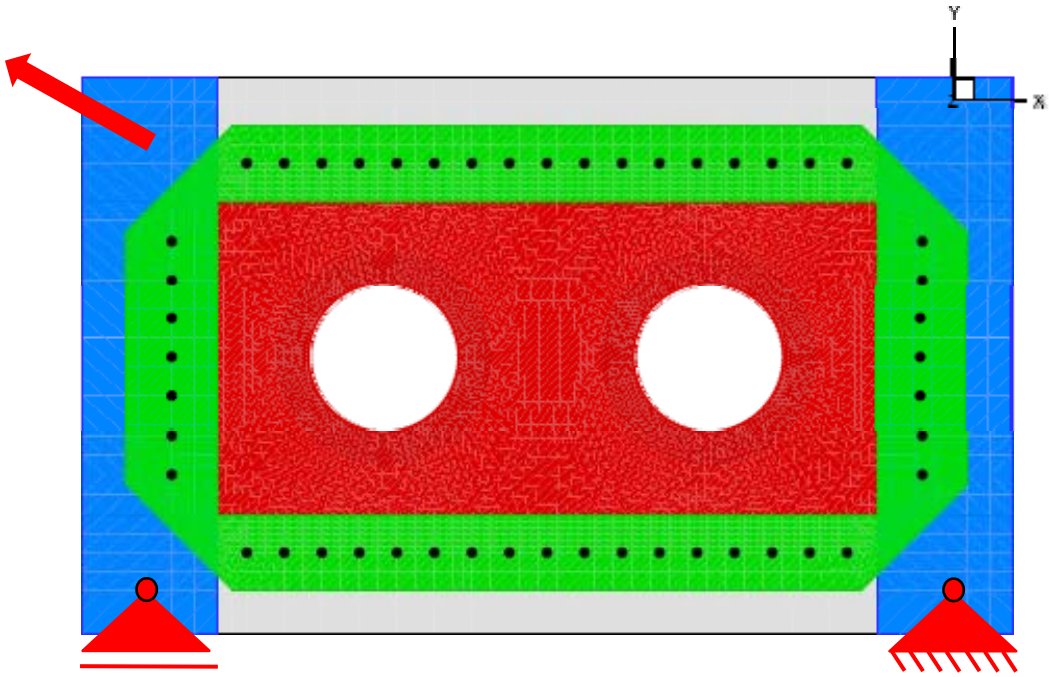
Shear test on Composite rib panel. Use of RIVET elements.



This tutorial provides general details to perform the numerical simulation of the shear test on a composite rib panel. One of the main issues explained in this tutorial is the use of special elements like SPOT or RIVET. The basic methodology described here may be applied to more complex processes. Some details are not explained here since they may be found in previous tutorials or in the help online.

Important aspects:
- setting of the problem (material, BC's and load definition)
- special materials and layup definition
- groups and curves definition
- special elements/nodes definition
- stage definition & running the simulation

A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate

D) Post-processor

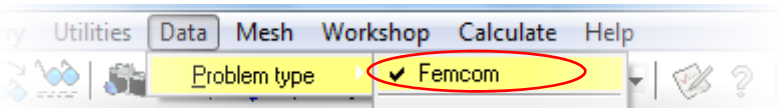
This tutorial presents the basic methodology to be followed to prepare the simulation of the **shear test on a composite rib panel**, employing a steel frame as test rig. The panel and the frame are joined employing special type of elements: the **RIVETS**.

The same methodology may then be applied to more complex geometries or processes.

The use of layers is fundamental to identify the parts that will be used in the simulation.

In this page it is explained the use of geometry layers.

- Open the **Femcom system**.
- Check that **Femcom problem type** is select.



Use of layers:

The geometrical layers are used to identify the imported geometries. Management of these Layers is done using the Layers menu

The following buttons are available:

Layer to use: indicates the active layer. Any geometrical entity that is created (lines, entities copy ...) will automatically be assigned to the current layer in use.

On - Off : visualize or hide the selected layer.

Colour: assign or change the colour of the selected layer.

Freeze: freeze the selected layer (layer is seen but no actions are assigned)

Unfreeze: unfreeze the selected layer

Opaque: turns the selected layer opaque

Transparent: turns the selected layer transparent.

Delete: delete the selected layer (only for layers containing no entities)

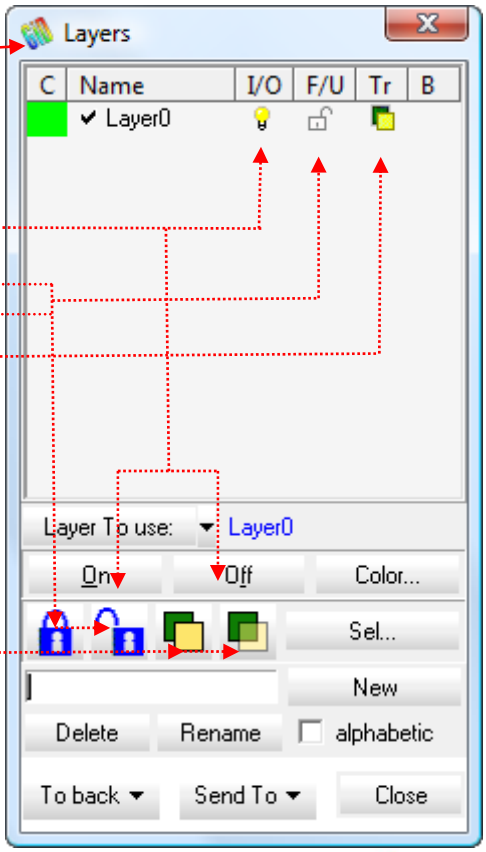
Rename: change the name of the selected layer.

Send To: move entities from one layer to another. The entities selected when using the **Send To** option will be included within the current layer in use.

New: create a new layer.

Use the **New** command for the creation of all the necessary layers for importing the model.

The **Send To** command should be used to move entities (points, lines, surfaces, volumes, points, elements) from one layer (original layer) to another (Layer-to-use). By default "also lower entities" are moved.



NOTE: Depending on the version of the **Femcom system**, the images used in this tutorial may be slightly different to the ones the user may find during the use of the software.

A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

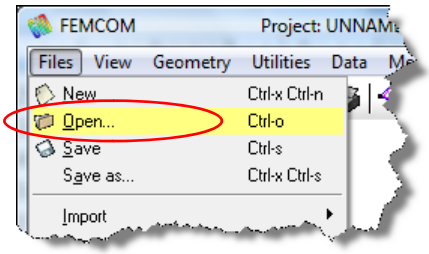
C) Calculate

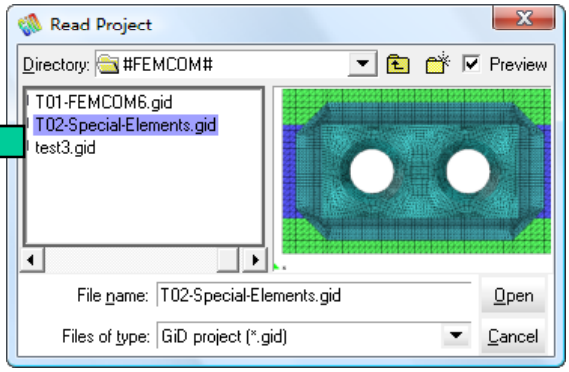
D) Post-processor

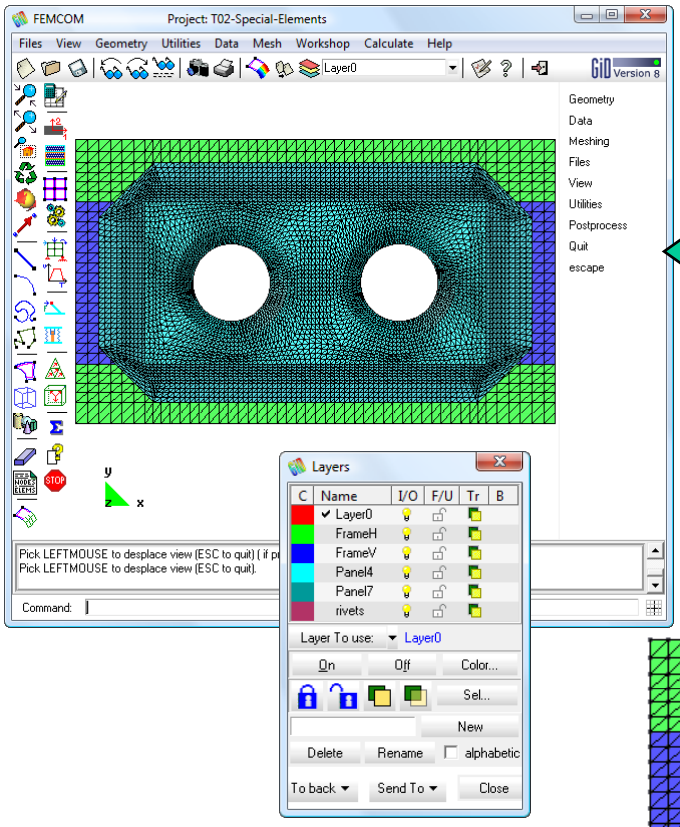
In this tutorial a previously prepared example will be used so the user must directly open a project where the necessary FEM meshes have already been imported.

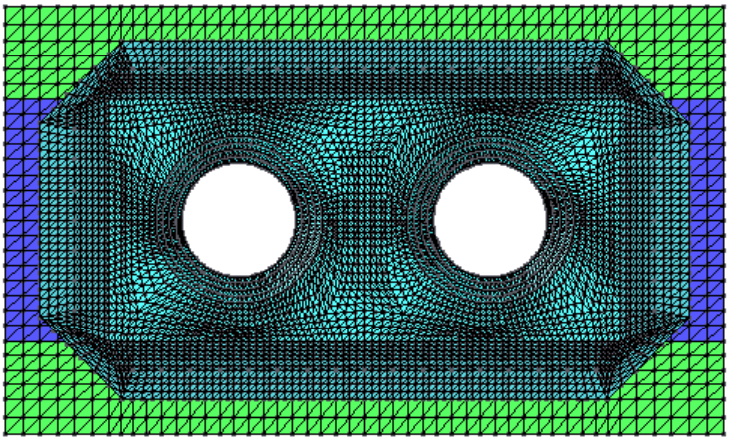
In order to do so, use **Files** menu to read the project **T02-Special-Elements.gid** from the Examples folder:

Files>Open









NOTE: To enable or disable the visualization of the nodes, the user must type the following instruction in the bottom command line: **"View Entities Nodes"** and then press "Esc" button twice.

A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

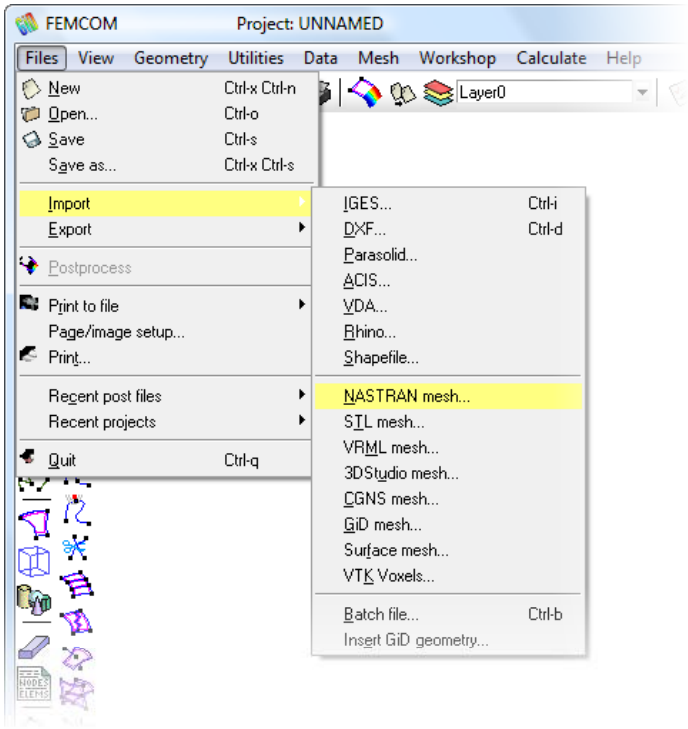
C) Calculate

D) Post-processor

This page briefly describe how to import a mesh created externally.

As an example, this page shows how to import a NASTRAN mesh.

NOTE: In this tutorial, we are not going to import any external mesh. To simplify the user's work, we use an already imported mesh that was already separated in different geometry layers. **You may skip this page**.



With the option **File>Import>Nastran mesh…** it is possible to read a file in NASTRAN format (version 68), with **Femcom** accepting most of its entities, which are:

- ✓ Entity name (Notes)
- ✓ CBAR CBEAM CROD CCABLE CBUSH CELAS1 CELAS2 CELAS3 RBAR (translated as 2 node bars)
- ✓ CQUAD4 CQUADR
- ✓ CHEXA
- ✓ CTETRA
- ✓ CPENTA
- ✓ CTRIA3 CTRIAR
- ✓ CONM1 CONM2 (translated as 1 node element)
- ✓ CORD1C CORD1R CORD1S
- ✓ CORD2C CORD2R CORD2S
- ✓ GRID

There are two options that can be used when reading a mesh if **Femcom** already contains a mesh:

a) Erasing the old mesh (Erase);

b) Adding the new mesh to the old one without sharing the nodes; the nodes will be duplicated although they may occupy the same position in the space (AddNotShare).

NOTE: The properties and materials of elements are currently ignored, because of the difficulties in associating the NASTRAN file properties with the requirements of the analysis programs. Therefore, you have to assign the materials "a posteriori" accordingly. However, in order to make this easier, the elements will be partitioned in different layers, each with the name PIdn, where n is the property identity number associated with the elements as defined in the NASTRAN file. Note also that CELAS2 elements do not have associated property identities so these will be created by default when the file is read.

A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

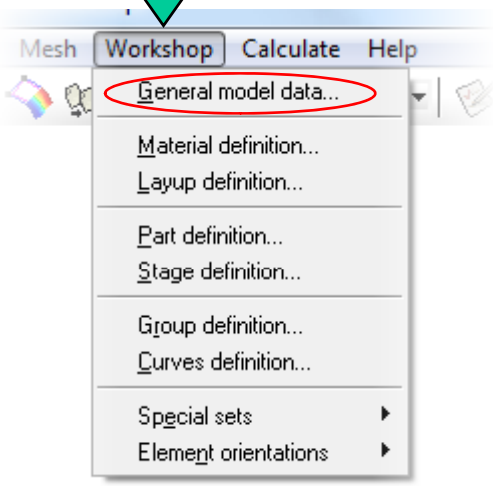
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

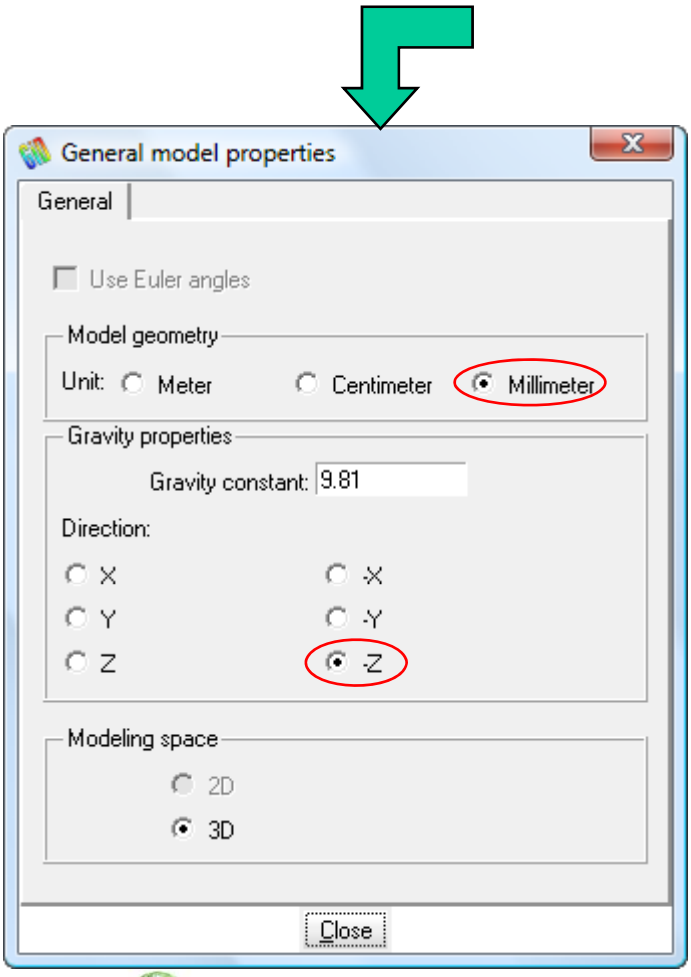
C) Calculate
D) Post-processor

The **Workshop** menu will help the user to describe the process and to run the simulation.

The first option is to define general properties related with the model.

Workshop>General model data…





In this window, the general properties of model are specified.

The option **Use Euler angles** must be enabled when local system bases are employed. At the moment this option is not available.

The mesh/geometry may be drawn in different scales according to size of the model. In this case **millimetres** are employed.

Select the appropiate value and direction of the gravity field.

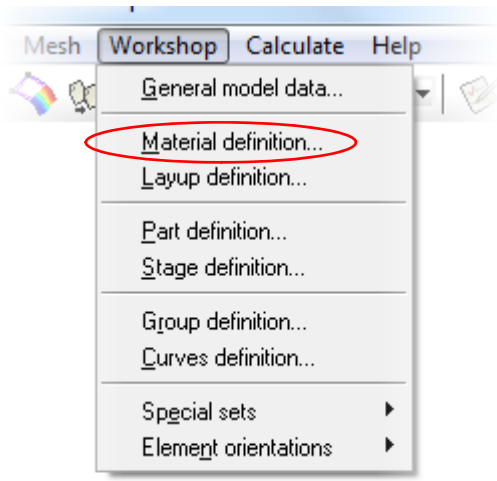
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

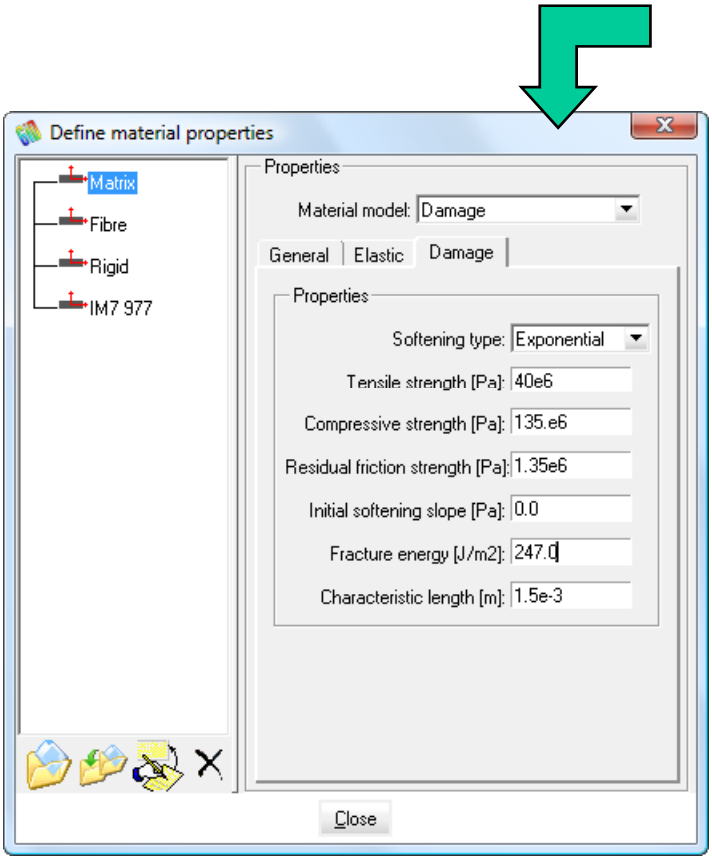
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

The second option in the **Workshop** menu is to define the material properties of the rigid and composite parts that will be used in the model.

Workshop>Material definition…





This window contains the **database of materials**. They may be of different types according to its **constitutive model**:

- Rigid
- Elastic
- Elasto-Plastic
- Damage
- UD-Composite
- Spot-Rivet

The following buttons may be used to manage the material database:

Add a new material.

Modify the name of a material.

Copy an existent material.

Delete the selected material.

Same options are available by rigth-clicking over the material name:

- Add
- Copy
- Modify
- Delete

A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

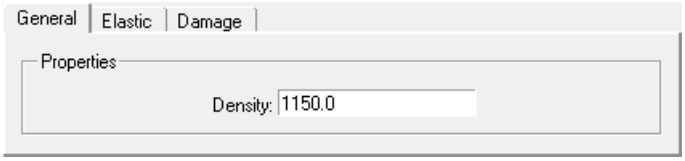
The **properties of the materials** must be edited directly by clicking over its value.

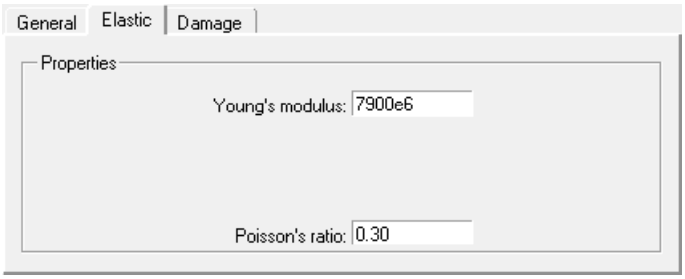
When a **new material** is created default values are suggested. The quantity of parameters to be entered depends on the material model selected.

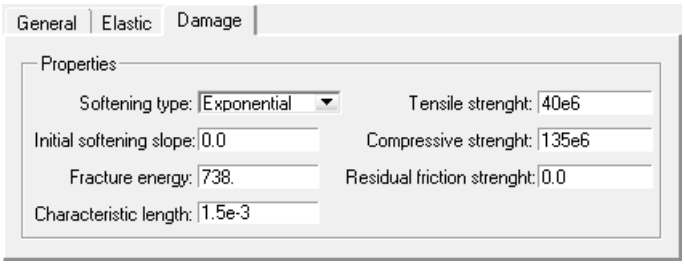
Every time you change a value, its is stored in the database automatically. Undo is not possible.

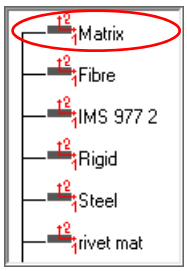
Here are some values that can be used to define **matrix and fiber properties**.when employing a **damage model**. They correspond to the epoyy resin CYCOM977-2 and to the carbon fiber IM7.

Matrix material

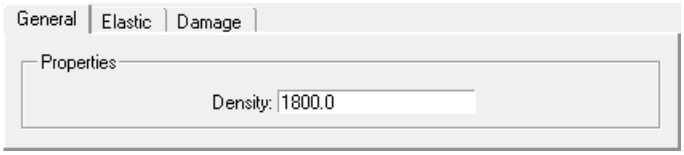


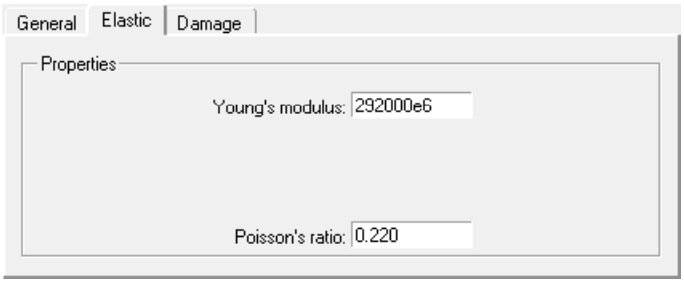


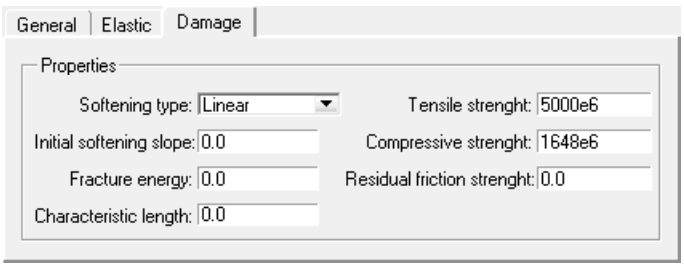


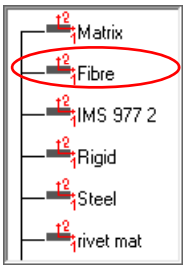


Fiber material









A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

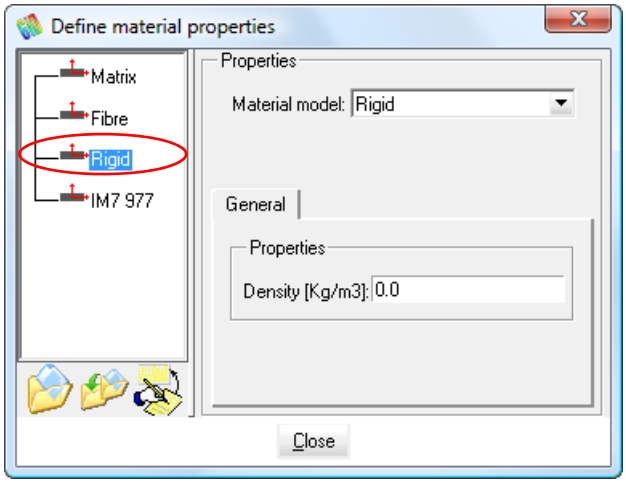
C) Calculate
D) Post-processor

There are also **special material types** that can be defined and added to the material database:

the **rigid material**,

the **UD-composite material**,

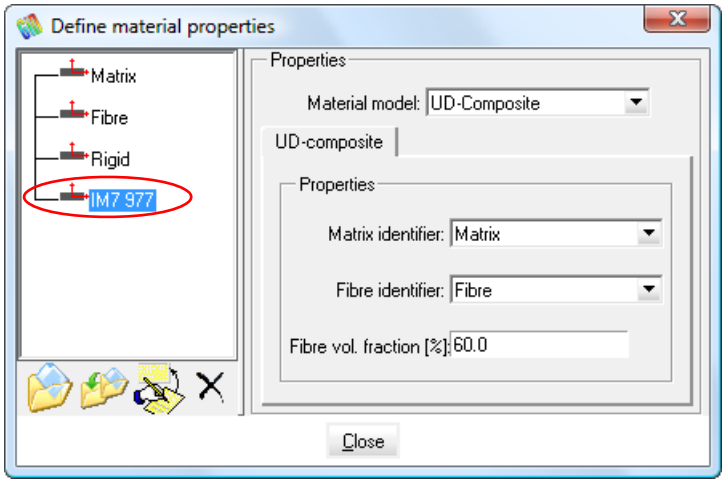
the **Spot-Rivet material**.



The **rigid material model** must be employed in those structural parts that will act as rigid bodies during the numerical simulation.

The density of the rigid material will be used by the FEMCOM solver to calculate the mass of the rigid body in terms of its volume.

In the present example, we do not plan to define any rigid body, so we will not need any rigid material.



The **UD-composite model** must be selected as a constitutive model of a unidirectional ply. This single ply will be then used to build up a laminate.

This model requires the selection of a material identifier for matrix and another for the fibre together with the fibre volume fraction.

Note: The component materials must be defined in advance, otherwise they will not be available for selection.

A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

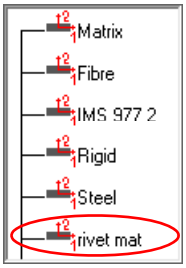
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

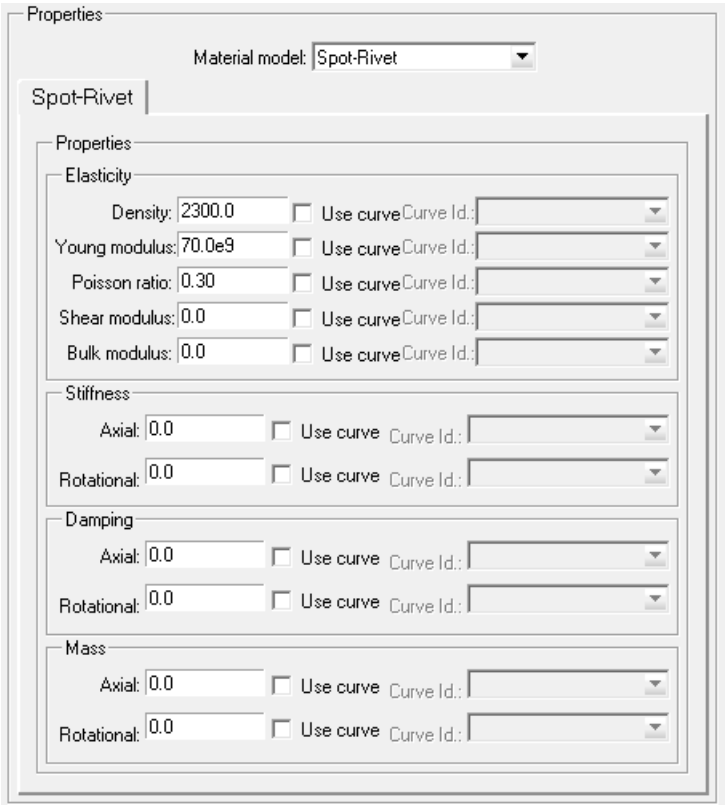
C) Calculate
D) Post-processor

The **Spot-Rivet material type** must be employed together with the definition of special sets of nodes and special sets of elements that are to be used to join or link structural parts between them.

This material type is very versatile since allow the definition of material parameters by setting a fixed value or by the selection of a curve (previously defined).

Note that not all the parameters are required at the same time. All parameters with zero value are disabled. The parameters must be enabled in accordance with its use: Spot elements will require different properties than the Rivets.





In the present example a **rivet material** is defined employing the Spot-Rivet model.

The elastic parameters defined here will be used later in the definition of the special rivet elements that will join the composite panel with the steel frame.

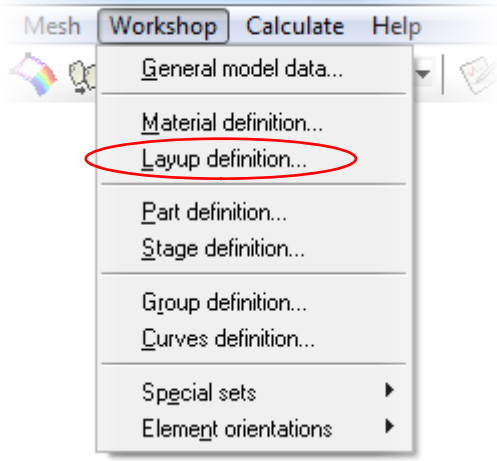
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

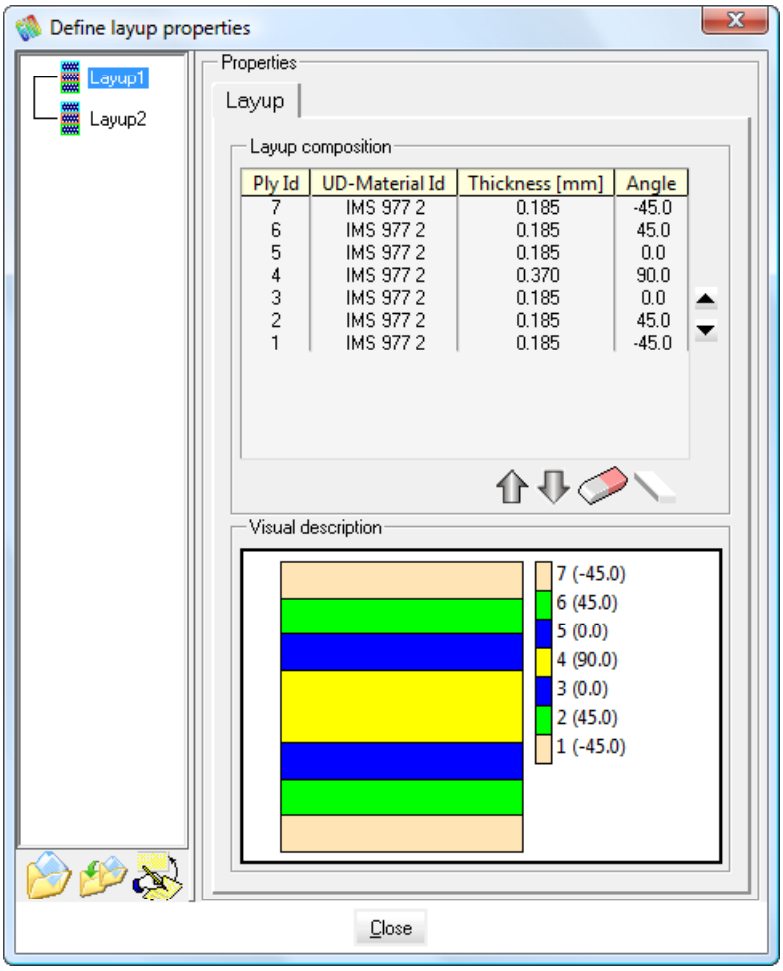
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

The third option in the **Workshop** menu is to define the different layups that will be employed in the composite parts.

Workshop>Layup definition…





This window contains the **database of layups**. The stacking sequence of the laminate can be constructed in the section layup composition. Each line in the table represents a ply. The material, thickness and angle must be specified by directly editing the default given value.

The following buttons may be used to manage the layup composition:

Add a new ply at the top.

Remove the top ply.

Delete the selected ply.

Delete all plies list.

A visual description of the laminate is given at the bottom. Different colours means different ply angle.

In this case, same ply material and thickness is given but different orientation angle is used.

A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

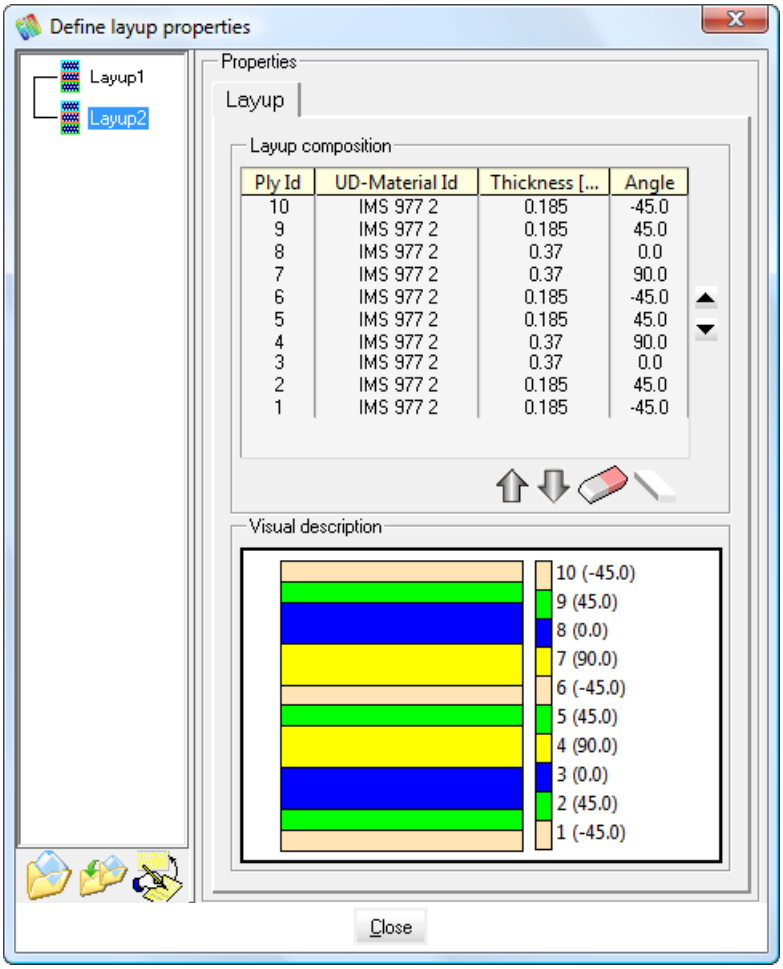
C) Calculate

D) Post-processor

In the previous page, the **Layup1** was defined with seven layers of composite material.

In this page, the **Layup2** is defined with ten layers of the same material but with different thickness and orientation, resulting in a thicker laminate.

Note that the virtual defined layers may not coincide with the real material stacking of the laboratory, but they are equivalent.



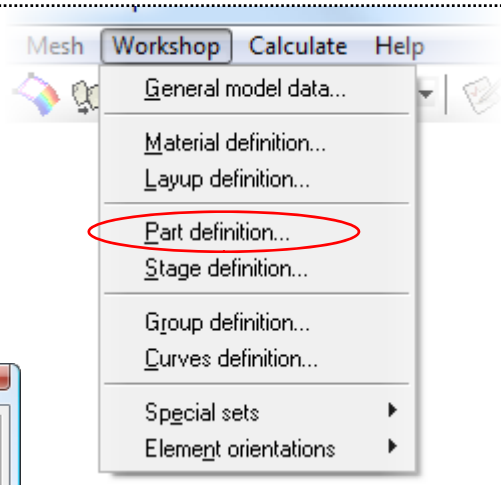
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

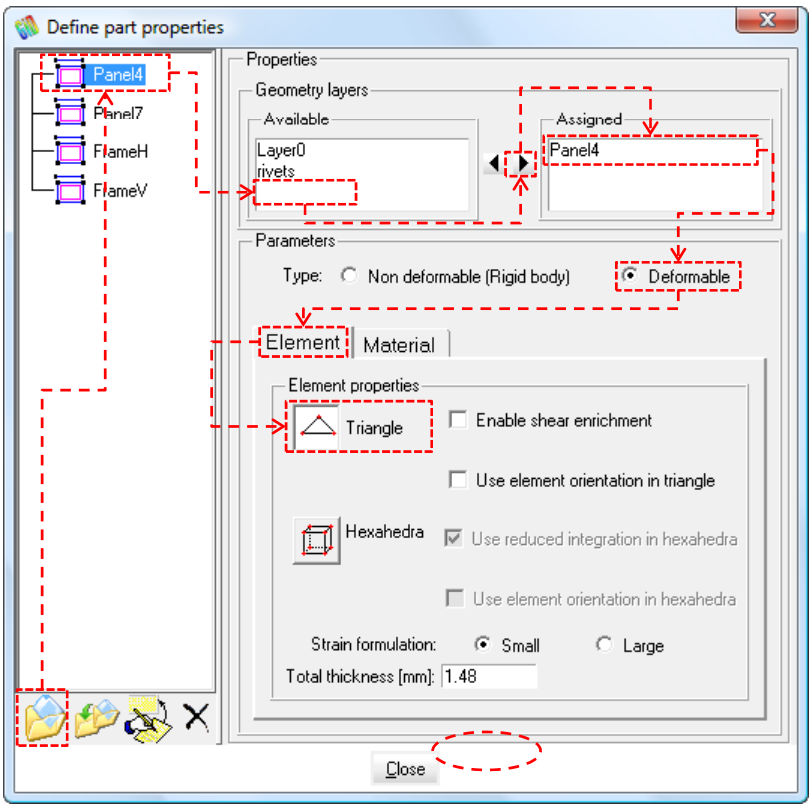
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

The next option in the **Workshop** menu is to specify the structural parts that will be employed in the simulation, and their properties.

Workshop>Part definition…



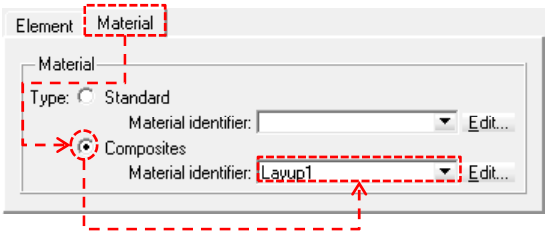


Create a new part definition by clicking on "Add new part" button: 

Type the part name: "**Panel4**".
Then select the corresponding layer from the "Available" list of layers and click on ▸ to assign it .
Tick the "**Deformable**" option.

In the "**Element**" tab, select the type of element for this part: "**Triangle**", and provide the total thickness: **1.48mm**.

In the "**Material**" tab, select the type of material to be used (**Composite**) and finally select the specific material ID from the list (**Layup1**).



Follow analogous steps to define the part named: "**Panel7**" but selecting the composite laminate "**Layup2**" and **2.59mm** of thickness.

For the parts "**FrameH**" and "**FrameV**" select the **standard** material type named **Steel**, and assign **50mm** of thickness (to create a stiff steel frame).

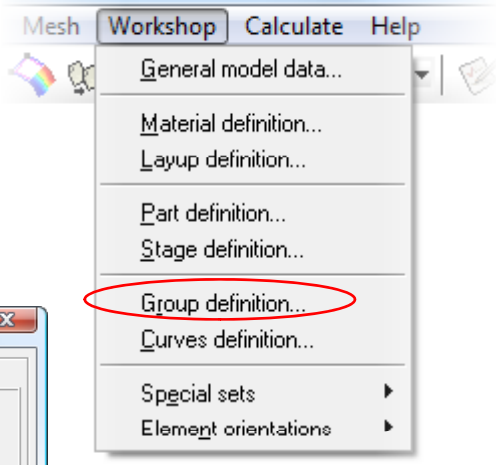
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

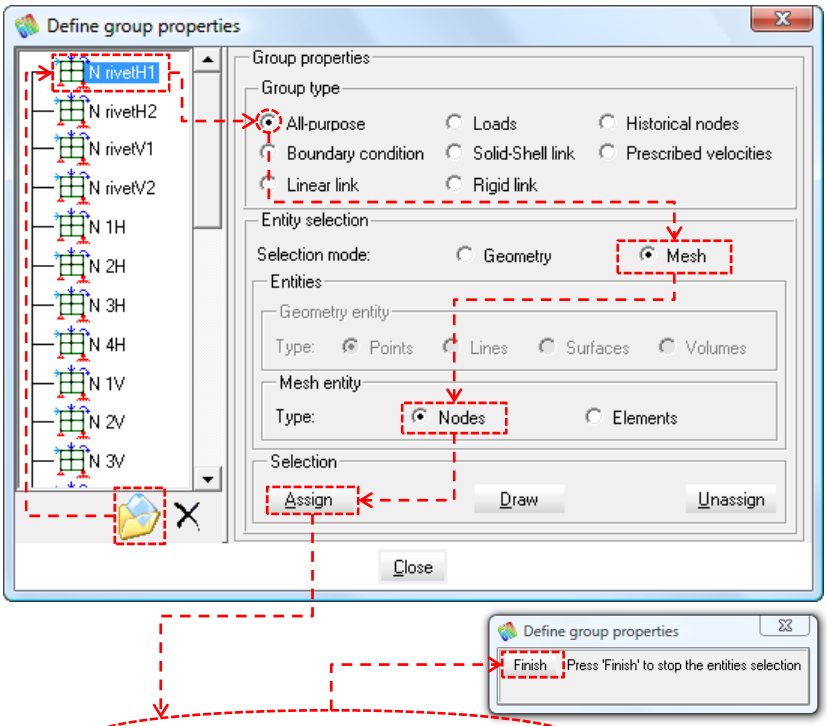
Another important option in the **Workshop** menu is: "**Group definition…**". It allows the user to give a name to a group of entities.

This will easy the future reference to them, for example to apply an specific constraint or property.



NOTE: To turn ON/OFF the nodes entities visualization, type the following instruction in the bottom command line: "**View Entities Nodes**" and then press "Esc" button three times.

Previous steps: Open the Layers window: Turn off all layers except "**rivets**".

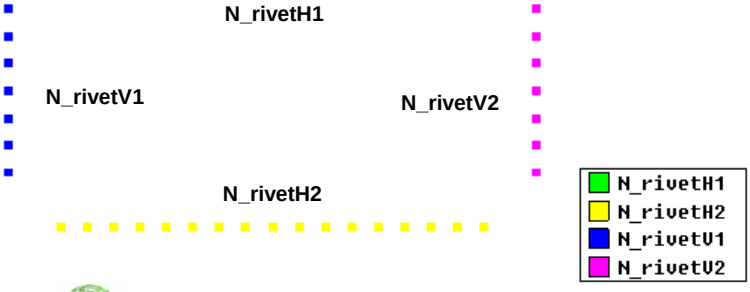


Create a new group by clicking on "Add new group" button:

Type the group name: "**N_rivetH1**".

Then select the group type. This will help the filtering of groups . If you use "**All-purpose**", this group will be available in all windows.

Tick the "**Mesh**" and "**Nodes**" options, and then click "**Assign**" button to select the corresponding **nodes** from the model geometry. Press "Finish" or "Esc" to stop entities selection.



Follow analogous steps to create the groups: "**N_rivetH2**", "**N_rivetV1**" and "**N_rivetV2**".

A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

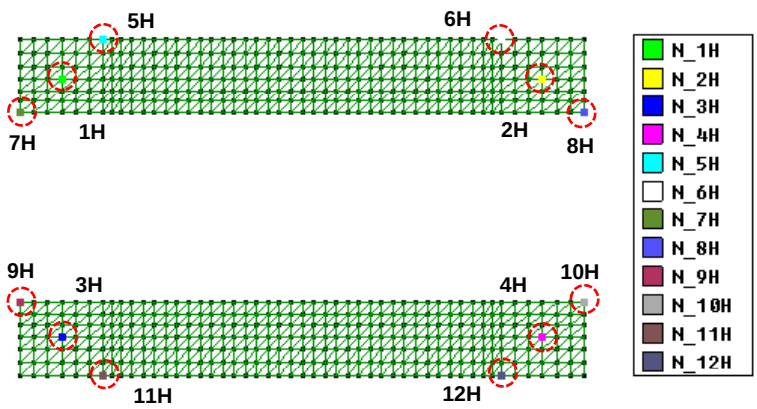
C) Calculate
D) Post-processor

Now we will define additional groups that will be employed to identify the nodes with special conditions.

In this case the group name is employed as a tag for a node. These tags will used later to establish kinematic conditions between them.

Turn off all layers except "**FrameH**".

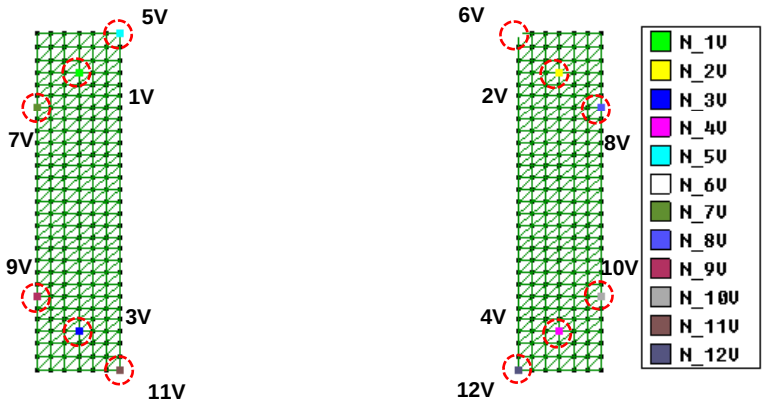
Define the following single-node groups:
N_1H, N_2H, N_3H, N_4H, N_5H, N_6H, N_7H, N_8H, N_9H, N_10H, N_11H, N_12H (to create each group follow analogous steps to previous page).

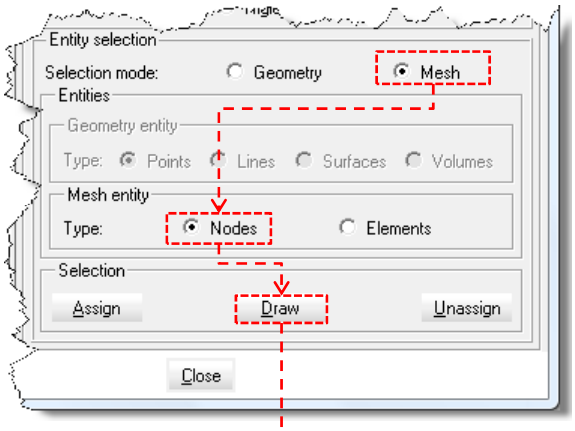


Turn off all layers except "**FrameV**".

Define the following single-node groups:
N_1V, N_2V, N_3V, N_4V, N_5V, N_6V, N_7V, N_8V, N_9V, N_10V, N_11V, N_12V (to create each group follow analogous steps to previous page).

To check that groups were appropriately defined, the user may click on the "**Draw**" button.
When **Mesh** and **Nodes** are selected, it will only be shown the "nodes groups" which layers are "on".





Note: To quit these views, click on the drawing and then press 'Escape'.

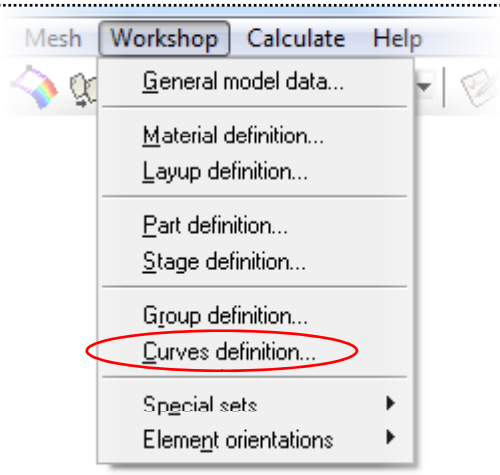
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

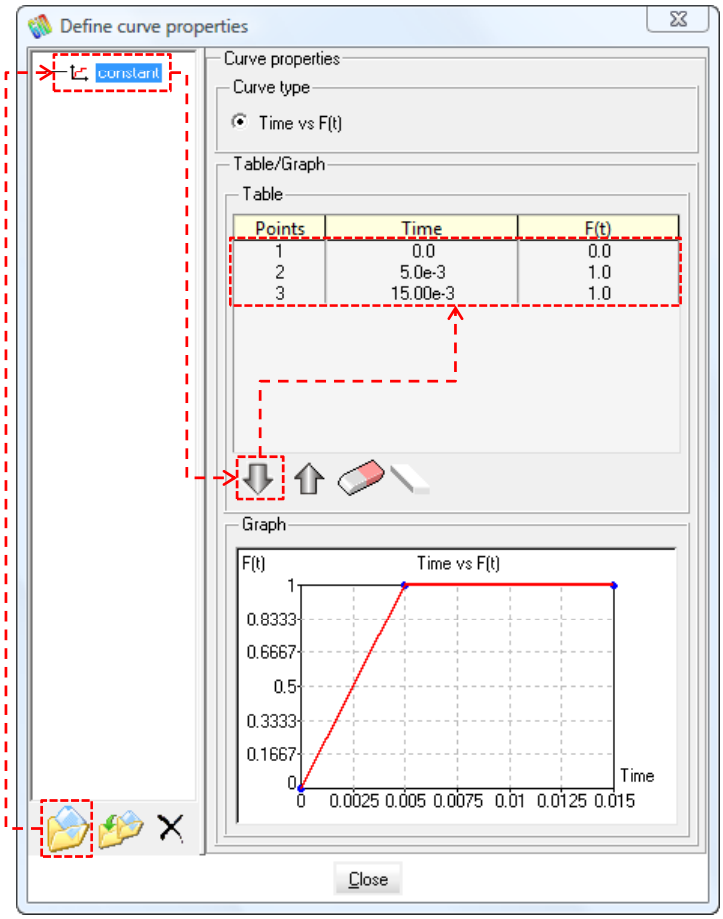
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

There is an useful option in the **Workshop** menu named: "**Curves definition…**". It allows the creation of curves defined by points.

These curves may be employed afterwards to describe the evolution of properties or conditions in terms of the time.





Create a new curve definition by clicking on "Add new group" button:

Type the curve name: "**constant**".

Add two lines to the table of points by twice clicking the "Add line" button:

Edit each line in the points list to define the curve that will be later used to apply the loading forces.

The resulting curve defined by the given points is shown in a graph below.

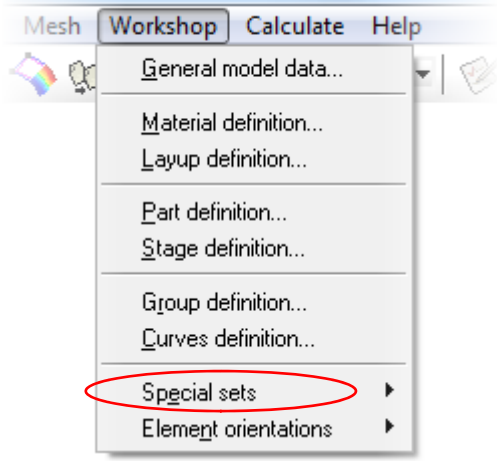
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

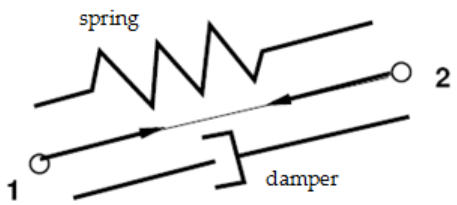
The "**Special sets**" option in the **Workshop** menu allows the user to define special sets of nodes and elements like: RIVET or SPOT.

Workshop>Special sets



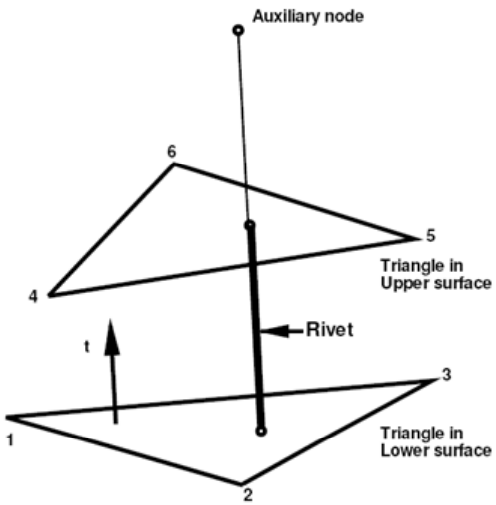
SPOT type (restriction using two nodal points)

- ✓ Forces may depend on:
  - points relative **position** (a non linear **spring**)
  - points relative **velocity** (**dampers**).



RIVET type (restriction between to triangular elements)

- ✓ It is assumed that the elements are **initially in contact** and joined by the rivet.
- ✓ The two surfaces must be **not coplanar** (the rivet length must be non zero).



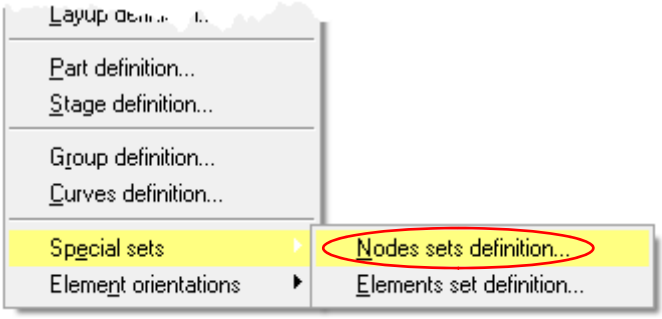
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

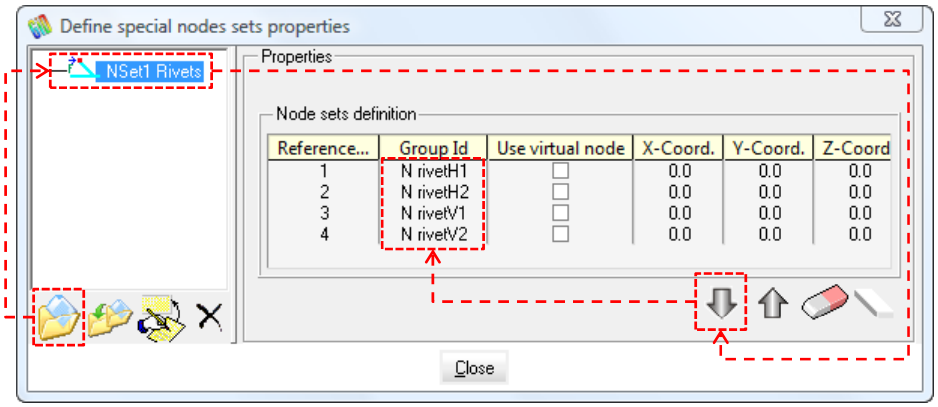
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

The "**Nodes sets definition…**" option let the user define special sets of nodes, that are not already included in the parts or tools. For example, the set of auxiliary points required to define the rivets location.

Workshop>Special sets>Nodes sets def…





Special nodes sets are those nodes that do not belong to any part or tool but that must be included in the strategy definition as auxiliary nodes.

Auxiliary nodes are required in the definition of special elements like spots or rivets.

Create a new set of special nodes named "**NSet1_RivetH1**".

This window allow the user to specify the nodes in two different ways:
1) by reference to an already defined group of nodes, or,
2) by employing the definition of virtual nodes.

Previously defined groups are employed in this case.

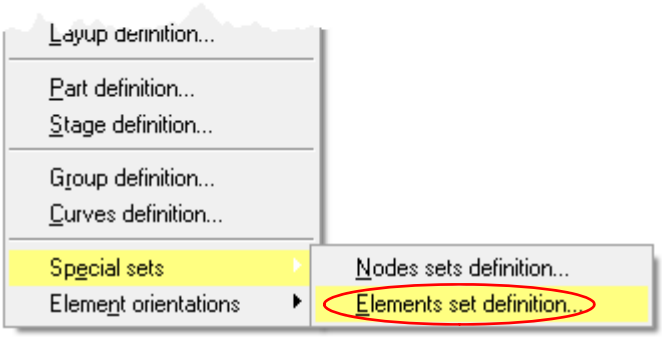
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

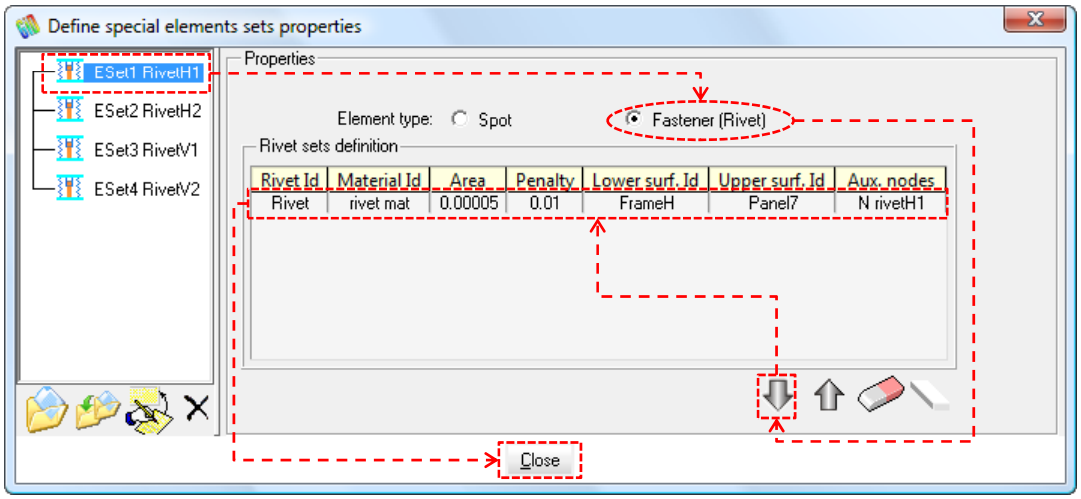
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

The "**Elements sets definition…**" option let the user define sets of special elements, that are not already included in the parts or tools. For example, the set of RIVET elements.

Workshop>Special sets>Elements sets def…





Create a new set of special elements named "**ESet1_RivetH1**".

Select the type of element: "Spot" or "Fastener". In this case, select "**Fastener**".

Add a line in the table specifying the following data:
- -Rivet Id: Id for this row of rivet elements set.
- -Material Id: Material Id for the rivets from the material database.
- -Area: Area of the cross section of an individual rivet [m^2].
- -Penalty: Penalty factor to be employed in contact algorithm.
- -Lower surf. Id: Part Id to be used as the lower surface of the joint.
- -Upper surf. Id: Part Id to be used as the upper surface of the joint.
- -Aux. nodes: Nodes set Id containing the auxiliary nodes that specify the rivets position.1

A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

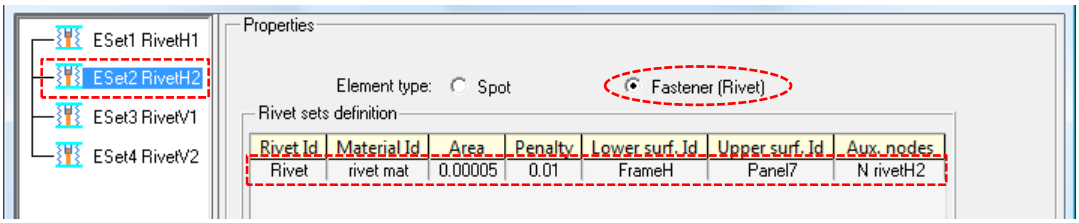
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

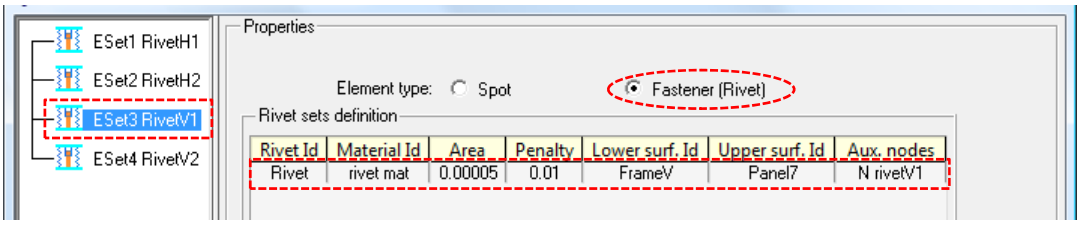
C) Calculate

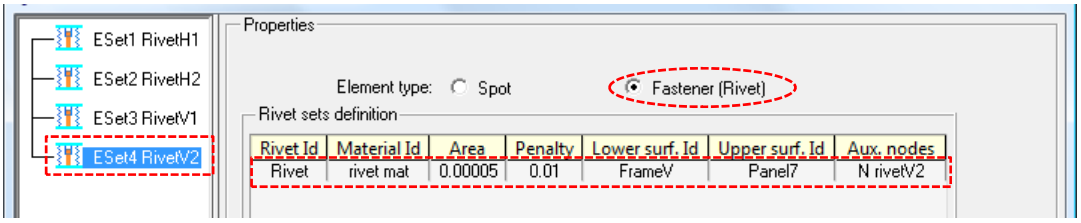
D) Post-processor

Define the remaining rivet element sets following analogous steps.

NOTE: You should take care in the correct definition of lower and upper surfaces, to avoid wrong contact definition in the elements joined by the rivets.







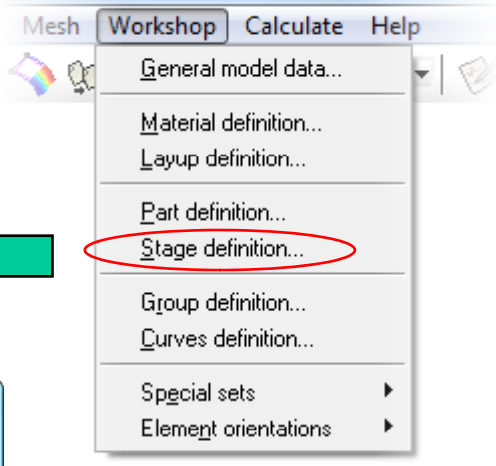
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

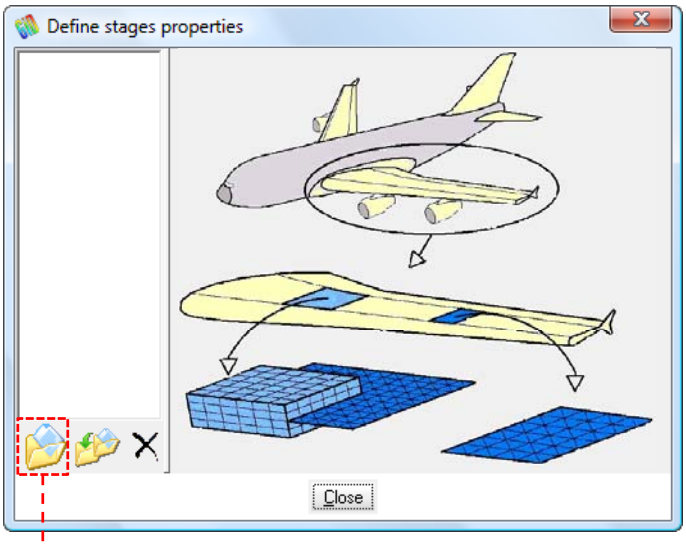
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

The "**Stage definition…**" option in the **Workshop** menu is the most important, since it will put together all the ingredients to perform the numerical simulation of the industrial process.

Workshop>Stage definition…



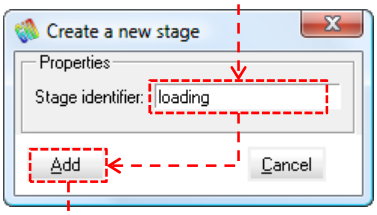


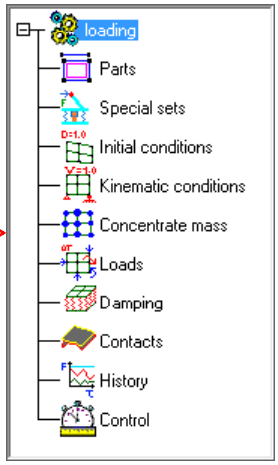
Create a **new stage definition** by clicking on "Add new stage" button:

Type the stage name: "**loading**".

This stage will contain the definition of the "loading" of the composite panel employing a steel frame test rig. The composite panel and steel frame will be joined with rivet elements.

The definition of a stage is subdivided into nine blocks that may be enabled or disabled depending on the strategy the user may follow to design the simulation.





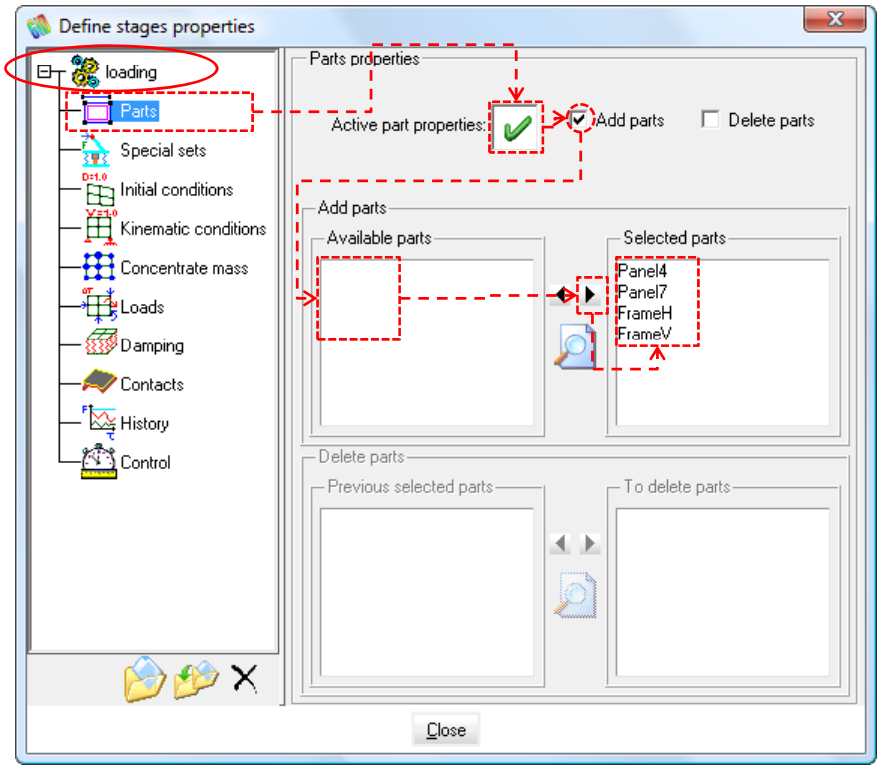
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

The "**Parts**" definition block is to establish which structural parts will be included in the simulation (in this stage and in the following ones).

Enable parts properties and then click "**Add parts**". Select the parts to be included (**Panel4, Panel7, FrameH, FrameV**) and then click on ▸ to finish the selection.



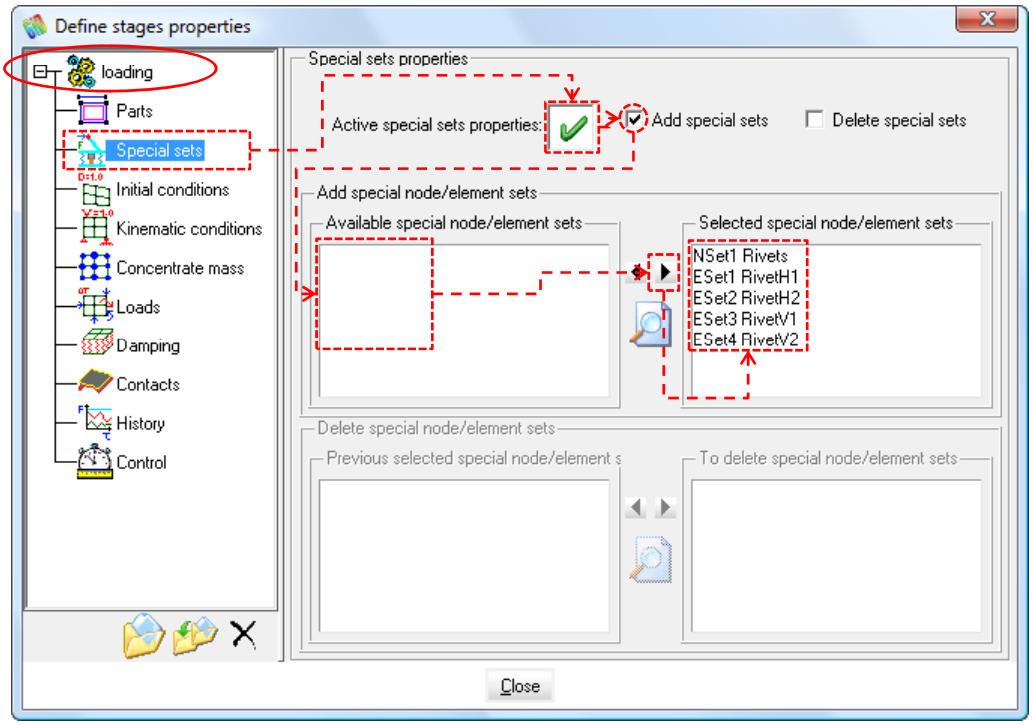
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

The "**Special sets**" definition block is to include extra nodes and elements in the simulation (in this stage and in the following ones if any).
These extra nodes do not belong to any part, and therefore should be specified here.

Enable parts properties and then click "**Add special sets**".
Select the sets of nodes and elements to be included and then click on ▸ to end the selection.



A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

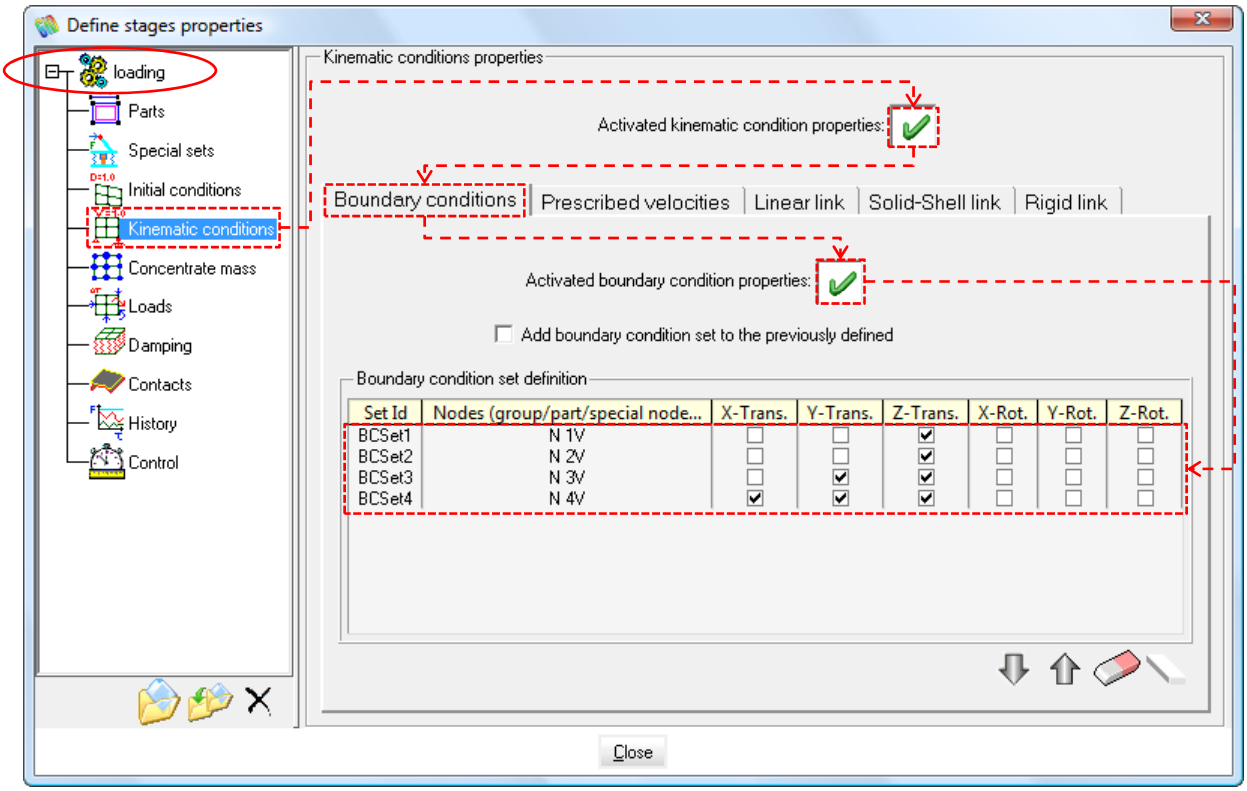
C) Calculate
D) Post-processor

The "**Kinematic conditions**" definition block is employed to specify Boundary conditions, Prescribed velocities, and other dependencies between master and slave nodes groups.

Enable the "Kinematic conditions" properties.

Click the "**Boundary conditions**" tab and enable this condition.

Add lines in the list to specify the BCs that fix the steel frame. Only the vertical parts of the frame are constrained since the horizontal parts will be later attached to them.



A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

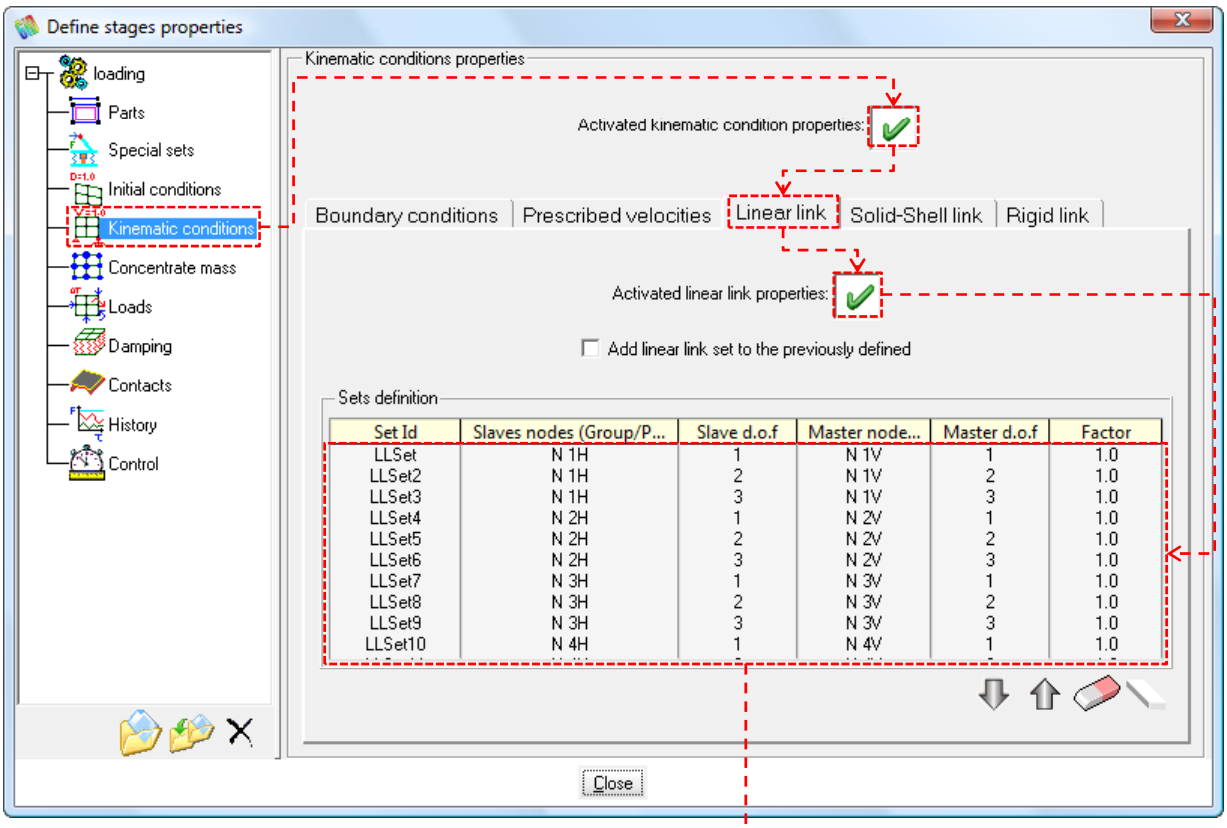
C) Calculate
D) Post-processor

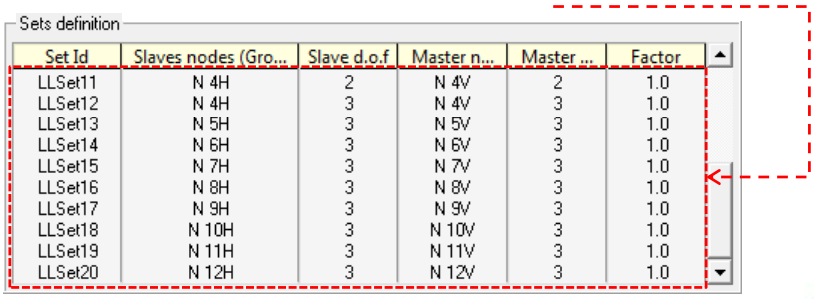
The "**Kinematic conditions**" definition block is employed to specify Boundary conditions, Prescribed velocities, and other dependencies between master and slave nodes groups.

Enable the "Kinematic conditions" properties.

Click the "**Boundary conditions**" tab and enable this condition.

Add lines in the list to specify the BCs that fix the steel frame. Only the vertical parts of the frame are constrained since the horizontal parts will be later attached to them.





A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

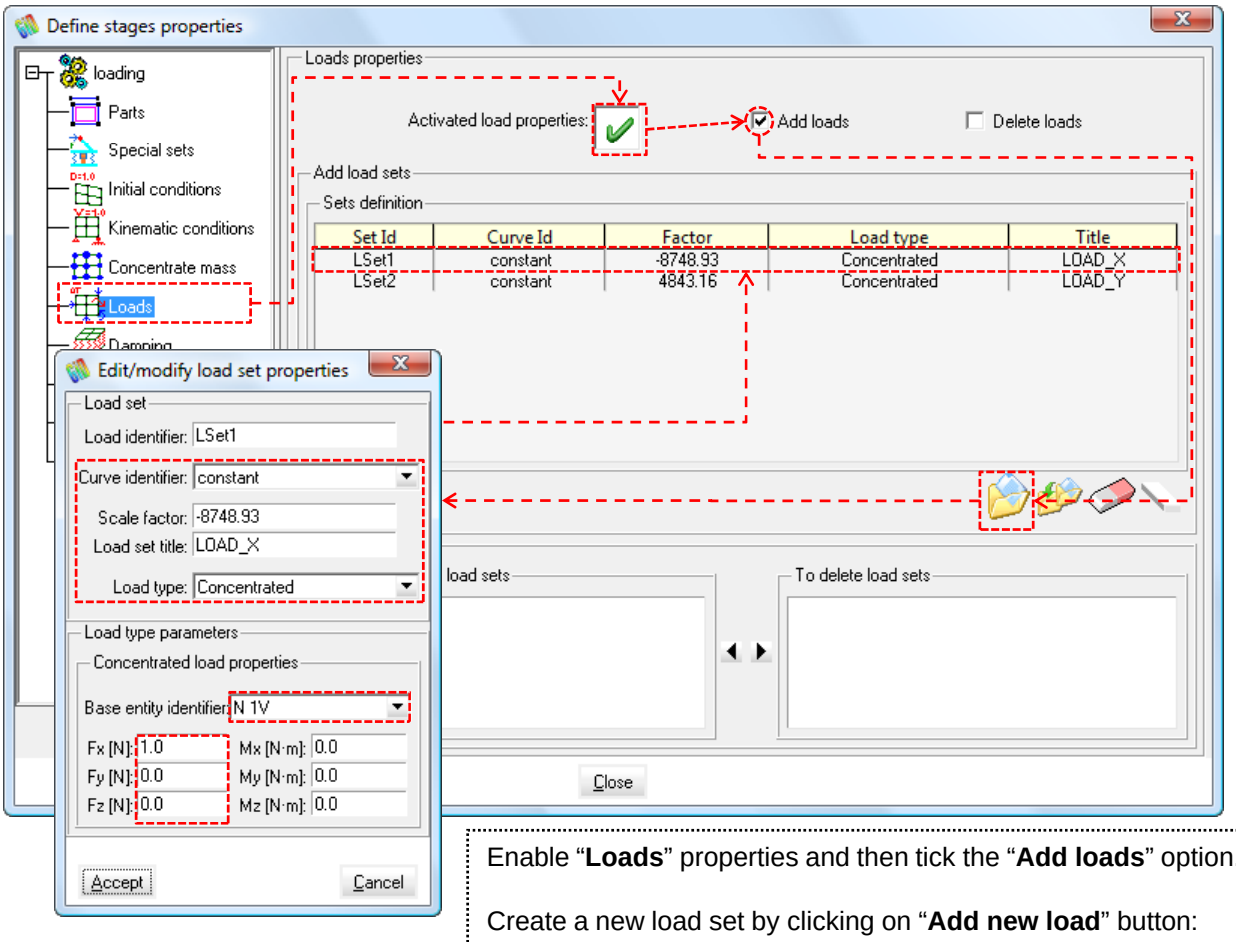
The "**Loads**" definition block is employed to apply loads on specific nodes defined by the user.

The load type may be gravitational or concentrated.

In the first case, volumetric load is applied taking into account the gravitational field defined in the window "General model data…".

In the case of concentrated load, a group or part must be selected and the force or momentum must be defined.

In all cases a curve ID must be selected and a scale factor must be given.



Enable "**Loads**" properties and then tick the "**Add loads**" option.

Create a new load set by clicking on "**Add new load**" button:
Then select the "**constant**" curve and provide the factor **-8745** [N]
Now select the "**concentrated**" load type and the "**N_1V**" group ID.
A unitary force in **X** direction is given in this case.
Close the window by clicking on the "Accept" button.

A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

To apply the load in Y direction, analogous steps should be followed:

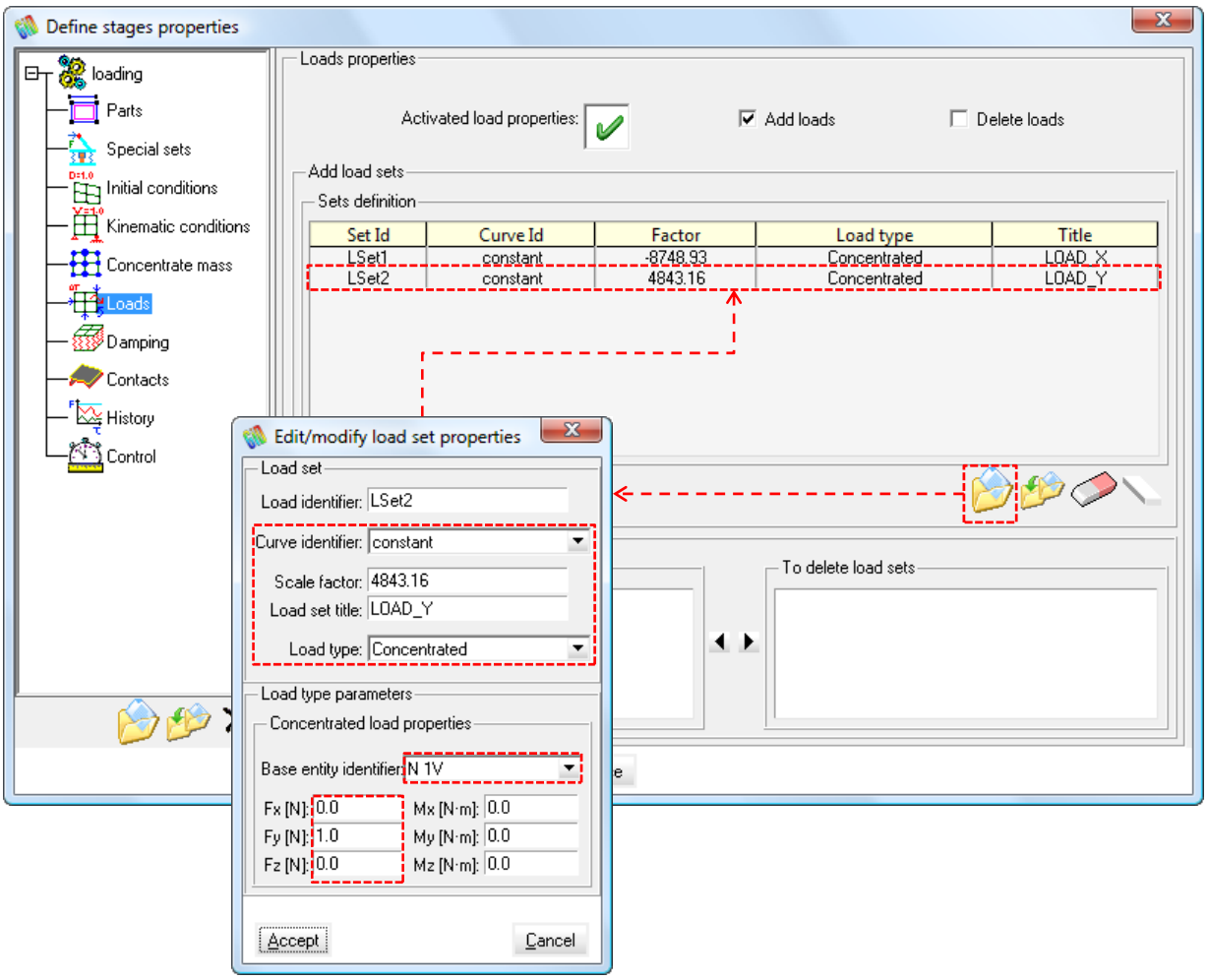
Create a new load set by clicking on "**Add new load**" button:

Then select the "**constant**" curve and provide the scale factor **4843.16** [N]

Now select the "**concentrated**" load type and the "**N_1V**" group ID.

A unitary force in **Y** direction is given in this case.

Close the window by clicking on "Accept".



A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

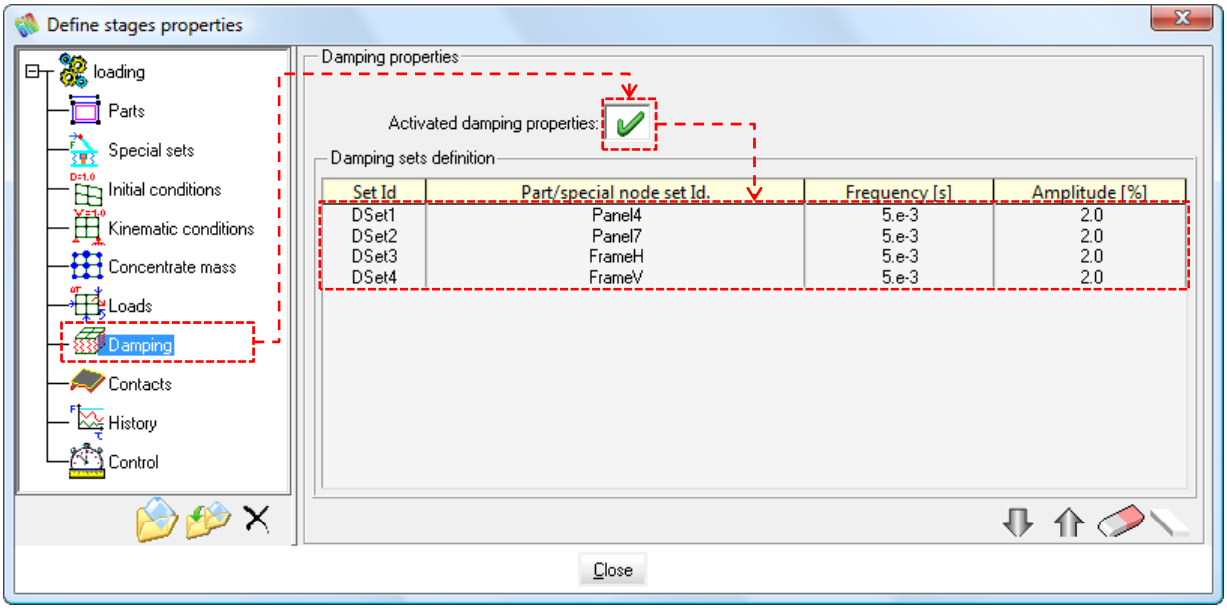
C) Calculate
D) Post-processor

The "**Damping**" definition block is employed to add numerical dumping to the nodes that comprises an specific part.

It is related with the natural frequency of the model.

Enable the "Damping" properties.

Add four lines in the list to assign the following damping to the **Panel** and to the **Frame**:
- ✓ Frequency = 5.0e-3 seconds, and
- ✓ Amplitude = 2 %.



A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

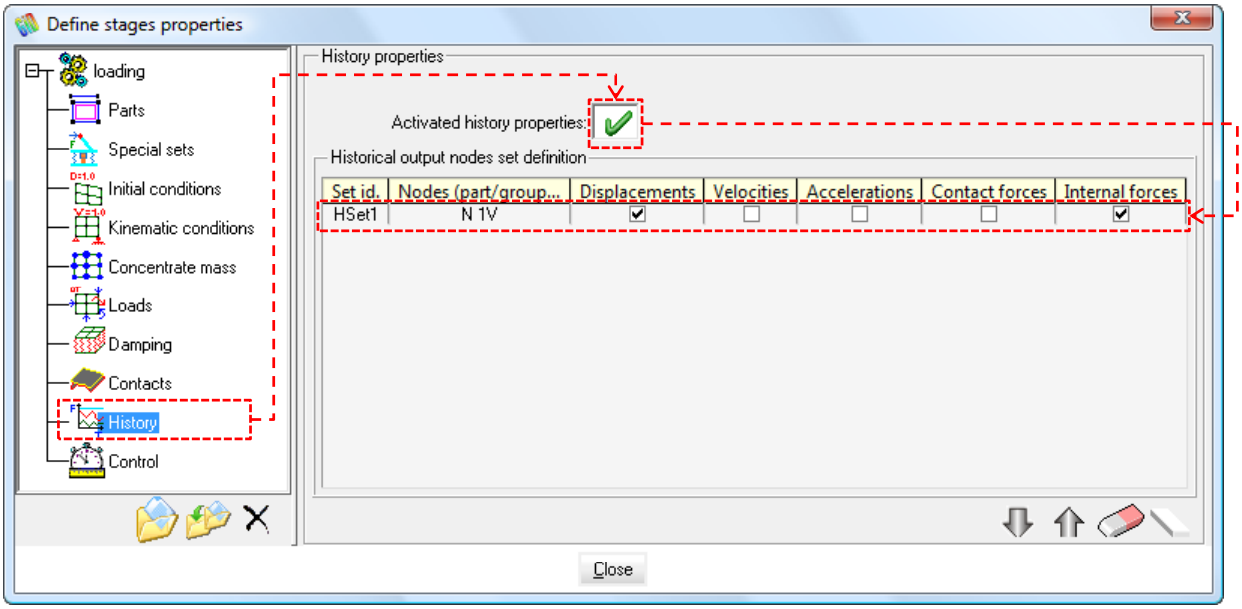
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

The "**History**" definition block is used to indicate which nodal information the user wants to trace in terms of the time.

Enable the "History" properties.

Add a line in the list to indicate that we want to see the evolution in time of the **displacements** and the **internal forces** for the loading node **N_1V**.



A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

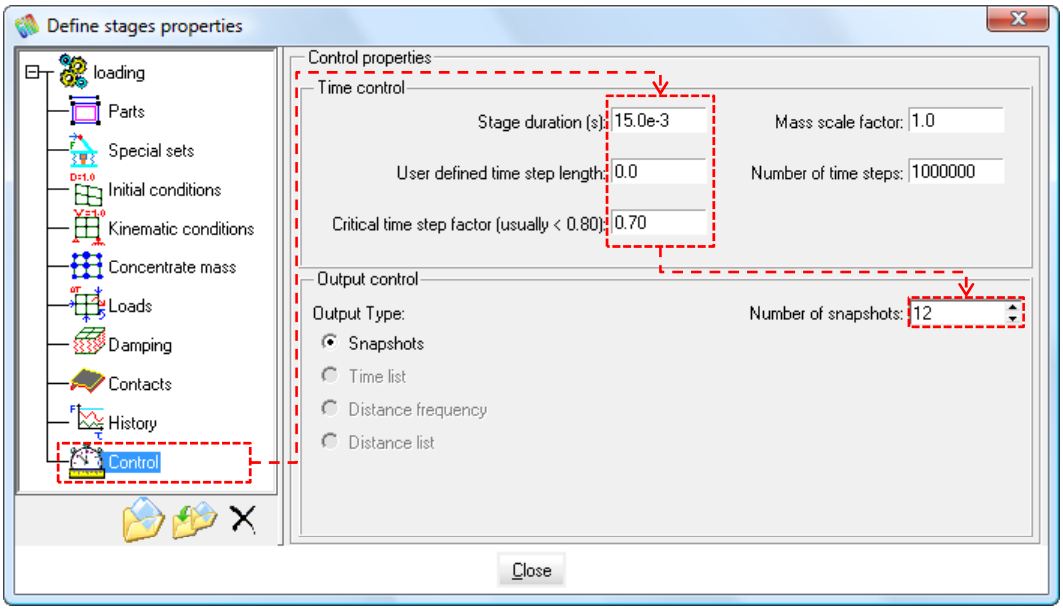
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate
D) Post-processor

The "**Control**" definition block is used to indicate the time and output control parameters.

This block is compulsory for each stage.

The following variables should be defined:
-stage duration,
-mass scale factor,
-user defined time step, (automatic if 0.0)
-number of time steps elapsed before recalculation,
-critical time step factor,
-number of desired snapshots for post-process.



A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

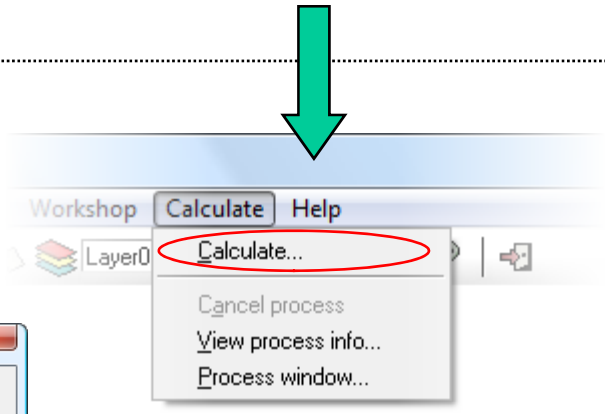
C) Calculate

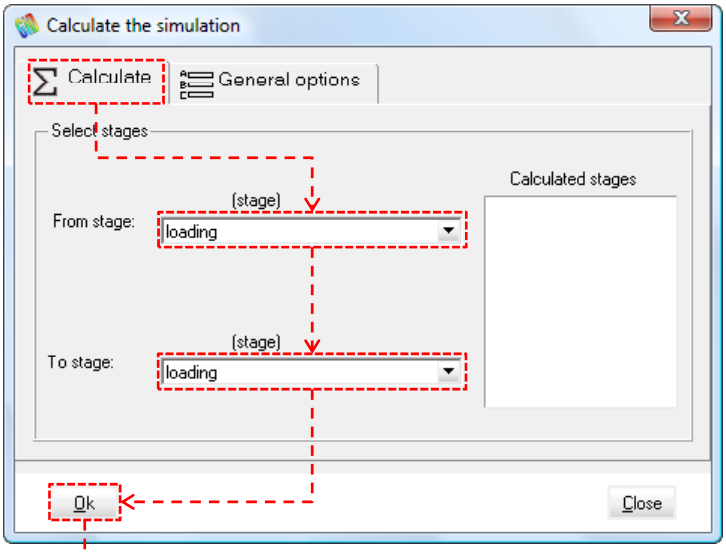
D) Post-processor

The **Calculate** menu is employed to run the model simulation, and to track the evolution of the calculation.

The first option opens the calculation windows to run the analysis.

Workshop>Calculate…

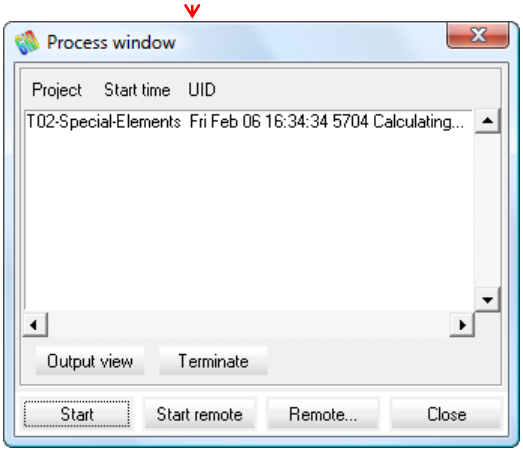




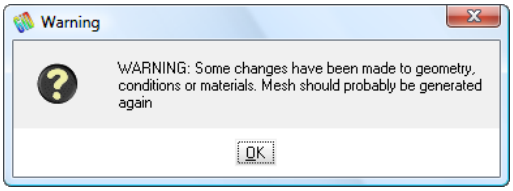
In the **Calculate** tab, select the initial and final stage to start the calculation.

In this case, initial and final stages coincide: "**loading**".

After clicking the "**Ok**" button, the "**Process**" window pops up.



Note: If a window pops up with a warning about the mesh, you should cancel the simulation with the button "Terminate", and then regenerate the mesh. This may happen if you apply some conditions to geometrical entities but you forget to generate a new mesh.



A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

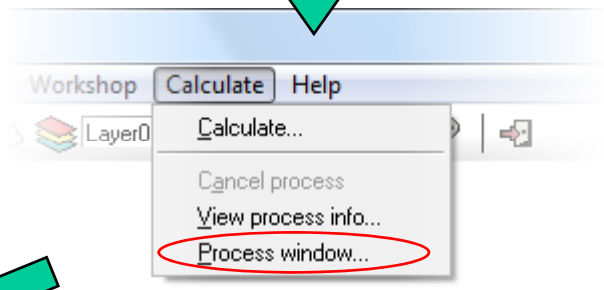
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

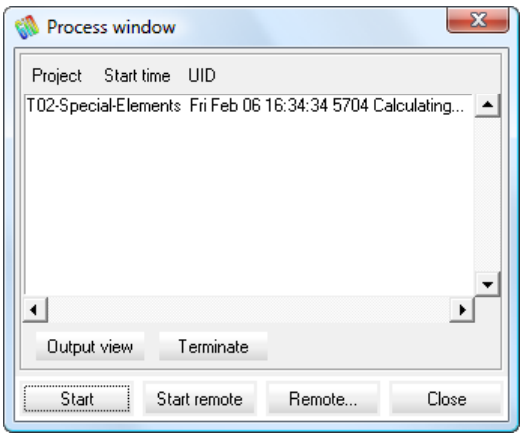
C) Calculate

D) Post-processor

Another way to open the window to track the calculation process is:

Calculate>Process window…

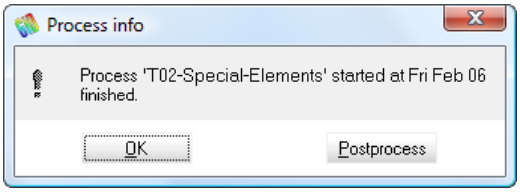




The "**Process**" window allows:

- ✓ to view the output info to track the process evolution with the button "**Output view**".
- ✓ to cancel the simulation process with the button "**Terminate**".

Note: The buttons at the bottom are not functional at the moment.



Once the simulation process is finished, an informative windows pops up.

It allows the switching to the post-process environment. This option is equivalent to: 

Note: If this windows pops up immediately after launching the simulation, an error in the definition of the problem may have been occurred. Please, check that the modelling was correctly defined. If the problem persists, please contact the **FEMCOM** support team.

A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

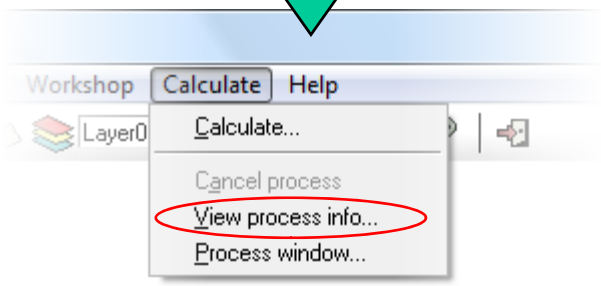
C) Calculate

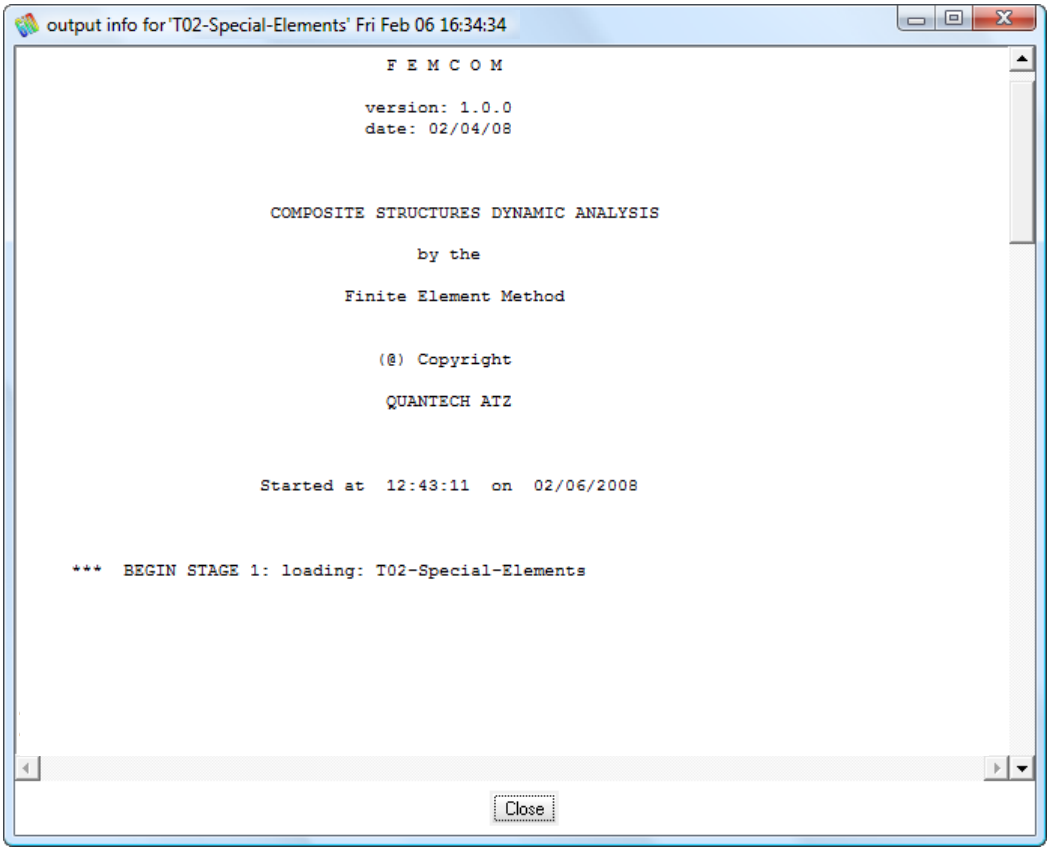
D) Post-processor

The **Output info** may be also displayed with the following option from the menu:

Calculate>View process info…

It is useful to track the evolution of the simulation and to have an estimation of the remaining CPU time to end the stage.





A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate

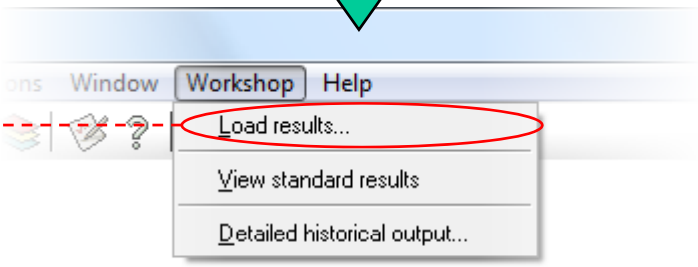
D) Post-processor

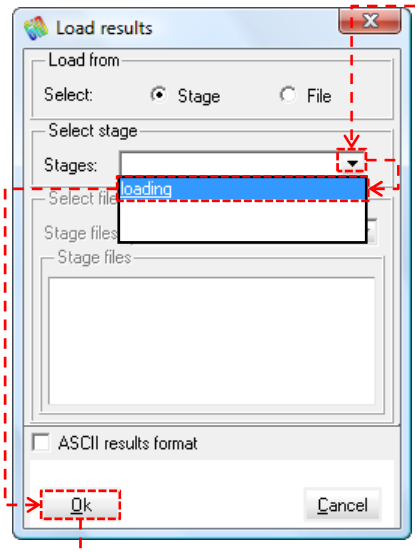
The results may be visualized either at the end of the calculation or while the simulation is in progress. To perform the visualization you must toggle to the **Post process environment**:

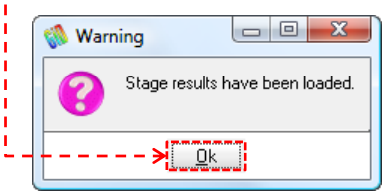
Now, the options in the workshop menu are related to the post-processing.

The first option is employed to load the results:

Workshop>Load results…







After the loading of the results, you will be ready to analyse the simulation results.

A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

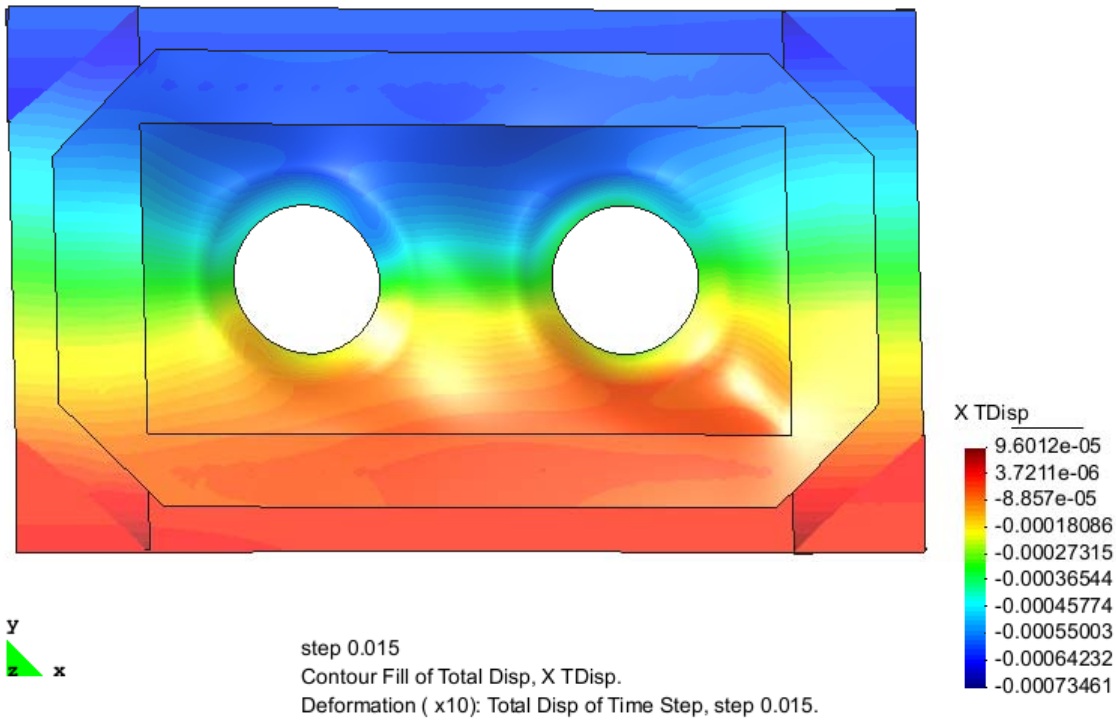
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...
- B.8) Special sets
- B.9) Element orientation

C) Calculate

D) Post-processor

The stage results have been loaded and the results can now be visualized.

NOTE: Depending on the **FEMCOM** version the results selection/visualization may differ.



Total displacement in X direction (m):

View results> Contour Fill> Total Disp> X TDisp