

GiD-Abaqus problemtyp

Interface with Abaqus

Abaqus

1 General Information	4
2 Prepare the geometry and generate the mesh	5
3 Create Sets	7
4 Define elastic materials	8
5 Define local coordinate systems	9
6 Output Abaqus file	10
7 Import Abaqus result database (ODB)	11

Abaqus



GiD-Abaqus problemtype

Interface with nonlinear analysis software ABAQUS

General Information

The **GiD-Abaqus** interface permits to extract mesh files suitable for input for the analysis software program **Abaqus**.

The interface is compatible from Abaqus 6.5. It is also able to import Abaqus **results** (.odb format), but require old Abaqus 6.5 Windows x32 dll libraries (otherwise this feature is disabled)

The data that can be output after the preprocessing is detailed below:

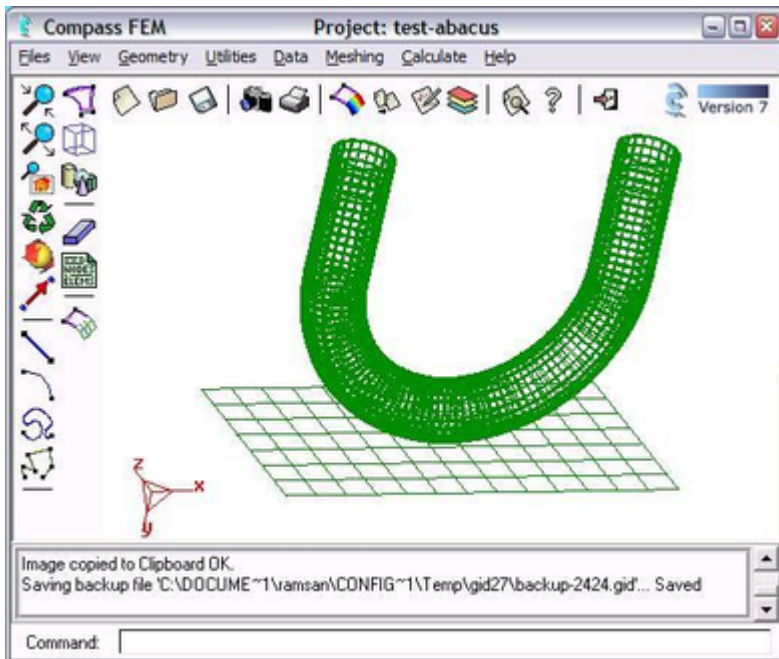
- The mesh: elements and nodes. All **GiD** supported elements can be printed, defining their formulation as an extra condition
- Sets of nodes
- Sets of elements
- Sets of surfaces (defined by nodes or face elements)
- Elastic materials
- Local coordinate systems

Before running the simulation, the file needs to be edited in order to identify each one of the created sets with the corresponding condition.

Note: GiD can import .inp mesh files in Abaqus format without this problemtype (the Abaqus mesh importer is implemented as another GiD plugin)

Prepare the geometry and generate the mesh

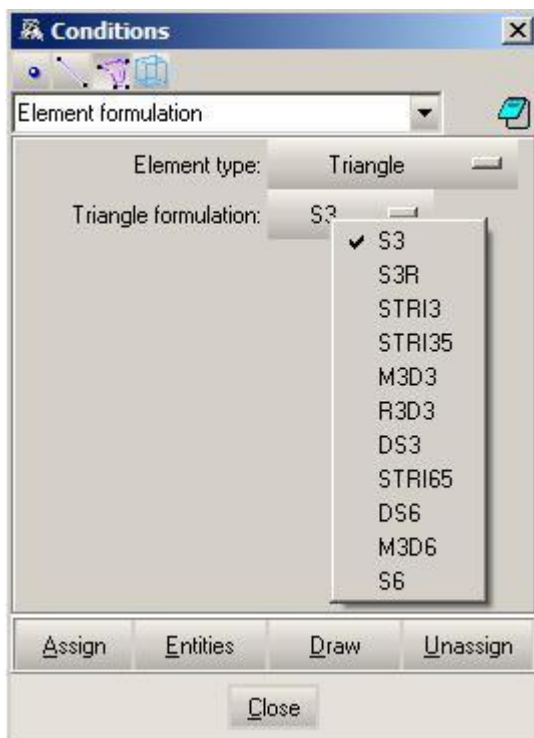
The geometry is created or imported using the standard **GiD** commands (See **GiD** Help for details). The mesh is also generated using standard **GiD** commands and all **GiD** element types are supported.



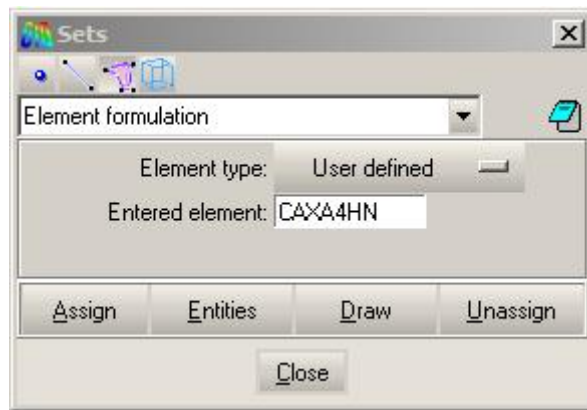
The following element formulation is set by default for each **GiD** element type (two values are presented because the mesh can also be quadratic):

- Linear Elements: B31,B32
- Triangle Elements: S3,S6
- Quadrilateral Elements: S4,S8,S9
- Tetrahedra Elements: C3D4,C3D10
- Hexahedra Elements: C3D8,C3D20,C3D27
- Prism Elements: C3D6,C3D15

The user can change the formulation of each element with the condition **Element formulation** (found in the menu **Data->Define Sets** for 1,2 and 3D elements), by selecting the different mesh or geometry entities of the model as in the sets assignment. Some options are suggested by the GUI but the user can also enter the formulation by hand (thanks to the *User defined* option):



Selecting an element formulation from the suggested list



Entering an element formulation by hand

Create Sets

To create a new set with GiD it is necessary to assign a condition to the geometry entities (recommended), before generating the mesh, or to the mesh entities, after generating it.

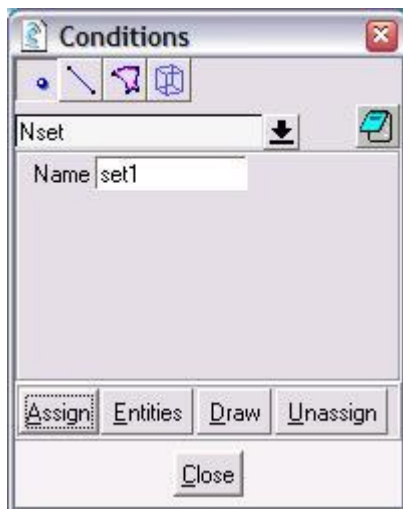
To create the sets, open the condition window:

Data->Define Sets

Or use the following icon of the Abaqus interface:



Use condition **Nset** to create nodes sets and the condition **Eset** to create element sets. Surface sets can also be defined with the help of **GiD-Abaqus** interface by selecting the surface nodes directly from the screen (**SurfaceNodeSet** condition). These surfaces can also be defined from 3D element faces (**SurfaceSet** condition). The choice of the surface definition scheme can affect its behavior during the computation.



Define elastic materials

User can define elastic materials from:

Data->Materials

Or by clicking the equivalent icon of the interface toolbar:



All GiD tools about material edition and **material database** importing/exporting are also available in this interface (see GiD Help). If a material is related with one or more elements of the mesh, they will be printed in the output file within a set named "Materialname_Set"



Define local coordinate systems

Local coordinate systems can be defined and edited from the GiD menu:

Data->Local axes


An icon in the interface toolbar allows the user to create new local coordinate easier:



New axes will be printed in the output file as a rectangular coordinate system. Refer to GiD Help to know all the possibilities of local coordinate system definition and edition.

Output Abaqus file

After preprocessing, the user can output the **Abaqus input file** using:
Files->Export->Write calculation file

Or by clicking on the equivalent icon in the interface toolbar: 

Only the conditions of the current interval will be printed in the **.inp** format output file (see GiD help to get more information of this point).

Import Abaqus result database (ODB)

To **import** an Abaqus analysis **results database**, there is an option which is available from the GiD menu:

Files->Import->ABAQUS ODB

It is necessary to have the ABAQUS dynamic libraries installed in the computer before attempting to import data to GiD.

The first time that the import filter is used, it asks for the directory where the libraries are installed (if it needs to be changed, the option *Files->Import->ODB libraries...* remains in the GiD menu).

The results are directly loaded from the **.odb** format and converted to a file of GiD results format, and all the GiD postprocessing capabilities are available.