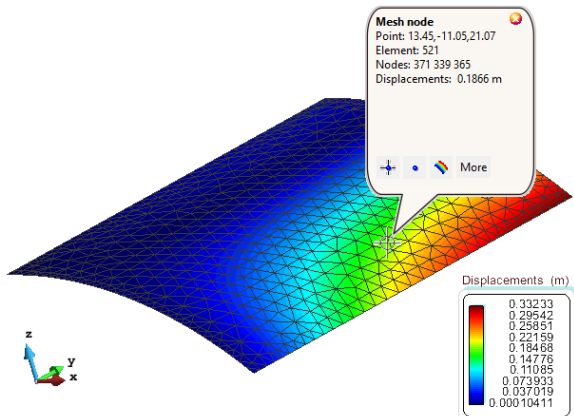


To see the numerical values of the deformation choose at any point, node or element:

Postprocess ► Mesh information

Try also pressing mouse left button over any structure element in the screen.

Make the appropriate zooms to visualize properly the values.



To switch off the labels press:

Mouse right button ► Label ► Off

In order to choose the visualization style for the shell, try:

Postprocess panel ► Meshes ► Display style

It is quite usual that there are local concentrations of strengths near the constraint points or near the nodal forces. In these cases, the scale of color has not enough contrast in the rest of the shell. To avoid it, it is possible to change the limits of the values of the contour fill. The zones of the shell that have values higher than the maximum limit or lower than the minimum one, will appear either black, transparent or in a defined colour, depending on the user preferences.

To define the maximum and minimum limits activate the corresponding checkbox, and define the limits:

Postprocess panel ► Results ► Limits

Right-click over the "Limits" label, leads to the following available options:

- Recalculate limits

Recalculate limits-all steps (only dynamic analyses)

- Multiple results
- Only active meshes
- Legend colours
- Redraw
- **Strength results**

All the strengths are reported referred to the local axes system of every shell (except the strengths in main axes). The visualization will be generally with a Contour Fill of every component of the strength although there are other ways to visualize them. To see the numerical value of the strength that is being visualized, choose:

Mouse right button ► Label ► All

To hide the numerical results select:

Mouse right button ► Label ► Off

Both commands can be found in the menu that appears when pressing the secondary mouse button over the screen.

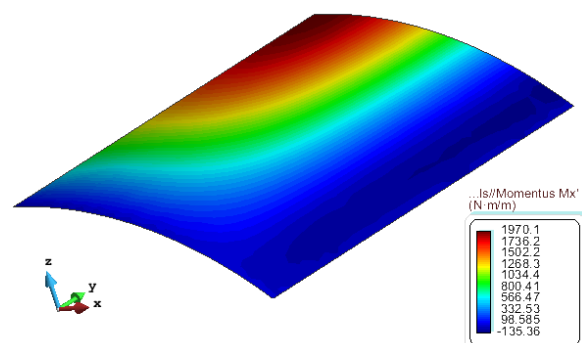
The strengths are the following:

Name		Default units	Remarks
Axial force	N_x, N_y, N_{xy}	N/m	Axial force in all the shell thickness per unit width of the shell
Momentum	M_x, M_y, M_{xy}	N·m/m	Momentum per unit width of the shell
Shear	Q_x, Q_y	N/m	Shear strength per unit width of the shell

Note: See that M_x momentum is contained inside plane $X'Z'$. This criteria is different from the strength criteria in beams or in constraints, where the rotation in X' is the one that rotates around X' axis. The rotation axis criteria for shell momentum are inherited from the plate criteria. In that case, as there are only two rotations in a plate, this choosing looked appropriate. For rotations in 3D, as there are 3, it is necessary to use the other criteria.

As a example, this is the M_x momentum on the shell:

Postprocess panel ► Results ► Static ► Shells ► Momentum ► M_x ► Contour fill



Shear, momentum and axial forces

The shear forces do not follow the same tensor approach because they are two forces, one for each local axis. As they can be represented in any local axes system, it is possible to see in RamSeries the shear represented in the same local axes base than the *Main momentum*. To do so, select:

Postprocess panel ► Results ► Static ► Shells ► Shear in main ► Qm1 -- Qm2

Following the theory of elasticity, the axial force is represented in a given base X'Y' by the following tensor:

$$\begin{pmatrix} N_{x'} & N_{x'y'} \\ N_{x'y'} & N_{y'} \end{pmatrix}$$

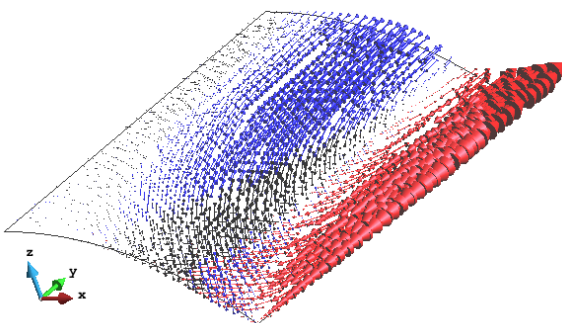
It is always possible to find another base X''Y'' where the tensor is represented as:

$$\begin{pmatrix} N_{11} & 0 \\ 0 & N_{22} \end{pmatrix}$$

These are the eigenvalues of the matrix and the new base is made with the eigenvectors. Its values represent the maximum and the minimum axial value for that point of the shell.

It is possible to see these values with:

Postprocess panel ► Results ► Static ► Shells ► Axial force ► Main stresses



Note: As the local axes system for the main axial forces change from element to element, in some cases where this change is too big the smoothing in the contour fill is not being done. For this reason, some discontinuities of the results can be appreciated.

The same ideas than for the *Main axial force* can be applied to the momentum. In this case, to display the main momentum use:

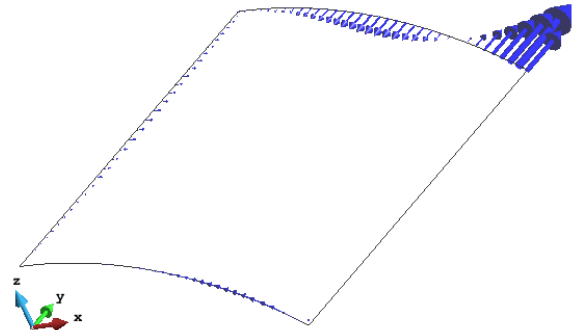
Postprocess panel ► Results ► Static ► Shells ► Momentum ► Mi -- Mii

This is the reaction momentum. It is expressed as a vector showing the rotation axe and a module that means the rotation amount. Default units are N·m.

Postprocess panel ► Results ► Static ► M Reactions ► Vectors

These are the reactions that appear in the constraints. All nodes that have no constraint will have a null reaction. Default units are Newtons.

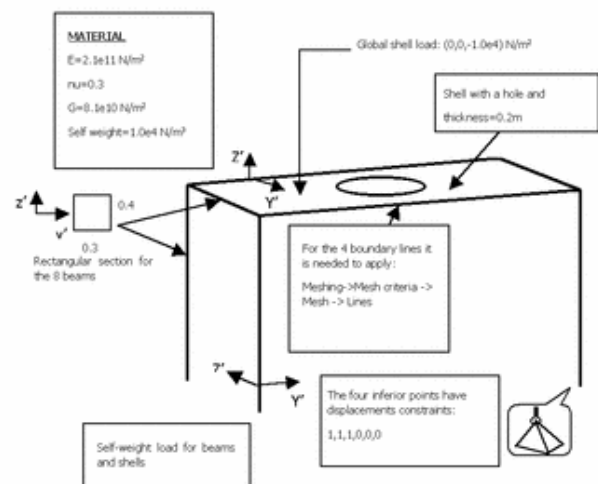
Postprocess panel ► Results ► Static ► M Reactions ► Vectors



If labels are displayed, four numbers appears. The first three are the reaction vector and the last is the module of this vector.

4. Tutorial 3 - Coupled beam and shell analysis

This example represents the analysis of a spacial frame supporting a plate with a hole. A schematic representation of the problem is shown in the figure below. Section, material properties and constraints are also indicated.



4.1. General ideas

The analysis of a coupled beam and shell analysis comes quite straightforward from the analysis of beams and the analysis of shells separated. Some points to take into care are:

- Beams and shells must share the connecting points when a physical connection needs to be defined. Also, a beam that is boundary of a shell must be represented with a line that is boundary of a surface.
- If a beam is represented by a line that is the boundary of one surface that represents a shell, the line is not going to be meshed, by default, with beam elements. It is necessary to explicitly tell GiD to mesh it. To do so, use:

Meshing ► Mesh criteria ► Mesh ► Lines

And select the lines that represent these beams.

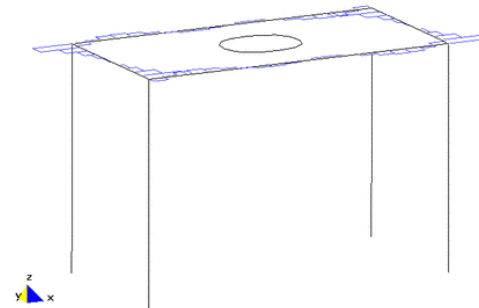
- It is not accepted to mesh the mixed elements with quadratic elements. So, linear elements must be used. It is advisable to set the option *Internal 6-noded elements* in order to have higher precision for the shell analysis.

For the beams that are boundary of a shell surface, it is necessary to mesh as many beam elements as boundaries of the triangle elements. So, every beam will be subdivided in several line elements. In these cases, it is better to use a small number for the option *Beam res granularity*. For example 2 or 4.

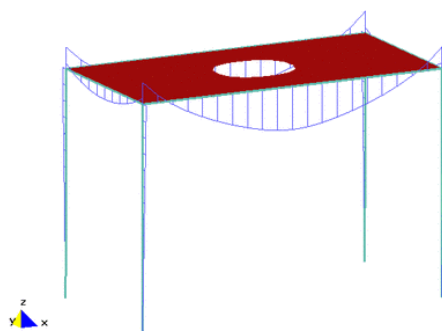
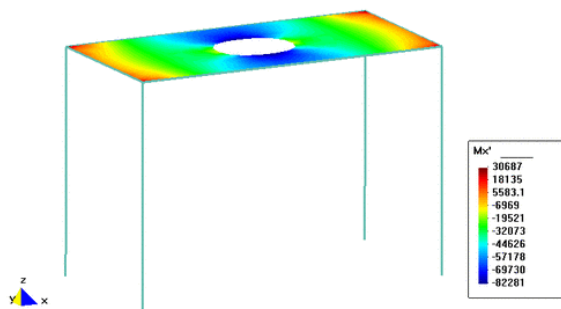
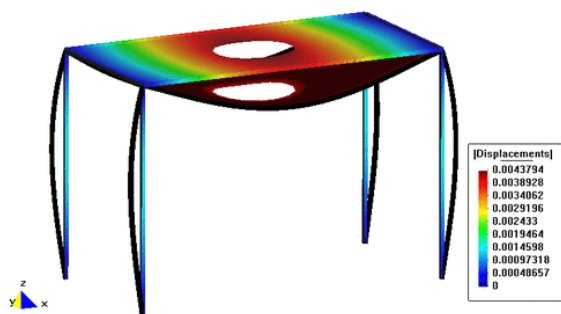
- Results are visualized in the same way that both analyses do.

4.2. Results from the example of coupled beams and shells

The following pictures are several results for the defined analysis:



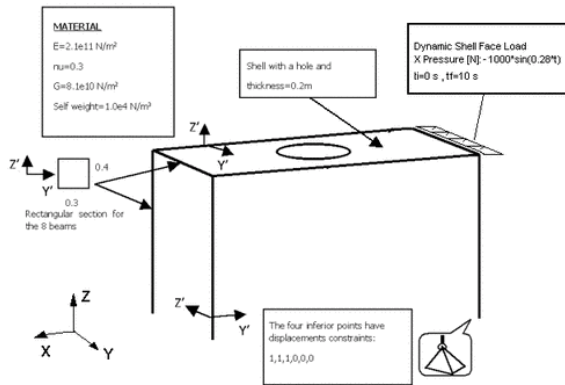
Torque on the beams



Momentum $M_{x'}$ on the shell Momentum M_y on the beams

5. Tutorial 4 - Dynamic analysis of a coupled beam and shell structure

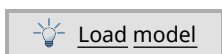
This example performs a dynamic analysis of the structure of the previous example.



The geometry corresponding to this tutorial can be found here (IGES format):

[RamSeries-Tutorials4-dynamic_beams_and_shells.igs](#)

Alternatively, the model ready to be meshed and run, can be automatically loaded into the GUI by left-clicking on the button below.



This example uses the same materials and displacements constraints as the previous example.

5.1. Analysis data

In this example the problem data information is similar to that of the previous example. The required information for the dynamic analysis can be defined in the analysis options:

General data ► Dynamic analysis data ► General

Type	Modal analysis
Δt	0.1 s
Number of steps	25
Type of modal analysis	Number of modes
Number of modes	5

General data ► Dynamic analysis data ► Integration data

Integration method	Implicit (Newmark)
Initial conditions	None

General data ► Dynamic analysis data ► Damping data

Damping type	Modal damping
Damping ratio	0.05

5.2. Dynamic Loads

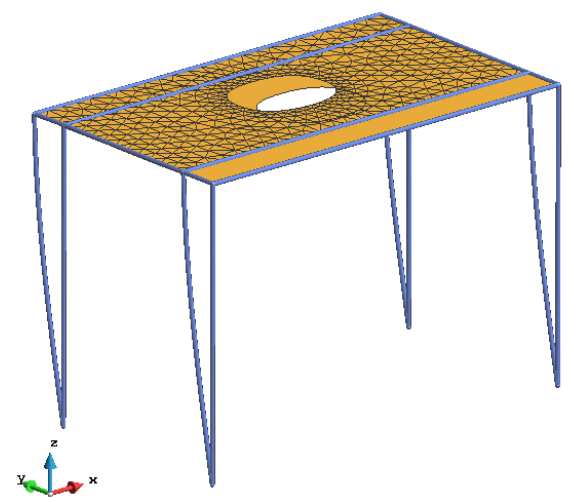
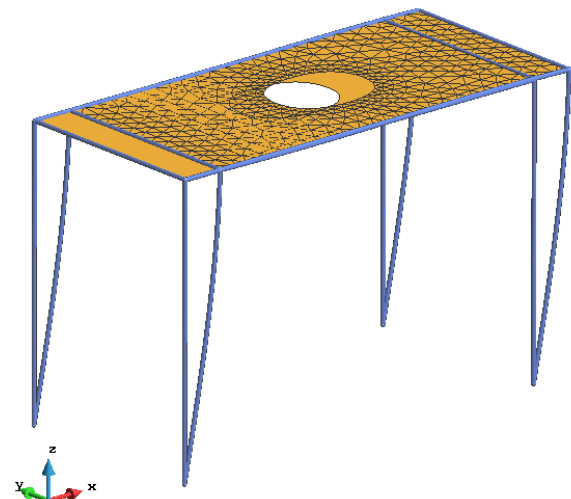
One face of the shell carries a distributed load of 1000 N/m along the negative global X-axis. A **Beam pressure Load** will be assigned to such face.

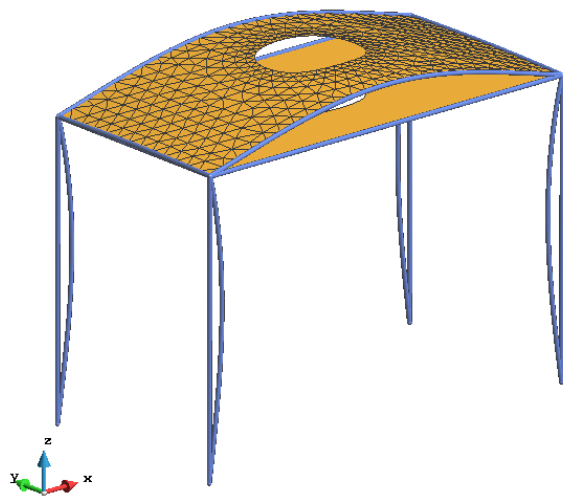
Loadcases ► Loadcase 1 ► Beams ► Pressure load

Factor	Click Function icon to insert the following info:
Function on time	Sinusoidal load
Amplitude	1.0
Frequency	0.5 Hz
Phase angle	0.0 deg
Initial time	0.0 s
End time	4 s
X pressure	-10000 N/m
Y pressure	0.0 N/m
Z pressure	0.0 N/m ²

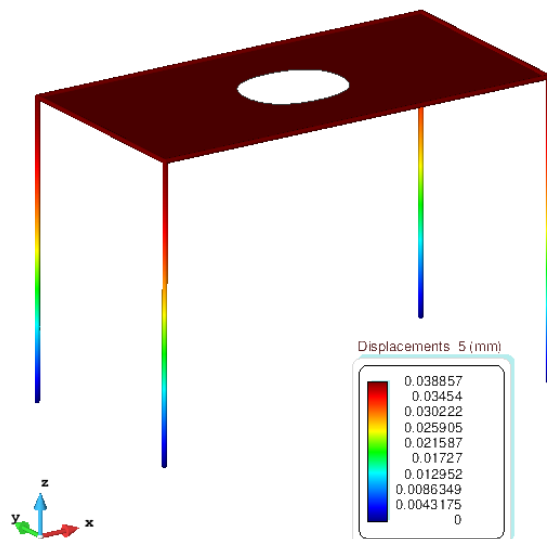
5.3. Results of the example "Dynamic analysis of coupled beam and shell structure"

The following pictures show several results for the performed analysis:

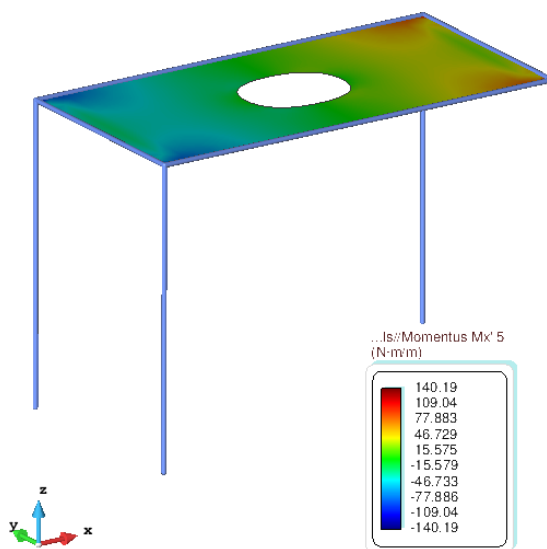




Vibration mode # 3 - 19.33 Hz



Displacement field after 5 seconds



Momentum Mx after 5 seconds

6. Tutorial 5 - Fatigue Damage Assessment

In this example the dynamic analysis of a shell is performed. The fatigue damage caused over the structure is also evaluated by using the tools provided in Tdyn for fatigue damage assesment. The shell structure under analysis is similar to that of example Tutorial 4 - Dynamic analysis of a coupled beam and shell structure but without the supporting beams since the shell is completely constrained at two of its boundary edges.

The model, ready to be meshed and run, can be automatically loaded into the GUI by left-clicking on the button below.



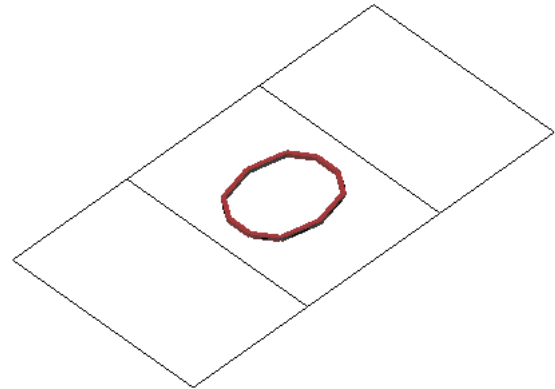
6.1. Beams properties

To assign the section and material properties to the beams choose:

Materials and properties ► Beams ► Rectangular section

In this example the beam is considered to have a rectangular section with an area $A = 0.01 \text{ m}^2$. The actual material being used is the predefined S-355N steel. Hence, beam properties must be defined as indicated in the figure below and must be applied to the boundary of the hole located at the center of the shell structure.

Section and material properties to be assigned to the beams located at the boundary of the shell structure



Beam applied to the boundary of the circular hole located at the center of the shell structure

- Local axes system 'Automatic' is chosen. Following the criteria given before, X' axes will go along the boundary and Y' axe will be horizontal. Z' axe will have the same direction and sense than Z axe.
- The 'Width y' is the width of the rectangular section following the Y' direction. So, it is the horizontal width and its value is 0.1 m. 'Width z' is the width in the Z' direction. In this case its value is also 0.1 m.
- G is the torsion modulus ($8.1e10 \text{ N/m}^2$) and E is the Young modulus ($2.1e11 \text{ N/m}^2$). Remember that for an isotropic material $G = \frac{E}{2 \cdot (1 + \nu)}$, where $\nu=0.3$
- If the Specific Weight is left to 0.0, then the self-weight of the beam is not considered. Its default units are N/m^3 .

Once the values are introduced in the 'Rectangular section' definitions window, the condition must be assigned to line that defines the contour of the circular hole at the center of the shell.

Assigned conditions can be verified by right-clicking on the group under consideration and selecting the following option in the contextual menu.

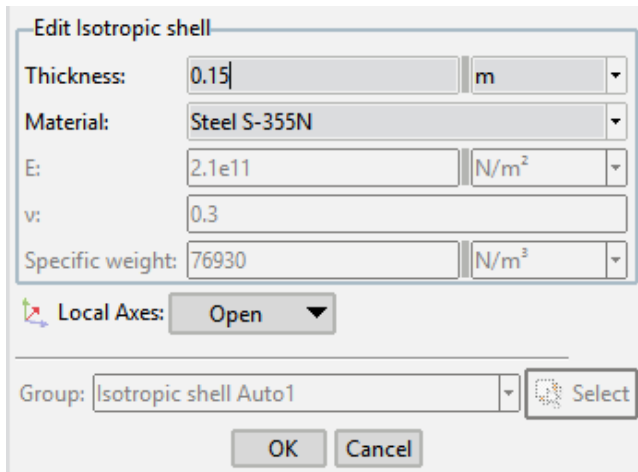
Draw ► Draw values / symbols / groups

6.2. Shell properties

To assign the section and material properties to the shell choose:

Properties ► Shells ► Isotropic shell

Shell properties for the present example read as follows:



Edit Isotropic shell

Thickness: 0.15 m

Material: Steel S-355N

E: 2.1e11 N/m²

ν: 0.3

Specific weight: 76930 N/m³

Local Axes: Open

Group: Isotropic shell Auto1

OK Cancel

Thickness and material properties to be assigned to the shell structure under analysis

- Local axes system 'Default' is chosen. Following the criteria given before, X' axe will point towards positive global Y axe. Z' axe will be normal to the elements and Y' will be contained in the plane XZ.
- 'Thickness' is the width of the shell in the orthogonal direction to the shell surface. In this case its value is 0.2 m.
- E is the Young modulus ($4.32 \text{ e}8 \text{ N/m}^2$) and ν is the Poisson coefficient with a value 0.3 in this case.
- If the Specific Weight is left to 0.0, then the self-weight of the shell is not considered. Its default units are N/m^3 .

Once the values are filled in, the condition must be assigned to the surface that defines the shell.

The conditions that have been assigned can be verified at any moment by right-clicking on the group under consideration.

Draw ► Draw values / symbols / groups

6.3. Fatigue assesment data

To perform a fatigue damage assesment analysis it is first necessary to choose the zone where the analysis is actually going to be performed. Note that we are usually not interested on doing the fatigue damage assesment over the entire structure but only on a given local area. Note also that the fatigue damage assesment can be performed over lines, thus representing welded joints, or over surfaces, then representing steel shells. To this aim, first activate the 'Fatigue Damage Assesment' capability by selecting the following option in the data tree:

General data ► Analysis ► Damage assesment

If a SN curves file is available, it can be loaded from:

General data ► Analysis ► SN curves file

Actually, a SN curve file is already provided with the software distribution in the scripts directory of the problemtype.

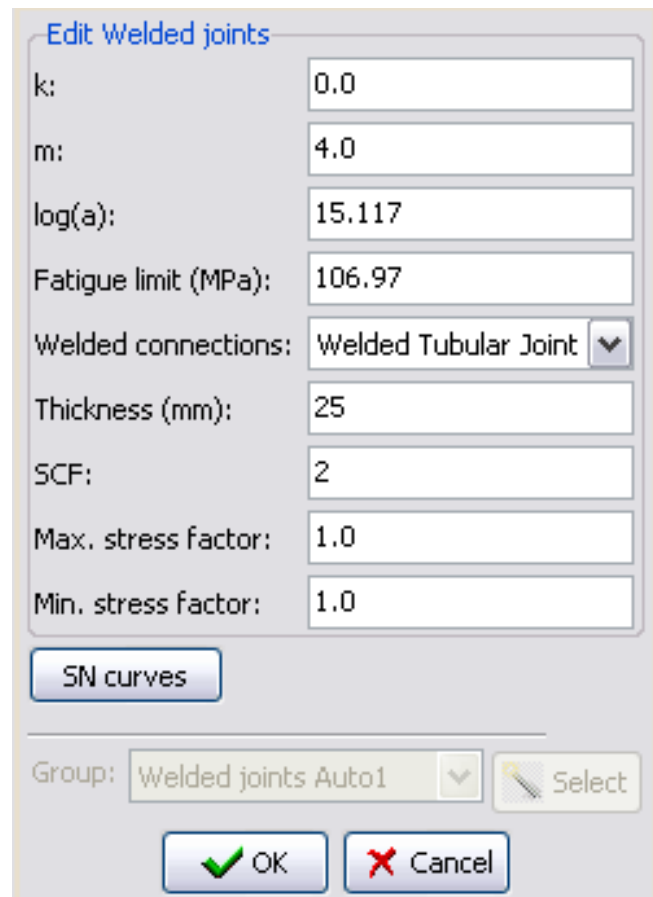
Finally, in order to be able to perform the fatigue damage analysis, it is necessary to define the analysis time for each combined load.

New combined	type	ncicles	Loadcase_1
New combined 1	ELU	5.0e6	1.0
New combined 2	ELU	4.0e6	0.8
New combined 3	ELU	1.0e7	0.6
New combined 4	ELU	5.0e7	0.4
New combined 5	ELU	1.0e6	1.2
New combined 6	ELU	3.0e6	1.4
New combined 7	ELU	2.0e7	0.9

Welded joints

Materials and properties ► Beams ► Damage assesment ► Welded joints

Assign this condition to any line of the geometry which is to be treated as a welded joint and be under fatigue check.



Edit Welded joints

k: 0.0

m: 4.0

log(a): 15.117

Fatigue limit (MPa): 106.97

Welded connections: Welded Tubular Joint

Thickness (mm): 25

SCF: 2

Max. stress factor: 1.0

Min. stress factor: 1.0

SN curves

Group: Welded joints Auto1

OK Cancel

Shells


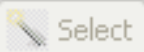
Materials and properties ► Shells ► Damage assesment ► Shells



Assign this condition to any surface of the geometry which is to be checked for fatigue damage under cyclic loads.

Edit Shells


k:	0.200
m:	3.000
log(a):	12.010
Fatigue limit (MPa):	46.780
Reference thickness (mm):	25
Thickness (mm):	25
SCF:	3
Max. stress factor:	1.0
Min. stress factor:	1.0


SN curves


Group: Shells Auto1  

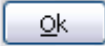

 

In both cases, the coefficients regarding SN curves can be either inserted manually, or automatically by choosing the desired curve to use from the curves file (this SN curves file is included in the example, *DNV_SNcurves.csv*), if it has been inserted:

GD SN Curves 

B1 

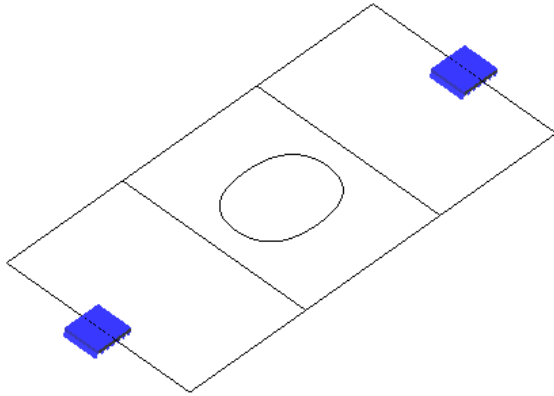
In air 

SN curves  

For more information regarding **Fatigue Damage Assessment**, please review: [Appendix 10: Fatigue Damage Assessment](#).

6.4. Displacement constraints

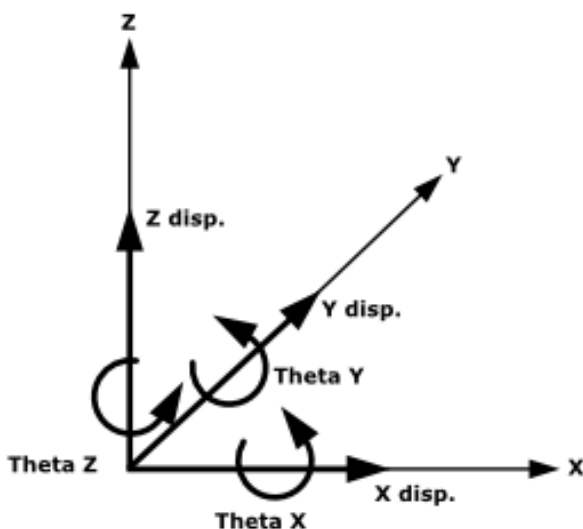
Constraints for shells can be either applied to points or lines. In this example, the constraints are applied to the two short boundary edges of the shell. Both lines will have all six degrees of freedom restrained.



The symbols indicate that the corresponding edges are constrained

In this condition, the local axes have no relationship with the shell local axes defined in the properties section. The GLOBAL option means to prescribe related to the global axes of the problem. This is used in this case. Local axes are used to prescribe the displacement or rotation in a direction not coincident with any of the global axes. The values part of the condition is used to prescribe a fixed amount of displacement or rotation. Default units are meters for the X, Y and Z displacements and radians for the prescribed rotations. In this case, the prescribed displacements and rotations are all zero.

X Constraint, Y Constraint and Z Constraint mean the displacements along the axes. Theta x Constraint, theta y constraints and theta z constraints mean the rotations around the axes. Signs are as follows (right hand rule):



This condition must be assigned to the lines described in the picture. In all cases, the prescribed displacements and rotations are zero.

6.5. Dynamic Loads

The shell is subjected to the action of a distributed load perpendicular to the surface of the shell with a magnitude $P = -50000$ Pa.

Pressure load definition window

Definition of the sinusoidal signal used to modulate the pressure load

6.6. Problem data

In this example the problem data information is similar to that of the previous example. The required information for the dynamic analysis can be defined in the analysis options:

General data ► Dynamic analysis data ► General

General data ► Dynamic analysis data ► Integration data

Integration data

Integration method: Implicit (Energy Conserving/Decaying) ▼

Alpha E-C/D (α): 0.0

Matrix storage: Consistent ▼

Initial conditions: None ▼

OK Cancel

General data ► Dynamic analysis data ► Damping data

Damping Data

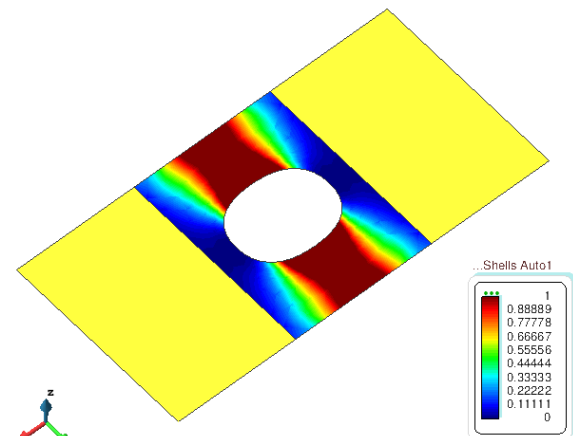
Damping type: Rayleigh damping ▼

Damping ratio: 0.05

αM : 0.1

αK : 0.0

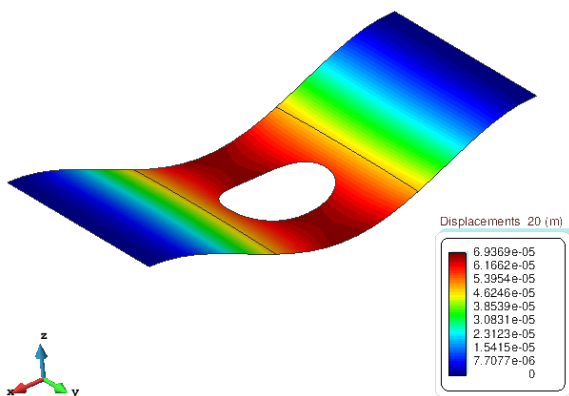
OK Cancel



The contour fill maps above show the critical zones where this fatigue is more likely to occur. The possible range for this result is 0-1. If the value is 1 -corresponding to red in the image above- the structural element is likely to fail under fatigue loads.

6.7. Results

Due to the cyclic load applied, the stresses variation occurred on the shells and the joints, may lead these structural elements to suffering from early crack creation/propagation.



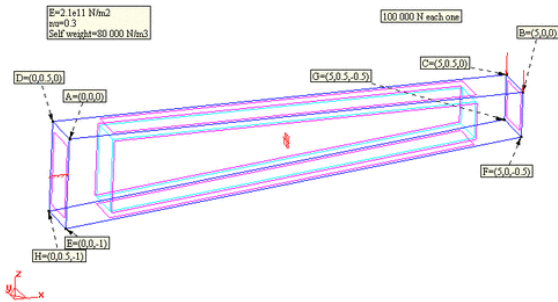
For performing the fatigue check, go to:

Postprocess ► Damage assesment

- Fatigue damage in welded joints
- Fatigue damage in shells

7. Tutorial 6 - Analysis of a solid

The example used to describe the solid analysis will be a cantilever beam with diminishing cross section.



The geometry corresponding to this tutorial can be found here (IGES format):

[RamSeries-Tutorials6-solids.igs](#)

Alternatively, the model ready to be meshed and run, can be automatically loaded into the GUI by left-clicking on the button below.



If you prefer to build the model from scratch, the geometry can be simply constructed in GiD using the following usefull pre-processing commands which are described in detail in the GiD User's Manual. The final model will be composed of 1 volume, 6 surfaces, 12 lines and all of its connecting points. Some useful commands are:

Geometry ► Create ► Line

Geometry ► Create ► NURBS surface ► By contour

Geometry ► Create ► Volume ► By contour

Be careful to position the structure related to the global axes as seen in the picture. Then, it will be easier to follow the example. The eight points will be:

A	0.000000 0.000000 0.000000
B	5.000000 0.000000 0.000000
C	5.000000 0.500000 0.000000
D	0.000000 0.500000 0.000000
E	0.000000 0.000000 -1.000000
F	5.000000 0.000000 -0.500000
G	5.000000 0.500000 -0.500000
H	0.000000 0.500000 -1.000000

7.1. Loading Solids analysis

To load the RamSeries type of analysis, choose:

Simulation type ► Structural analysis

For this kind of problem, the Solids analysis can be set up from the Start Data window, as shown in the Beams tutorial, with the following options in this case:

Data ► Start data

- Simulation dimension: 3D

- Structural analysis: Solids

- Analysis type: Static analysis

- Material constitutive model: Linear-elastic model

- Geometrical constitutive model: Linear geometry

7.2. Displacement constraints

Constraints for solids can be either applied to points or lines or surfaces. In this example, the constraints are applied to one surface of the boundary of the solid.

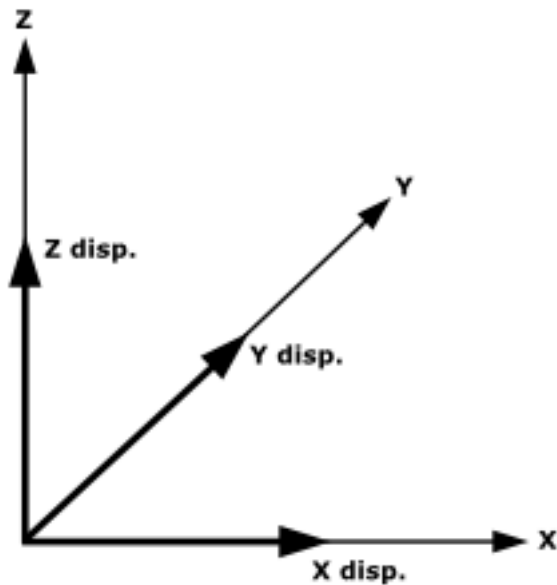
Constraints ► Fixed constraints

Activation		Values	
X constraint	YES	X value	0.0
Y constraint	YES	Y value	0.0
Z constraint	YES	Z value	0.0

The condition must be assigned to the vertical surface with minimum X value.

In the local axes field, The GLOBAL option means to prescribe related to the global axes of the problem. This is used in this case. Local axes are used to prescribe the displacement in a direction not coincident with any of the global axes. The values part of the condition is used to prescribe a fixed amount of displacement. Default units are meters for the X, Y and Z displacements. In this case, the prescribed displacements are all zero.

X Constraint, Y Constraint and Z Constraint mean the displacements along the axes. Signs are as follows:



7.3. Solid properties

To assign the section and material properties to the solid choose:

Materials and properties ► Solid ► Isotropic solid

-E= 2.1e11 N/m² (Young modulus)

-ν= 0.3 (Poisson coefficient)

-Specific Weight= 8.0e4 N/m³

Note: Remember that for an isotropic material: $G=E/(2\cdot(1+\nu))$

Once the values are filled in, the condition must be assigned to the volume that defines the solid.

The conditions that have been assigned can be viewed right-clicking on the created group:

Draw ► Draw values / Draw groups / Draw symbols

7.4. Punctual Loads

Points B and C in the solid have a punctual load of 100000 N each one in the negative Z direction. So the condition **Punctual Load** will be used.

Loadcases ► Loadcase 1 ► Solids ► Punctual load

Factor	1.0
X pressure	0.0 [N]
Y pressure	0.0 [N]
Z pressure	-1.0e5 [N]

This condition will be assigned to the two points B and C.

7.5. Mesh generation

RamSeries accepts 4-noded and 10-noded tetrahedron as the elements that define the solid. So, it is possible to mesh the

volumes with either *Normal* or *Quadratic* tetrahedrons (see).

Meshing ► Quadratic elements

Use always quadratic tetrahedrons except when, due to the complexity of the geometry, the minimum final amount of nodes is too big for the computer capacity. They give much more precision in the results for the same amount of nodes.

If more elements are used to mesh the geometry, more accuracy in the results will be obtained. At the same time, more computer time and RAM memory is needed. Use

Calculate ► View process info

after doing a preliminary analysis, in order to obtain information of the memory requirements and the computer time needed.

It is also interesting to mesh smaller elements in the zones of the solid that will have a bigger gradient in the results.

To estimate the precision of the results related to the number of nodes in the mesh and the type of tetrahedrons, check the test graphics given in following sections.

To obtain the desired sizes of the elements, use GiD options to control the mesh size like:

Meshing ► Assign unstructured sizes ► Points

Meshing ► Assign unstructured sizes ► Lines

Meshing ► Assign unstructured sizes ► Surfaces

Meshing ► Assign unstructured sizes ► By chordal error

To generate the mesh use:

Meshing ► Quadratic elements ► Quadratic

Meshing ► Generate ► 0.2

In this model, 10-noded tetrahedrons are used and a default size of 0.1 is chosen for the elements. The total amount of nodes is 27166 and the total amount of tetrahedrons is 18041.

7.6. Calculate

If the model has not been saved yet, use:

Files ► Save

And give a name to the model.

To begin the analysis choose:

Calculate ► Calculate

The analysis runs as a separate process. Then, it is possible to continue working with GiD or exit the program. When the analysis is finished, a window will appear notifying it. If it has not run successfully, one window showing some error info will be supplied. Correct the error and run again.

It is possible to visualize, when the process is running, some information about its evolution. Press:

Calculate ► View process info

To get it. One of the most important information that can be obtained in this window is the amount of RAM memory, expressed in Megabytes, needed by the solver. The total amount of memory required by the analysis code is somewhat bigger. For skyline solver, the total amount is about 10% more.

For Sparse solver, it is about the double of memory. This data gives an idea of the maximum problem that can be solved in a given computer.

This specific analysis needs about 30Mb of RAM memory to execute in the direct solver. In the conjugate-gradients solver, it would take about 180 iterations to finish and much less memory.

7.7. Postprocess

The part of the program dedicated to visualize the results of the analysis is called Post-process. Once the calculation is finished, to enter in the post-process part and load automatically the results select:

Postprocess ► Start

To perform the operation correctly, it is necessary to have the model loaded inside the reprocessing part before going to post-process.

Choose the view style *Body bound* if it is not still selected. With this type of view, *Contour fills* on solids are visualized better:

Postprocess panel ► Meshes ► Display style

7.8. Deformation of the structure

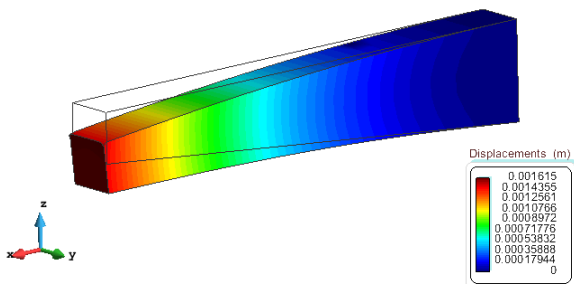
To visualize the deformed of the structure check:

Postprocess panel ► Results ► Static ► Displacements ► Contour fill

Postprocess panel ► Preferences ► Deformed ► Draw:[Yes]

Postprocess panel ► Preferences ► Deformed ► Factor:[200]

Postprocess panel ► Preferences ► Deformed ► Result:["Static Displacements"]

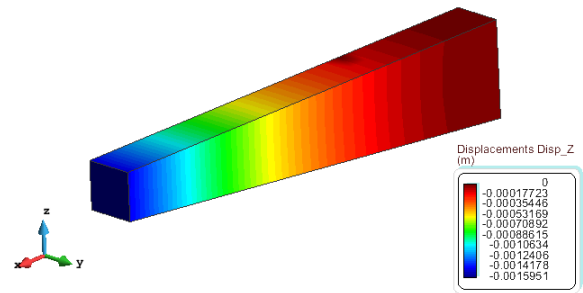


The deformed of the structure is drawn magnified in the screen. The magnifying factor is automatically calculated if "Adimensional factor" option is selected.

7.9. Contour fill of the displacements

Another way to see the magnitude of the deformation of the solid is to use:

Postprocess panel ► Results ► Static ► Displacements ► Disp z ► Contour fill



To see the numerical values of the deformation choose:

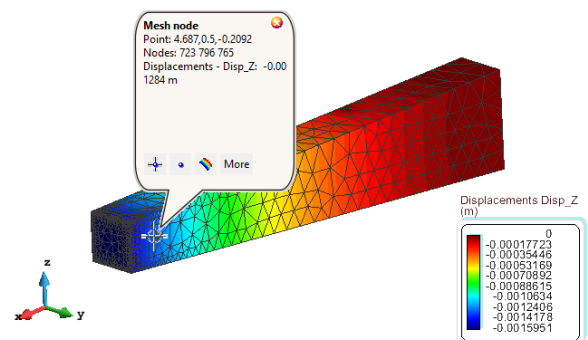
Postprocess ► Mesh information

Try also pressing mouse left button over any structure element in the screen.

Make the appropriate zooms to visualize properly the values.

To switch off the labels press:

Mouse right button ► Label ► Off



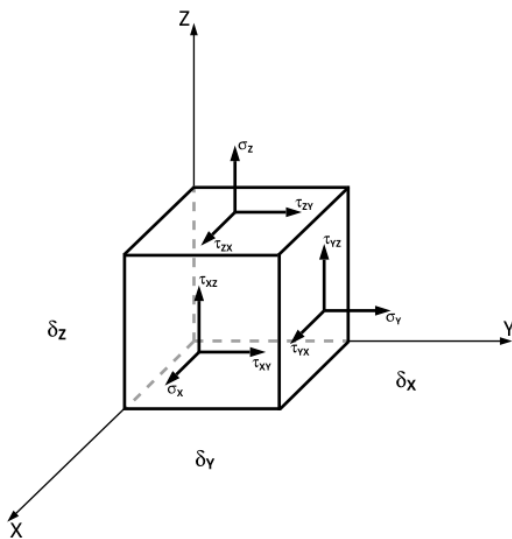
7.10. Visualization of strengths

All of the strengths are represented related to the global axes systems of the model (except Von Misses that do not depend on the axes). The visualization will be generally with a Contour Fill of every component of the strength although there are other ways to visualize them.

The strengths are the following:

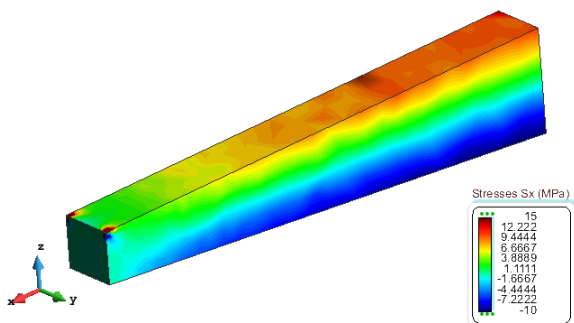
Name		Default units	Remarks
Normal strengths	Sx,Sy,Sz	N/m ²	Also called σ_x σ_y σ_z are the stresses for every global axe
Tangential strengths	Txy, Txz, Tyz	N/m ²	Also called τ_{xy} τ_{xz} τ_{yz}
Main stresses	Si, Sii, Siii	N/m ²	Main stresses expressed in the main axes

The sign criteria for these strengths is:



As an example, this is the σ_x strength on the solid:

Postprocess panel ► Results ► Static ► Stresses ► Sx ► Contour fill



7.11. Self defined Contour fill limits for strengths

It is quite usual that there are local concentrations of strengths near the constraint points or near the nodal forces. In these cases, the scale of color has not enough contrast in the rest of the shell. To avoid it, it is possible to change the limits of the values of the contour fill. The zones of the shell that have values higher than the maximum limit or lower than the minimum one, will appear either black, transparent or in a defined colour, depending on the user preferences.

To define the maximum and minimum limits activate the corresponding checkbox, and define the limits:

Postprocess panel ► Results ► Limits

Right-click over the "Limits" label, leads to the following available options:

- Recalculate limits
- Recalculate limits-all steps (only dynamic analyses)
- Multiple results
- Only active meshes
- Legend colours
- Redraw

7.12. Main stresses

Following the theory of elasticity, the strengths in a differential volume can be expressed in global axes by this tensor:

$$\begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix}$$

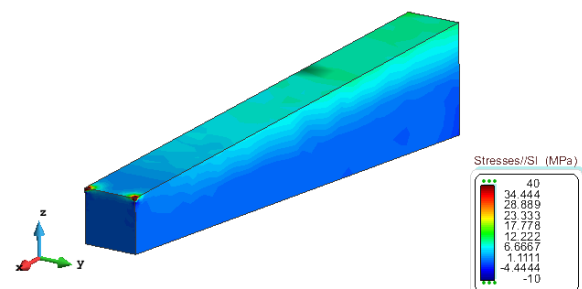
It is always possible to find another base $X'Y'Z'$, different for every node, where the tensor is represented as:

$$\begin{bmatrix} \sigma_i & 0 & 0 \\ 0 & \sigma_{ii} & 0 \\ 0 & 0 & \sigma_{iii} \end{bmatrix}$$

These are the eigenvalues of the matrix and the new base is made with the eigenvectors. Its values represent the maximum and the minimum strengths values for that point of the solid.

It is possible to see these values with:

Postprocess panel ► Results ► Static ► Stresses ► Si ► Contour fill



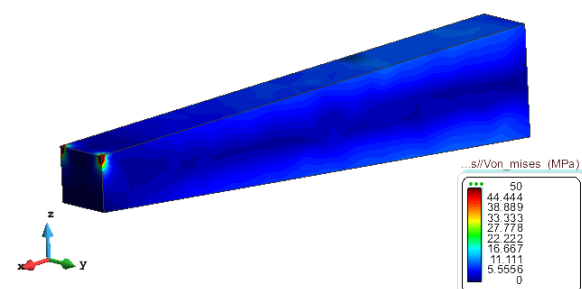
7.13. Von Mises

Von Mises strength is a scalar tension that gives a mean value of all the strengths in a given point of the solid. It can be compared with the maximum acceptable strength for that material. Its expression is given by:

$$\sigma_{vm} = \sqrt{\frac{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}{2}}$$

To see a contour fill of its value, choose:

Postprocess panel ► Results ► Static ► Stresses ► Von Mises ► Contour fill



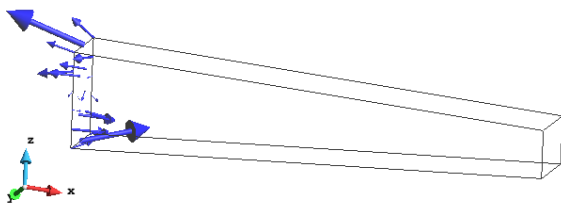
Note: The maximum value to visualize has been set manually to $2.1e7$ in order to appreciate a good contrast in the colors. The zones that have a bigger Von Mises value (near the application of the point loads), are in the same colour as the maximum (red).

7.14. Reactions

These are the reactions that appear in the constraints. All nodes that have no constraint will have a null reaction. Default units are Newtons.

To see the reaction vectors, choose:

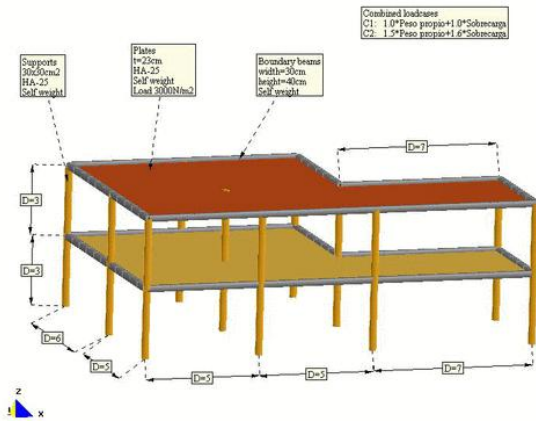
Postprocess panel ► Results ► Static ► M Reactions ► Vectors



If labels are displayed, four numbers appears. The first three are the reaction vector and the last is the module of this vector.

8. Tutorial 7 - Analysis of a concrete building

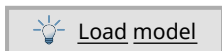
This example will consist in the analysis of a simple concrete building introducing the concepts of combined load cases and the concept of concrete dimensioning.



The geometry corresponding to this tutorial can be found here (IGES format):

[RamSeries-Tutorials7-concrete_building.iges](#)

Alternatively, the model ready to be meshed and run, can be automatically loaded into the GUI by left-clicking on the button below.



If you prefer to build the model from scratch, the geometry can be simply constructed in GiD using the following usefull pre-processing commands which are described in detail in the GiD User's Manual.

Geometry ► Create ► Line

Utilities ► Copy ► Traslation (Do extrude lines)

Geometry ► Create ► NURBS surface ► By contour

View ► Rotate ► Trackball

View ► Render ► Smooth

The final model will be composed of 8 surfaces, 22 lines for the supports, 28 lines for the boundary beams, 8 interior help lines (not meshed) and all of its connecting . Be careful to position the structure related to the global axes as seen in the picture. Then, it will be easier to follow the example. The six points that define the base projection of the building are the following:

A->	(-5,0,0)
B->	(12,0,0)
C->	(12,5,0)
D->	(5,5,0)
E->	(5,11,0)
F->	(-5,11,0)

As a convenience, it is interesting to subdivide the geometrical entities into several layers. This will help to assign the properties and to visualize easily the results.

To transfer geometrical entities to one layer, use button

entities in the layers window.

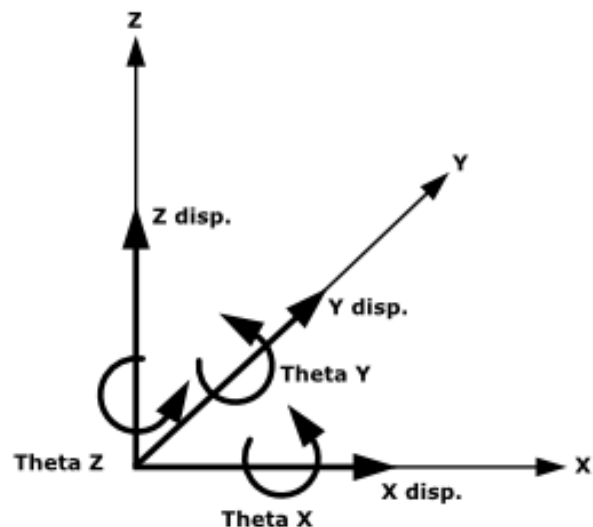
8.1. Displacement constraints

Constraints for shells can be either applied to points or lines. In this example, the constraints are applied to the 11 points that are the base of the supports. These points have prescribed both, the displacements and the rotations.

Constraints ► Fixed constraints

Activation		Values	
X constraint	YES	X value	0.0
Y constraint	YES	Y value	0.0
Z constraint	YES	Z value	0.0
θ_x constraint	YES	θ_x value	0.0
θ_y constraint	YES	θ_y value	0.0
θ_z constraint	YES	θ_z value	0.0

If there were prescribed displacements, signs would be the following:



This condition must be assigned to the 11 points that are the base of the building. In all cases, the prescribed displacements and rotations are zero.

8.2. Shell properties

To assign the section and material properties of an isotropic the shell, choose:

Materials and properties ► Shells ► Isotropic shell

-Thickness= 230 mm

-Material= HA-25 (Its characteristic resistance is 25 N/mm²)

-E= 27.264e9 N/m² (Young modulus)

-v= 0.2 (Poisson coefficient)

-Specific Weight= 25000 N/m³

Local axes system **Default** is chosen. Following the criteria given before, X' axis will point towards positive global Y axis. Z' axis will be normal to the elements and Y' will be contained in the plane XZ.

Once the values are filled in, the condition must be assigned to the surface that defines the shell.

The conditions that have been assigned can be viewed right-clicking on the created group:

Draw ► Draw values / Draw groups / Draw symbols

8.3. Support properties

To assign the beam properties to the support, disconnect all layers except the layer called *pilares*. Then choose:

Materials and properties ► Beams ► Rectangular section

The beams will have the following rectangular section properties:

- Width Y= 300 mm (width of the rectangular section along the Y' direction)
- Width Z= 300 mm (width of the rectangular section along the Z' direction)
- Material= HA-25 (Its characteristic resistance is 25 N/mm²)
- E= 27.264e9 N/m² (Young modulus)
- ν= 0.2 (Poisson coefficient)
- Specific Weight= 25000 N/m³
- Maximum stress= 25 N/mm²

To assign properties to the beams, it is first necessary to define local axes. To this aim, in the present case **Automatic** local axes are used. Hence, following the standard criteria, X' axes (prime refers to local axes) will remain aligned along the horizontal beams longitudinal axis. On the other hand, Y' will be horizontal and perpendicular to the beam longitudinal axis, while Z' will have the same direction and sense than the global Z axis.

Once the values of all rectangular section parameters have been provided, the condition must be assigned to all the active vertical beams.

When assigning the properties on the vertical beams, the Local axes system **Automatic** is chosen, following the criteria given before. In this case the Y' axis points to the direction of the X axis.

The conditions that have been assigned can be viewed right-clicking on the created group:

Draw ► Draw values / Draw groups / Draw symbols

8.4. Boundary beams properties

To assign the properties to the boundary beams, disconnect all layers except the layer called *vigas_de_borde*. Then choose:

Materials and properties ► Beams ► Rectangular section

The beams will have the following rectangular section properties:

- Width Y= 300 mm (width of the rectangular section along the Y' direction)
- Width Z= 400 mm (width of the rectangular section along the Z' direction)
- Material= HA-25 (Its characteristic resistance is 25 N/mm²)
- E= 27.264e9 N/m² (Young modulus)
- ν= 0.2 (Poisson coefficient)
- Specific Weight= 25000 N/m³
- Maximum stress= 25 N/mm²

To assign properties to the beams, it is first necessary to define local axes. To this aim, in the present case **Automatic** local axes are used. Hence, following the standard criteria, X' axes (prime refers to local axes) will remain aligned along the horizontal beams longitudinal axis. On the other hand, Y' will be horizontal and perpendicular to the beam longitudinal axis, while Z' will have the same direction and sense than the global Z axis.

Once the values of all rectangular section parameters have been provided, the condition must be assigned to all the active horizontal beams.

When assigning the properties on the horizontal beams, the Local axes system **Automatic** is chosen, following the criteria given before.

The conditions that have been assigned can be viewed right-clicking on the created group:

Draw ► Draw values / Draw groups / Draw symbols

8.5. Loadcases

When a new model is created, RamSeries has already defined a simple load case. Before applying loads, it is necessary to have created the load case where the loads will be included. To rename the first load case, choose **Rename** in the menu displayed after doing right-click on

Loadcases ► Loadcase 1

and enter the name '*Peso propio*' for the new simple load case.

To create a new load case, select **Create new Loadcase** in the menu displayed after doing right-click on

Loadcases

Enter name '*Sobrecarga*' for the created load case.

8.6. Self weight load

As we want this load included in the *Peso propio* load case, select it in the menu defined in the previous section for applying the load:

Loadcases ► Peso propio ► Beams ► Self weight

And assign this load to all the supports and boundary beams.

Then choose:

Loadcases ► Peso propio ► Shells ► Self weight

And assign it to all the surfaces that define the shells.

8.7. Distributed Loads in the shells

As we want this load included in the *Sobrecarga* load case, select it in the menu defined in the previous section for applying the load:

Loadcases ► Sobrecarga ► Shells ► Pressure load

The entire shell has a distributed load of 3000 N/m² along the negative global Z-axis:

Factor	1.0
Loadtype	Global
X pressure	0.0 [N/m]
Y pressure	0.0 [N/m]
Z pressure	-3000.0 [N/m ²]

This condition will be assigned to all the surfaces that define the shells.

8.8. Combined loadcases

Open the window:

Loadcases ► Combined loadcases

RamSeries will give a different result for every combined load case. To change the name of one load case, double click over it and write the new name. To add a new combined load case, press right button mouse over it and select **insert after**.

The field ELU is not used for the analysis. It will be used later, when dimensioning the steel, to decide if this combined load case is used to calculate the section against collapse or against service states, like fissuration. Enter the coefficients that will amplify the loads. The combined load cases used for this problem are:

Combined loadcase	Amplification factor for Peso propio	Amplification factor for Sobrecarga	Will be used for section collapse checking
Peso propio+sobrecarga servicio	1.0	1.0	NO
Peso propio+sobrecarga último	1.5	1.6	YES

8.9. Mesh generation

When combining beams and shells, option *quadratic* must be deactivated (*Meshing>Quadratic* elements must be set to *Normal*). Triangles are meshed as 3-noded triangles but will be calculated internally as 6-noded triangles if problem data option is set. It is necessary to obtain good results.

If more elements are used to mesh the geometry, more accuracy in the results will be obtained. At the same time, more computer time and RAM memory is needed. Use *Calculate>View process info* after doing a preliminary analysis in order to obtain information of the memory requirements and the computer time needed.

It is also interesting to mesh smaller elements in the zones of

the shell that will have a bigger gradient in the results.

To estimate the precision of the results related to the number of nodes in the mesh, check the test graphics given in the appendixes.

To obtain the desired sizes of the elements, use GiD options to control the mesh size like:

Meshing ► Assign unstructured sizes ► Points

Meshing ► Assign unstructured sizes ► Lines

Meshing ► Assign unstructured sizes ► By chordal error

Before generating remember to force the meshing of the beam boundary lines by choosing:

Meshing ► Mesh criteria ► Mesh ► Lines

To generate the mesh use:

Meshing ► Generate ► 1

In this model, a default size of 1m is chosen for the triangles. The total amount of nodes is 381 (1373 after the internal conversion to 6-noded triangles).

8.10. Calculate

If the model has not been saved yet, use:

Files ► Save

And give a name to the model.

To begin the analysis choose:

Calculate ► Calculate

The analysis runs as a separate process. Then, it is possible to continue working with GiD or exit the program. When the analysis is finished, a window will appear notifying it. If it has not run successfully, one window showing some error info will be supplied. Correct the error and run again.

It is possible to visualize, when the process is running, some information about its evolution. Press:

Calculate ► View process info

To get it. One of the most important information that can be obtained in this window is the amount of RAM memory, expressed in Megabytes, needed by the solver. The total amount of memory required by the analysis code if somewhat bigger. For skyline solver, the total amount is about 10% more. For Sparse solver, it is about the double of memory. This data gives an idea of the maximum problem that can be solved in a given computer.

This specific analysis needs about 25Mb of memory to execute (it should be necessary a RAM of 64Mb or 96Mb to run it correctly). Using an Sparse solver, it would need much less memory.

8.11. Postprocess

The part of the program dedicated to visualize the results of the analysis is called Post-process. Once the calculation is finished, to enter in the post-process part and load automatically the results select:

Postprocess ► Start

To perform the operation correctly, it is necessary to have the

model loaded inside the reprocessing part before going to post-process.

8.12. Active load case

To select the active load case choose:

Postprocess panel ► Results ► Pp sobrecarga último

or

Postprocess panel ► Results ► Pp sobrecarga servicio

All the results will be displayed for each active load case.

8.13. Deformation of the structure

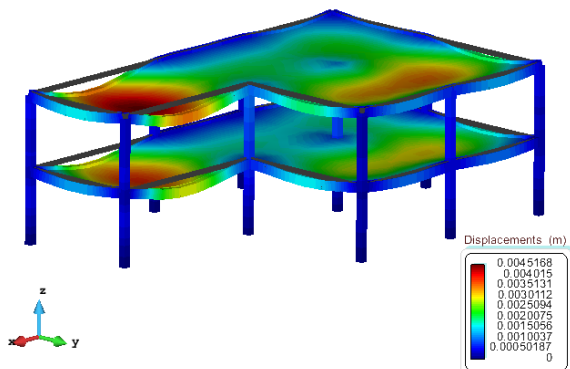
To visualize the deformed of the structure check:

Postprocess panel ► Results ► Pp sobrecarga servicio ► Displacements ► Contour fill

Postprocess panel ► Preferences ► Deformed ► Draw:[Yes]

Postprocess panel ► Preferences ► Deformed ► Factor:[200]

Postprocess panel ► Preferences ► Deformed ► Result:["Static Displacements"]

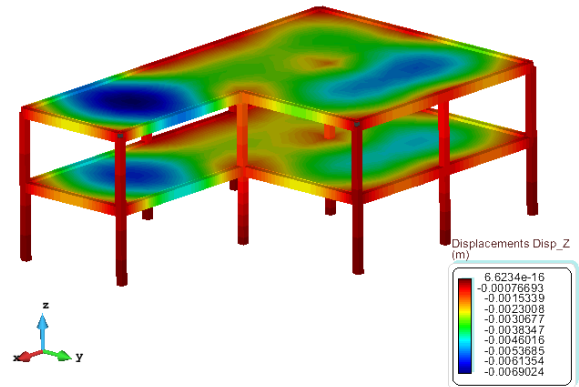


The deformed of the structure is drawn magnified in the screen. The magnifying factor is automatically calculated if "Adimensional factor" option is selected. To change the factor, write a new factor and unselect "Adimensional factor" checkbox.

8.14. Contour fill of the displacements

Another way to see the magnitude of the deformation of the shell is to use:

Postprocess panel ► Results ► Pp sobrecarga último ► Displacements ► Disp z ► Contour fill

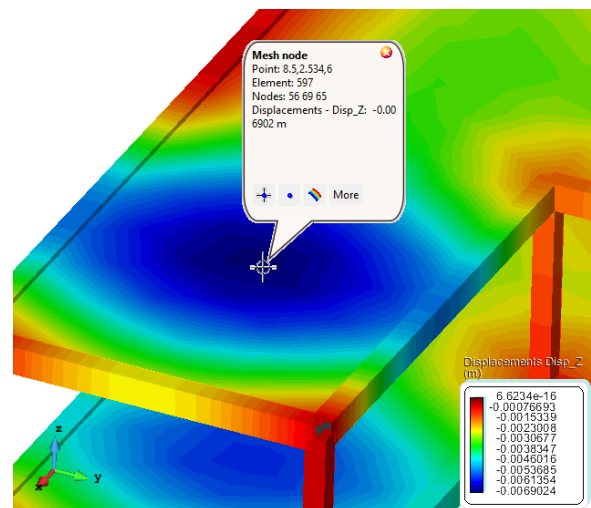


To see the numerical values of the deformation choose at any point, node or element:

Postprocess ► Mesh information

Try also pressing mouse left button over any structure element in the screen.

Make the appropriate zooms to visualize properly the values.



To switch off the labels press:

Mouse right button ► Label ► Off

In order to choose the visualization style for the shell, try:

Postprocess panel ► Meshes ► Display style

8.15. Momentum reactions

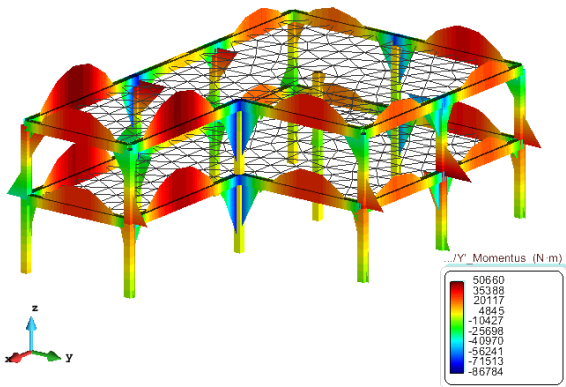
This is the reaction momentum. It is expressed as a vector showing the rotation axe and a module that means the rotation amount. Default units are N·m.

Postprocess panel ► Results ► Static ► M Reactions ► Vectors

8.16. Momentum in Y' axe

This is the momentum that rotates around the Y' axe. Default units are N·m

Postprocess panel ► Results ► Pp sobrecarga último ► Beams ► Y' momentus ► Line diagram



Sign criteria for momentum in beams is:

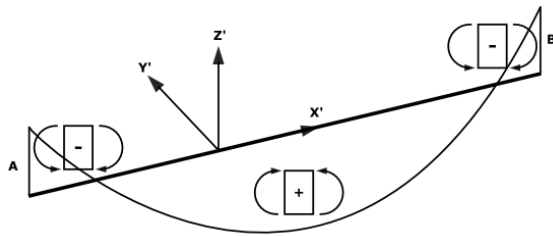
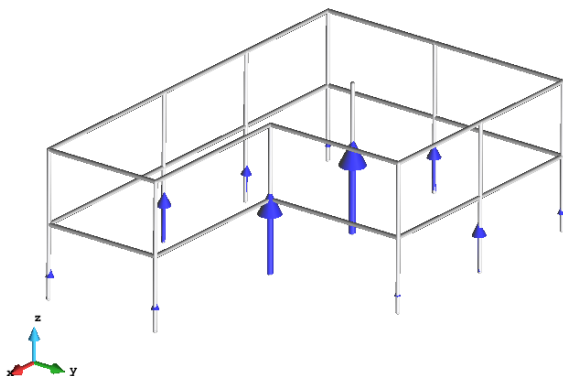


Diagram is drawn in the plane $X'Z'$ and in the side of the beam where the traction is. Positive values of the momentum mean that traction is in the $-Z'$ side (in the negative side of Z').

Other strengths in beams are described in the **Analysis of a beam structure** chapter.

8.17. Reactions

These are the reactions that appear in the constraints. All nodes that have no constraint will have a null reaction. Units are N.



If labels are displayed, four numbers appears. The first three are the reaction vector and the last is the module of this vector.

To see the colored vectors can be seen with option:

Postprocess panel ► Results ► Static ► M Reactions ► Vectors

To see the resulting reactions:

Postprocess ► Compose reactions

8.18. Self defined Contour fill limits for strengths

It is quite usual that there are local concentrations of strengths near the constraint points or near the nodal forces. In these cases, the scale of color has not enough contrast in the rest of the shell. To avoid it, it is possible to change the limits of the values of the contour fill. The zones of the shell that have values higher than the maximum limit or lower than the minimum one, will appear either black, transparent or in a defined colour, depending on the user preferences.

To define the maximum and minimum limits activate the corresponding checkbox, and define the limits:

Postprocess panel ► Results ► Limits

Right-click over the "Limits" label, leads to the following available options:

- Recalculate limits
- Recalculate limits-all steps (only dynamic analyses)
- Multiple results
- Only active meshes
- Legend colours
- Redraw

Other strengths in shells are described in the **Analysis of a shell structure** chapter.

8.19. Visualization of strengths in the beams

All of the strengths are represented related to the local axes systems of every beam. The visualization is made with a diagram that will be more separated from the beam as the magnitude of the strength increases.

Postprocess panel ► Results ► Pp sobrecarga último ► Beams ► Max stresses ► Sx ► Line diagram

It will be drawn in either side of the beam depending on the sign.

To see the numerical values of the strengths choose:

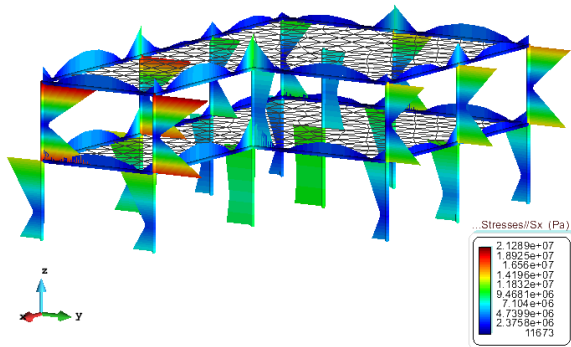
Mouse right button ► Label ► All

or

Postprocess ► Mesh information

To hide the numerical results select:

Mouse right button ► Label ► Off



8.20. Visualization of strengths in the shell

All of the strengths are represented related to the local axes systems of every shell (except the strengths in main axes). The visualization will be generally with a Contour Fill of every component of the strength although there are other ways to visualize them.

Postprocess panel ► Results ► Pp sobrecarga último ► Shells ► Axial force/Shear/Momentus

To see the numerical values of the deformation choose (in the menu that appears when pressing the secondary mouse button over the screen):

Mouse right button ► Label ► All

Or

Postprocess ► Mesh information

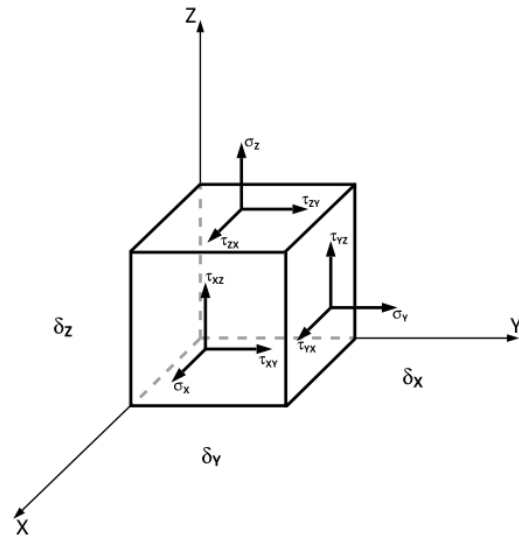
To hide the numerical results select:

Mouse right button ► Label ► Off

The strengths are the following:

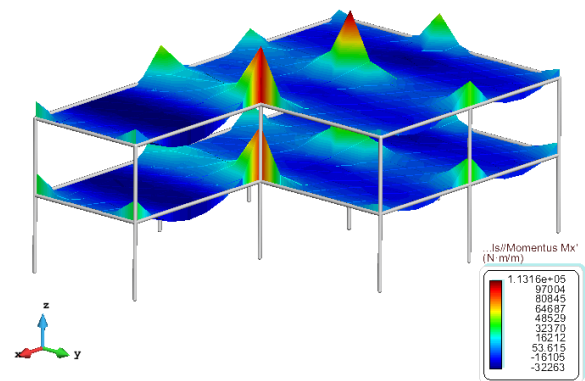
Name		Units chosen	Remarks
Axial force	N_x, N_y, N_{xy}	kN/m	Axial force in all the shell thickness per unit width of the shell
Momentum	M_x, M_y, M_{xy}	kN·m/m	Momentum per unit width of the shell
Shear	Q_x, Q_y	kN/m	Shear strength per unit width of the shell

The sign criteria for these strengths is:



Note: See that M_x momentum is contained inside plane $X'Z'$. This criteria is different from the strength criteria in beams or in constraints, where the rotation in X' is the one that rotates around X' axis. The rotation axis criteria for shell momentum are inherited from the plate criteria. In that case, as there are only two rotations in a plate, this choosing looked appropriate. For rotations in 3D, as there are 3, it is necessary to use the other criteria.

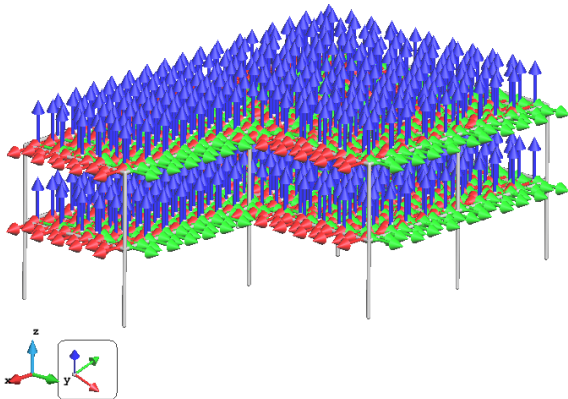
As a example, this is the M_x momentum on the shells:



8.21. Visualization of the shells Local axes

To see the local axes for Shells, choose:

Postprocess panel ► Results ► Pp sobrecarga último ► Shells ► Local axes ► Local axes

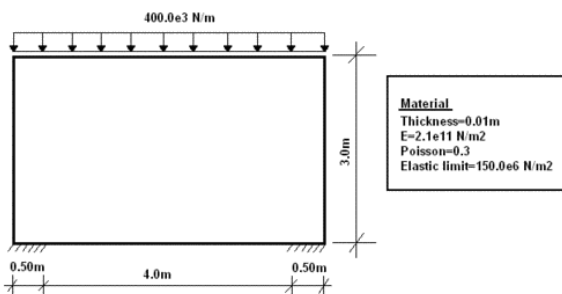


Local X' axe is drawn in color blue, Y' axe in red and Z' axe in green.

If they are different than supposed, go to the pre-processing part, change the properties accordingly and calculate again.

9. Tutorial 8 - Elastoplastic analysis of a deep beam

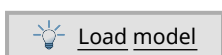
Next example shows the elastoplastic analysis of a deep beam. Geometric and mechanic characteristics are shown below.



The geometry corresponding to this tutorial can be found here (IGES format):

[RamSeries-Tutorials4-dynamic_beams_and_shells.igs](#)

Alternatively, the model ready to be meshed and run, can be automatically loaded into the GUI by left-clicking on the button below.



9.1. Analysis Data

The required information for the elastoplastic analysis can be defined in the analysis options:

General data ► Analysis

Analysis type Incremental loads analysis

Material Constitutive model Plasticity on materials

General data ► Non-linear analysis data ► General

Use default data

General data ► Non-linear analysis data ► Advanced

Use default data

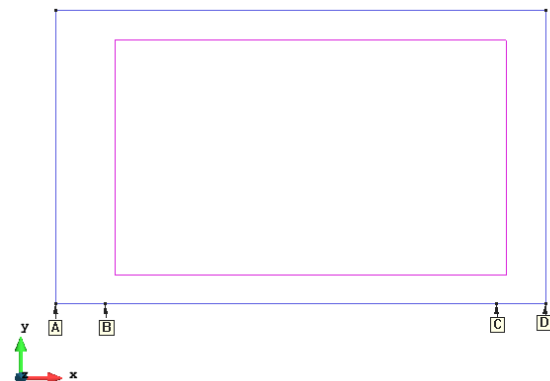
General data ► Incremental analysis data

Num increments 10

9.2. Displacement constraints

Constraints for shells can be either applied to points or lines. In this example, the constraints are applied to 2 segments of the bottom boundary of the shell.

Constraints ► Fixed constraints

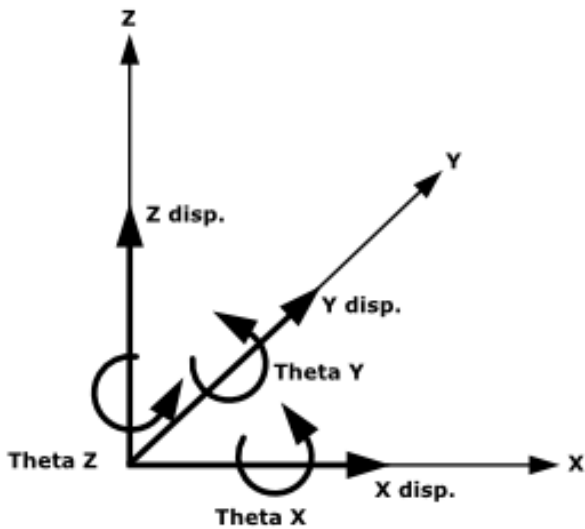


Segments **AB** and **CD** are completely restrained:

Activation	Values		
X constraint	YES	X value	0.0
Y constraint	YES	Y value	0.0
Z constraint	YES	Z value	0.0
θ_x constraint	YES	θ_x value	0.0
θ_y constraint	YES	θ_y value	0.0
θ_z constraint	YES	θ_z value	0.0

In this condition, the local axes have no relationship with the shell local axes defined in the properties section. The GLOBAL option means to prescribe related to the global axes of the problem. This is used in this case. Local axes are used to prescribe the displacement or rotation in a direction not coincident with any of the global axes. The values part of the condition is used to prescribe a fixed amount of displacement or rotation. Default units are meters for the X, Y and Z displacements and radians for the prescribed rotations. In this case, the prescribed displacements and rotations are all zero.

X Constraint, Y Constraint and Z Constraint mean the displacements along the axes. Theta x Constraint, theta y constraints and theta z constraints mean the rotations around the axes. Signs are as follows (right hand rule):



This condition must be assigned to the lines described in the picture. The prescribed displacements and rotations are zero.

9.3. Static load

The top face of the beam carries a distributed load of $400e3$ N/m along the negative global Y-axis. Therefore, the setting **Contour pressure** will be used.

Loadcases ► **Loadcase 1** ► **Shells** ► **Boundary pressure load**

Factor	1.0
Loadtype	Global
X pressure	0.0 [N/m]
Y pressure	-400e3 [N/m]
Z pressure	0.0 [N/m]

9.4. Materials

The following elastoplastic properties of the material of the beam will be assigned,

Materials and properties ► **Shells** ► **Plasticity shell**

-Thickness= 0.01 m

-Material= Steel A52 P

-Num layers= 10

Such material *Steel A52 P* has the following properties:

-E= $1.1e11$ N/m²

-G= $4.6e10$ N/m²

-ν=0.2

-Specific weight= 42112 N/m³

-Maximum stress= 0.0 N/m² (not accounted for in the calculation)

-Constitutive model= J2_plasticity

-Tensile Yield Stress= $150.0e6$ N/m²

-Isotropic hardening= None

-Kinematic hardening= None

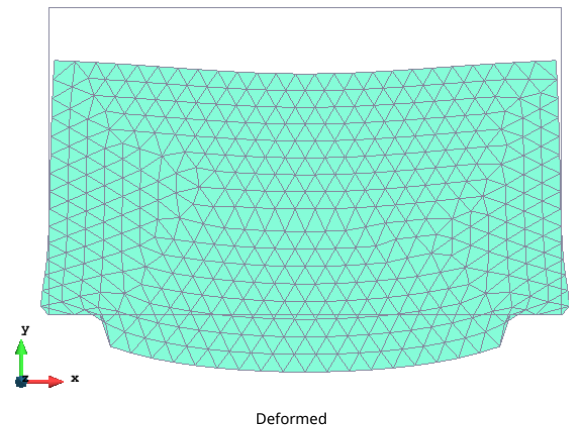
9.5. Results

The following pictures show some results for the performed analysis:

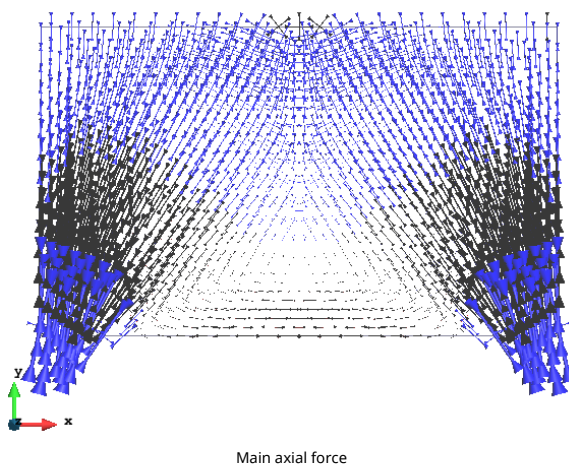
Postprocess panel ► **Preferences** ► **Deformed** ► **Draw:[Yes]**

Postprocess panel ► **Preferences** ► **Deformed** ► **Factor:[200]**

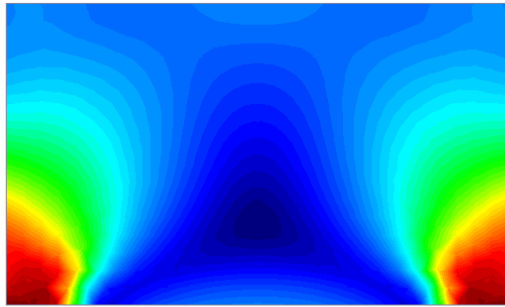
Postprocess panel ► **Preferences** ► **Deformed** ► **Result:["Static Displacements"]**



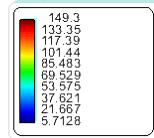
Postprocess panel ► **Results** ► **Static** ► **Shells** ► **Axial force** ► **Main stresses**



Postprocess panel ► **Results** ► **Static** ► **Shells** ► **Von mises top** ► **Contour fill**



...on Mises_Top 10
(MPa)



Von Mises stress Top [MPa]

10. Tutorial 9 - Analysis of a laminated composite shell

Introduction

The application of laminated composite materials in various engineering areas (from aeronautics to naval and civil engineering applications) is continuously increasing over the years. These laminated materials are mainly used in the form of shells and plates and also thin-walled beams built as an assembly of flat panels. In this tutorial, a very simple laminated shell clamped at the end edges and subjected to its self-weight is analyzed.

In this case, the analysis is performed using the classical lamination theory (CLT) constitutive model. Alternatively, the more sophisticated heterogeneous and serial/parallel rule of mixture (RoM) models are also available in RamSeries to undertake for instance non-linear analysis of composite structures.

Model definition

The model geometry is shown in Figure 1. It includes a shell of dimensions 1x1 meter. The shell is actually divided into two halves, each one corresponding to a different laminated composite material. Laminate 1 has a total thickness $t_1 = 9.38$ mm., while laminate 2 has a total thickness $t_2 = 13.73$ mm.

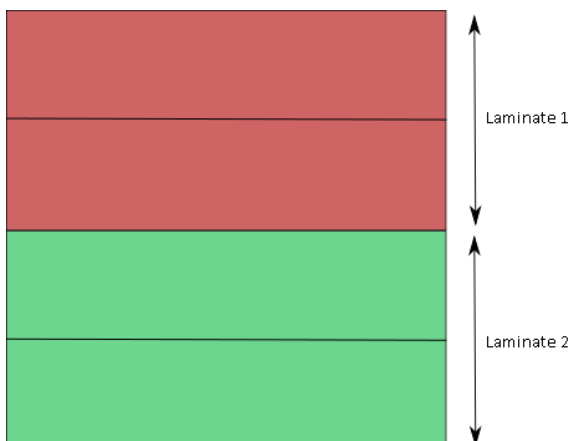
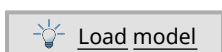


Fig.1. Laminated composite shell geometry

The geometry corresponding to this tutorial can be found here (IGES format):

[RamSeries-Tutorials9-laminate_shell.igs](#)

Alternatively, the model ready to be meshed and run, can be automatically loaded into the GUI by left-clicking on the button below.



Modelling instructions

If this is preferred, the model can be built from scratch. To this aim, the geometry can be simply constructed in GiD using the following useful pre-processing commands which are described in detail in the GiD User's Manual.

Geometry ► Create ► Object ► Rectangle

- 1.- In the command line enter the coordinates (0.0, 0.0, 0.0) of one of the vertices of the shell and press the <Enter> key.
- 2.- In the command line enter the coordinates (1.0, 0.25, 0.0) of

the opposite vertex of the first quarter of the shell and press the <Enter> key.

Utilities ► Copy (surfaces) ► Translation (Multiple copies: 3)

- 1.- In the *Copy* window select the *Surfaces* entity type.
- 2.- Select the *Translation* transformation.
- 3.- Keep the coordinates (0.0, 0.0, 0.0) for the *First point* of the transformation.
- 4.- Introduce the coordinates (0.0, 0.25, 0.0) for the *Second point* of the transformation.
- 5.- Check the *Collapse* checkbox to ensure coincident lines and not duplicated after applying the transformation.
- 6.- Select *No* in the *Do extrude* option.
- 7.- Check the *Maintain layers* checkbox to ensure, for convenience, that all the new geometrical entities created during the transformation are located in the same original layer.
- 8.- In the *Copy* window click the *Select* button and proceed to select the previously created rectangular surface. Finish the copy operation by pressing the <Escape> key.

As a result, you should obtain two adjacent rectangles representing half of the 1x1 m. shell. Repeat the Copy transformation operation by selecting both surfaces and applying a 0.5 meters translation in the Y direction. You should finally obtain the complete 1x1 shell divided for convenience into 4 equally sized parts. The final geometry of the model is hence composed by 4 rectangular surfaces, 13 lines and all the corresponding connection points.

10.1. Analysis data

In order to undertake the analysis of a composite material structure, it is necessary to explicitly activate the 'Laminate/Composite material' feature in RamSeries. This is done in the 'Analysis' section of the data tree:

General data ► Analysis

Use Laminate/Composite materials Yes

By doing this, composite materials become available in the 'Materials and properties' section of the data tree. Specific composite results become available as well, so that failure index (FI) and reserve factor (RF) results can be specified to be computed at run time.

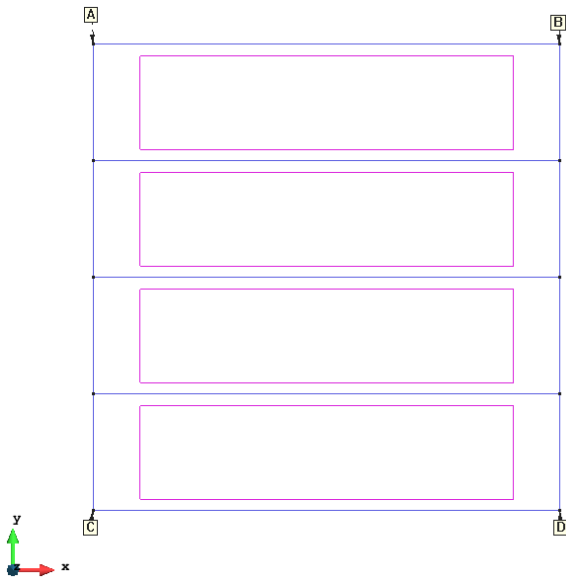
General data ► Results ► Composite sec. factor

Output composite results	Yes
Full Tsai-Wu FI	Yes
Tsai-Wu RF	Yes

10.2. Displacement constraints

The laminated shell under analysis is clamped by its two end edges. Hence, constraints must be applied to the lines corresponding to these edges, AB and CD segments in the figure below.

Constraints ► Fixed constraints

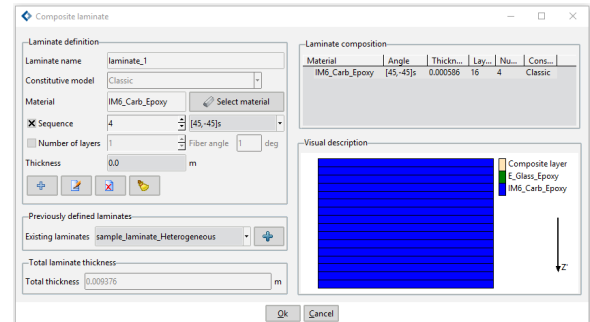


DOF	Constrained	Values
X	YES	0.0
Y	YES	0.0
Z	YES	0.0
θ_x	YES	0.0
θ_y	YES	0.0
θ_z	YES	0.0

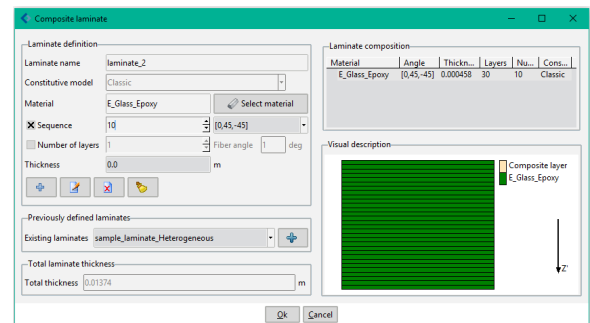
In this case, the constraints are applied in the global reference system and all displacements and rotations are fixed to zero. Local axes can be used if necessary to prescribe the displacements and/or rotations in a direction not coincident with any of the global axes.

10.3. Shell properties

The shell under analysis consists in two laminated composites joined together. Hence, two different laminate materials must be created in RamSeries. To this aim, the unitary composite materials need to be defined first. Nevertheless, few composite materials are already included by default in the RamSeries' materials library. In particular, IM6-Carb_Epoxy and E-Glass_Epoxy composite materials are available and will be used to build the laminates for the present tutorial. Laminate characteristics, such as the number of layers, the material of each layer(or set of layers) the fiber angle within each layer and the corresponding stacking sequence, must be specified for the definition of a given laminate. The first laminate in this tutorial consists in a 4 x [45,-45]_s sequence of IM6-Carb_Epoxy. This results in a laminate with 16 layers for a total thickness $t = 0.009378$ m. The second laminate consists on a 10 x [0,45,-45] sequence of E-Glass_Epoxy that results in a laminate with 30 layers and a total thickness $t = 0.013728$ m. Each laminate description can be found in the figure below.



Laminate 1 definition: 4 x [45,-45]_s sequence of IM6-Carb_Epoxy

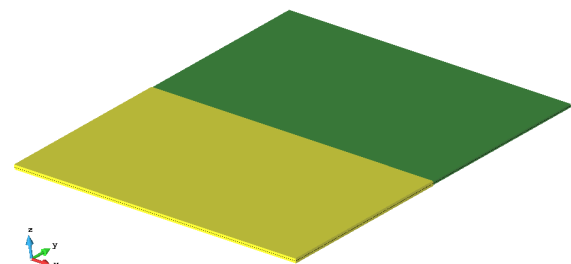


Laminate 2 definition: 10 x [0,45,-45] sequence of E-Glass_Epoxy

Once the various laminates have been defined, the actual material properties can be assigned to the different parts of the shell.

Materials and properties ► Shells ► Laminate shell

In this case, laminate 1 is assigned to half of the shell and laminate 2 is assigned to the other half. This implies the creation of two material groups that are assigned to the geometry as shown in the figure below:



Material properties assigned to the laminated shell. The yellow region of the shell has assigned laminate 1 material properties while the green region has assigned laminate 2 properties.

10.4. Distributed loads

The shell under analysis is subjected only to its own self-weight. Self-weight condition can be simply applied in RamSeries using the following option that must be applied to the entire geometry.

Loadcases ► Loadcase 1 ► Shells ► Self weight load

10.5. Mesh generation

Shells can be meshed in RamSeries using triangular and quadrilateral elements. In both cases linear and quadratic element formulations are also available. In this case, the shell is meshed using linear triangles with a maximum element size $h = 0.05$. It results in a mesh with 748 triangular elements and 415 nodes.

10.6. PostProcess

The part of the program dedicated to visualize the results of the analysis is called Post-process. Once the calculation is finished, load the post-process by using the following option:

Postprocess ► Start

To perform the operation correctly, it is necessary to have the model loaded inside the pre-processing part before going to post-process. Choose the view style *Body bound* if it is not still selected. With this type of view, contour fill results on shells are visualized better.

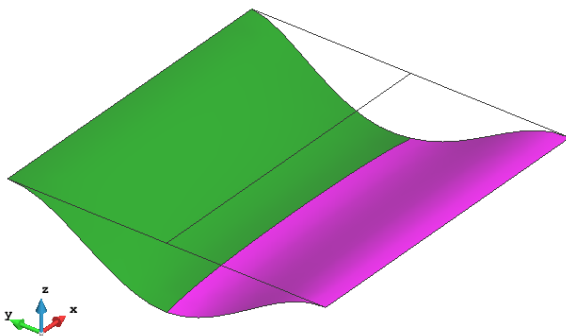
10.6.1. Deformation of the structure

To visualize the deformed mesh of the structure check the following options within the preferences panel of the post-process data tree:

Postprocess panel ► Preferences ► Deformed ► Draw:[Yes]

Postprocess panel ► Preferences ► Deformed ► Factor:[1.0e3]

Postprocess panel ► Preferences ► Deformed ► Result:["Static Displacements"]

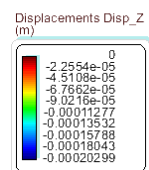
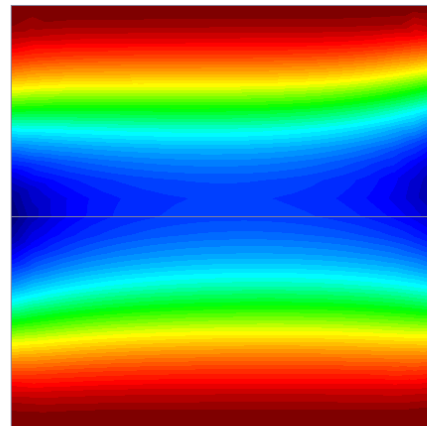


The deformation of the structure is drawn magnified in the screen. The magnifying factor is automatically calculated if "Adimensional factor" option is selected. To change the factor, write a new factor and unselect "Adimensional factor" checkbox.

10.6.2. Contour fill of the displacements

Another way to see the magnitude of the deformation of the shell is to use the displacements contour fill result.

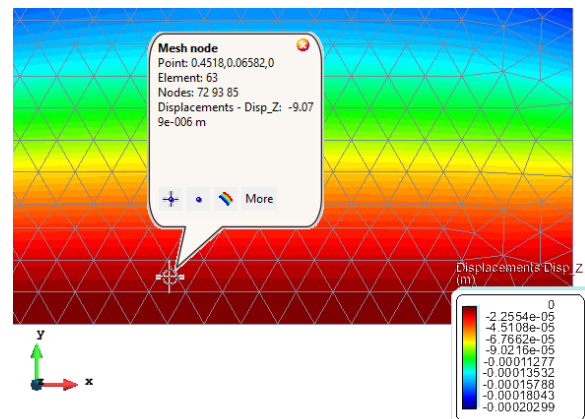
Postprocess panel ► Results ► Static ► Displacements ► Disp z ► Contour fill



To see the numerical values of the deformation at any point, node or element use the following option:

Postprocess ► Mesh information

Try also pressing the mouse left button over any point in the screen. Make the appropriate zooms to visualize properly the values.



To switch off the labels press:

Mouse right button ► Label ► Off

In order to choose the visualization style for the shell, try:

Postprocess panel ► Meshes ► Display style

10.6.3. Visualization of strengths

All of the strengths are represented related to the local axes systems of every shell (except the strengths in main axes). The visualization will be generally with a Contour Fill of every component of the strength although there are other ways to visualize them. To see the numerical value of the strength that is being visualized, choose:

Mouse right button ► Label ► All

To hide the numerical results select:

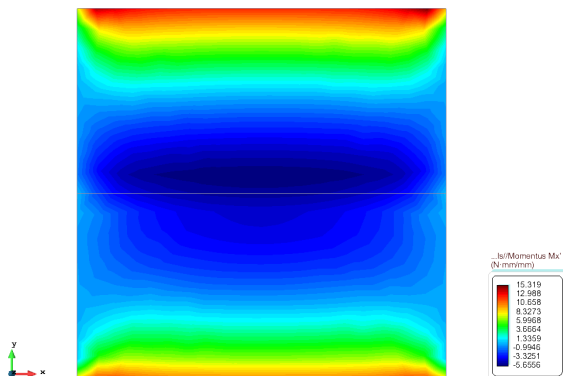
Mouse right button ► Label ► Off

Both commands can be found in the menu that appears when pressing the secondary mouse button over the screen. The strengths are the following:

Name		Default units	Remarks
Axial force	N_x, N_y, N_{xy}	N/m	Axial force in all the shell thickness per unit width of the shell
Momentum	M_x, M_y, M_{xy}	N·m/m	Momentum per unit width of the shell
Shear	Q_x, Q_y	N/m	Shear strength per unit width of the shell

Note: See that M_x momentum is contained inside plane $X'Z'$. This criteria is different from the strength criteria in beams or in constraints, where the rotation in X' is the one that rotates around X' axe. The rotation axe criteria for shell momentum are inherited from the plate criteria. In that case, as there are only two rotations in a plate, this choosing looked appropriate. For rotations in 3D, as there are 3, it is necessary to use the other criteria. As an example, the M_x momentum on the shell is shown in the figure below:

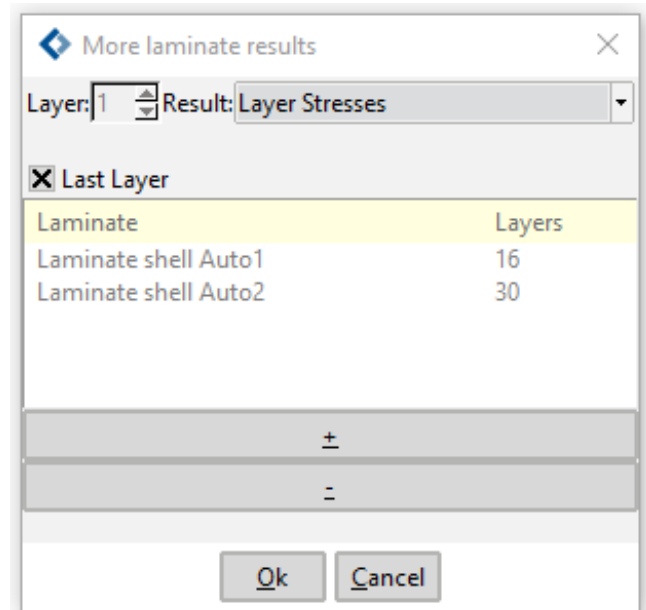
Postprocess panel ► Results ► Static ► Shells ► Momentum ► M_x ► Contour fill



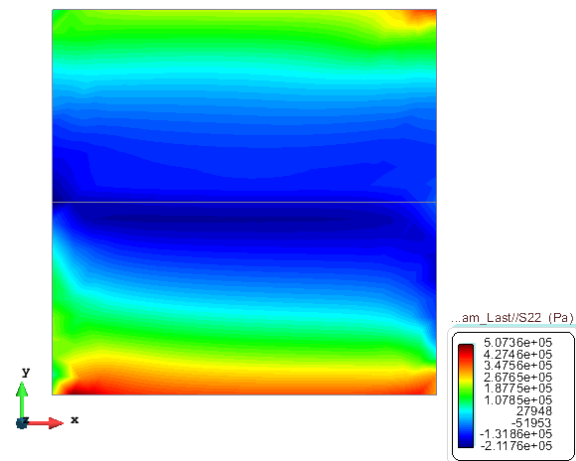
10.6.4. Visualization of Security factors

If the user needs to visualize the different results (stress, strain, shear strain, shear stress or security coefs.) among each individual layer of the laminate, a specific tool exists in the post-process of RamSeries:

Postprocess ► Laminates



In the displayed window, choose the desired layer and result to be viewed. It is also possible to visualize the results for the last layer of all the existing laminates in the analysis, for all laminates need not have the same number of total layers. So, if an user selects the last layer of one of the laminates and one or various of the others have less layers, only the results for this selected laminate will be displayed. Using the option "Last layer", results for the last layer of each of the laminates will be displayed. In addition, the user could see the results for the rest of layers counting down from the last of every laminate, by means of clicking on the '+' or the '-' button.



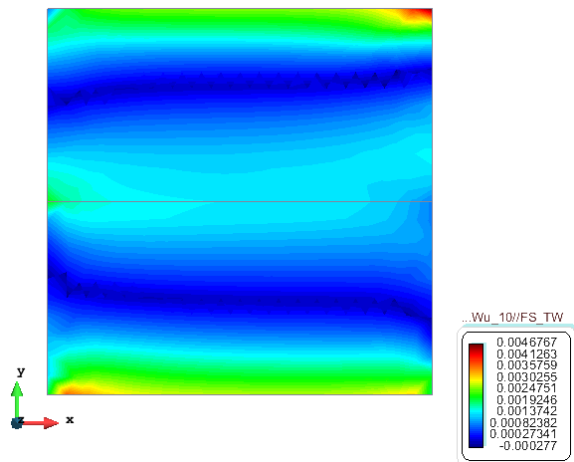
The security factors, (Tsai-Wu or LaRC04 criteria) can be visualized for the global laminate or for each layer separately, providing maximum stresses values have been inserted in the material definition.

Also, it is necessary to have choosed to output this security factors results, and select which one of them is to be actually output.

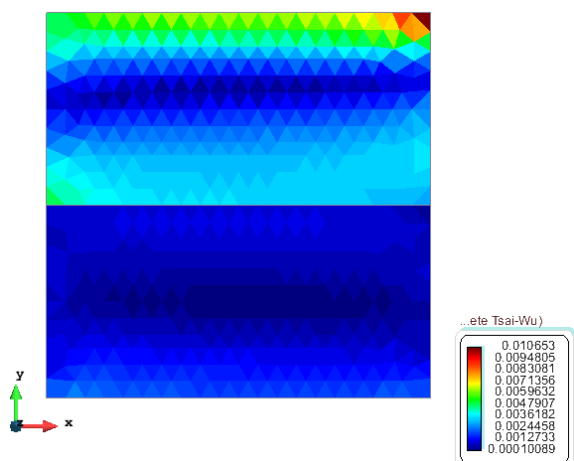
• Tsai-Wu criteria:

Security factor following complete Tsai-Wu criteria can be visualized, either for every single layer using the specific laminates tool:

Postprocess ► Laminates



Or for the complete layer:



♦ LaRC04 criteria (NASA):

Security factors following LaRC04 criteria can be also visualized (Ref. [10]). This criteria includes six different failure modes for matrix and reinforcement.

Visualization of security factors can also be done for the full composite, or for each layer separately.

11. Tutorial 10 - Non-Linear Analysis with SeaFEM Loads

This tutorial is a simple analysis produced in order to show RamSeries Non-Linear (geometric) dynamic capabilities, combined with wave pressure loads coming from previously performed SeaFEM simulations.

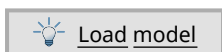
For running this analysis, it is necessary to have previously run the correspondent SeaFEM analysis, which results will be the loads (wave pressure loads) for RamSeries model. Though some description of SeaFEM model will be showed in this tutorial, please refer to the section "Application example" in SeaFEM reference manual (*Help>Help Seakeeping* or <http://www.compassis.com/downloads/Manuals/SeaFEMManual.pdf>) for details on the set-up of the simulation.

The geometries corresponding to this tutorial can be found here (IGES format):

[RamSeries-Tutorials10-SeaFEM_loads_nonlinear_Ram.igs](#)

[RamSeries-Tutorials10-SeaFEM_loads_nonlinear_Sea.igs](#)

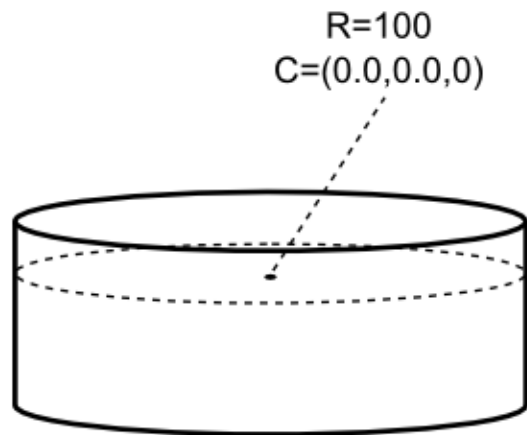
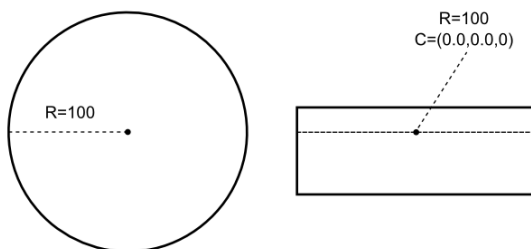
Alternatively, the model ready to be meshed and run, can be automatically loaded into the GUI by left-clicking on the button below.



The model has a simple geometry, consisting on a cylinder, with the following dimensions:

$$R_{cyl} = 1.0 \text{ m}$$

$$h_{cyl} = 0.7 \text{ m}$$



11.1. Analysis Data

In order to be able to perform this analysis, the user must select the following simulation options:

General data ► Analysis ► Element types: Shells

General data ► Analysis ► Analysis type: Dynamic analysis

General data ► Analysis ► Geometric constitutive model: Non-Linear geometry

The following dynamic analysis parameters are chosen:

General data ► Dynamic analysis data ► General

Type	Direct integration
Δt	0.1 s
Number of steps	600

General data ► Dynamic analysis data ► Integration data

Integration method	Implicit (Energy Conserving/Decaying)
α_{E-C/D}	0.0

General data ► Dynamic analysis data ► Damping data

Damping type	Rayleigh
α_K	0.0
α_M	0.4415

The damping coefficient has been calculated as the **5%** of the **critical damping**, as showed next:

$$\xi = C/C_c = 1$$

with:

$$C = M \cdot \alpha'_M$$

$$C_c = 2 \cdot M \cdot \omega_c$$

So:

$$\alpha'_M = 2 \cdot \omega_c$$

The frequency is calculated as:

$$\omega_c = \sqrt{(K_{33}/M)} = 4.415 \text{ s}^{-1}$$

with:

$$K_{33} = \rho \cdot g \cdot A_{\text{float}} = 31557.3 \text{ N/m}$$

M: Total mass of the floater

$$\text{In this case: } \alpha_M = 0.05 \cdot (2 \cdot \omega_c) = 0.05 \cdot \alpha'_M = 0.04415$$

11.2. Constraints

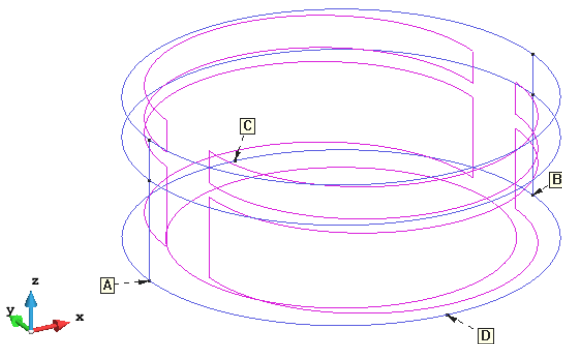
Constraints ► Fixed constraints

The model has its displacement restrained in the direction transversal to the advancing direction of the wave system (Y axis). This condition is applied to the points $A(0.0,-1.0,-0.5)$ and $B(0.0,1.0,-0.5)$, together with a restriction in the X axis rotation (to prevent the model from having a roll movement, and make it coherent with SeaFEM model).

Activation		Values	
X constraint	NO	X value	0.0
Y constraint	YES	Y value	0.0
Z constraint	NO	Z value	0.0
θ_x constraint	YES	θ_x value	0.0
θ_y constraint	NO	θ_y value	0.0
θ_z constraint	NO	θ_z value	0.0

Also points $C(-1.0,0.0,-0.5)$ and $D(1.0,0.0,-0.5)$ have restrained the displacement in the Y axis direction, in order to prevent the rotation in the Z axis (yaw movement), and be coherent with the SeaFEM model.

Activation		Values	
X constraint	NO	X value	0.0
Y constraint	YES	Y value	0.0
Z constraint	NO	Z value	0.0
θ_x constraint	NO	θ_x value	0.0
θ_y constraint	NO	θ_y value	0.0
θ_z constraint	NO	θ_z value	0.0



Constraints ► Elastic constraints

Also, an **elastic constraint** has been applied at the point $A(-1.0,0.0,-0.5)$. The **stiffness value** is **K=100 N/m**. This condition is also applied in the SeaFEM model, to make them equivalent.

Activation		Values	
X constraint	YES	X value	100.0 N/m
Y constraint	NO	Y value	0.0
Z constraint	NO	Z value	0.0
θ_x constraint	NO	θ_x value	0.0
θ_y constraint	NO	θ_y value	0.0
θ_z constraint	NO	θ_z value	0.0

11.3. Material properties

The shells are assigned with a material of the following properties (the material is assumed to be isotropic and linear elastic). To assign the section and material properties of an isotropic the shell, choose:

Materials and properties ► Shells ► Isotropic shell

-Thickness= 0.1 m

-E= 2.1e11 N/m² (Young modulus)

-ν= 0.3 (Poisson coefficient)

-Specific Weight= 20821 N/m³

Local axes system **Default** is chosen. Following the criteria given before, X' axe will point towards positive global Y axe. Z' axe will be normal to the elements and Y' will be contained in the plane XZ.

Once the values are filled in, the condition must be assigned to the surface that defines the shell.

The conditions that have been assigned can be viewed right-clicking on the created group:

Draw ► Draw values / Draw groups / Draw symbols

The hidrostatic equilibrium is previously stablished, by checking de displacement calculated in **SeaFEM**, and the reactions in **RamSeries** due to the self-weight load.

Therefore, the specific weight (ρ) has been adjusted so the cylinder has its floating line in $z = 0$ (this means a depth of $d=0.5$ m).

Displacement (SeaFEM): $\Delta_{SF} = 15689.31$ N

Cylinder area: $S_{cyl} = 7.535$ m²

$\rho_{cyl} = \Delta_{SF} / (S_{cyl} \cdot t_{cyl}) = 20821$ N/m³

Reactions (RamSeries, self-weight): $R_{sw} = 15643.3$ N

Reactions (RamSeries, hidrostatic): $R_{hyd} = 15689.3$ N

**Note:* An slight difference between weight and lift exists and it is assumed (0.3%).

11.4. Loads

The model is loaded with the **dynamic pressures** coming from the SeaFEM analysis results:

Loads ► Loadcase 1 ► Shells ► Seakeeping wave load

The file with the corresponding SeaFEM model results must be selected.

This load is applied in the same sense and direction of the normal to the surface, so it must be checked that the normals points towards the inside of the cylinder. Seakeeping Wave Loads are applied to the surfaces below the floating line (**Z = 0**).

The previously described load reads and applies only the dynamic part of the wave pressure coming from SeaFEM result files. Therefore the **hydrostatic pressure load** is needed to be applied:

Loadcases ► Loadcase 1 ► Shells ► Pressure load

Factor	Click Function icon to insert the following info:
	Function on geometry (Hydrostatic load)
	Reference coordinate = 0.0
	Water specific weight =- 10045.0 N/m ³

Note that for the water specific weight, the same density as in SeaFEM has been applied (density corresponding to sea water, 1025 kg/m³). This load is applied to the same surfaces as the Seakeeping load, and **also to the surfaces above the water line**, to account for the hydrostatic pressure acting on the geometry when it sinks below Z=0. Also note that a negative signed has been included in the specific weight due to the fact that hydrostatic loads in RamSeries Non-linear geometry analysis are applied in the opposite sense to the normal to the surface.

The load corresponding to the **self-weight** of the structure is also applied, to all the model surfaces, so the forces are equilibrated.

Loads ► Loadcase 1 ► Shells ► Self weight load

11.5. SeaFEM model

The geometry of the model for SeaFEM is the same, but for the part of the cylinder over the free surface ($Z=0$), which is neglected. So:

$$R_{cyl} = 1.0 \text{ m}$$

$$h_{cyl} = 0.5 \text{ m}$$

- Enviroment data.

The simulation is performed for a monochromatic wave system in the X axis direction (0.0 deg), with the following parameters:

$$A_{wave} = 0.1 \text{ m}$$

$$T_{wave} = 6.0 \text{ s}$$

- Body data:

The body properties (center of gravity and radii of gyration) have been set after obtaining them with the tools available in RamSeries, whose output is shown in what follows:

Calculating Center of Gravity and Inertia Momentum....

(*Note: Results are given in model length units [Geometry units])

Center of Gravity [L]

$(xG,yG,zG) = (-1.83949e-007,-2.6283e-011,-0.295692) [L]$

Total weight = 15680 [N]

Moments of Inercia respect to ortogonal axis passing through the Center of Gravity

$(I_{xg},I_{yg},I_{zg}) = (7112.24,7110.87,12476.4) [F*L^2]$

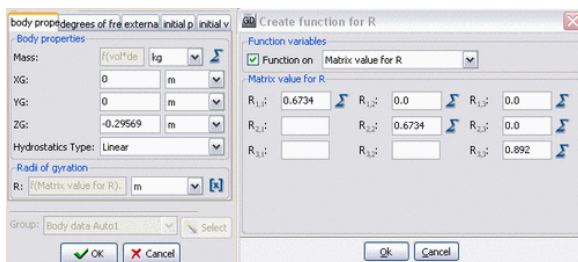
Radii of gyration respect to ortogonal axis passing through the Center of Gravity

$(r_{xg},r_{yg},r_{zg}) = (0.673488,0.673423,0.892014) [F*L^2]$

Moments of Inercia respect to ortogonal axis passing through the origin)

$(I_{xo},I_{yo},I_{zo}) = (8483.2,8481.84,12476.4) [F*L^2]$

Total time = 0.018 min. = 1.1 seg.



The object has its center of gravity set to **CDG=(0.0,0.0,-0.295)**.

Only heave, surge and pitch degrees of movement are left free.

External forces have been added:

Elastic spring (same as in RamSeries):

$$F_x = -100 \cdot dx[-1.0,0.0,-0.5]$$

2. Compensating pitch moment:

This moment is applied so it compensates the moment which will appear in RamSeries due to the elastic restriction.

$$M_y = -100 \cdot dx[-1.0,0.0,-0.5] \cdot (0.5-0.295)$$

- Irregular frequency removal = **0.05**.

A **5%** of the critical damping factor has been considered here, in order to make the analysis equivalent to the RamSeries one.

11.6. Mesh

An unstructured mesh of linear triangles has been used.

Mesh ► Generate mesh ► [Maximum element size]=0.2

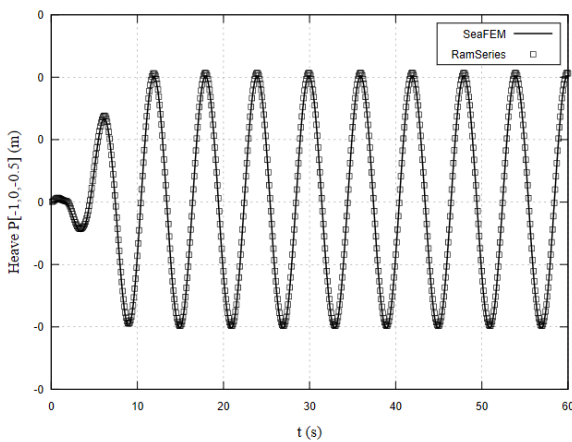
11.7. Results

The main goal of this tutorial analysis is to validate that the dynamics of RamSeries follows the dynamic imposed by the pressure results obtained in SeaFEM.

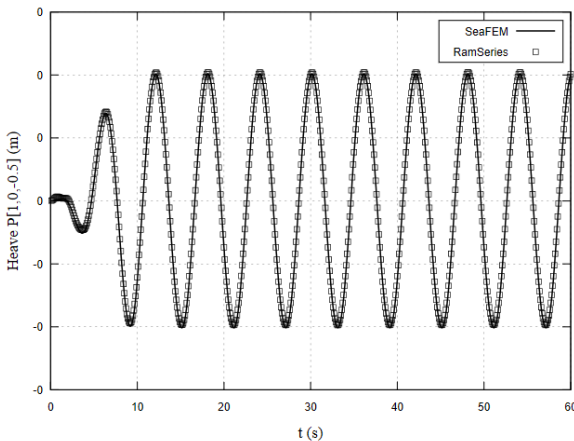
The following images show the comparison of both codes solution for the main movements: *heave*, *surge* and *pitch*. Pitch movement is compared verifying that two points, at bow and stern ([1,0,-0.5] and [-1,0,-0.5]), have the same heave movement for both codes solution.

The coincidence of both signals grant that the pitch motion (rotation around the Y axis) is also coincident.

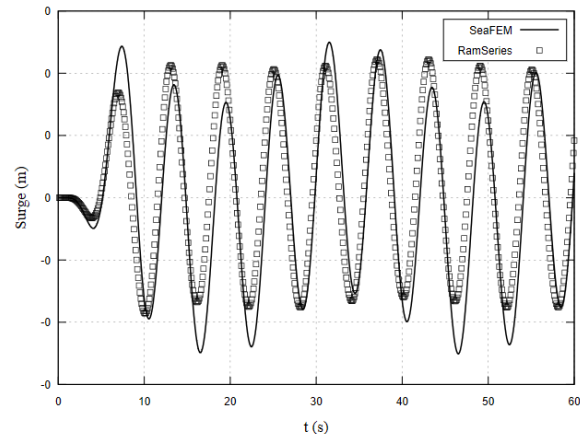
• Heave movement of a point located at stern:



• Heave movement of a point located at bow:



• Surge movement of center of gravity (0,0,-0.296):



For this movement, an small offset appears in the amplitude of the signals. This is produced by the initial transitory which is not exactly the same in both simulations.

Nevertheless, it is very important to remark that both signals follow the two main movements periods:

- $T_w = 5$ s (wave period)
- $T_k = 25.33$ s (spring natural period)

The spring period is calculated as follows (it includes the critical damping percentage inserted in RamSeries model):

$$\omega_K = \sqrt{(K/M) - (\xi/2)^2}$$

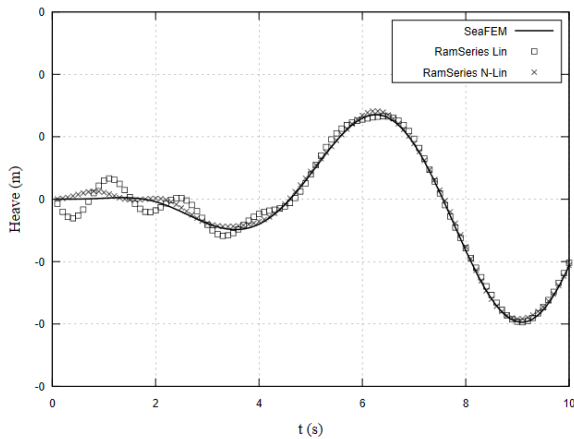
$$T_K = 1/\omega_K$$

Being M the total floater mass, $K=100$ N/m the spring constant, and $\xi = 0.05$ (5%) the percentage of the critical damping considered.

11.7.1. Linear results

A linear analysis has also been performed, in order to compare results. If small movements are expected, it may be quite convenient and advisable to perform a linear analysis instead of the non-linear, for it runs faster.

The obtained signal for heave is very similar, but for the initial transitory (showed in the following image):



12.1. Analysis data

General data ► Analysis ► Element types: Solids

General data ► Analysis ► Analysis type: Incremental loads analysis

General data ► Analysis ► Material constitutive model: Plasticity on materials

The following dynamic analysis parameters are chosen:

General data ► Non-linear analysis data ► General

Solver control	Load control
Conv. tolerance	1.0e-4
Iteration type	Full Newton-Raphson
Max iterations	40

General data ► Incremental analysis data

Num increments 10

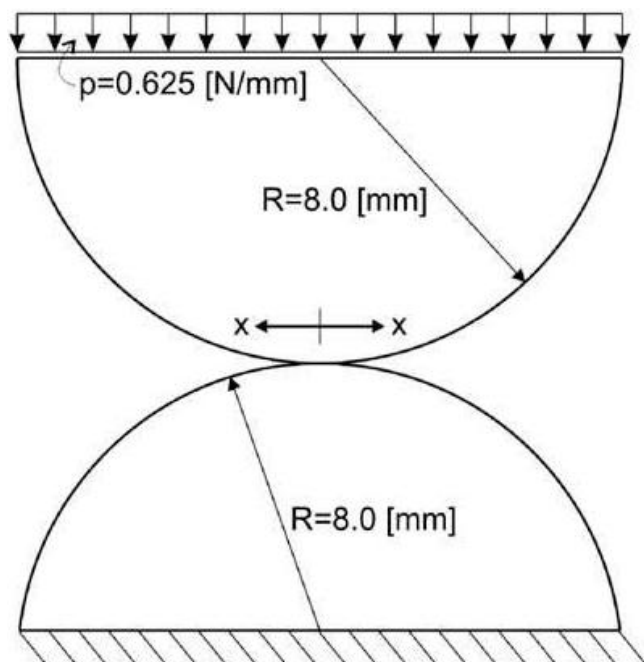
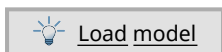
12. Tutorial 11 - Contacts analysis

The example is used to describe the contact analysis will be a cylinder on cylinder Hertzian contact problem.

The geometry corresponding to this tutorial can be found here (IGES format):

[RamSeries-Tutorials11-solids_contact.igs](#)

Alternatively, the model ready to be meshed and run, can be automatically loaded into the GUI by left-clicking on the button below.



12.2. Constraints

Constraints ► Fixed constraints

The bottom surface has a prescription in its movements in the three global directions X, Y and Z.

Activation		Values	
X constraint	YES	X value	0.0
Y constraint	YES	Y value	0.0
Z constraint	YES	Z value	0.0

For applying symmetry to the model, the following constraints must be defined. Slave nodes cannot have fix constraints, so elastic constraints must be used instead of fix constraints. If stiffness of elastic constraints is thousand times greater than materials' stiffness, elastic constraint works better than a fix constraint:

Constraints ► Elastic constraints

Constraint in plane X=0:

Activation		Values	
X constraint	YES	X value	2.0e5 N/mm ³
Y constraint	NO	Y value	0.0
Z constraint	NO	Z value	0.0

Constraint in planes Z=0 and Z=2:

Activation		Values	
X constraint	NO	X value	0.0
Y constraint	NO	Y value	0.0
Z constraint	YES	Z value	2.0e5 N/m ³

Constraint in lines Z=0/X=0 and Z=2/X=0:

Activation		Values	
X constraint	YES	X value	2.0e5 N/m ²
Y constraint	NO	Y value	0.0
Z constraint	YES	Z value	2.0e5 N/m ²

12.3. Contacts definition

For defining contacts, it is necessary to select which surfaces are master and which are slave. Type of contact must also be chosen.

Contacts ► Contacts structural ► Master-Slave Surf-Surf

It is recommended to rename master and slave groups to make easy to identify them in **Groups window**.

Data ► Groups

There is not a fix rule about which surface must be **master** and which must be **slave**. However, it is advisable to choose the more rigid as master surface. On the other hand, stresses in slave surface will be more accurate than in master slave, so, it is a good option to choose as slave the surface where a better accuracy is requested.

12.4. Material properties and loads definition

The solids are assigned with a material of the following properties (the material is assumed to be isotropic and linear elastic). To assign the section and material properties of an isotropic solid, choose:

Materials and properties ► Solids ► Isotropic solid

-Thickness= 0.1 m

-E= 200 N/mm² (Young modulus)

-ν= 0.3 (Poisson coefficient)

-Specific Weight= 0.0 N/mm³

Once the values are filled in, the condition must be assigned to the volumes which defines the solid.

The two volumes corresponding to the master and the slave solids are each one assigned with a different material group, though both have the same properties.

The conditions that have been assigned can be viewed right-clicking on the created group:

Draw ► Draw values / Draw groups / Draw symbols

12.5. Mesh generation

For improving stress accuracy in contacts elements, it is recommended to use more elements in slave surface than in master surface. Doing so, every master element will have more than one slave node in contact.

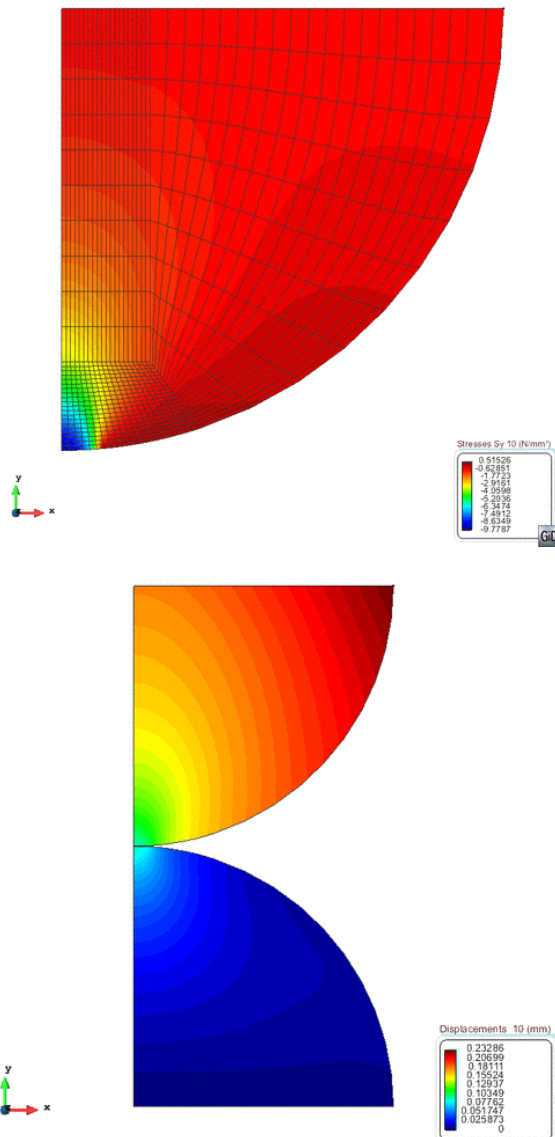
In this example, a structured mesh is used and master and slave mesh are equal, so stress results will be very accurate.

Use the following Gid options to get a structural mesh:

Mesh ► Structured ► Volumes ► Assign number of cells

12.6. Results

The following pictures show results for the defined analysis:



13. Tutorial 12 - Full Body Coupling with SeaFEM

This example is used for describing the coupled analysis with **SeaFEM** (seakeeping analyses) and **RamSeries** (structural analyses). This coupling can be of two types:

- 3.. **HFS** (Height Free Surface), which is performed via an special free surface pressure condition.
- 4.. **Body**, this is the case showed in this tutorial. The coupling is performed using the mesh of the floating body.

The geometry corresponding to this tutorial can be found here (IGES format):

[RamSeries-Tutorials12-coupling_SeaFEM_RamSeries.igs](#)

Alternatively, the model ready to be meshed and run, can be automatically loaded into the GUI by left-clicking on the button below.



13.1. Loading the coupled analysis

To load the RamSeries type of analysis, choose:

Simulation Type ► Coupled Seakeeping-Structural Analysis

For this kind of problem, the analysis can be set up from the Start Data window, as shown in the Beams tutorial, with the following options in this case:

Data ► Start data

- Simulation dimension: 3D
- Structural analysis: Shells
- Analysis type: Dynamic: Direct integration analysis
- Material constitutive model: Linear materials
- Geometrical constitutive model: Non-Linear geometry
- Analysis domain: (Time) First order diffraction radiation
- Environment: Waves
- Type of analysis: Seakeeping

Also from the data tree in the GUI:

Simulation data ► Simulation type

13.2. Geometry generation

The first step is to create the geometry. It can be defined using the pre-processing commands described in the GiD manual. It is important to take in consideration that both geometries for RamSeries and SeaFEM models must be generated. Some parts of the geometry will need to be duplicated in the coincident zones, therefore, it is more than advisable to have a good layer organization for the geometry.

So, the first step is to open the layers window and create the layers to be used:

Utilities ► Layers

RamSeries geometry creation

In this case, the geometry for RamSeries is very simple: it consists in a rectangular barge of 20 m length, 5 m beam and 3.0 m height.

5. Open the layers window and create the layer called **RamSeries>Hull**:

6. Create the base rectangle:

Geometry ► Object ► Rectangle ► Corner 1: (-10,-2.5,-1.0) ► Corner 2: (10,2.5,-1.0)

7. Extrude de contour lines of the rectangle to create the barge:

First up to the free surface:

Utilities ► Copy, Lines ► Traslation, Extrude: Surfaces ► P1: (10,2.5,-1.0), P2: (10,2.5,0.0)

and finally up to complete the total height of the barge:

Utilities ► Copy, Lines ► Traslation, Extrude: Surfaces ► P1: (10,2.5,0.0), P2: (10,2.5,2.0)

SeaFEM geometry creation

The dimensions are the following:

Body (wet part of the barge):

- $L_b = 10$ m
- $B_b = 5$ m
- $d_b = 1.0$ m

Domain cylinder:

For the cases when no currents are imposed, like this, the most indicated shape for computational domain is a cylinder.

Computational depth should be no larger than physical depth. Moreover, computational depth might be smaller than physical and/or recommended if the bottom boundary condition is used. The dimensions for the domain in this case are:

- $H_d = 20$ m
- $R_d = 50$ m

Analysis area cylinder:

A zone with more mesh definition is needed around the floating object, in order to correctly capture the scattered waves (waves radiated and diffracted by the floating object). Moreover, as explained before, the border of this area will indicate the start of the damping/absorption area (beach). The absorption area should be at least as long as the maximum wave length. Recommended length is twice the maximum wave length. Nevertheless, if monochromatic wave is used along with Sommerfeld radiation condition, the absorption area might be reduced to half the wave length.

Therefore, the dimension chosen is:

- $H_{aa} = 10$ m
- $R_{aa} = 20$ m

All geometries for computational domains in SeaFEM must be located in $Z \leq 0$. In this case it is also centered in the origin (0,0,0). A separated layer can be created for the body, some more for the free surface, analysis area, external contours of the domain, and finally, another for the domain volumes.

In the following steps, one of the several ways for generating the geometry will be explained in detail:

1. Open the layers window and create the layer called **SeaFEM>cyl_ext**:

Utilities ► Layers

Right click on the layers window + *New layer*

Create a circle for the base:

Geometry ► Create ► Object ► Circle

Enter center: 0 0 -20

Enter normal: Positive Z

Enter radius: 50.0

2. Create a new layer called **SeaFEM>cyl_int**.

Create another circle for the base of the analysis area

Enter center: 0 0 -10

Enter normal: Positive Z

Enter radius: 20.0

3. Create a new layer called **SeaFEM>Body_hull**.

Create a rectangle for the base of the body.

Geometry ► Object ► Rectangle ► Corner 1: (-10,-2.5,-1.0) ► Corner 2: (10,2.5,-1.0)

Extrude de contour lines of the rectangle to create the barge:

Utilities ► Copy, Lines ► Traslation, Extrude: Surfaces ► P1: (10,2.5,-1.0), P2: (10,2.5,0.0)

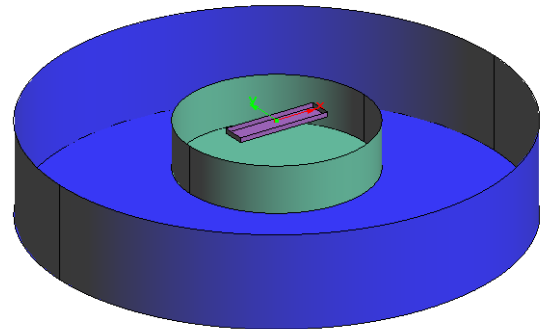
4. Copy lines in **SeaFEM>cyl_ext** creating the surfaces by extrusion in the same operation:

Utilities ► Copy, Lines ► Traslation, Extrude: Surface ► P1: (50,0,-20), P2: (50,0,0)

5. Repeat the operation for **SeaFEM>cyl_int**:

Utilities ► Copy, Lines ► Traslation, Extrude: Surface ► P1: (20,0,-10), P2: (20,0,0)

Up to this point, the geometry should look as in the following image:



6. Create a new layer called **SeaFEM>free_surface_ext**

Create a surface using the circumferencial contour with radius 50 m, at z=0:

Geometry ► Create ► NURBS surface ► By contour

7. Create a hole in the previously created surface:

Geometry ► Edit ► Hole NURBS surface

Enter surface: Select surface at to z=0 created in the previous step.

Enter lines to define the hole: Select lines of the analysis area at z=0.

8. Create a new layer called **SeaFEM>free_surface_int**

Create a surface using the circumferencial contour with radius 20 m, at z=0:

Geometry ► Create ► NURBS surface ► By contour

9. Create a hole in the previously created surface:

Geometry ► Edit ► Hole NURBS surface

Enter surface: Select surface at to z=0 created in the previous step.

Enter lines to define the hole: Select lines of the barge at z=0.

8. Create a new layer called **SeaFEM>vol_int**

Create a new volume for the inner part of the domain:

Geometry ► Create ► Volume ► By contour

Select the surfaces in **SeaFEM>cyl_int**, **SeaFEM>free_surface_int** and **SeaFEM>body_hull**.

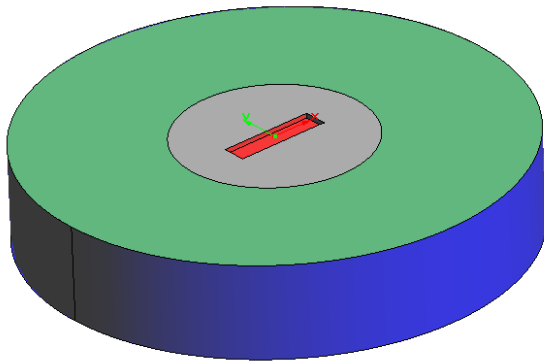
9. Create a new layer called **SeaFEM>vol_ext**

Create a new volume for the outer part of the domain:

Geometry ► Create ► Volume ► By contour

Select the surfaces in *SeaFEM>cyl_ext* and
SeaFEM>free_surface_ext and
SeaFEM>cyl_int.

The final geometry should look as in the next image:



13.3. Coupling data

First of all, for performing this analysis the correct simulation type must be selected:

Simulation data ► Simulation type: Coupled Seakeeping-Structural analysis

To edit the data regarding to the coupling of both structural and seakeeping analysis, please go to the Data tree corresponding area:

Simulation data ► Coupling data

Δt	0.1
Number of steps	200
Max FSI iterations	20
FSI tolerance	1.0e-2
FSI relaxation factor	0.25

13.4. Seakeeping data

The following pages explain how to set the data corresponding to the seakeeping part of the analysis.

13.4.1. General data

The seakeeping analysis general data can be edited at:

Seakeeping data ► General data

The seakeeping problem setup is performed editing the following fields:

Seakeeping data ► General data ► Problem setup ► Analysis type ► Time domain type: 1st order diffraction radiation

Seakeeping data ► General data ► Problem setup ► Environment: Waves

Seakeeping data ► General data ► Problem setup ► Type of analysis: Seakeeping

13.4.2. Problem description

Here, the data concerning domain depth and waves absorption is defined:

Seakeeping data ► Problem description

Bathymetry:	Infinite depth
Wave absorption	Yes
X absorption reference	0 m
Y absorption reference	0 m
Absorption factor	1.0
Beach	20 m
Sommerfeld radiation condition	Yes

13.4.3. Environment data

Here, the data concerning waves environment is set and defined:

Seakeeping data ► Environment data ► Waves

Wave spectrum	Monochromatic wave
Amplitude	0.5 m
Period	3.58 s
Direction	0.0 deg

13.4.4. Time data

The simulation time data is to be set in this section.

Seakeeping data ► Time data

As the time step is already defined in the coupling data, the only data to be set is the following:

Output step	0.5 s
Start time recording	0.0 s
Initialization time	5 s

13.4.5. Body coupling

This condition is to be assigned to the body, so as to couple it with structural analysis:

Seakeeping data ► Body coupling

13.4.6. Numerical data

The relevant numerical data for this seakeeping problem can be set at:

Seakeeping data ► Numerical data

Solver	Conjugate gradient
Preconditioner	ILU
Free surface stability factor	1.0
Dynamic solver relaxation	0.25
Damping factor	0.05

13.4.7. Boundary conditions

The following boundary conditions are to be applied to the domain

To the top domain boundary surfaces:

Seakeeping data ► Boundary conditions ► Free surface

To the lateral domain boundary surfaces:

Seakeeping data ► Boundary conditions ► Outlet

To the domain volumes:

Seakeeping data ► Boundary conditions ► Fluid volume

13.5. Structural data

The following pages explain how to set the data corresponding to the structural part of the analysis.

13.5.1. General data

The structural analysis general data can be edited at:

Structural data ► General data

As a summary, the relevant data to be correctly set is the following:

Structural data ► General data ► Analysis

Element types	Shells
Analysis type	Dynamic Analysis
Material constitutive model	Linear materials
Geometric constitutive model	Non-linear geometry
Boundary conditions	Linear boundary conds

The structural non-linear analysis data can be edited at:

Structural data ► General data ► Non-Linear analysis data

In this case, the default data is used.

The structural dynamic analysis data can be edited at:

Structural data ► General data ► Dynamic analysis data ► Integration data

Integration method	Implicit (Energy conserving/decaying)
$\alpha_{E-C/D}$	0.0
Initial conditions	None

Structural data ► General data ► Dynamic analysis data ► Damping data

Damping type	Rayleigh damping
α_M	0.31315
α_K	0.0

Regarding the chosen damping coefficient (α_M), it is obtained as follows, in order to make it consistent with the damping used in SeaFEM. The damping coefficient has been calculated as the **5%** of the **critical damping**, as showed next:

$$\xi = C/C_c = 1$$

with:

$$C = M \cdot \alpha'_M$$

$$C_c = 2 \cdot M \cdot \omega_c$$

So:

$$\alpha'_M = 2 \cdot \omega_c$$

The frequency is calculated as:

$$\omega_c = \sqrt{(K_{33}/M)} = 3.13155 \text{ s}^{-1}$$

with:

$$K_{33} = \rho \cdot g \cdot A_{\text{float}} = 1.00518 \times 10^6 \text{ N/m}$$

M: Total mass of the floater

In this case: $\alpha_M = 0.05 \cdot (2 \cdot \omega_c) = 0.05 \cdot \alpha'_M = 0.31315$

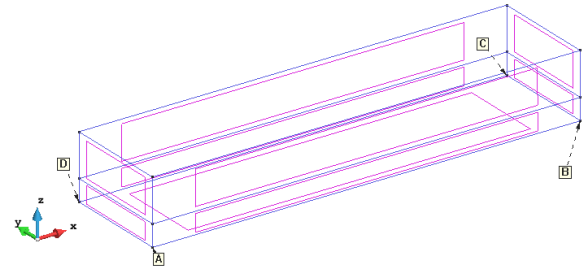
13.5.2. Constraints

The barge has been restrained in order to prevent its lateral movement. Therefore, only δ_y component has been restrained:

Structural data ► Constraints ► Fixed constraints

Activation		Values	
X constraint	NO	X value	0.0
Y constraint	YES	Y value	0.0
Z constraint	NO	Z value	0.0
θ_x constraint	NO	θ_x value	0.0
θ_y constraint	NO	θ_y value	0.0
θ_z constraint	NO	θ_z value	0.0

This constraint has been applied over the four lower corners of the barge (A, B, C, D):



13.5.3. Materials properties

An isotropic steel shell material condition has been applied to the barge geometry.

Structural data ► Materials and properties ► Shells ► Isotropic shell

-Thickness= 0.0523 m

-E= 2.1e11 N/m² (Young modulus)

-ν= 0.3 (Poisson coefficient)

-Specific Weight= 76900 N/m³

This condition must be applied to all the surfaces corresponding to the barge geometry in layer *RamSeries>hull*, as defined in Geometry generation.

The shell thickness has been adjusted in order to adjust the weight of the barge to the actual floating line corresponding displacement:

$$\Delta_V = L \cdot B \cdot d = 20 \cdot 5 \cdot 1 = 100 \text{ m}^3$$

$$\Delta_M = \rho_w \cdot \Delta_V = 1025 \cdot 100 = 102500 \text{ kg}$$

$$\rho_w \cdot g \cdot \Delta_V = \rho_M \cdot S \cdot t$$

$$t = \frac{\rho_w \cdot g \cdot \Delta_V}{\rho_M \cdot S} = \frac{1025 \cdot 9.8 \cdot 100}{76900 \cdot 250} = 0.0523 \text{ m}$$

13.5.4. Loads

In this case, three different loads must be applied.

- Hydrostatic load:

Structural data ► Loadcases ► Loadcase 1 ► Shells ► Pressure load

Factor	Click Function icon to insert the following info:
	Function on geometry (Hydrostatic load)
	Reference coordinate = 0.0
	Water specific weight =10055.3 N/m ³

This load is to be applied to all the hull surfaces.

- Self-weight:

Structural data ► Loadcases ► Loadcase 1 ► Shells ► Self weight load

This load is to be applied to all the hull surfaces.

- Seakeeping load (dynamic pressure coming from SeaFEM)

Structural data ► Loadcases ► Loadcase 1 ► Shells ► Body coupling load

This load is to be applied to all the hull surfaces which are under the water line.

13.6. Mesh generation

The mesh is generated with **linear triangle and tetrahedra** elements.

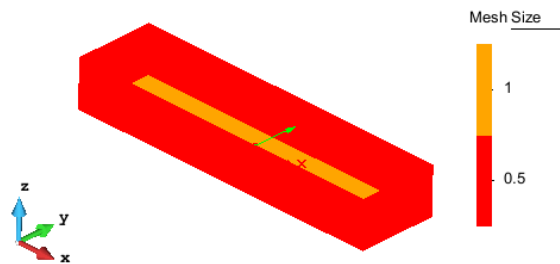
The global mesh has a total of **42612 nodes**, **18978 triangle elements**, and **219846 tetrahedra elements**.

13.6.1. Structural mesh

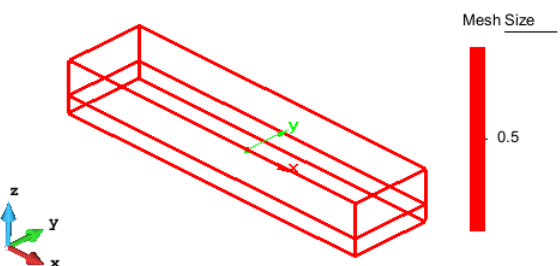
For the structural part of the geometry, the following mesh sizes have been assigned:

Menu ► Mesh ► Unstructured ► Assign sizes on Surfaces/Lines/Points

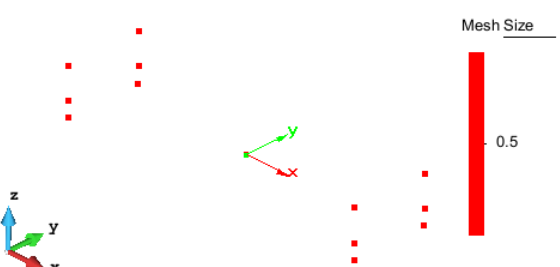
- Sizes on surfaces:



- Sizes on lines:



- Sizes on points:

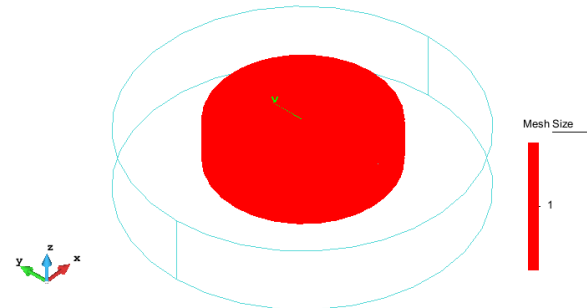


13.6.2. Seakeeping mesh

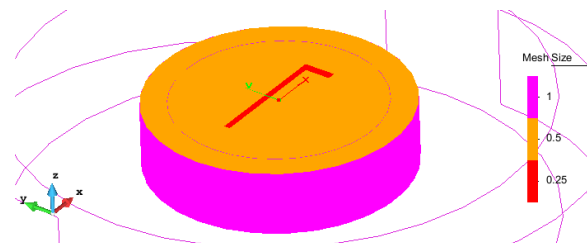
For the seakeeping analysis geometry, the following mesh sizes have been assigned:

Menu ► Mesh ► Unstructured ► Assign sizes on Volumes/Surfaces/Lines/Points

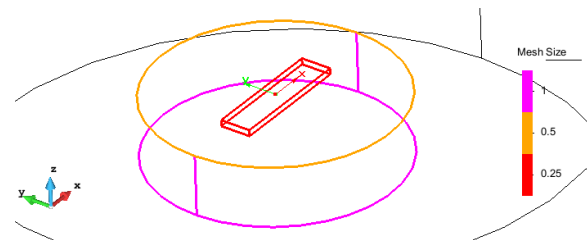
- Sizes on volumes:



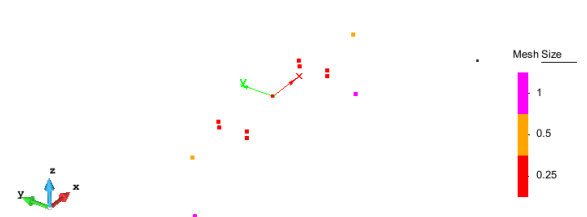
- Sizes on surfaces:



- Sizes on lines:



- Sizes on points:



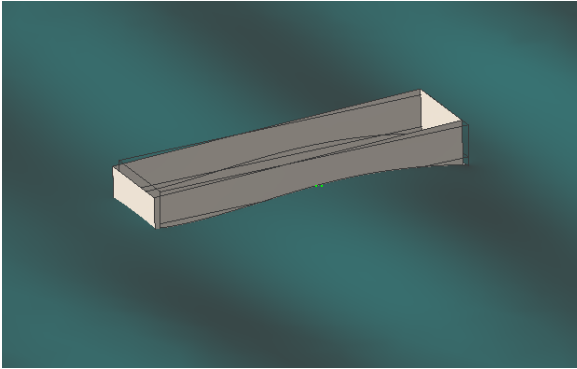
13.7. Postprocess

Once the analysis has finished, or even during the simulation, it is possible to access the postprocess, and visualize results:

Postprocess ► Start

One interesting aspect of this coupled simulation is that results of both analyses can be visualized simultaneously.

For example, it is possible to visualize the free surface deformation, and the structural shells deformation:



To do so, in the Postprocess menu/panel, do the following (after activating the corresponding meshes: free surface and shell structural mesh):

Postprocess panel ► Meshes

Postprocess panel ► Preferences ► Deformed ► Draw:[Yes]

Postprocess panel ► Preferences ► Deformed ► Factor:[1 -1]

Postprocess panel ► Preferences ► Deformed ► Result (multiple):["Static Displacements"] ["Total elevation"]

Note that the sign of the deformation factor for the free surface depends on the orientation of the normal to the free surface. For a correct visualization, if the normal is pointing inwards, the factor shall be negative, and viceversa.

An animation of the results can be created from this deformation view:

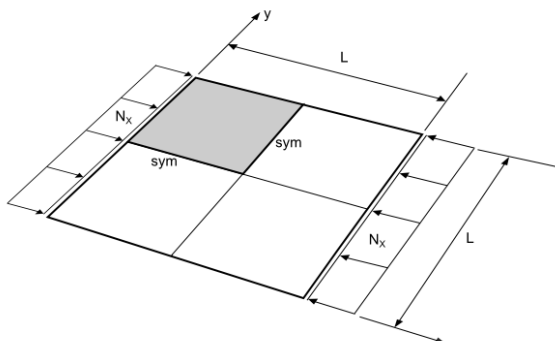
Postprocess ► Animations

Visualization of contour fill of the different results (for structural or seakeeping analyses) can be done, in the usual way.

14. Tutorial 13 - Buckling analysis

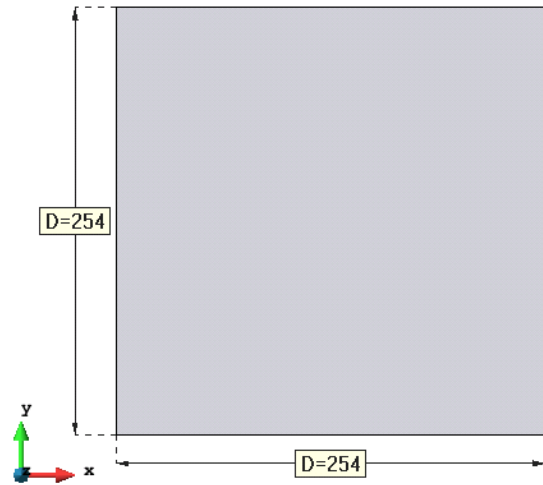
This example performs a buckling analysis of a shell simply supported along all edges with in-plane uniaxial compression.

The next figure shows the geometry of the complete model.



$L=508 \text{ mm}$

Since this model is symmetric, only a quarter of the model will be used to perform the buckling analysis.



The geometry corresponding to this tutorial can be found here (IGES format):

[Ramseries-Tutorials13-buckling_shells.igs](#)

Alternatively, the model ready to be meshed and run, can be automatically loaded into the GUI by left-clicking on the button below.



14.1. Analysis data

It is possible to run an **Incremental Load Analysis** combined with buckling analysis but for this example, an Static analysis is perfectly suitable.

General data ► Analysis ► Analysis type: Static

General data ► Analysis ► Linearized prebuckling analysis: Yes

Some particular parameters must be defined to perform the buckling analysis.

General data ► Buckling data

Num. of buckling modes 3
Imperfections factor 0.0

At least, the number of buckling modes must be one. The imperfections factor represents the amplitude of geometric imperfection related to the first buckling mode. In this example, the shell is considered ideal so a value of 0.0 is chosen.

All dimensions in this example are expressed in mm, so that, it is important not to forget to defined them.

Simulation data ► Units ► Geometry units: [mm]

14.2. Constraints

Since the plate is simply supported along all edges, the edges AD and DC shown in the next figure, must have Z direction fixed.

Structural data ► Constraints ► Fixed constraints

Activation		Values	
X constraint	NO	X value	0.0
Y constraint	NO	Y value	0.0
Z constraint	YES	Z value	0.0
θ_x constraint	NO	θ_x value	0.0
θ_y constraint	NO	θ_y value	0.0
θ_z constraint	NO	θ_z value	0.0

For applying symmetry condition, the following constraints must be added to AB:

Activation		Values	
X constraint	NO	X value	0.0
Y constraint	YES	Y value	0.0
Z constraint	NO	Z value	0.0
θ_x constraint	YES	θ_x value	0.0
θ_y constraint	NO	θ_y value	0.0
θ_z constraint	YES	θ_z value	0.0

To CB:

Activation		Values	
X constraint	YES	X value	0.0
Y constraint	NO	Y value	0.0
Z constraint	NO	Z value	0.0
θ_x constraint	NO	θ_x value	0.0
θ_y constraint	YES	θ_y value	0.0
θ_z constraint	YES	θ_z value	0.0

And to point B:

Activation		Values	
X constraint	YES	X value	0.0
Y constraint	YES	Y value	0.0
Z constraint	NO	Z value	0.0
θ_x constraint	YES	θ_x value	0.0
θ_y constraint	YES	θ_y value	0.0
θ_z constraint	YES	θ_z value	0.0

It is important to apply this point restrain after the previous line restrains have been applied.

14.3. Material properties

An isotropic shell material condition has been applied to the geometry.

Structural data ► Materials and properties ► Shells ► Isotropic shell

-Thickness= 3.175 mm

-E= 2.062e5 N/mm² (Young modulus)

- ν = 0.3 (Poisson coefficient)

-Specific Weight= 0.0 N/m³

14.4. Load

In this example, a compression load equal to the critical force (Nx) is applied to laterall contour line AD. Therefore, the setting **Contour pressure** will be used.

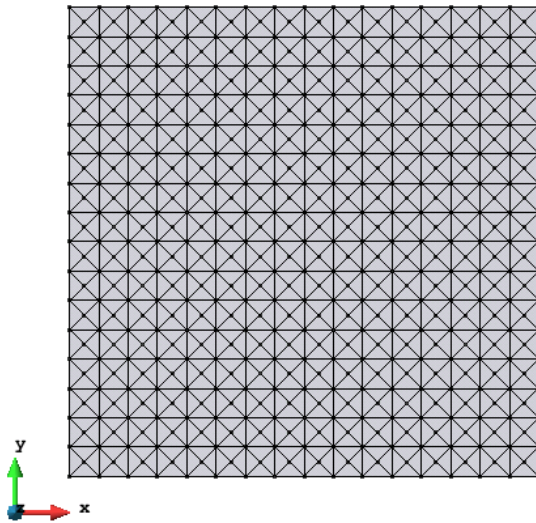
Loadcases ► Loadcase 1 ► Shells ► Boundary pressure load

Factor	1.0
Loadtype	Global
X pressure	92.45 [N/mm]
Y pressure	0.0 [N/m]
Z pressure	0.0 [N/m]

14.5. Mesh

Buckling analysis can only be performed using meshes with 3-node triangles. In this example, a structural mesh with 16 cells for each edge has been used.

Mesh ► Structured ► Surfaces ► Assign number of cells



For obtaining a symmetric mesh like is shown in the previous figure, the following preference option must be set up:

Utilities ► Preferences ► Meshing ► Symmetrical structured: [Triangles] (Yes)

14.6. Results

After the analysis is calculated, the following results will be obtained:

• Critical factor and buckling modes

The critical factor and buckling modes can be read from the process information of the analysis:

Calculate ► View process info

The amount of Eigenvalues calculated are= 3

The first 3 critical buckling load factors are:

Mode Fact_crit

1 1.001

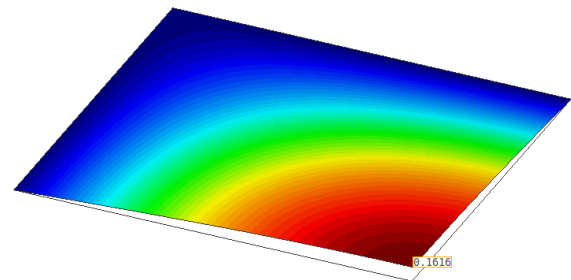
2 2.785

3 6.8

• Postprocess

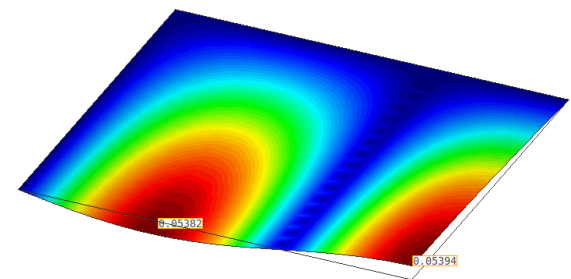
The following pictures show the three buckling modes for the performed analysis:

Postprocess panel ► Results ► Static ► Pre linearized buckling mode 1 ► Contour fill



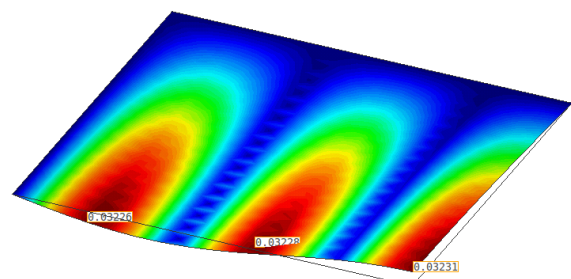
Pre linearized buckling mode 1

Postprocess panel ► Results ► Static ► Pre linearized buckling mode 2 ► Contour fill



Pre linearized buckling mode 2

Postprocess panel ► Results ► Static ► Pre linearized buckling mode 3 ► Contour fill



Pre linearized buckling mode 3

15. Tutorial 14 - Impact between bottles

The aim of this tutorial is explaining the recommended steps to perform a simulation of impact between two bottles. It can also be used as a guide for impacts among other kind of objects.

This tutorial's manual is available in LogNoter format within the distribution of the software itself and can be accessed at any time during the working session of Ramseries by using the following option of the main menu:

Help ► Learning Tdyn ► Ramseries tutorials

This tutorial provides brief instructions to construct the geometry of the model from a bottle profile. The geometry of bottle's profile is included in the LogNoter version of this tutorial. This bottle's profile can be extracted from the following internal link:

[bottle_profile.igs](#)

This geometry can be also found and downloaded from the

support section of the compass website:
<http://www.compassis.com/compass/en/Soporte>

The igs files can be imported within Ramseries by using the following option of the main manu:

Files ► Import ► IGES

On the other hand, the geometry of this tutorial model can be extracted from the following internal link and can be used by the user as the starting point to reproduce this tutorial:

[bottle_impact_shoulder.igs](#)

Alternatively, the model ready to be meshed and run, can be automatically loaded into the GUI by left-clicking on the button below.



15.1. Introduction

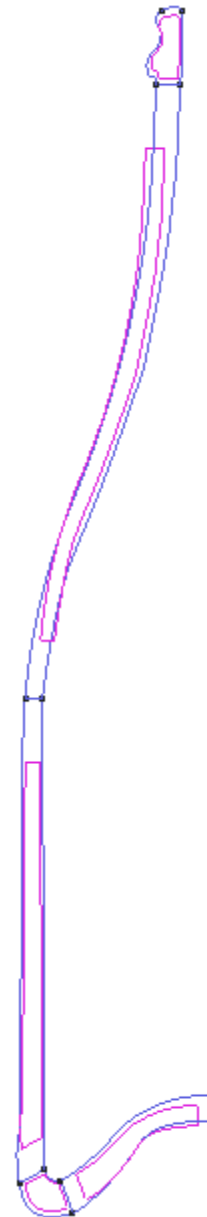
This example simulates an impact between two glass bottles on the shoulder. Both bottles are on a conveyor belt. One moves toward the other, that stands still, at a velocity of 0.6 m/s. In the following image the model is shown:



Impact between both bottles on the shoulder. Geometric model

15.2. Model's geometry

For performing any simulation, first you must define the geometric model. To start to draw the geometric model, the bottle's profile must be imported. At the beginning of this tutorial, it is explained how getting and importing the profile's model.

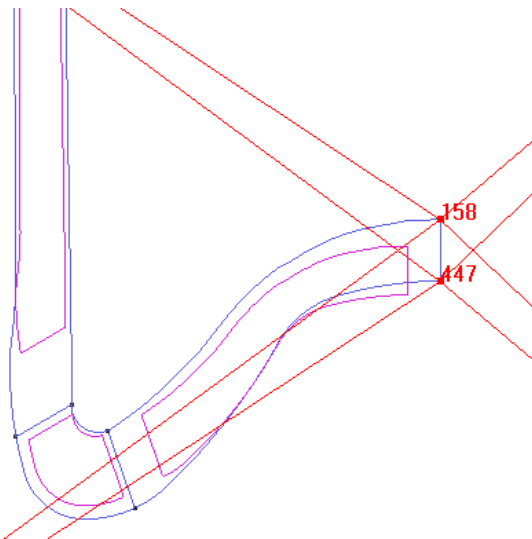


Bottle's profile

The surface have been splitted in the impact point, in this case, in shoulder. For doing so:

Geometry ► Edit ► Divide ► Surfaces ► Near point

First, a rotation extrude must be done. The points that define the rotation axis are shown in the following picture:

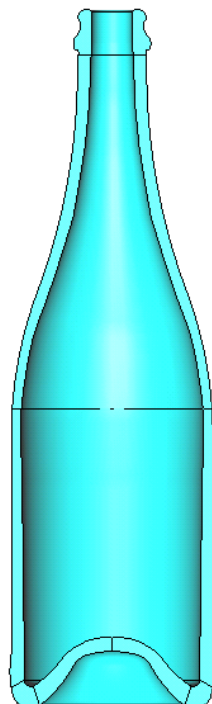
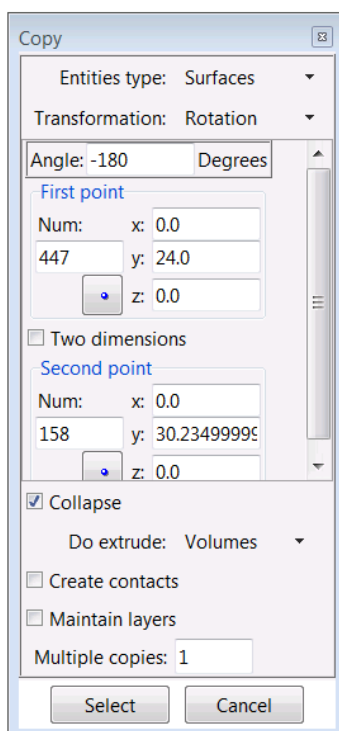


Points that define the rotation axis to apply the extrusion to create the first bottle.

For doing the extrusion:

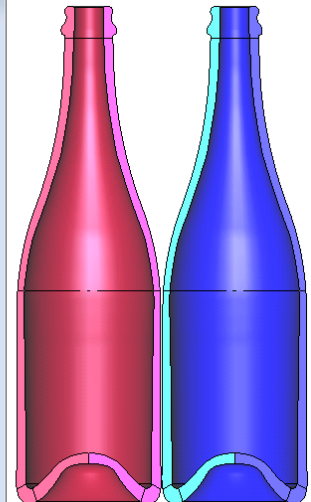
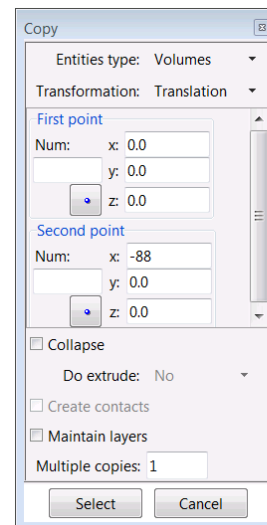
Utilities ► Copy

In the next picture the required parameters to create the first half bottle are shown. Only half bottle is created: symmetry constraints will be applied for performing an efficient simulation.

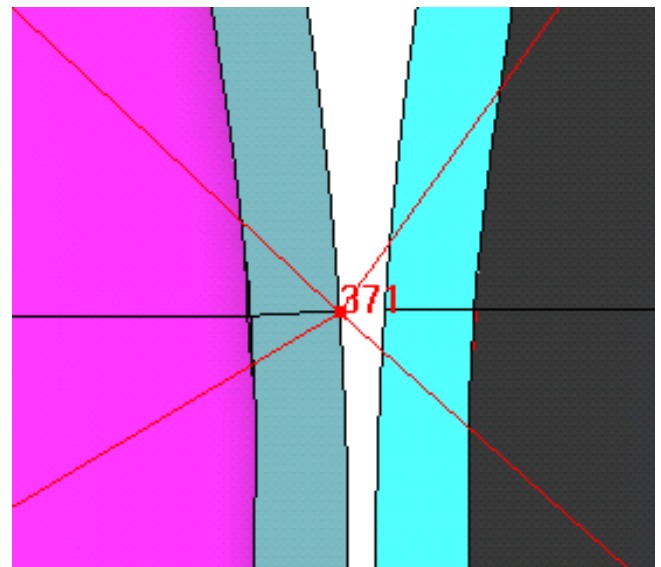


Extrusion operation using the bottle profile

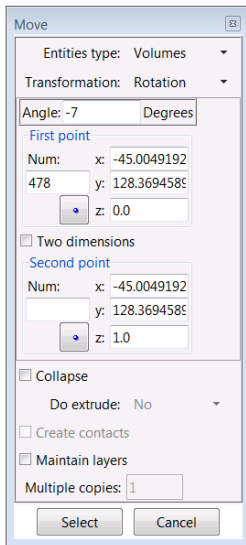
For creating the second half bottle, the previous half bottle is copied applying a translation.



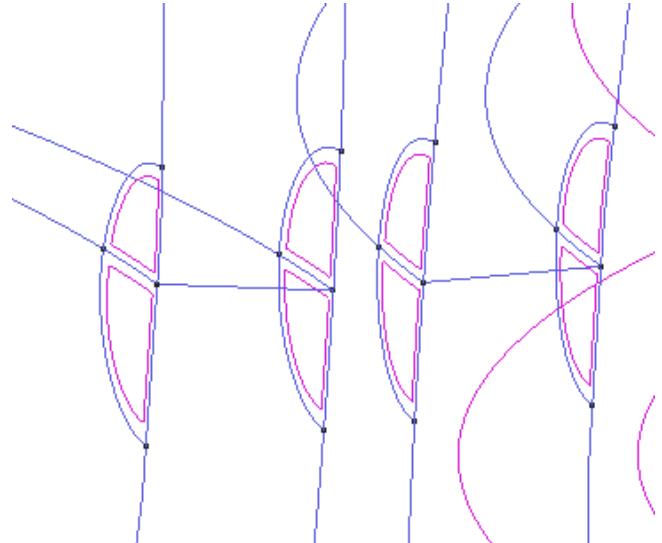
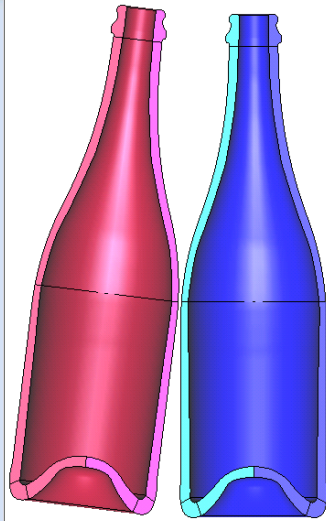
A rotation of 7° is applied in the left bottle to assure the impact in the shoulder. This operation can be unnecessary depending on the impact zone. The rotation center is shown in the following image:



Center used to apply the rotation to the left bottle



Parameters of rotation applied to the left bottle



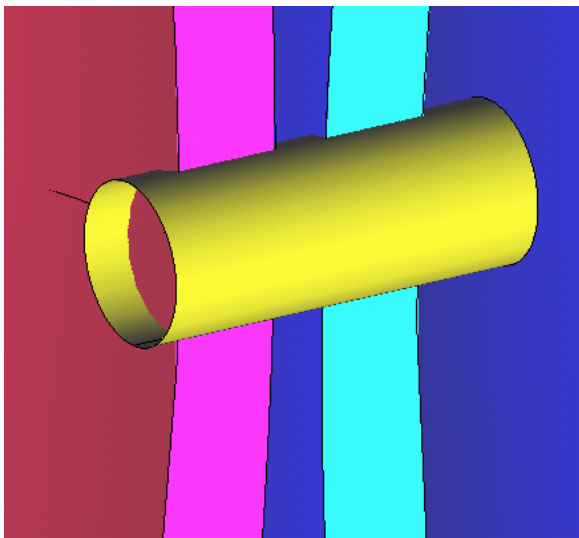
Defined areas to mesh with smaller elements

The geometric modes is already finished. However, it is recommended to create surfaces in contact area, and opposite areas inside bottles, for meshing these areas using smaller elements than in the remainder of the model.

Creation of contact area

For creating the contact area, it is recommended to apply the following steps:

Create a cylinder in the contact area as the following picture shows:



Cylinder for creating the surfaces in contact area and the opposite to them inside the bottle.

The radius of cylinder must be a little higher than the radius of contact area when the contact stress reaches the maximum value (3 mm for example).

Using the menu option

Geometry ► Edit ► Intersection ► Superficies

the contact area and the opposite to them inside the bottles are created.

It is **critical** that the normals in the contact zone surfaces are coherent. For doing so:

Utilities ► Swap normals ► Surfaces ► Make coherent

15.3. Loading Solids dynamic analysis

The first step to prepare a simulation using Tdyn is to select the type of analysis to be performed. This can be done within the **Start Data** window that appears by default the first time the user starts Ramseries. If the window does not appear by default, you can open it using the following option of the menu:

Data ► Start data

For this kind of problem, the Solids analysis can be set up from the Start Data window, as shown in the Beams tutorial, with the following options in this case:

- Simulation dimension: 3D
- Structural analysis: Solids
- Analysis type: Dynamic, Direct integration analysis
- Material constitutive model: Linear-elastic model
- Geometrical constitutive model: Linear geometry

15.4. Analysis data

The parameters that must be defined in the data tree to perform the simulation are explained below. The rest of parameters can remain with the default values.

Simulation data ► Units ► Geometry units: mm

Gravity is vector is set towards the negative Y axis for this

example.

Simulation data ► Gravity

General data ► Analysis ► Element types: Solids

General data ► Analysis ► Analysis type: Dynamic analysis

General data ► Analysis ► Boundary conditions: Non-linear boundary conds

The following non-linear analysis parameters are chosen:

General data ► Non-linear analysis data ► General

Solver control	Load control
Conv. tolerance	1.0e-3
Iteration type	Full Newton-Raphson
Max iterations	40

The following dynamic analysis parameters are chosen:

General data ► Dynamic analysis data ► General

Type	Direct integration
Δt	(function)
Number of steps	600

The time steps are defined as a function:

Interpolation function at ΔSteps

ΔSteps	Δt (s)
0	8.0e-7
600	5.0e-7

General data ► Dynamic analysis data ► Integration data

Integration method	Implicit (Bossak-Newmark)
α_{B-N}	-0.05
Initial conditions	User defined

If **Energy conserving** method is chosen as integration method, it is ensured that energy will be constant during all simulation. It can be checked if **Global parameters** are activated. On the other hand, it is possible to use **Bossak-Newmark** method too.

General data ► Advanced ► Dynamic output

If **Global parameters** are activated, *kinetic, elastic, potential* and *total energy* will be calculated and written as results.

Since the number of steps to be calculated is large, the size of results file can be heavy so, it is recommended an *output step* of 8.

General data ► Advanced ► Contacts data

Maximum admissible penetration (m)	0.025e-6
Normals sense swap	1
Contacts analysis method	Full
Maximum admissible force (N)	0.5

Contacts analysis method **Full** is the most efficient option in this case because both bottles are hard, but when bodies are soft analysis may not converge. **Simplified** option improves the convergence, but needs more calculation time.

15.5. Constraints

Constraints in the plane of symmetry and in the base of bottles must be applied.

► Constraints in the plane of symmetry

The constraint in the plane of symmetry must be elastic and its stiffness is $7e15 \text{ N/m}^2$ in Z direction. To define this constraint:

Constraints ► Elastic constraints

Activation		Values	
X constraint	NO	X value	0.0
Y constraint	NO	Y value	0.0
Z constraint	YES	Z value	$7e15 \text{ N/m}^2$

This constraint must be assigned to the surfaces in the symmetry plane (Z=0)

► Constraint in the base of bottles

A constraint that avoids displacement downwards but allows displacement upwards must be used to simulate a conveyor. Therefore, a non-linear elastic constraint must be applied in the base of the bottle.

For this reason, Y direction must be fixed and base surface must be selected.

Constraints ► Elastic constraints

Activation		Values	
X constraint	NO	X value	0.0
Y constraint	YES	Y value	(function)
Z constraint	NO	Z value	0.0

The function to be inserted is:

Function on Δy: Interpolation function at Δy

(Single value)

Δy (mm)	Ky (N/mm ²)
-1.0	-1000
0.0	0.0
1.0	0.0

This condition must be assigned to the heel of the bottle

Remember that it is necessary to select **Non-Linear Boundary conditions** in *General data* in order to be able to apply this constraint.

15.6. Contacts definition

In order to perform an impact simulation, it is required to define the type of contact (frictionless or sticking) and to assign the surfaces's bottles where the impact will be happen. For doing this, it is necessary to press the next Tdyn data tree option:

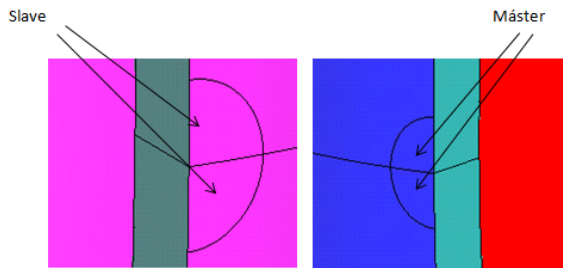
Contacts ► Contacts structural ► Master-Slave contacts Surf-Surf

This option is available only a type of non-linearity (material,

geometry or boundary conditions) has been activated. In this example, non-linear boundary conditions are selected.

If contact is **frictionless**, contact surfaces can slide but not if contact is **sticking**. In this case, contact must be frictionless.

Moreover, master and slave surfaces must be selected. It is recommended to choose surface of static bottle as master and the surface of bottle that is moved as slave.



Detail of master and slave surfaces. In this case, bottle on the left will move to the right one.

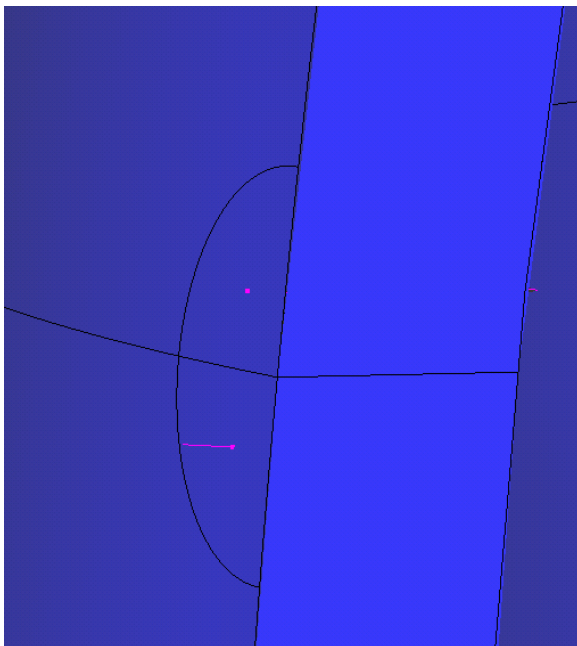
Checking normals of contact areas

In a simulation with contacts, it is very important that all master surfaces's normals must be coherent.

To see surfaces's normals the menu option

View ► Normals ► Surfaces ► Normal

must be used and then, master surfaces must be selected.

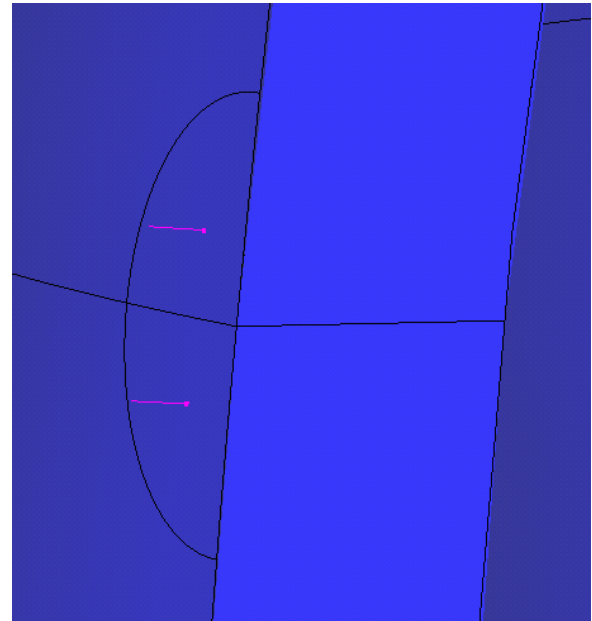


Master surfaces's normals are not coherent. One vector points to inside the bottle and other to outside.

As the previous image shows, both vectors are not coherent and one of them must be swapped. The menu option

Utilities ► Swap normals ► Surfaces ► Make coherent

swaps normals of the selected areas. In this case, the upper surface is selected to swap its normal but the lower could be selected too.



Now, both normals are coherent.

15.7. Initial conditions

Initial conditions of displacement and velocity must be applied to the bottle that moves to the static bottle.

Only it is possible to define initial conditions if "User defined" is chosen in the parameter

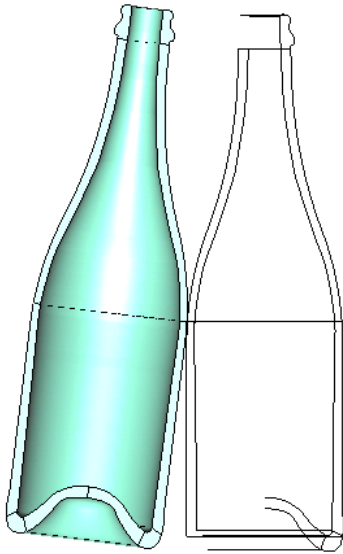
General data ► Dynamic analysis data ► Integration data ► Initial conditions

Initial displacement to be applied depends on the distance between both bottle.

Dynamic conditions ► Initial conditions

In this case, an initial velocity of 0.6 m/s and initial displacement of 2.00275 mm to bottle on the left as the following picture shows.

Displacements x	2.00275 mm
Displacements y	0.0 mm
Displacements z	0.0 mm
Velocity x	0.6 m/s
Velocity y	0.0 m/s
Velocity z	0.0 m/s



An initial velocity and displacement are applied to bottle on the left.

15.8. Material properties

To assign the properties to both bottles the next option must be chosen in *Tdyn data tree*:

Material and properties ► Solids ► Isotropic solid

Material of both bottles is glass:

-E= 70.0 GPa(Young modulus)

- ν = 0.22 (Poisson coefficient)

-Specific Weight= 25000 N/m³

In this case, since glass is a material defined in the materials library it can be chosen and all properties are introduced automatically.

If material properties are apply to both bottles separately, as it is shown in the picture below, then each bottle will have its own mesh in Postprocess and their mesh properties (visible, display style, etc...) can be changed separately too.

15.9. Loads

Self weight load must be assigned to both bottle using the next *Tdyn data tree* option:

Loadcases ► Loadcase 1 ► Solids ► Self weight load

15.10. Mesh generation

When the geometry of the model is drawn and all conditions have been assigned to it, then the mesh can be creating applying the following procedure:

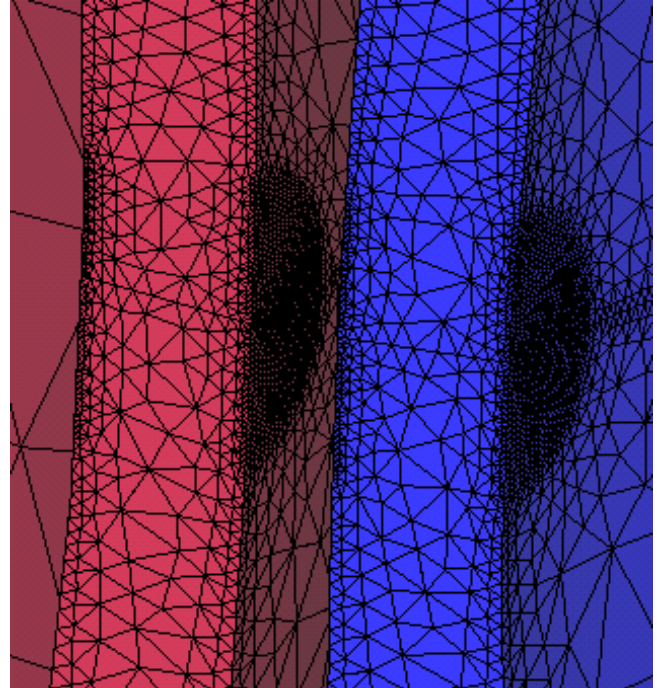
The contact area and the opposite of them inside the bottle can be meshed with a structured mesh of triangles with a size of 0,2 mm. For defining a structured mesh, the next menu option must be used:

Mesh ► Structured ► Surfaces ► Assign size

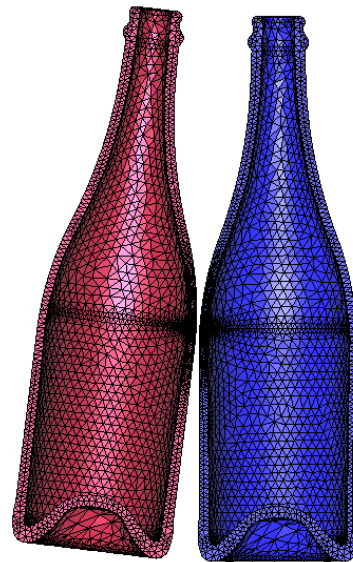
The remainder of the model can be meshed with a maximum

element size of 5 mm

The mesh generated using the options explained previously is shown below.



Detail of mesh in contact areas and the opposite to them inside the bottles.



General view of mesh

15.11. Calculate

When model's geometry is drawn, boundary conditions, material properties and the remainder simulation parameters are defined, and the mesh is generated, it is possible to run the simulation if the model has been saved. For saving the model use :

Files ► Save

And to run the simulation use the menu option

Calculate ► Calculate

The analysis runs as a separate process. Then, it is possible to continue working with GiD or exit the program. When the analysis is finished, a window will appear notifying it. If it has not run successfully, one window showing some error info will be supplied. Correct the error and run again.

It is possible to visualize, when the process is running, some information about its evolution. Press:

Calculate ► View process info

```
Increment:: 493 ... time step:0.0002465

Load norm check:0.00954192

Starting solution process
Total Hybrid-Sparse solver time = 0.043 min. = 2.6 seg.
Number of contacts added: 1
Number of active contacts before iterations: 114
Iteration 1: .... Load convergence factor=17022.1
Iteration 2: .... Load convergence factor=5329.12
Iteration 3: .... Load convergence factor=390.191
Iteration 4: .... Load convergence factor=4.58908
Iteration 5: .... Load convergence factor=0.0320858
Iteration 6: .... Load convergence factor=0.00102985
Iteration 7: .... Load convergence factor=4.45101e-005
Number of active contacts after iterations: 114
```

View of process info

In a simulation with contacts, the number of nodes contacted can change (new contacts created or old contacts removed) at the beginning of each step. When convergence has been reached, contacts are changed another time and new iterations are calculated until convergence is reached another time. When there are not changes in contacts (creation and disconnection), then this time step has been calculated totally. For instance, the previous image it is shown that a new contact has been created and after convergence has been reached there has been not changes in contacts so this time step is finished.

It is possible that convergence is not reached. Reducing time step can be a possible solution in order to improve convergence.

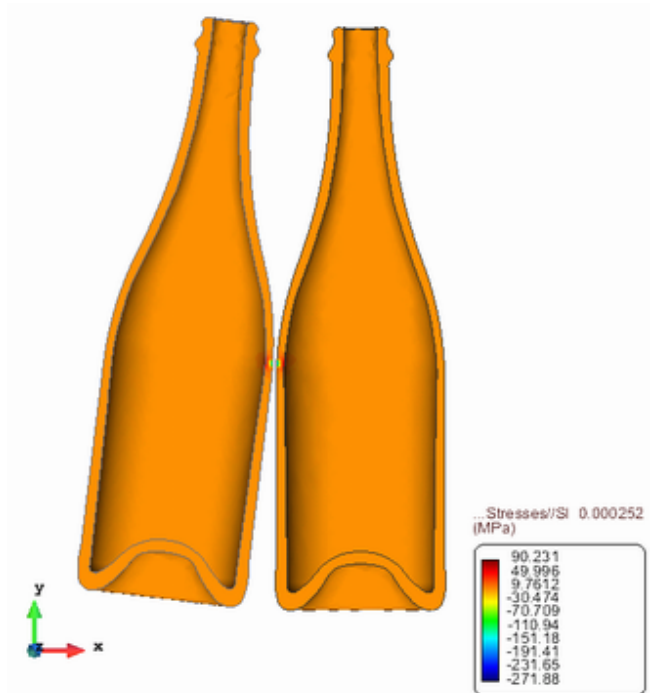
15.12. Results

When simulation has been executed, then it is possible to see the results with the next menu option:

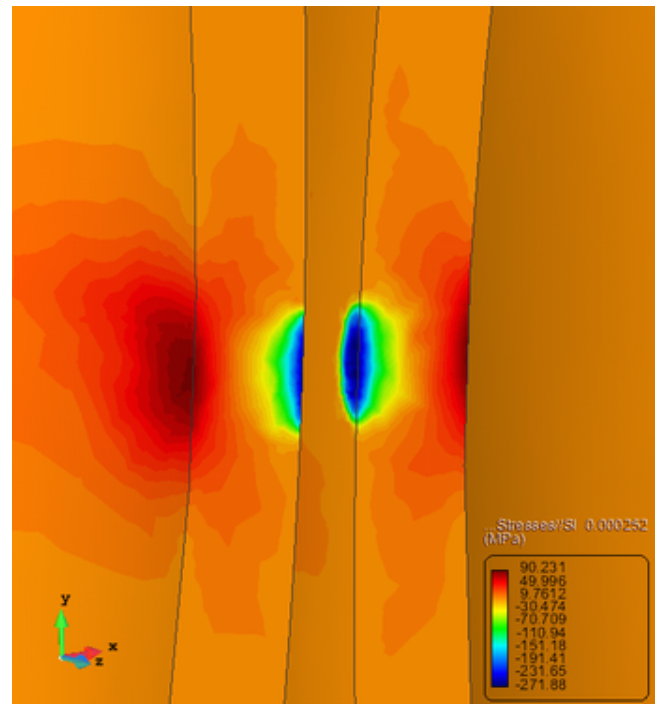
Postprocess ► Start

Some interesting results can be visualized, for example, Si-Stress:

Postprocess panel ► Results ► Peso propio dynamic ► Stresses ► Si ► Contour fill



Si-Stress result at time step = 0.000252



Si-Stress result at time step = 0.000252 s in contact area

And the **Animations control** allows to show results of a specific time step or to make animations.

Postprocess panel ► Time step

Deformed meshes

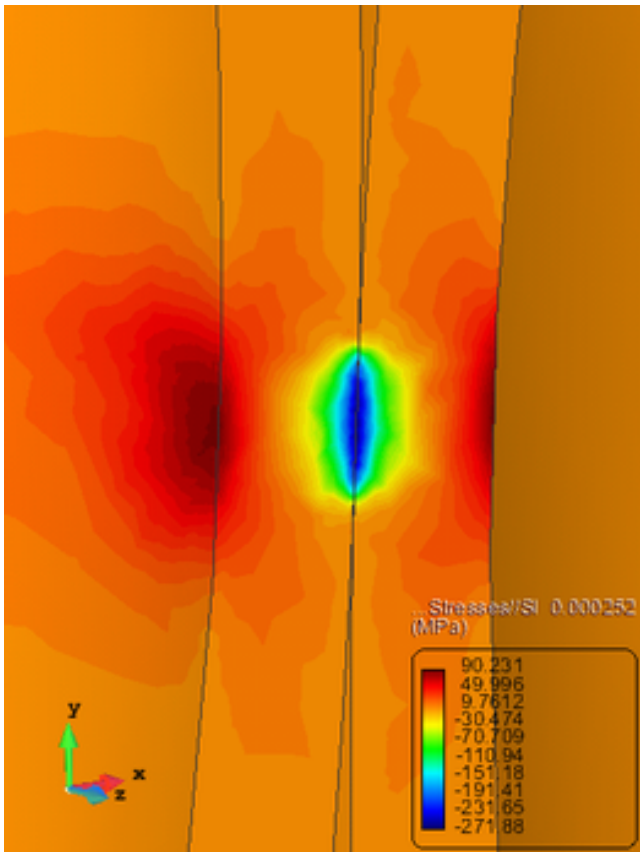
Meshes can be deformed according to a vector result and a factor. When doing this, all results are drawn on the deformed meshes. For instance, it is possible to move bottles according to displacement result calculated in the simulation. The next image shows the same result as the previous images but deforming

meshes.

Postprocess panel ► Preferences ► Deformed ► Draw:[Yes]

Postprocess panel ► Preferences ► Deformed ► Factor:[1]

Postprocess panel ► Preferences ► Deformed ► Result:["Peso propio dynamic Displacements"]



View of contact area with deformed mesh using Displacement result

Time graph

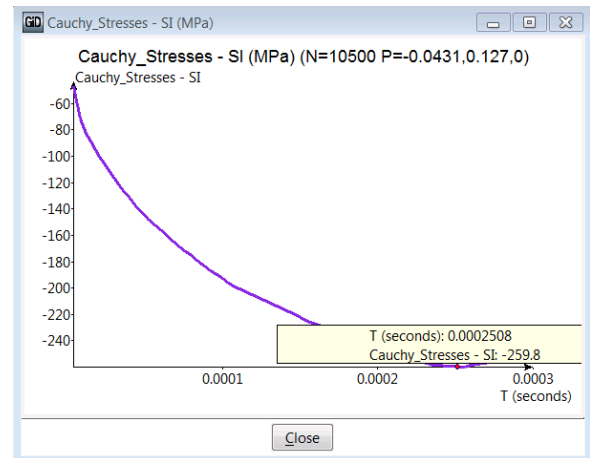
It is possible to create a time graph of a result in a node or point. The next procedure must be applied to create a time graph:

Postprocess ► Mesh information

Select a point or node (for example the node 10500)

Press **Time graph** in **Mesh information** panel.

As shown in the next image, it is possible to see a specific value selecting a point with the left button of mouse in the time graph. The number of points in the time graph depends on the number of time steps in **Animation control**.

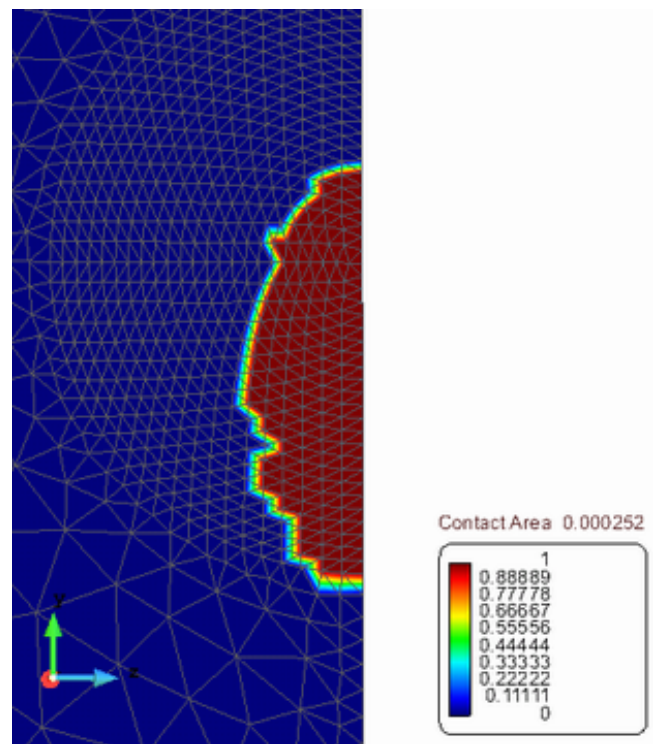


Time graph of SI-Stress of node 10500

Contact area

Contact area results is a specific result calculated in analysis with contacts. Nodes in slave surface and elements in master surface that are contacted, has a value of 1 and 0 the remainder of nodes and elements.

Postprocess panel ► Results ► Peso propio dynamic ► Contact area ► Contour fill



Contact area result in master surface at time step = 0.000252 s

Energy results

If **Global parameters** were activated in

General data ► Advanced ► Dynamic output

then it is possible to draw time graphs of kinetic, potential, elastic and total energy applying the next procedure:

Postprocess panel ► Results ► Peso propio dynamic ► Kinetic/Potential/Elastic strain/Total energy ► Contour fill

Select the menu option

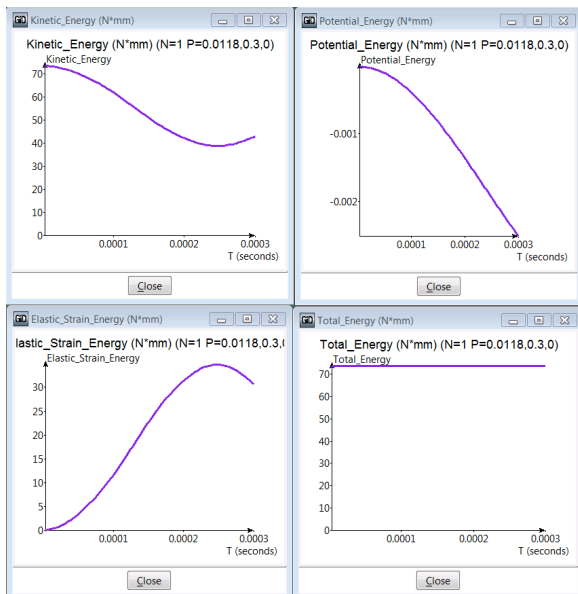
Postprocess ► Mesh information

Select the node 1 because this results are stored in this node.

Press **Time graph** option in **Mesh information** panel.

As the following images shows, total energy is constant during all time of simulation if **Energy conserving** was selected in

General data ► Dynamic analysis data ► Integration data



Time graphs of kinetic, potential, elastic strain and total energy

16. References

1. Barbat, A. H., Miquel, J. *Estructuras sometidas a acciones sísmica*, 2nd Ed., CIMNE 1994
2. Bathe, K.J. *Finite Element Procedures*, Prentice Hall, New Jersey, USA, 1996.
3. Clough, R. W., Penzien, J. *Dynamic of Structures*, McGraw-Hill, Inc. New York, 1975
4. Comisión permanente del Hormigón, *Instrucción de hormigón EHE*, 5^a Ed., Ministerio de Fomento 1999
5. Crisfield, M.A. "Non-linear finite element analysis of solids and structures", John Wiley & Sons, 1991.
6. Oñate, E. *Cálculo de estructuras por el método de los elementos finitos*, 2nd Ed., CIMNE 1995.
7. MacNeal, R.H., and Harder, R.L. *A proposed standard set of problems to test finite element accuracy*, Finite elements and Design, Vol. 1, pp. 3-20, 1985.
8. Zienkiewicz, O.C., and Taylor, R.L. *The finite element method*, 4th Ed., Mc Graw Hill, Vol. I, 1989, Vol. II, 1991.
9. Miravete, A., *Materiales Compuestos I*, Ed. Antonio Miravete, 2000
10. S. T. Pinho, C. G. Dávila, P.P. Camanho, L. Iannuzzi, P. Robinson. *Failure Models and Criteria for FRP under In-Plane or Three-Dimensional Stress States Including Shear Non-Linearity*. NASA 2005

