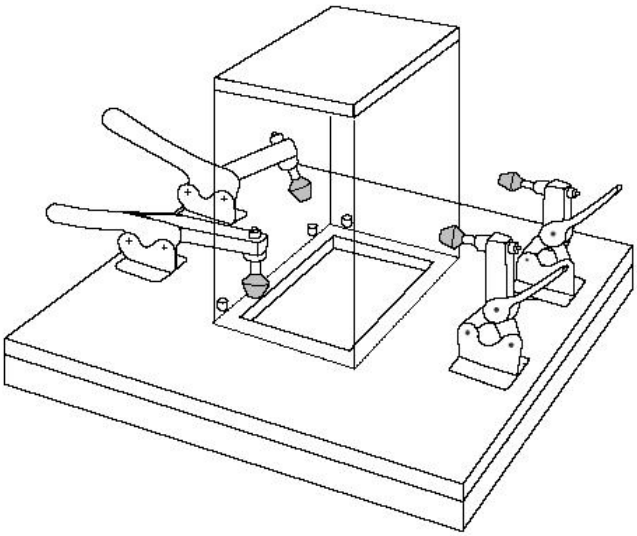
CAI test analysis with 3D solid model



This tutorial provides general details to perform a CAI test (compression after impact test) simulation employing a 3D solid model. The basic methodology described here may be applied to more complex processes. Some details are not explained here since they may be found in the user manual or in the help online.

Important aspects: - setting of the problem (material, BC's and load definition)

- operation and stage definition
- running the simulation
- post-processing

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...

C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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

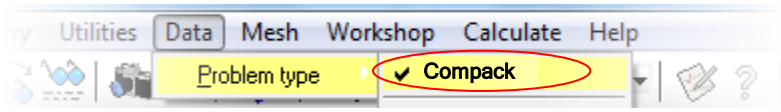
This tutorial presents the basic methodology to be followed to prepare the simulation of the CAI test (Compression After Impact) employing a 3D solid mesh. The same methodology may then be applied to more complex geometries or processes.

COMPACK is able to carry out many CAD functions (see the on-line Help), but it is not generally recommended for the creation of complex CAD models. Usually they are created externally and then imported into **COMPACK**.

In this page it is explained the use of geometry layers and a brief comment on how to import the geometries when they are previously created and stored in IGES format.

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

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



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)

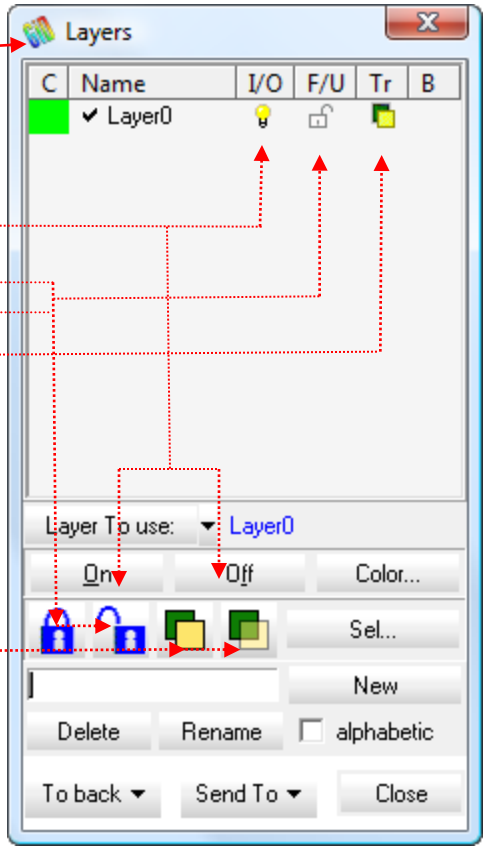
Rename: change the name of the selected layer.

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.

Prior to reading the IGES files create the necessary layers.



IMPORTANT NOTE: When importing IGES files, the user has the option to decide the procedure to be used by **COMPACK**, that is, whether to locate all the geometry inside the active layer or to distribute it between several layers, automatically created. More details may be found in online help.

CIMNE

QUANTECH ATZ

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...

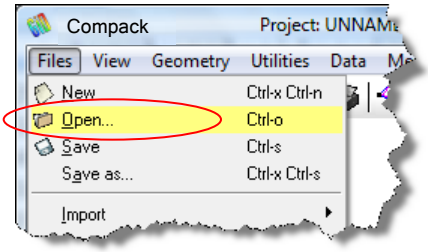
C) Calculate

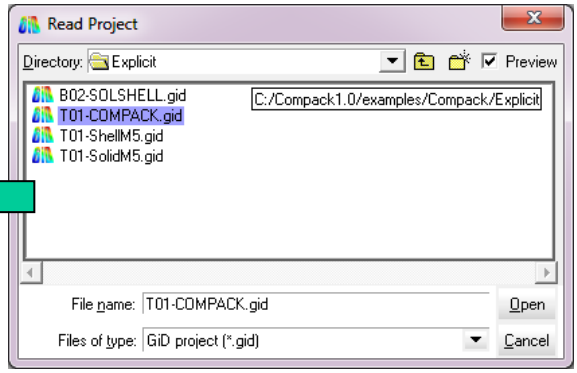
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

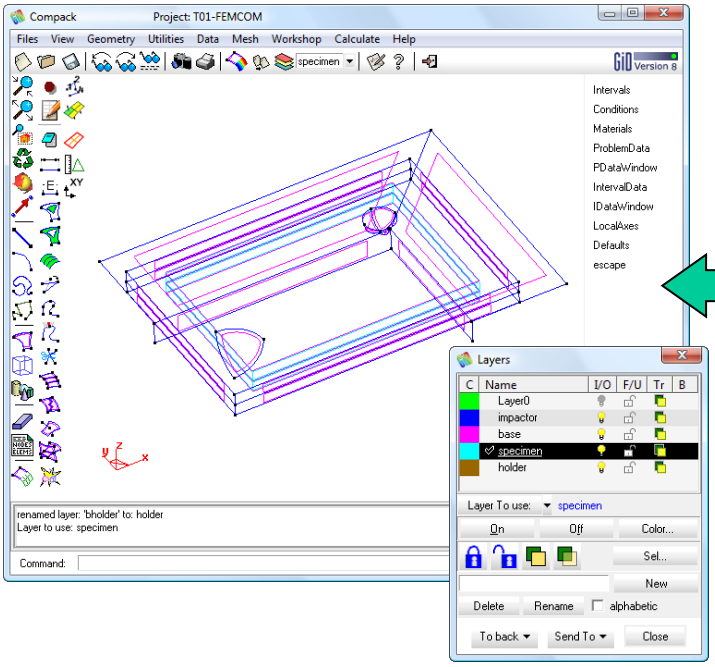
In this tutorial a previously prepared example will be used so the user must directly open a project where the necessary geometries have already been created.

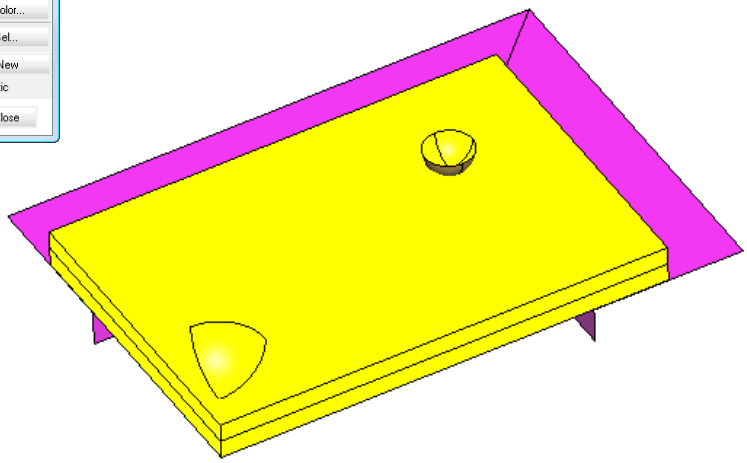
In order to do so, use **Files** menu to read the project **T01-FEMCOM.gid**:

Files>Open









Once the project has been read, check for the correct orientation of the normals.
All normals must be oriented towards the specimen (deformable body).

This is very important for the contact definition between different parts.

Yellow color will always indicate the backside.

View>Normals>Surfaces>Colored

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...

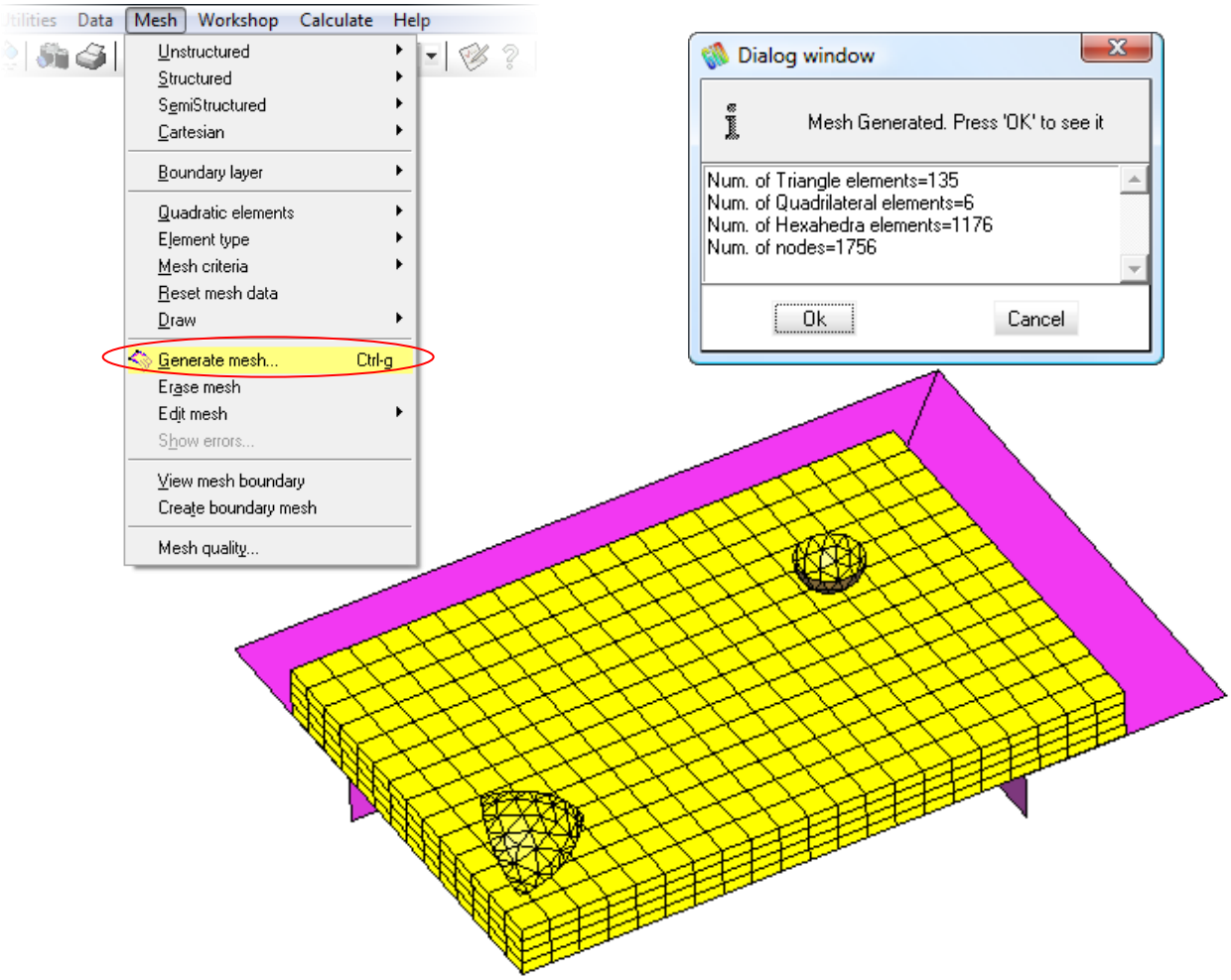
C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

COMPACK uses the Finite Element Method (FEM) for its calculations and therefore a finite element mesh must be created.

No details about mesh generation will be given in this tutorial. Users should consult **the online GiD tutorials** to learn about mesh generation in the **COMPACK** system.

Note: Alternatively, the mesh may be generated in an external software, and later imported into the project. The import mesh process is detailed in the following page, but you may skip it now and continue with the tutorial.



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...

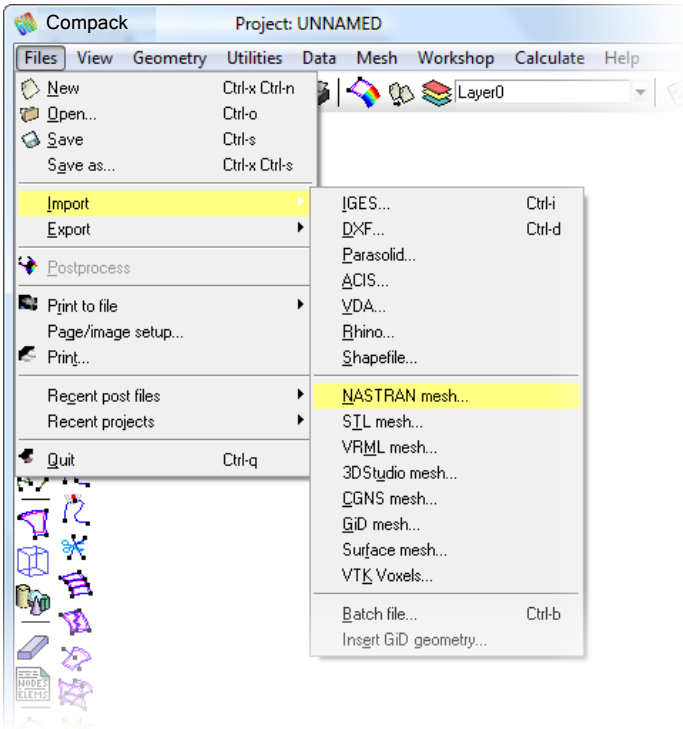
C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

The user may prefer creating the mesh externally and then importing it into **COMPACK**.

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. We use the **COMPACK** system to create the mesh (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 **COMPACK** 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 **COMPACK** 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...

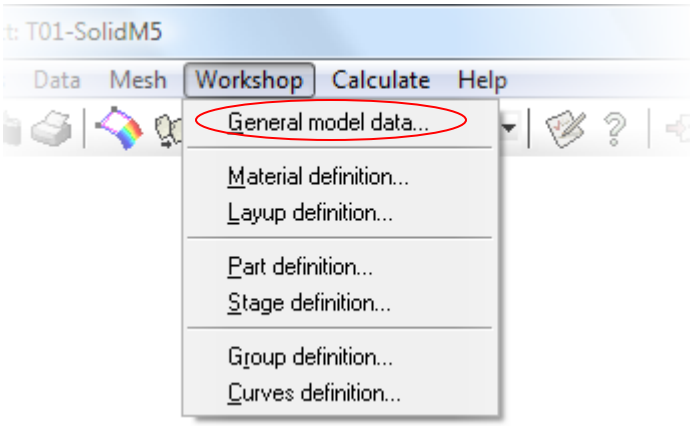
C) Calculate

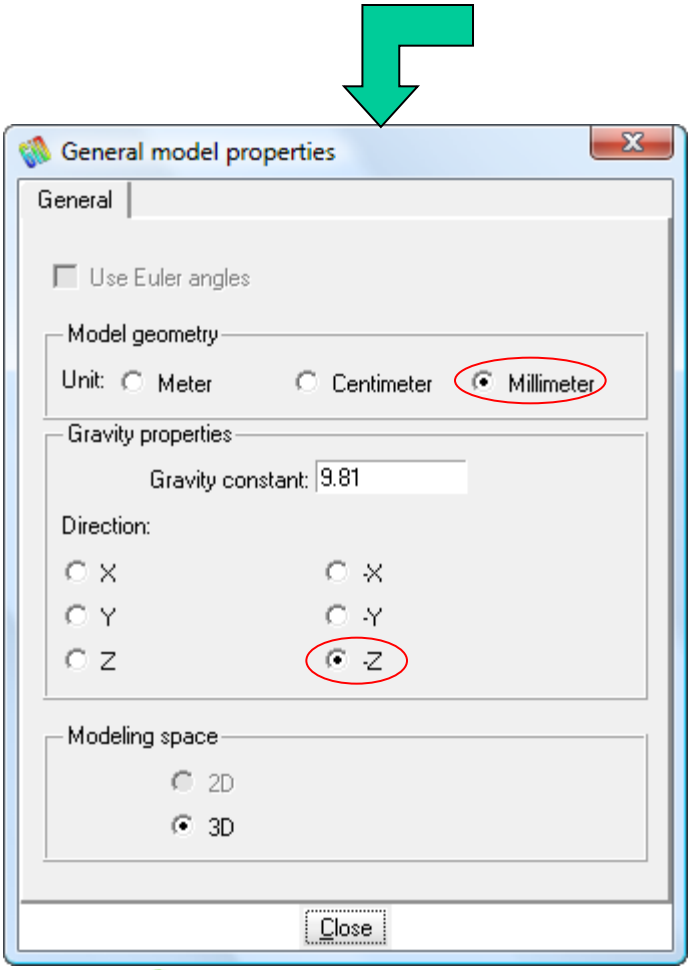
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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

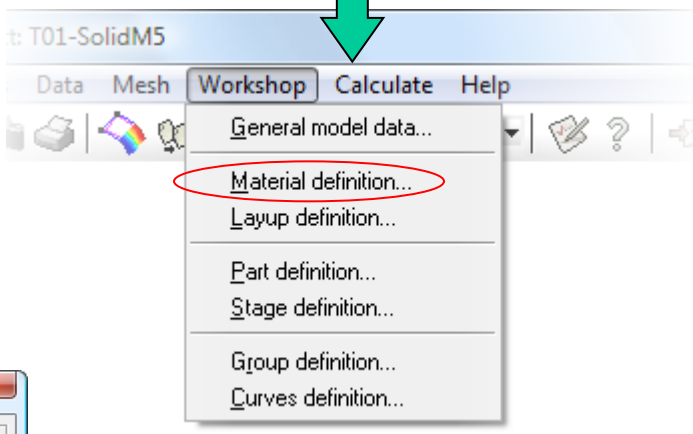
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...

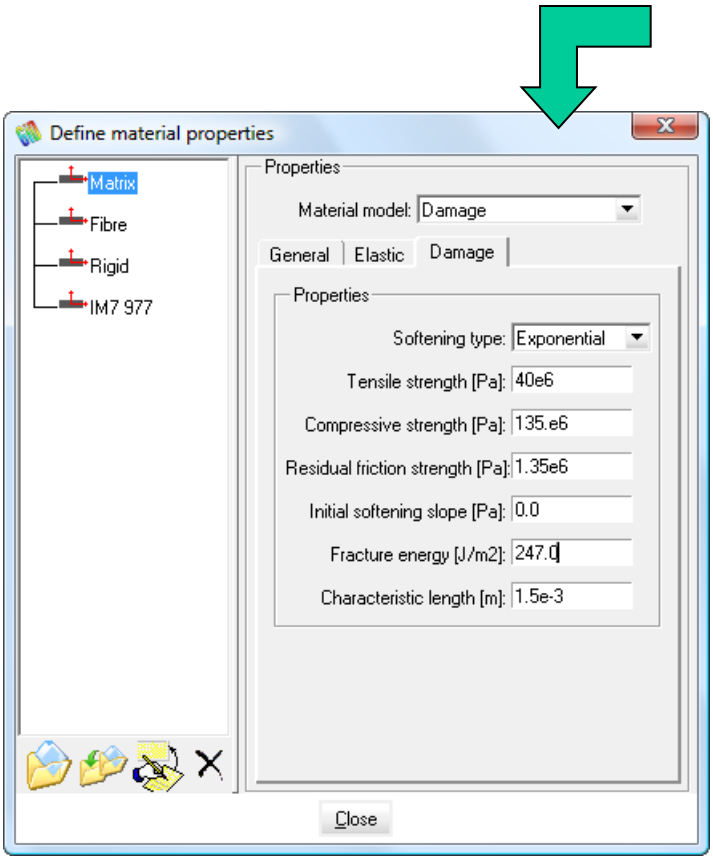
C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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

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...

C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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**. They correspond to the CYCOM977-2 epoyy resin and to the IM7 carbon fiber.

Matrix material

General | Elastic | Damage

Properties

Density: 1150.0

General | Elastic | Damage

Properties

Young's modulus: 7900e6

Poisson's ratio: 0.30

General | Elastic | Damage

Properties

Softening type: Exponential

Tensile strenght: 40e6

Initial softening slope: 0.0

Compressive strenght: 135e6

Fracture energy: 738.

Residual friction strenght: 0.0

Characteristic length: 1.5e-3

Fiber material

General | Elastic | Damage

Properties

Density: 1800.0

General | Elastic | Damage

Properties

Young's modulus: 292000e6

Poisson's ratio: 0.220

General | Elastic | Damage

Properties

Softening type: Linear

Tensile strenght: 5000e6

Initial softening slope: 0.0

Compressive strenght: 1648e6

Fracture energy: 0.0

Residual friction strenght: 0.0

Characteristic length: 0.0

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...

C) Calculate

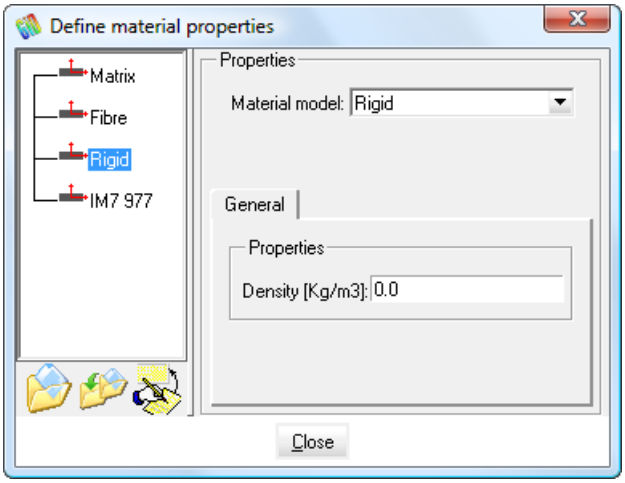
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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

the **rigid material** and,

the **UD-composite material**.

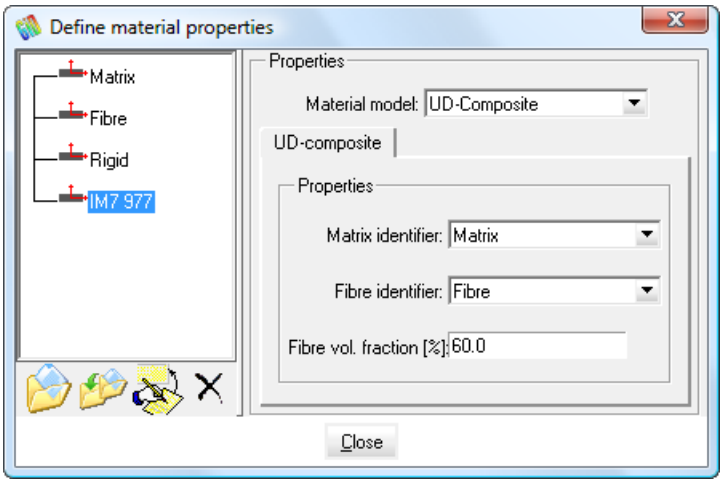
This material types are very useful for the definition of rigid bodies that will impact composite structures.



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 **COMPACK** solver to calculate the mass of the rigid body in terms of its volume.

In the present example a rigid material with zero density is defined because we plan to assign the mass of rigid bodies directly employing the "concentrated mass" option that will be explained later in this tutorial.



The **UD-composite model** must be selected as a constituive 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 fiber together with the fiber 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

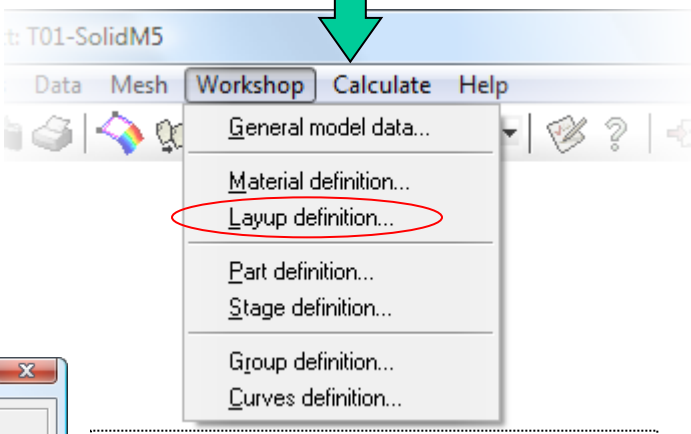
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...

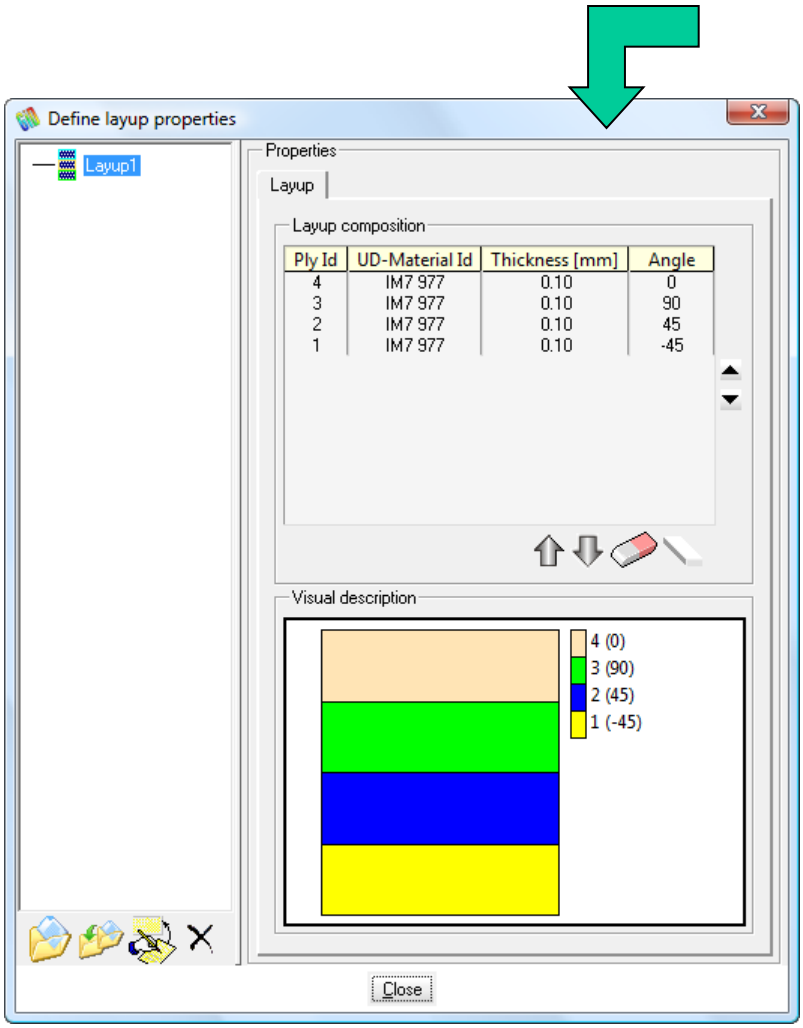
C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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

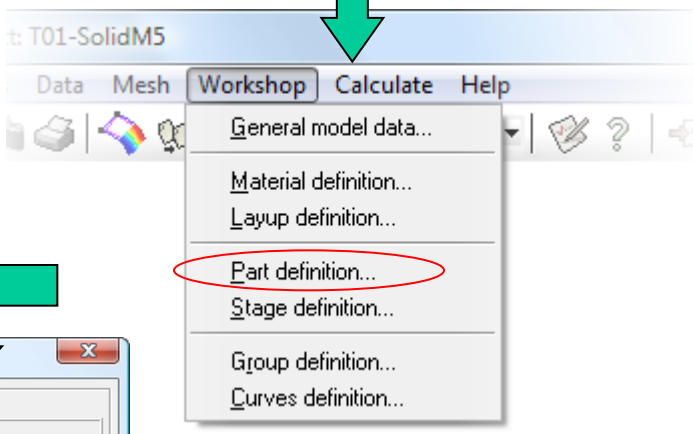
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...

C) Calculate

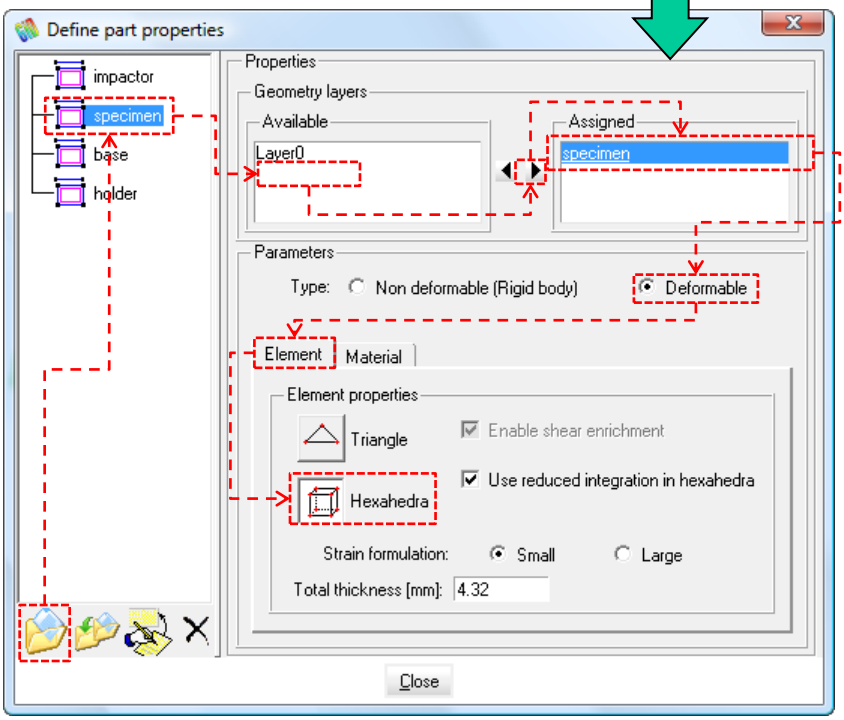
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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…



Rigid and Deformable parts may be defined. Let's see how to define the "**specimen**"…



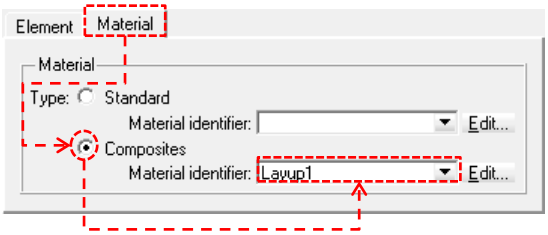
Create a new part definition by clicking on "Add new part" button: 

Type the part name: "**specimen**".

Then select the corresponding layer from the "Available" list of layers and click on ▸ to assign it .

Tick the "**Deformable**" option for this part.

In the "**Element**" tab, select the type of element used to create this part mesh: "**Hexahedra**" 



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

CIMNE

QUANTECH ATZ

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...

C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

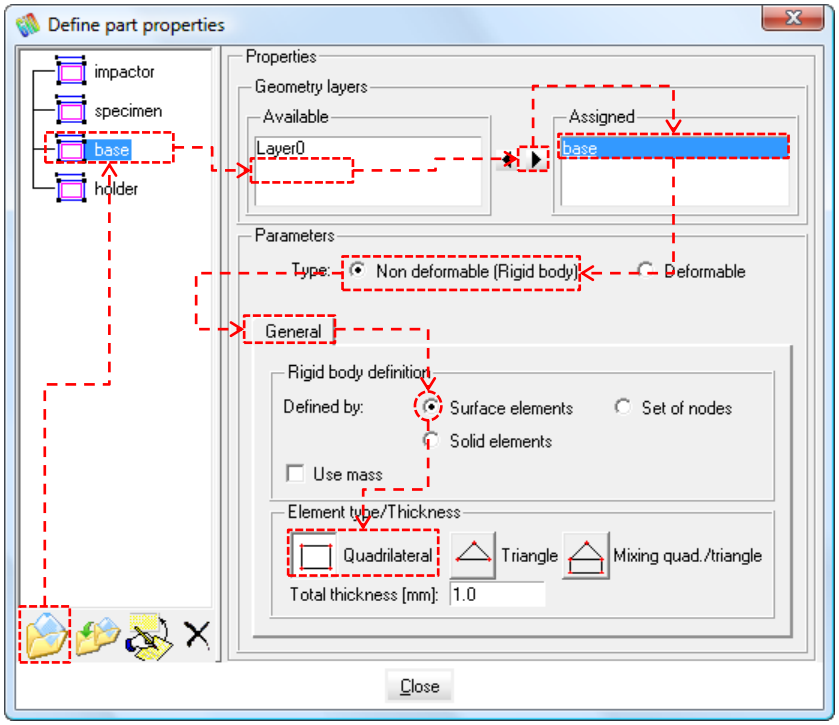
Now, let's see how to define the **Rigid Bodies** "RB" that will interact with the deformable body.

The rigid bodies may be divided in two sub-types: a) "RB without mass" or b) "RB with mass".

a) "**RB without mass**":

Static parts and those parts that will be moved with prescribed velocity do not require mass definition.

In this example, the "**steel base**" is static so the "use mass" option is disabled.



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

Type the part name: "**base**".

Then select the corresponding layer from the "Available" list of layers and click on ' ▶ to assign it .

Tick the "**Non deformable**" option for this part.

In the "**General**" tab, select the type of element used to create this part mesh: "**Quadrilateral**".

Note: The "**Use mass**" option must remain disabled.

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...

C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

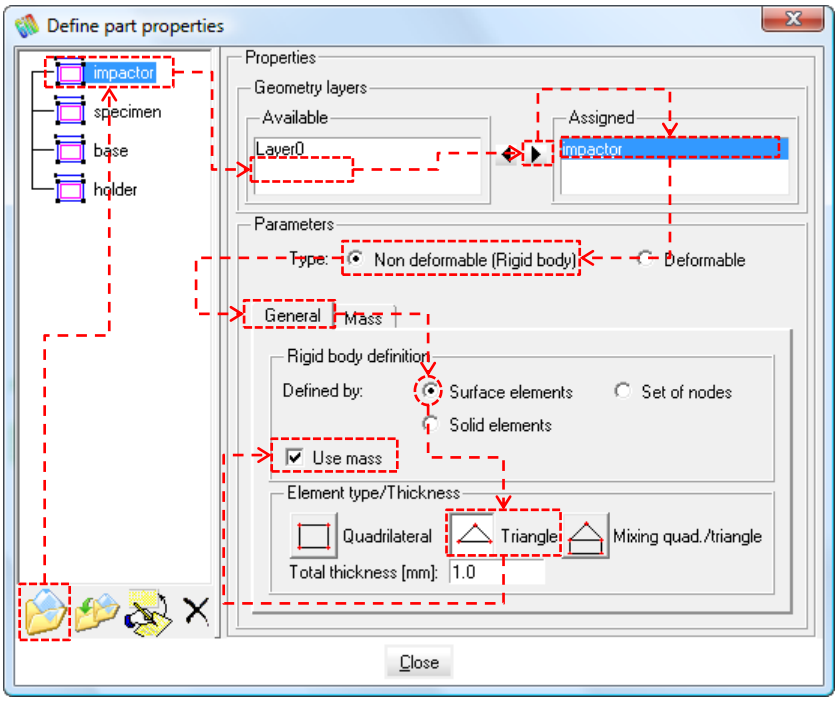
b) "**RB with mass**":

Moving parts require mass definition. "**Use mass**" option must be enabled.

In this example, the "**impactor**" and the "**holder**" parts will require mass definition.

The mass may be defined giving the total mass for the whole body, or by automatic calculation employing the material density and part volume.

Note: Another way to assign mass is through the option "concentrated masses" that will be later explained in this tutorial.



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

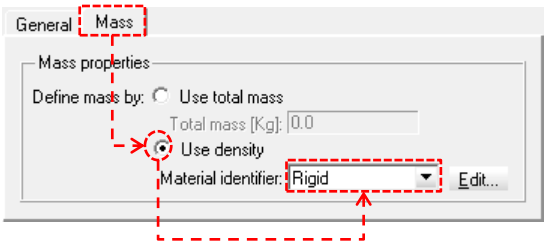
Type the part name: "**impactor**".

Then select the corresponding layer from the "Available" list of layers and click on ' ▶ to assign it .

Tick the "**Non deformable**" option for this part.

In the "**General**" tab, select the type of element used to create this part mesh: "**Triangle**". 

Enable the "**Use mass**" option.



In the "**Mass**" tab, select how the mass will be defined ("**Use density**") and finally select the specific material ID from the list (**Rigid**).

Analogous steps must be followed to define the "**holder**" rigid part.

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

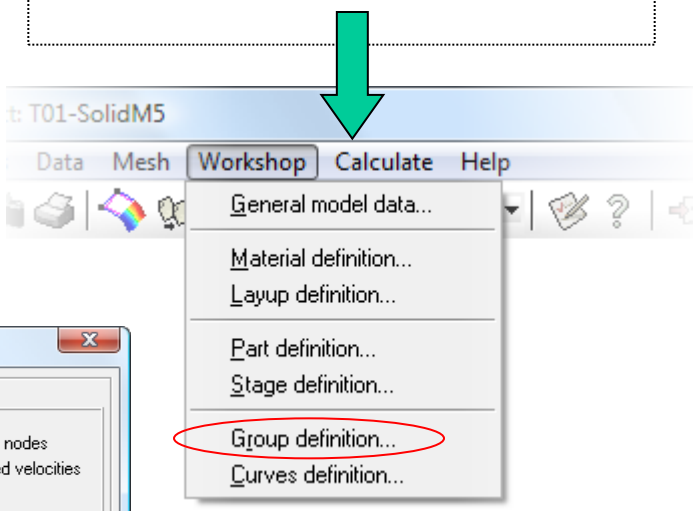
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...

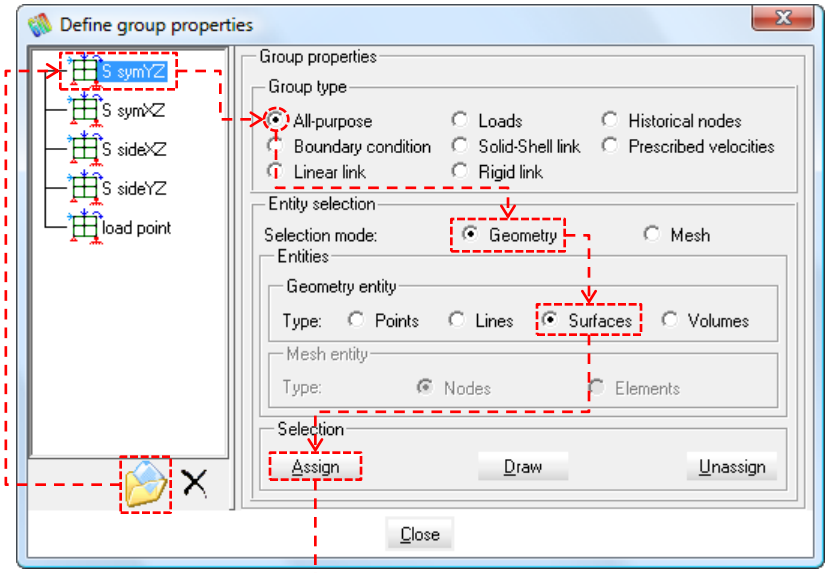
C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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.



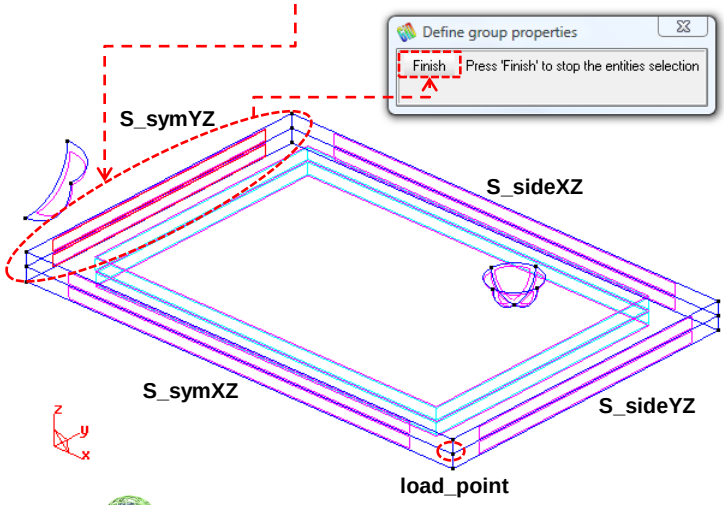


Create a new group by clicking on "Add new group" button:

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

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 "**Geometry**" and "**Surface**" options, and then click "**Assign**" button to select the corresponding surfaces from the model geometry. Press "Finish" or "Esc" to stop entities selection.



Follow analogous steps to create the groups: "**S_symXZ**", "**S_sideXZ**" and "**S_sideYZ**" and "**load_point**".

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...

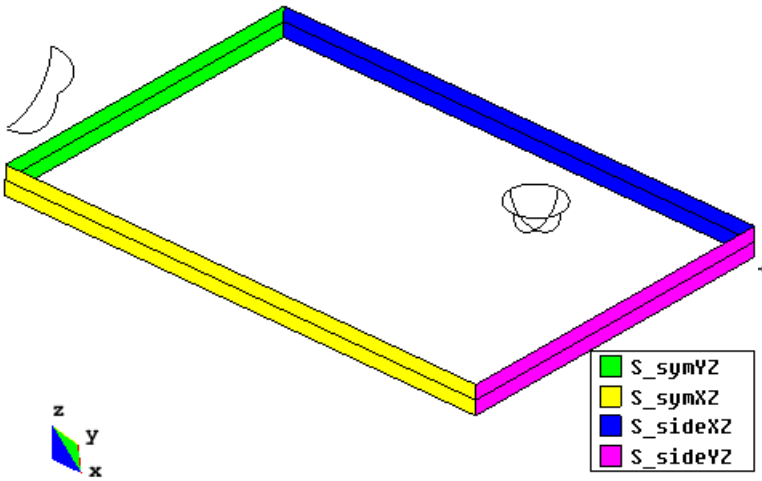
C) Calculate

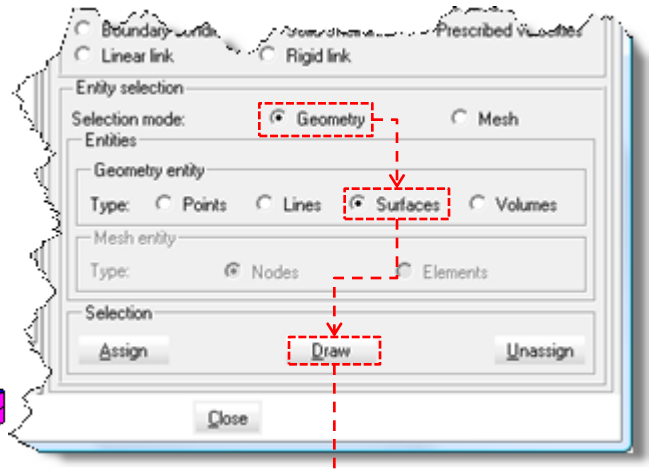
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

To check that groups were appropriately defined, the user may click on the "**Draw**" button.

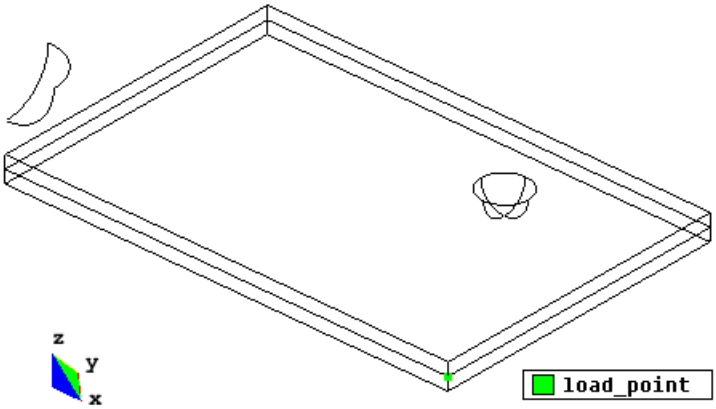
When **Geometry** and **Surfaces** are selected, **COMPACK** will show "surfaces groups" only.

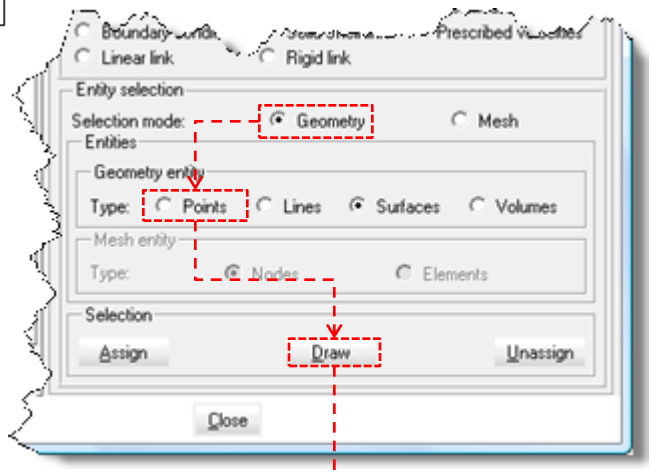
When **Geometry** and **Points** are selected, **COMPACK** will show "points groups" only.





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





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

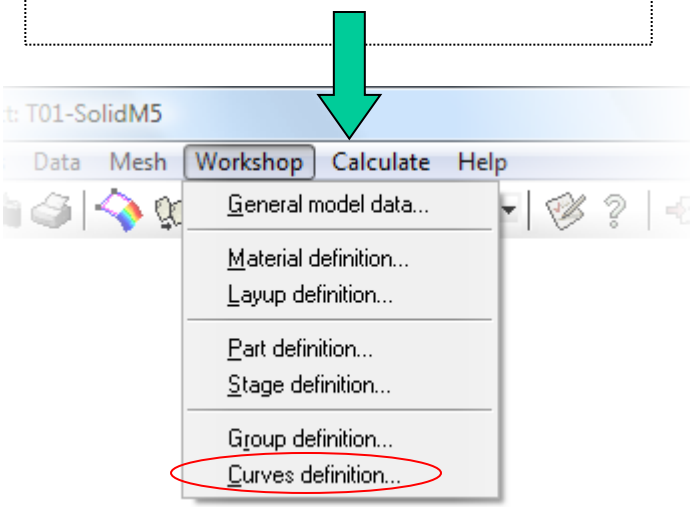
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...

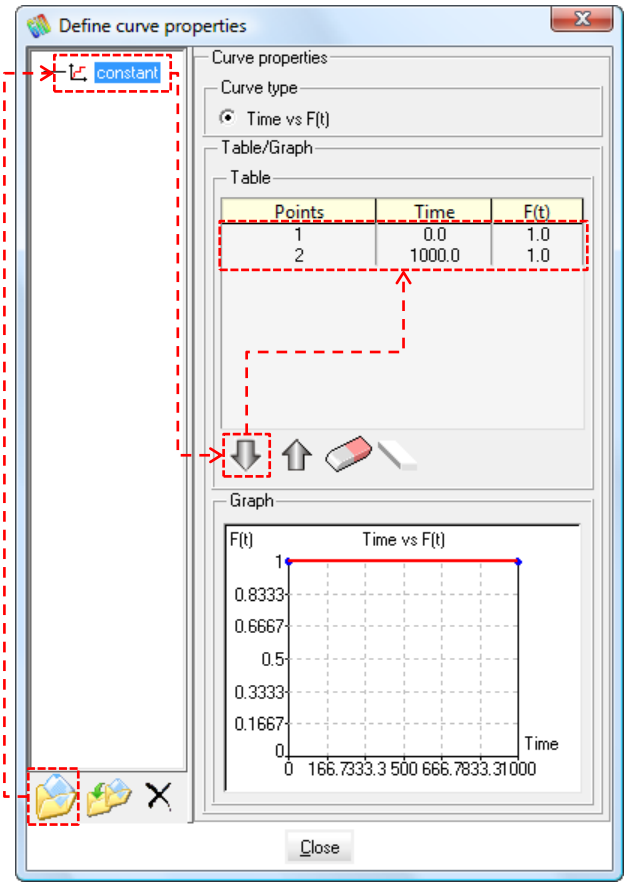
C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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 a constant curve with unitary value.

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

This "constant" curve will be later employed to assign a constant prescribed velocity to a group of nodes.

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

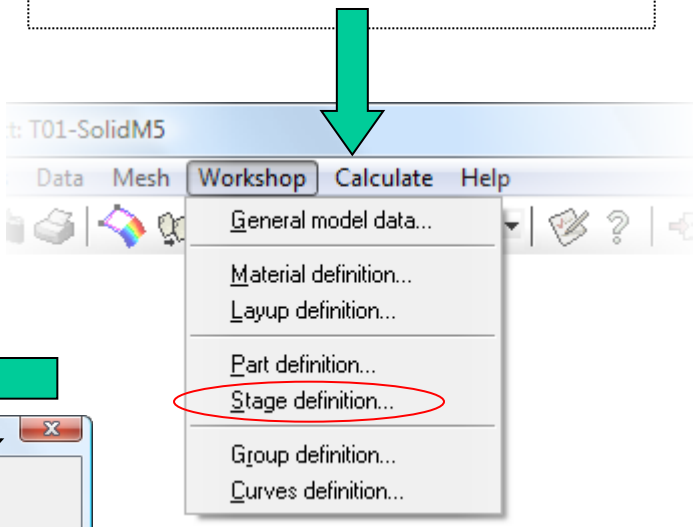
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...

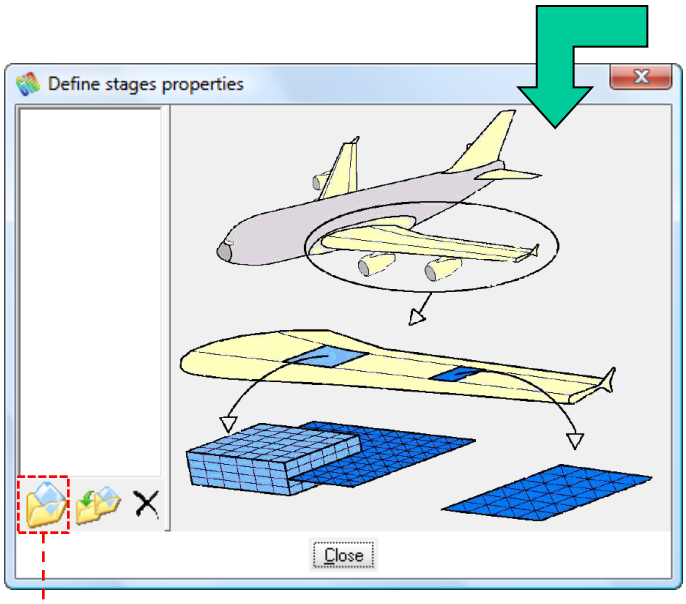
C) Calculate

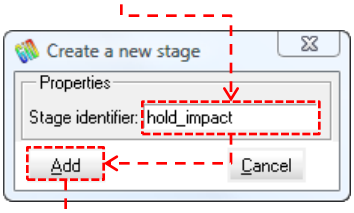
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

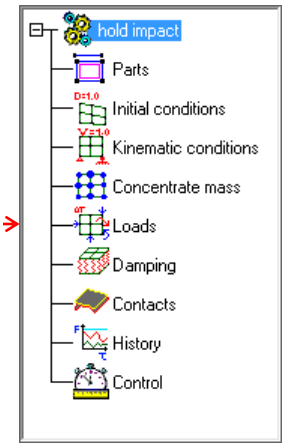
The "**Stage definition…**" option in the **Workshop** menu is the most important, since it will put together all the ingredients to perform the numerical simulation of the industrial process.

Workshop>Stage definition…









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

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

This stage will contain the definition of the "clamping" of the specimen plus the "impact" process.

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.

Add a new stage named "**damping**" that will be employed to quickly release the residual vibrations.

Add another stage: "**compression**" to model the **CAI test**.

CIMNE

QUANTECH ATZ

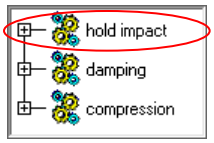
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

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...

C) Calculate

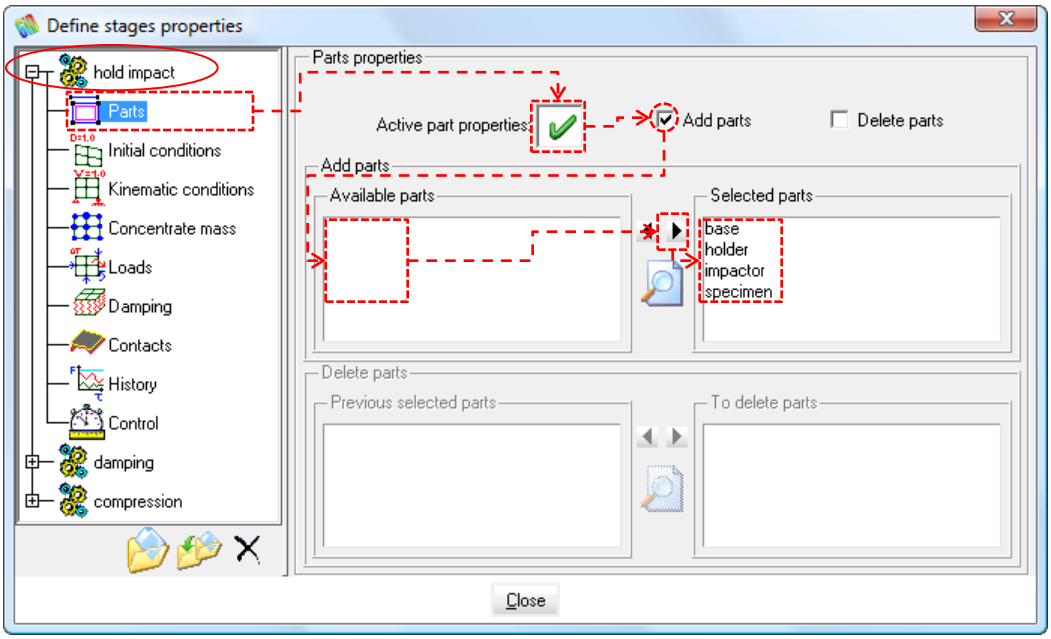
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

Let's see how to describe the "**hold_impact**" stage.



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 that we want to include (**base, holder, impactor, specimen**) 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...

C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

The "**Initial conditions**" definition block is to apply an initial displacement or velocity to a set of parts.

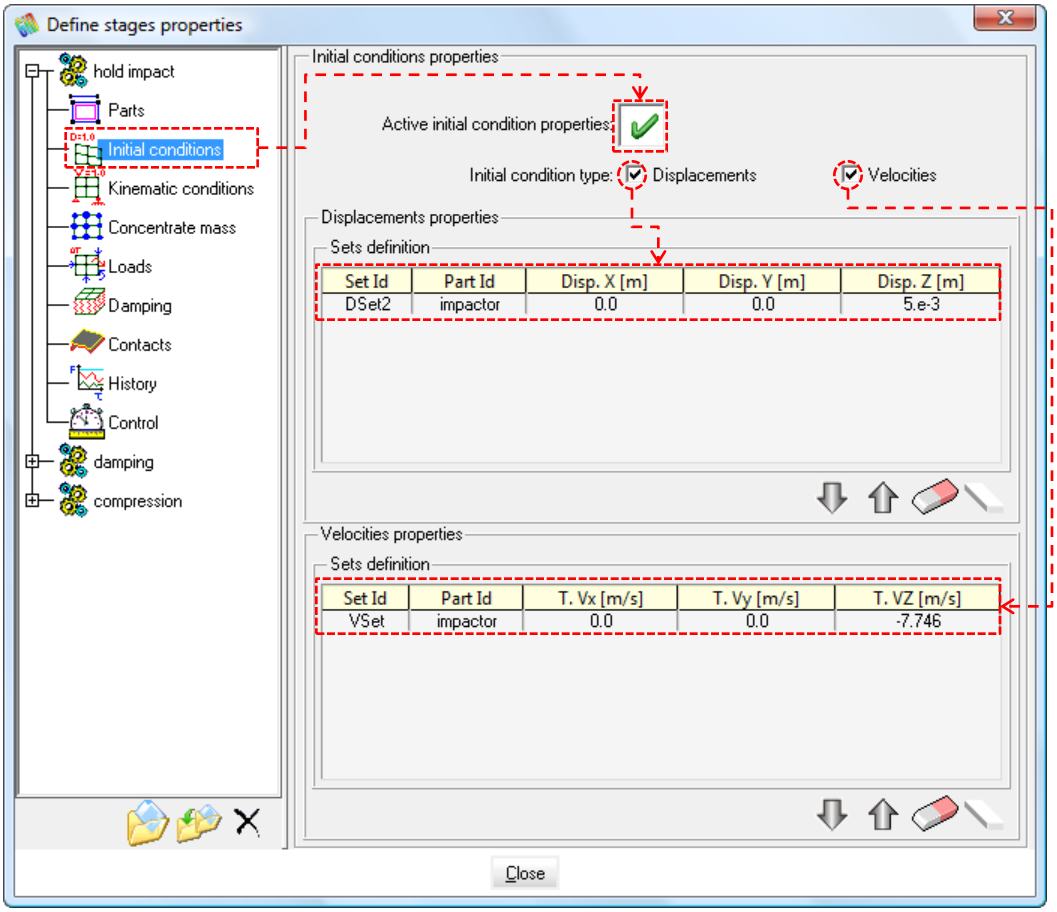
Initial displacements are useful to move a part without having to regenerate the mesh.

Initial velocities are employed to start the free moving of a part. For example, in an impact test.

Enable the "Initial conditions" properties.

Tick "**Displacement**" and move the impactor 5 mm in positive Z direction.

Tick "**Velocities**" and define the velocity of the impactor with a value of 7.746 m/s in negative Z direction.



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...

C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

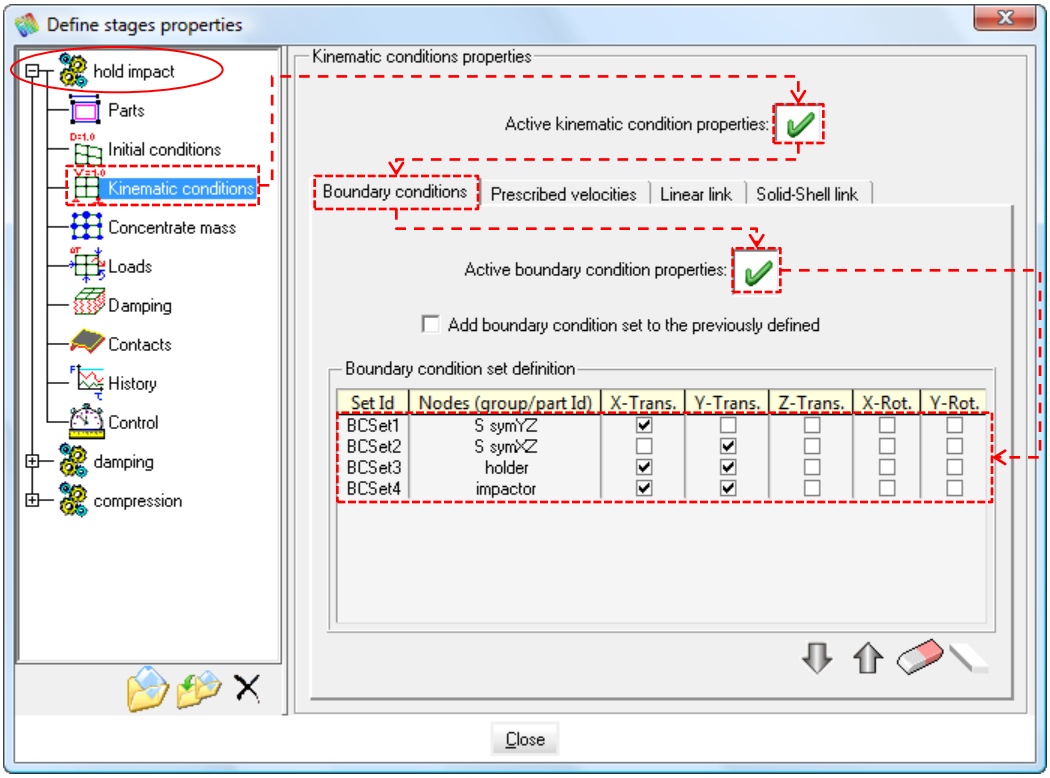
The "**Kinematic conditions**" definition block is employed to specify Boundary conditions, Prescribed velocities, and other dependencies between master and slave nodes groups.

Enable the "Kinematic conditions" properties.

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

Add lines in the list to specify the following BCs:

- ✓ Symmetry condition on planes YZ and XZ.
- ✓ Movement of the holder and the impactor is only allowed in Z direction.



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...

C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

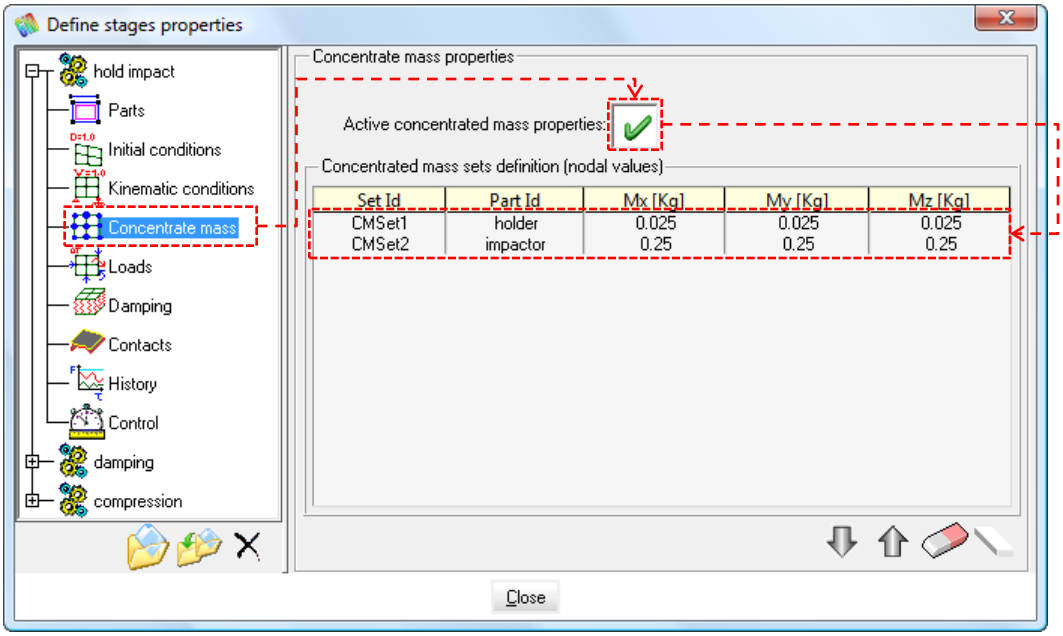
The "**Concentrate mass**" definition block is to assign mass on specific nodes defined by the user.

It is useful to define the mass of those parts that do not have mass already assigned beforehand.

Enable the "Concentrated mass" properties.

Add two lines in the list to assign the following masses:

- ✓ 0.25 Kg to the impactor (1/4 of the geometry),
- ✓ 0.025 Kg of mass to the holder.



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...

C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

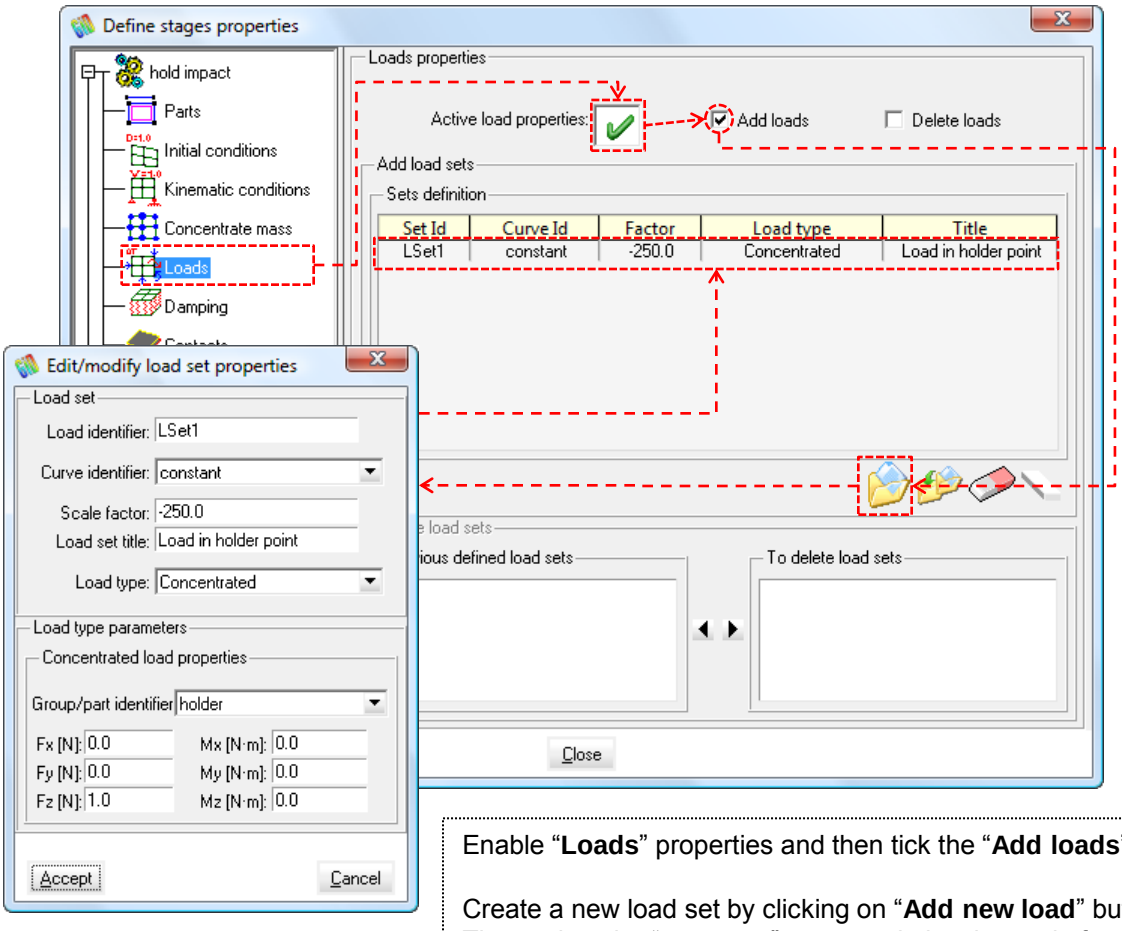
The "**Loads**" definition block is employed to apply loads on specific nodes defined by the user.

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 give the scale factor -250 [N]
Now select the "**concentrated**" load type and the "**holder**" part ID.
A unitary force in Z 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...

C) Calculate

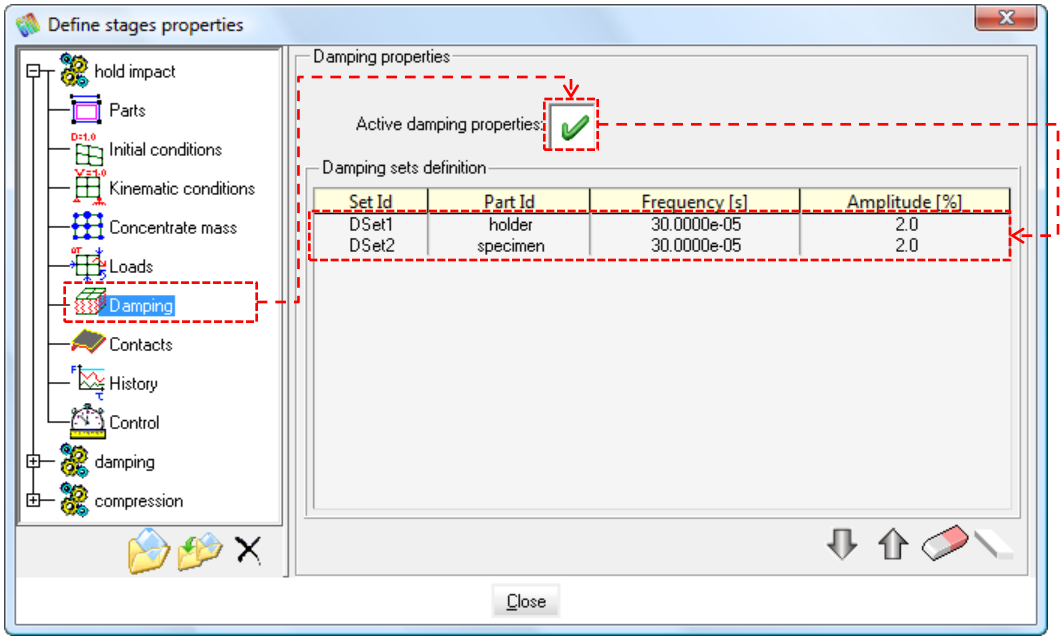
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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 two lines in the list to assign the following damping to the **holder** and to the **specimen**:
- ✓ Frequency = 30.0e-5 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...

C) Calculate

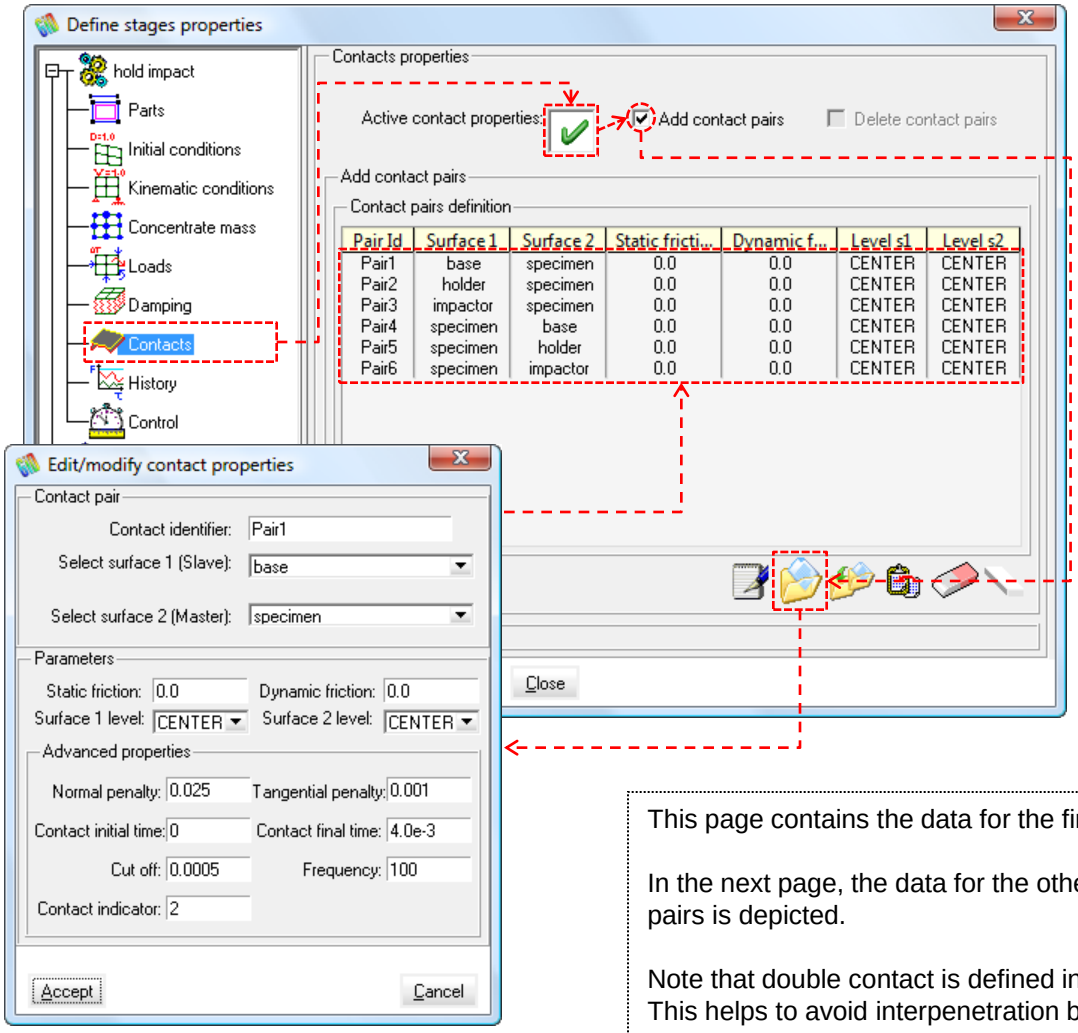
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

The "**Contacts**" definition block is employed to set up the set of contact pairs and their properties.

Enable the "Contacts" properties, and tick the "**Add contact pairs**" option.

Click on "**Add new contact pair**" button:  to create a new contact pair definition.

Select "**base**" as slave and "**specimen**" as master. Beware of employing the appropriate contact indicator in accordance to the following:
Use 0 if both surfaces contain mass defined.
Use 1 if only surface 1 (slave) has mass defined.
Use 2 if only surface 2 (master) has mass defined.



This page contains the data for the first contact pair.

In the next page, the data for the other five contact pairs is depicted.

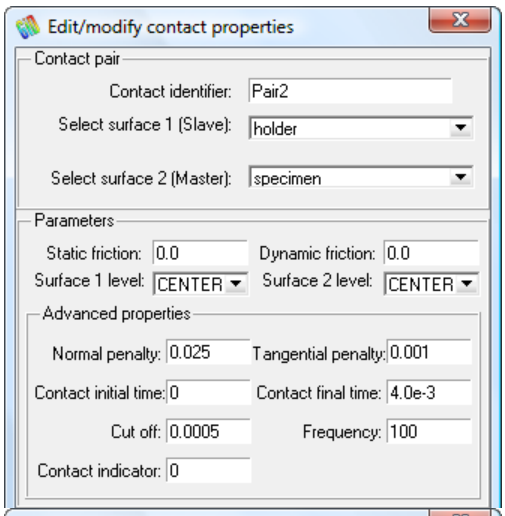
Note that double contact is defined in this example. This helps to avoid interpenetration between parts.

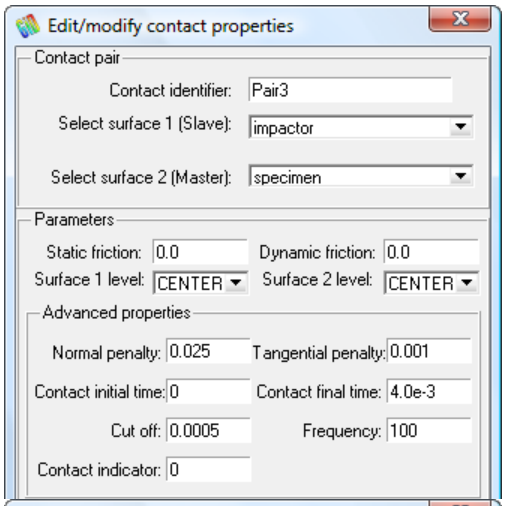
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

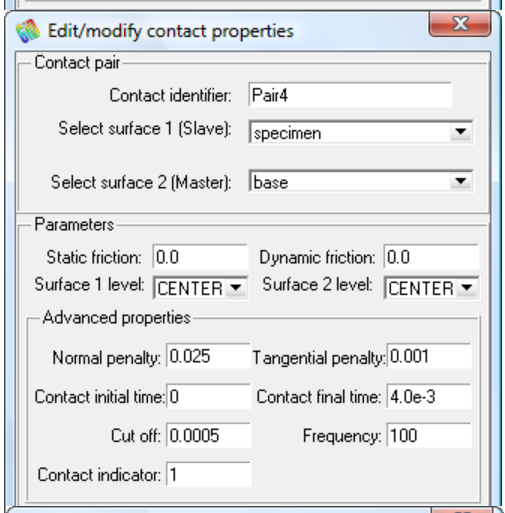
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...

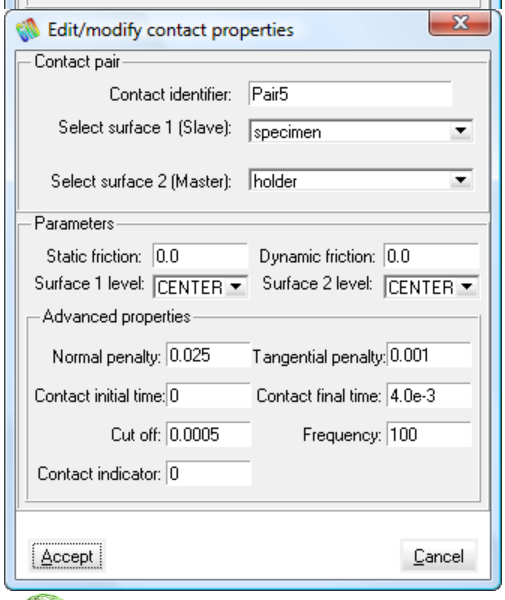
C) Calculate

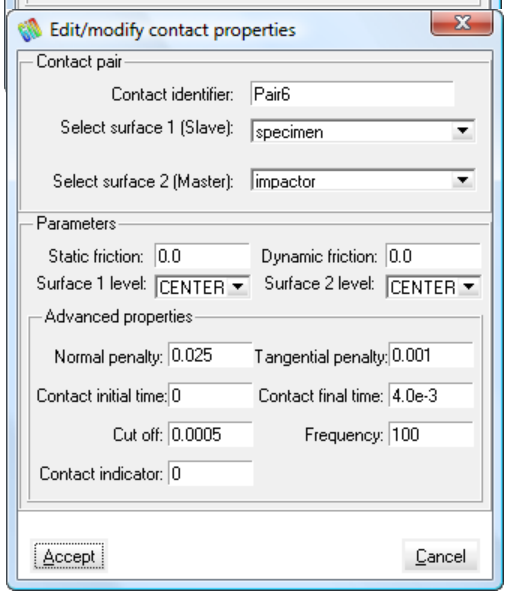
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output











CIMNE

QUANTECH ATZ

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...

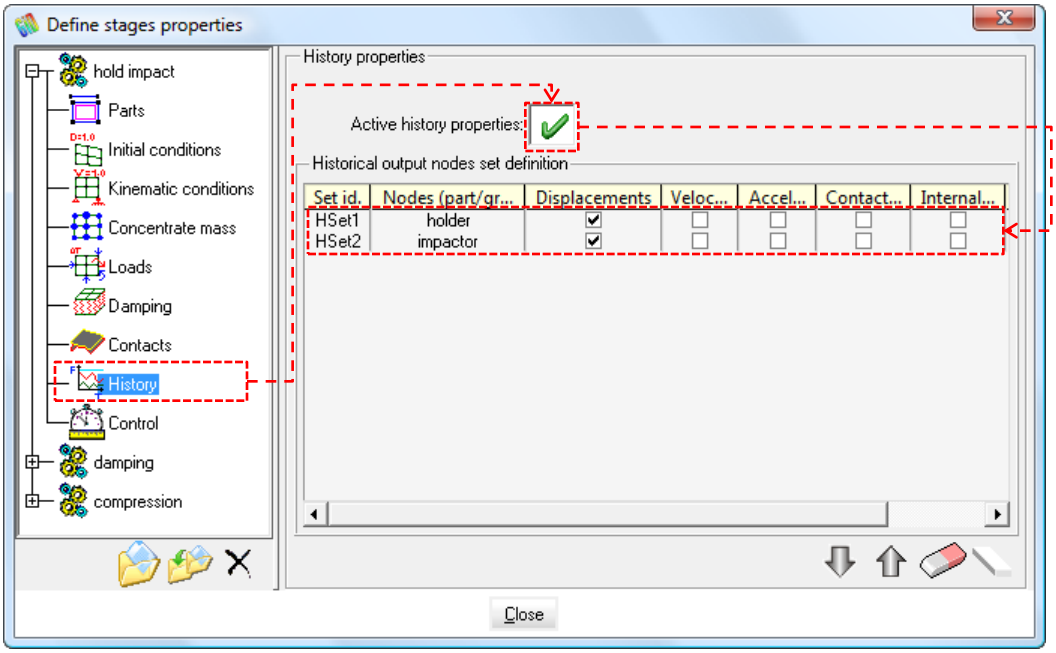
C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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 two lines in the list to indicate that we want to see the **displacements** as the historical output for the **holder** and **impactor** master node, in this 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...

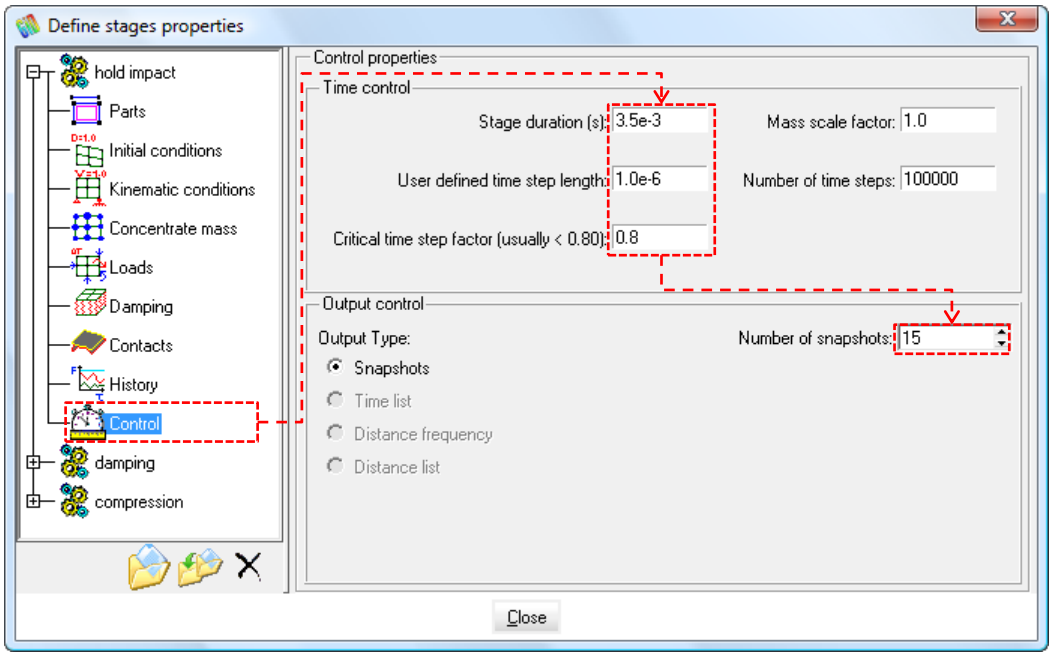
C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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

This block is compulsory for each stage.

It contains the stage duration, the user defined time step and number of desired snapshots for the postprocessor.



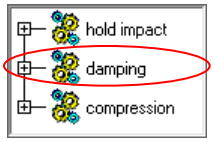
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

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...

C) Calculate

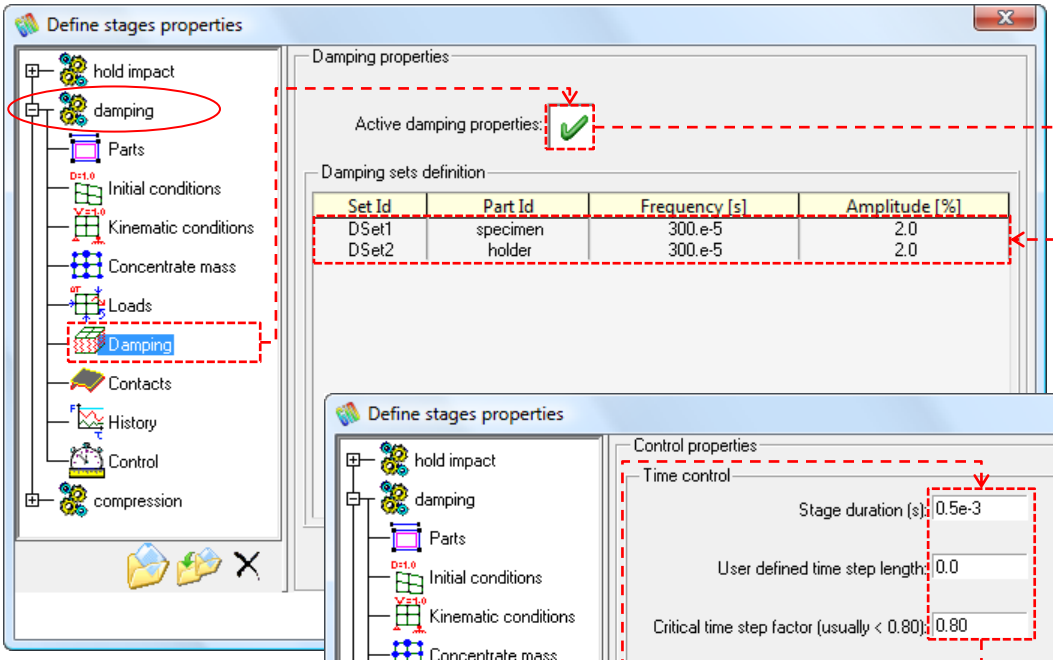
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

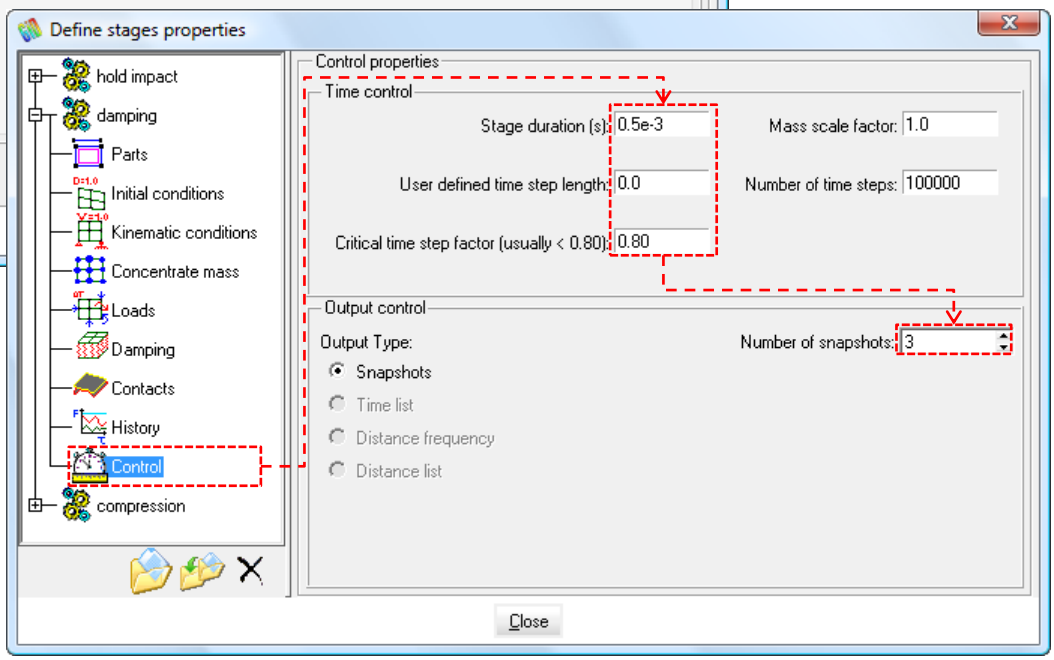
Now, let's see how to set the "**damping**" stage.



In this stage, only two blocks will be defined: the "**Damping**" and the "**Control**" definition blocks.

Click to enable **damping** properties and then add two lines the list to define the parameters for the "**specimen**" and for the "**holder**": **Frequency= 300e-5 seg** and **Amplitude=2%**.





In the "**Control**" block specify the stage duration, the user time step (0.0 for automatic evaluation) and factor, and the number of snapshots for postprocess.

CIMNE

QUANTECH ATZ

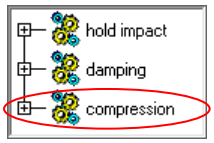
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

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...

C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

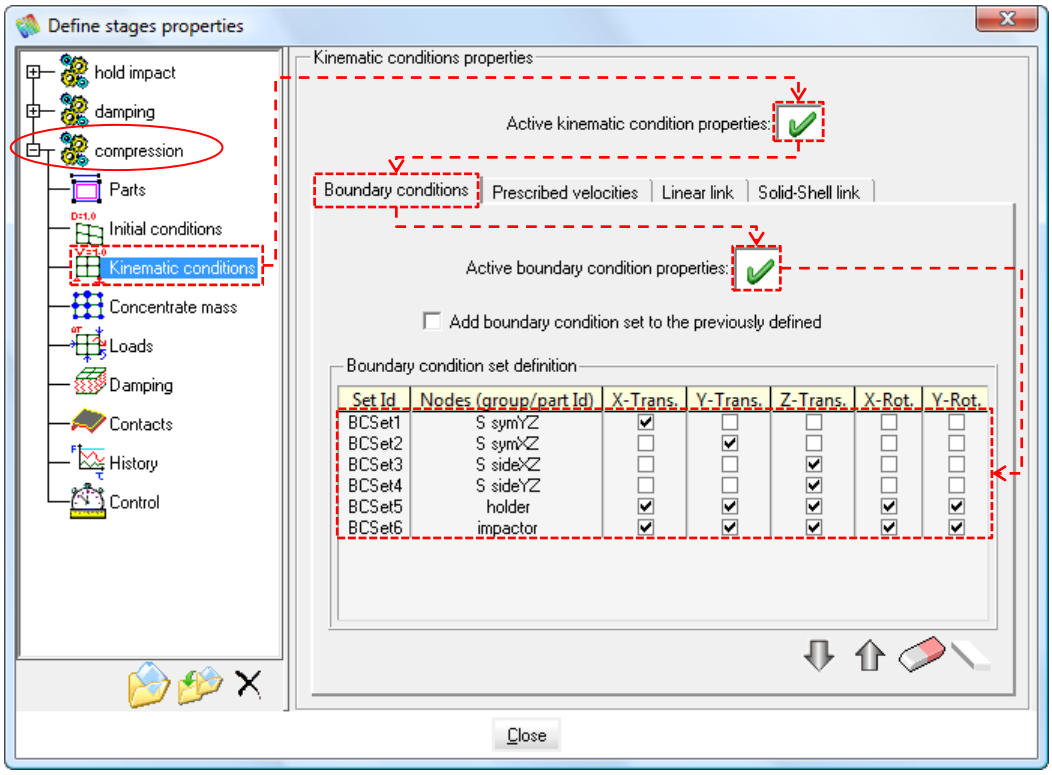
Finally, let's define the "**compression**" stage.



In this stage, the **Boundary conditions** will be redefined to match the new simulation scenario: the CAI test.

In the "**Kinematic conditions**" block, enable "kinematic" properties. And in the "**Boundary conditions**" tab, enable "boundary" properties.

Then, specify the symmetry and side conditions for the specimen employing appropriate nodes groups.



Note that the holder and the impactor must be fixed since they wont be employed during this stage.

Note also that the "Add boundary condition" option must remain unchecked.

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

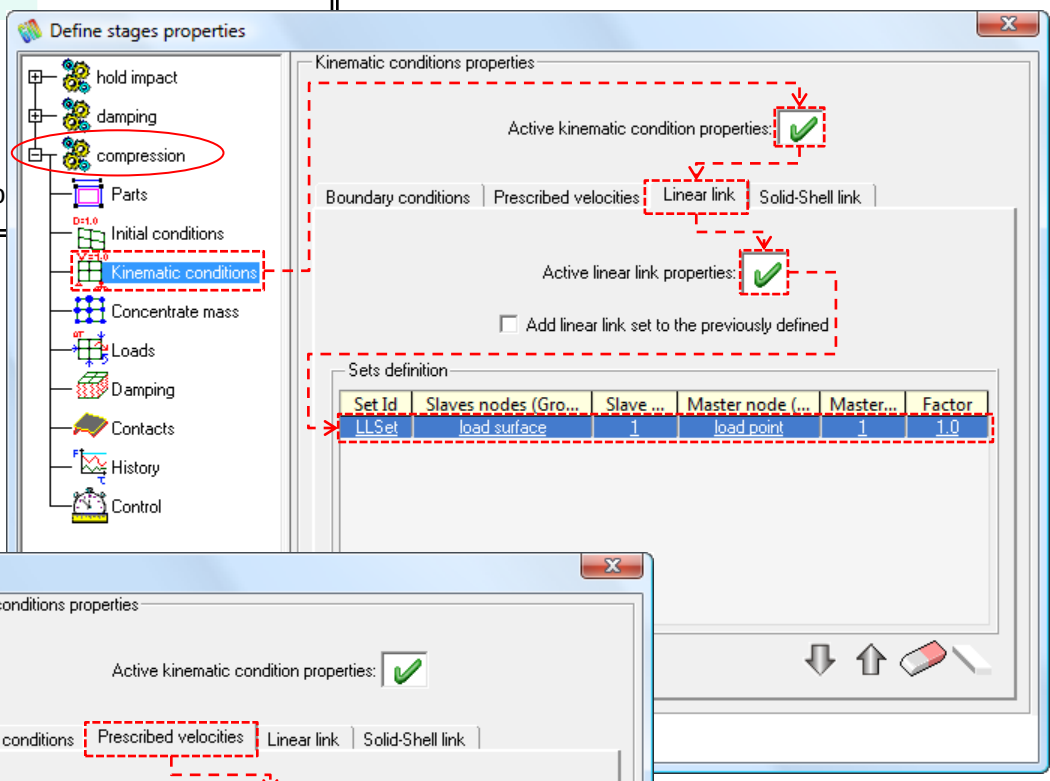
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...

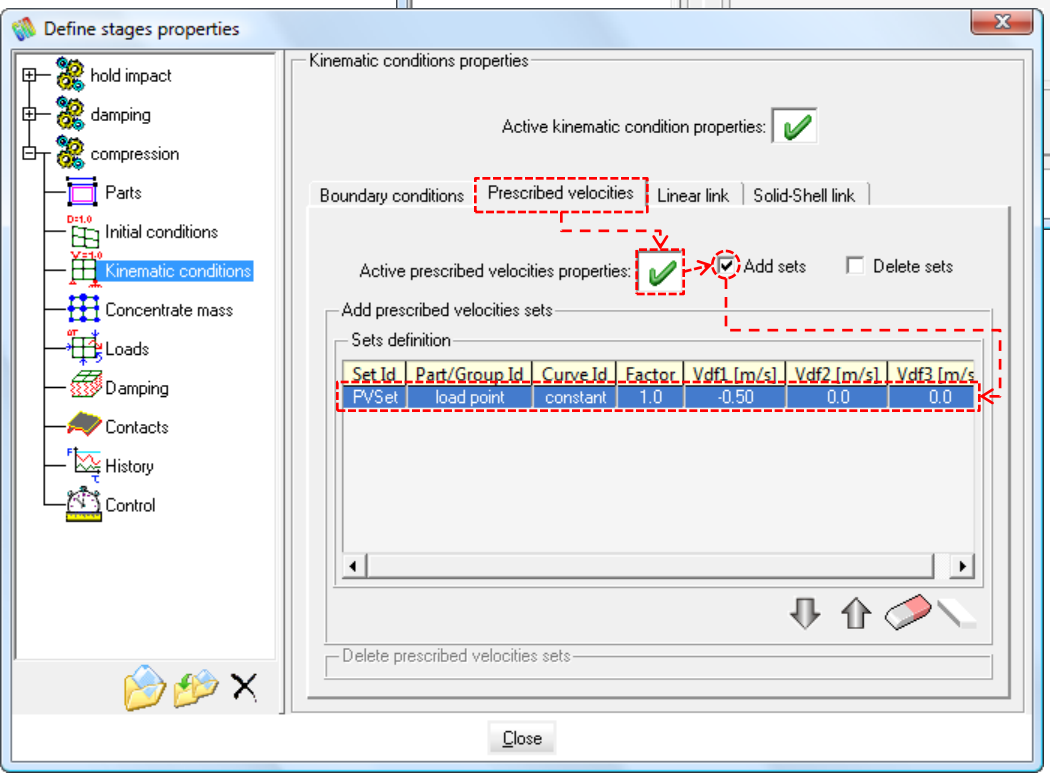
C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical o

The compressive loading will be applied employing "**Prescribed velocities**" & "**Linear link**" definition.

In the "**Linear link**" tab , make active the "linear link" properties. Then define the dependence between the "**load point**" (master node) and the "**load surface**" (slave nodes) in the X direction (first degree of freedom –d.o.f.-).





In the "**Prescribed velocities**" tab , make active the "prescribed velocities" properties and tick the "Add sets" option.

Then define the velocity in X direction (**-0.5 m/s**) for the "**load point**" and select the "**constant**" curve.

CIMNE

QUANTECH ATZ

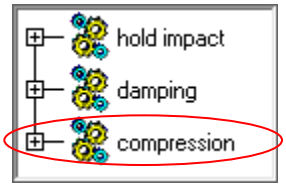
A) Pre-processor
- A.1) Import geometry (IGES)
- A.2) Import the mesh / Generate the mesh

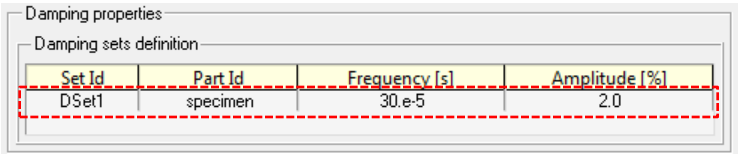
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...

C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

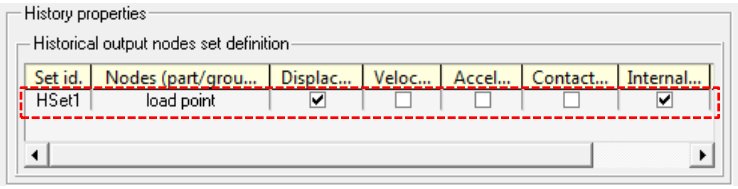
"**Damping**", "**History**" and "**Control**" properties must also be specified.





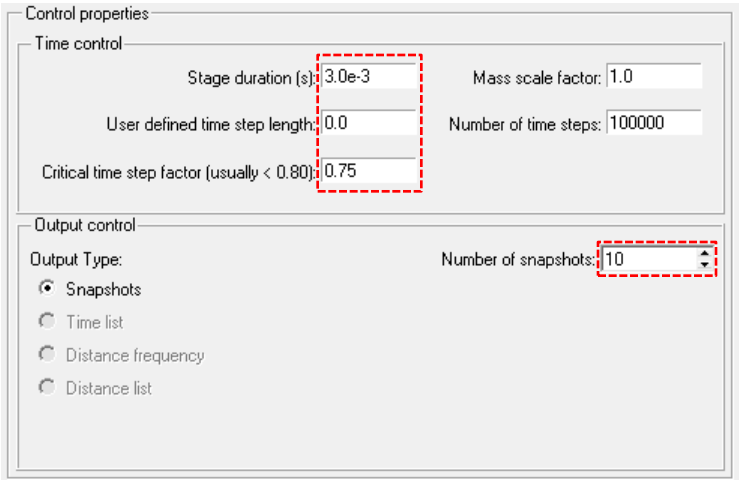
Enable the "Damping" properties.

Add two lines in the list to assign the following damping to the **holder** and to the **specimen**:
- ✓ Frequency = 30.0e-5 seconds, and
- ✓ Amplitude = 2 %.



Enable the "History" properties.

Add two lines in the list to indicate that we want to see the **displacements** as the historical output for the **holder** and **impactor** master node, in this stage.



In the "**Control**" block, specify the stage duration, the user defined time step and number of desired snapshots for the postprocessor.

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...

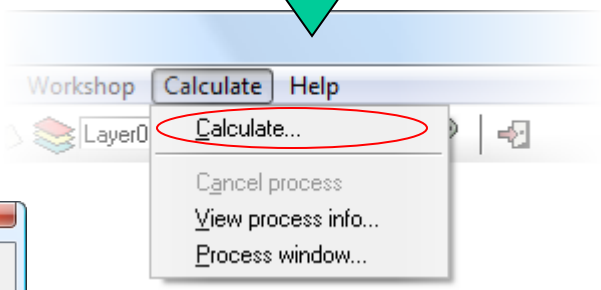
C) Calculate

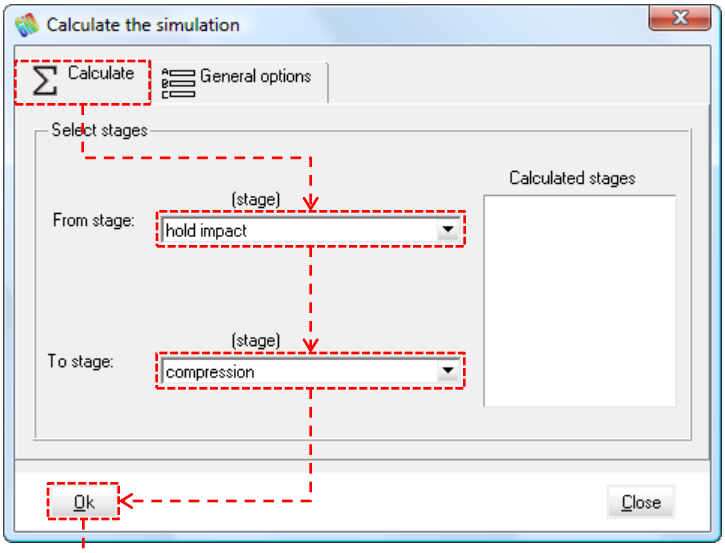
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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.

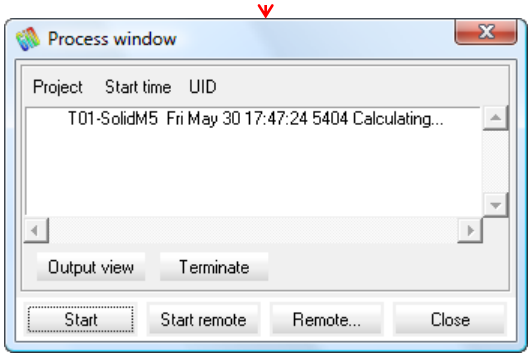
Workshop>Calculate…



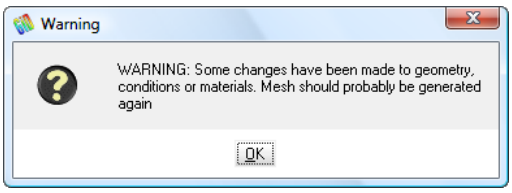


In the **Calculate** tab, select the initial and final stage to start the calculation. In this case, from "**hold_impact**" to "**compression**" stage.

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

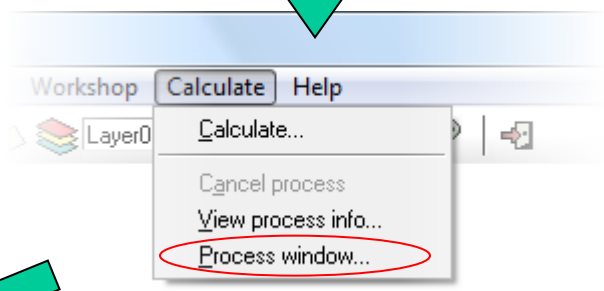
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...

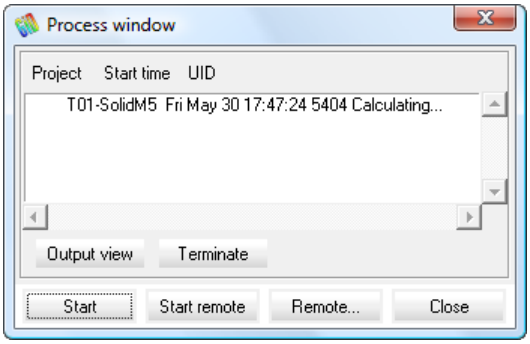
C) Calculate

D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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

Calculate>Process window…

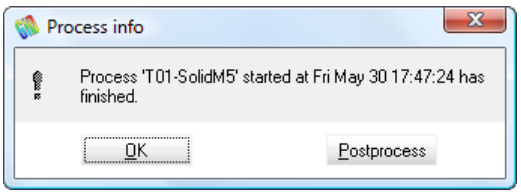




The "**Process**" window allows:

- ✓ 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 **COMPACK** 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...

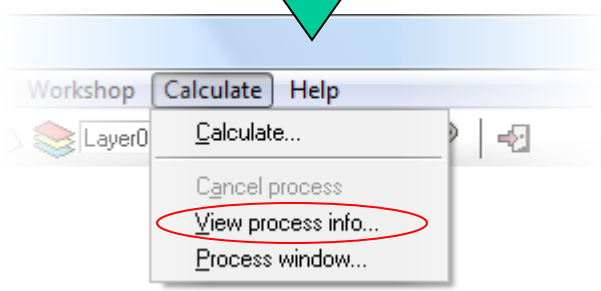
C) Calculate

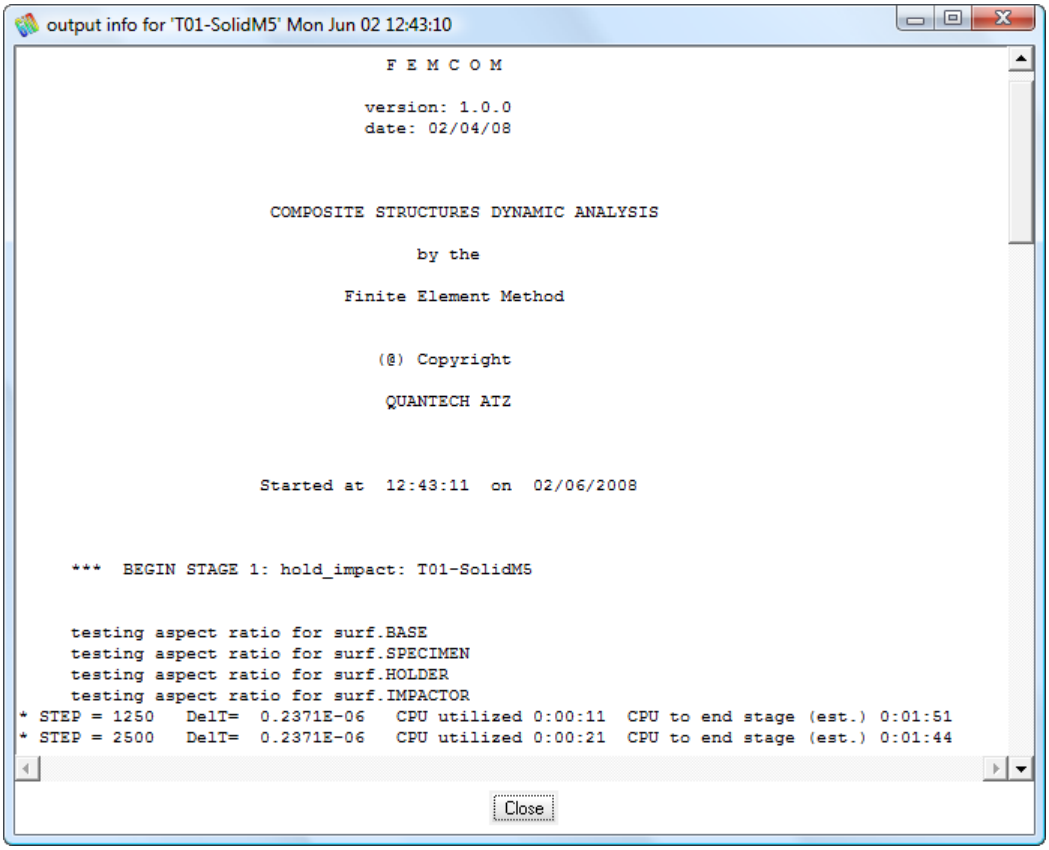
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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...

C) Calculate

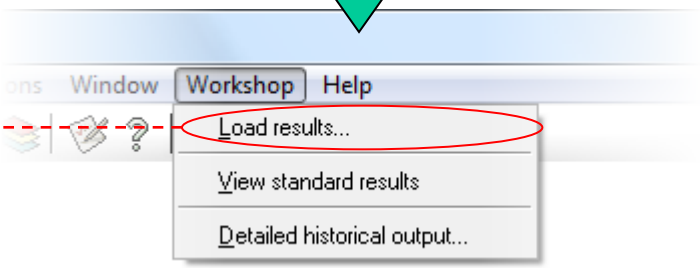
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

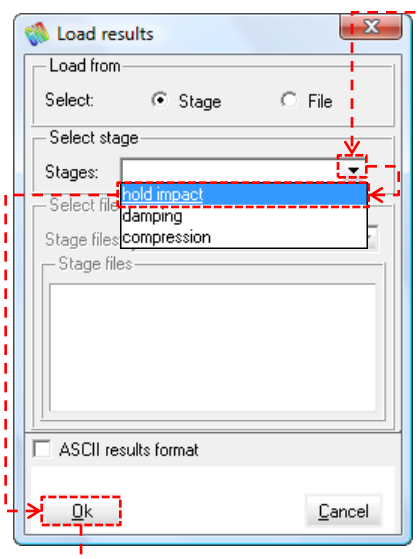
The results may be visualized either at the end of the calculation or while the simulation is in progress. To perform the visualization you must toggle to the **Post process environment**:

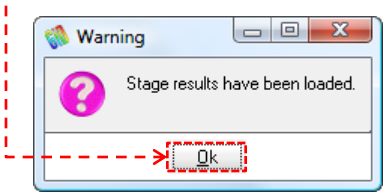
Now, the options in the workshop menu are related to the post-processing.

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...

C) Calculate

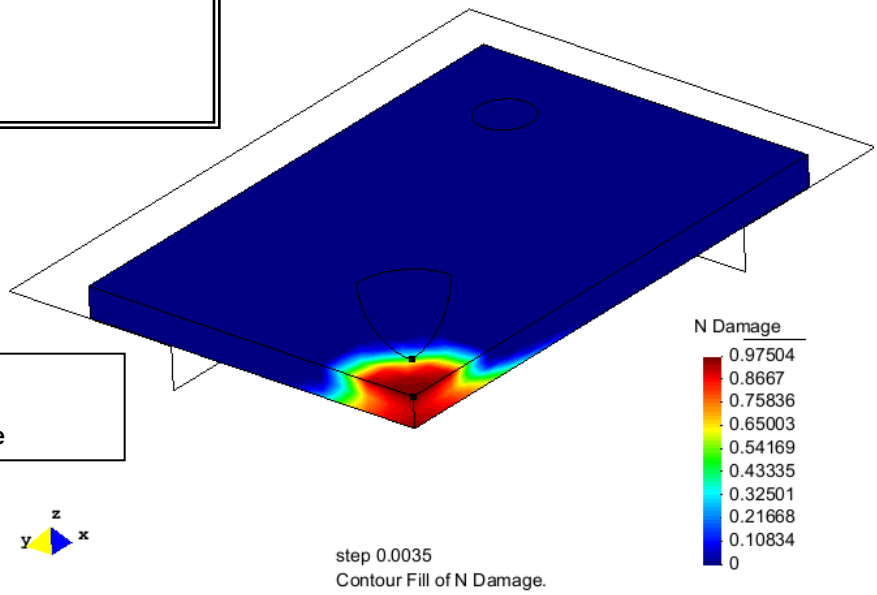
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

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

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

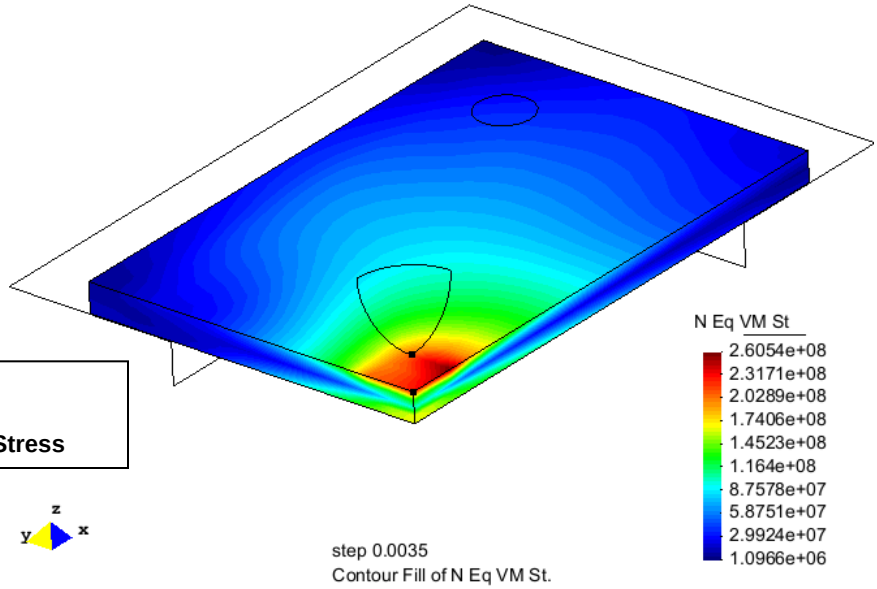
Average internal Damage:

View results>Contour Fill>N Damage



Equivalent Von Misses stresses (Pa):

View results>Contour Fill>Eqv. VM Stress



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...

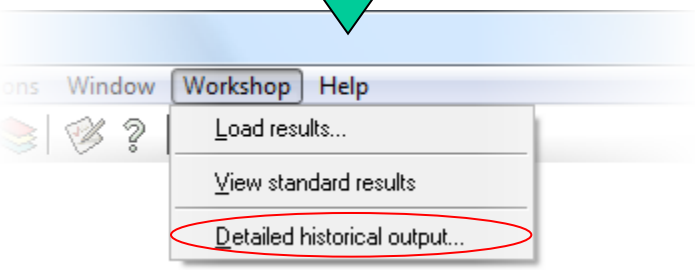
C) Calculate

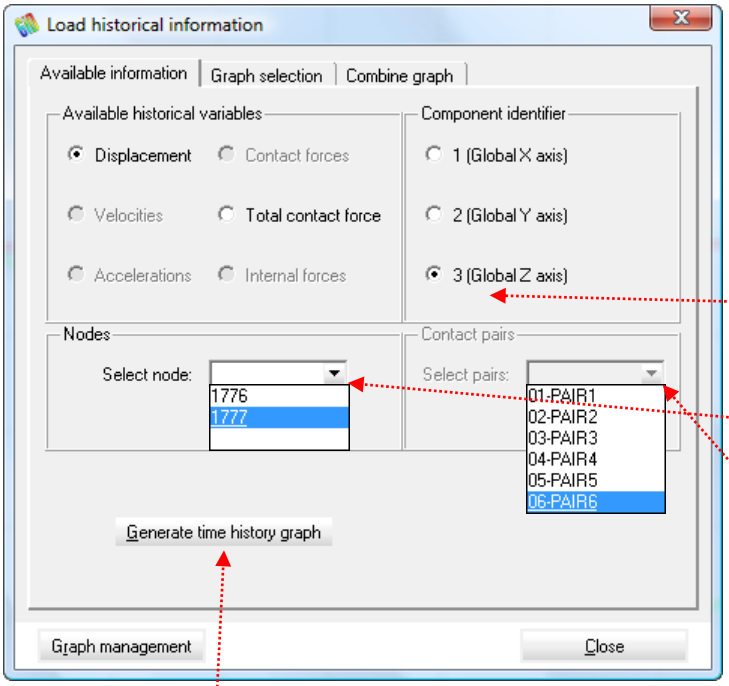
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

After loading the stage results, detailed historical variations at the nodes for the parameters selected in the pre-process can be graphically represented.

FIRST STEP: From the main menu click on

Workshop>Detailed history output…





A window pops up with the menu for generating graphs.

SECOND STEP:
In the **Available information** tab, the desired variable must be selected. Then select the component and finally the node.
In this case, **Displacement** component **Z** and node number **1777** of the **impactor** have been selected.

NOTE: In the case that the evolution of **Total contact force** is requested: a contact pair must be selected in place of a node number.

THIRD STEP: Click on **Generate time history graph**

FOURTH STEP: Click on the **Graph selection** tab to visualize the results as graphs.

CIMNE

QUANTECH ATZ

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

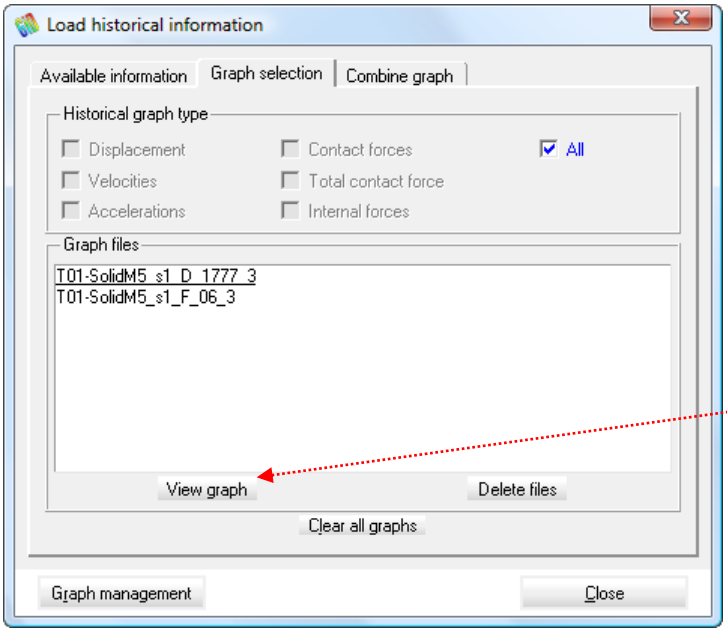
B) Workshop
- B.1) General model data...
- B.2) Material definition…
- B.3) Layup definition...
- B.4) Part definition...
- B.5) Stage definition...
- B.6) Group definition...
- B.7) Curves definition...

C) Calculate

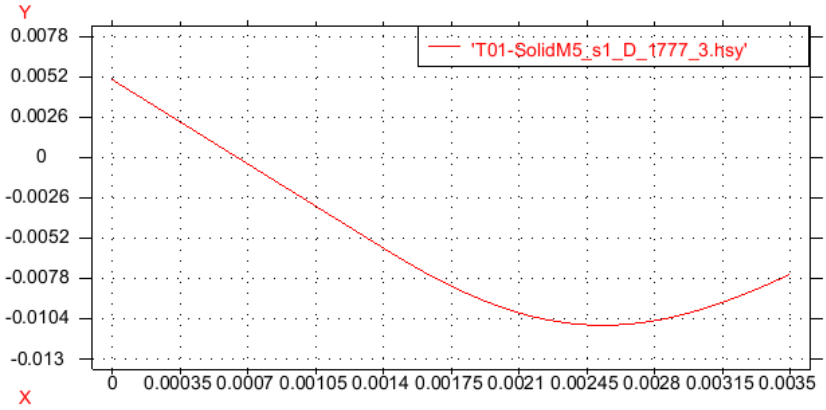
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

The detailed history outputs may be visualized as graphs.

The information generated in the **Available information** section is now available in files which can be visualized by clicking on **View graph**



NOTE: The name of the file is automatically generated by **COMPACK** and is formed by:
- ✓ the name of the project;
- ✓ the number of the stage
- ✓ the variable identifier (D - displacement, F – total contact force, L – internal force (load), etc.)
- ✓ node identification
- ✓ component



The procedure can be repeated as many times as necessary. For example, the historical variation of the total contact force at the pair 06 can be generated.

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...

C) Calculate

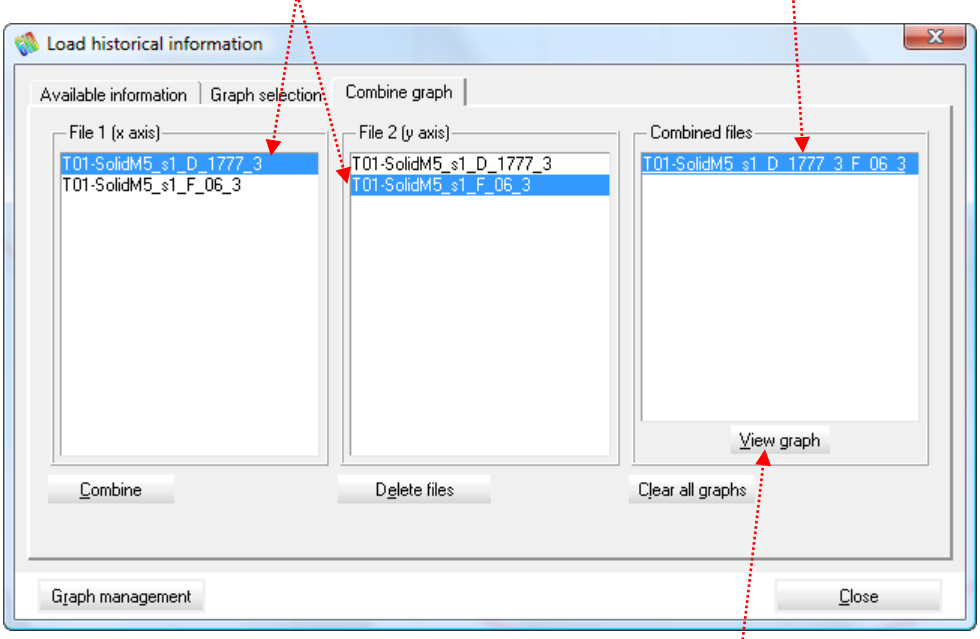
D) Post-processor
- D.1) Load stages
- D.2) Detailed historical output

Graphs can be combined. Graphs generated in previous sections (variable vs. time). Combining graphs enables graphs of the results with respect to each other (i.e. the time variable is eliminated).

Click on the **Combine graph** tab.

In the **Combine graph** section, select two graphs to be combined by clicking on their names. Then click on **Combine**.

A third file is generated containing the combination. The name of the new file is composed by merging the names of the two original files.



Click on **View graph** to visualize the new graph.