



Customization Manual

GiD

The universal, adaptative and user friendly pre and postprocessing system for computer analysis

Developers

Miguel Pasenau de Riera

Enrique Escolano Tercero

Abel Coll Sans

Adrià Melendo Ribera

Anna Monros Bellart

Javier Gárate Vidiella M^a

Rosa Peyrau Rubio

For further information please contact

International Center for Numerical Methods in Engineering Edificio C1,
Campus Norte UPC
Gran Capitán s/n, 08034 Barcelona, Spain
<http://www.gidsimulation.com> gid@cimne.upc.edu

Contents

FEATURES	10
INTRODUCTION.....	11
XML file	12
ValidatePassword node	13
PROBLEMTYPE SYSTEM	14
Structure of the problem type.....	15
Definitions	16
Data tree fields.....	16
value.....	18
proc	20
condition	21
blockdata	24
edit_command	24
dependencies	24
groups	25
groups_types	25
units.....	25
Style	30
function.....	30
Annex I: Using functions	32
Annex II: Using matrices	40
Access to tree data information	42
Xpath	43
Main procedures	45
Description of the local axes	46
Writing the input file for calculation.....	52
User preferences	58
Import export materials	59
About CustomLib	59
CustomLIB extras	60
include	60
EXECUTING AN EXTERNAL PROGRAM.....	64
Showing feedback when running the solver.....	64
Commands accepted by the GiD command.exe.....	65
Managing errors.....	71
•Examples	71
PREPROCESS DATA FILES	72
Geometry format: Modelname.geo	72
POSTPROCESS DATA FILES	80
Results format: ModelName.post.res.....	81
List file format: ModelName.post.lst	107
Graphs file format: ModelName.post.grf	108

Binary format.....	109
HDF5 format	110
TCL AND TK EXTENSION	112
Event procedures.....	113
Geometry.....	117
Copy / Move	118
Mesh.....	119
Dimensions.....	121
Layers.....	122
Groups.....	123
Start / End.....	125
Read / Write.....	126
Transform	127
Materials	128
Conditions.....	129
Intervals.....	130
Units	131
Calculation file	132
Run.....	133
Start/End	135
GraphsSet	136
Graphs.....	137
Sets	138
Cuts.....	139
Results	140
GUI.....	143
View.....	145
Licence	146
Login.....	147
DataManager.....	148
Other	149
GiD_Process function	150
GiD_Info function	151
Sets	160
Graphs.....	160
display style	161
results.....	161
backup.....	169
batchfile	170
set	171
view	172
transform_problemtype	173
Preprocess mesh.....	182
Cartesian grid	185

Postprocess mesh	186
Geometry.....	187
Mesh.....	188
Definition	189
Entities.....	190
Definition	191
Entities.....	192
Books	193
CreateData	194
AssignData	195
UnAssignData.....	196
AccessValueAssignedCondition	197
ModifyData	198
AccessValue.....	199
IntervalData	200
LocalAxes	201
Units	202
File	204
WriteCalculationFile.....	205
Definition	210
Entities.....	211
Managing menus	225
Custom data windows.....	228
Interaction with themes	232
GiD version	237
PLUG-IN EXTENSIONS.....	238
Tcl plug-in	239
GiD dynamic library plug-in.....	245
APPENDIX A (PRACTICAL EXAMPLES).....	251
APPENDIX B (classic problemtype system).....	252
PROBLEMTYPE 'CLASSIC'	252
Problem and intervals data file (.prb)	257
Materials file (.mat)	259
Special fields	260
Unit System file (.uni).....	266
Conditions symbols file (.sim)	268
Commands used in the .bas file.....	271
General description.....	286
Detailed example - Template file creation	287

FEATURES

GiD offers the following customization features:

- Complete menu's can be customised and created to suit the specific needs of the user's simulation software. Simple interfaces can be developed between the data definition and the simulation software.
- Simple interfaces based on scalar, vector and matrix quantities can be developed for the results visualisation.
- Menus for the results visualisation can be customised and created according to the needs of the application or analysis.

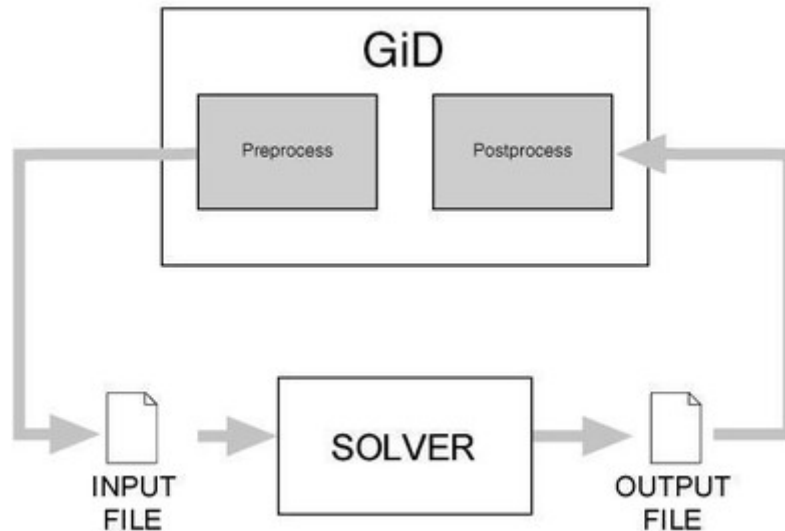
The customization in GiD is done by creating a **Problem Type**.

INTRODUCTION

When GiD is to be used for a particular type of analysis, it is necessary to predefine all the information required from the user and to define the way the final information is given to the solver module. To do so, some files are used to describe conditions, materials, general data, unit systems, symbols and the format of the input file for the solver. We give the name **Problem Type** to this collection of files used to configure GiD for a particular type of analysis.

Note: You can also learn how to configure GiD for a particular type of analysis by following the **Problem Type Tutorial**; this tutorial is included with the GiD package you have bought. You can also download it from the GiD support web page (<http://www.gidsimulation.com>).

GiD has been designed to be a general-purpose Pre- and Postprocessor; consequently, the configurations for different analyses must be performed according to the particular specifications of each solver. It is therefore necessary to create specific data input files for every solver. However, GiD lets you perform this configuration process inside the program itself, without any change in the solver, and without having to program any independent utility.



To configure these files means defining the data that must be input by the user, as well as the materials to be implemented and other geometrical and time-dependent conditions. It is also possible to add symbols or drawings to represent the defined conditions. GiD offers the opportunity to work with units when defining the properties of the data mentioned above, but there must be a configuration file where the definition of the unit systems can be found. It is also necessary to define the way in which this data is to be written inside the file that will be the input file read by the corresponding solver.

From the 13th version of GiD, a new system of problemtype has been implemented (based in the CustomLIB library). Although the 'classic' problem type system is still supported by GiD, it is considered deprecated, as the new one offers clear advantages in terms of usability, performance and integration capabilities. Documentation about the deprecated classic problem type system can be found in the annex of this manual [APPENDIX B \(classic problemtype system\)](#).

This new problem type definition uses a single .spd file to describe general properties, materials, conditions and units (as a tree with xml syntax). All this data is showed in a 'tree view', and materials and conditions are associated to groups of entities. About writing the input file, Tcl commands are used to write the data in files (optionally aided with the special function `GiD_WriteCalculationFile` for efficiency).

The new problem type creation system lean on a collection of tools, which facilitates the development of advanced problem types for customizing the personal pre and post processor system [GiD](#) for computer simulation codes. It is based on a XML hierarchical structure and an automatic physical tree view.

XML file

The file `problem_type.xml` declare information related to the configuration of the problem type, such name, version, file browser icon, password validation or message catalog location, history news, etc.

The data included inside the xml file should observe the following structure:

```
<Infoproblemtype version="1.0">
  <Program>
    <Name>XXX</Name>
    <Version>XXX</Version>
    ...
  </Program>
</Infoproblemtype>
```

By default GiD read this file when loading the problemtype and provide its key-value pairs parsed in a Tcl global array named 'problemtype_current' (e.g. `$::problemtype_current(version)` returns the version of the problemtype)

Compulsory nodes: (the values of these nodes are just examples)

- `<Name>cmas2d_customlib</Name>` to provide an identifier name for the problem type.
- `<Version>1.0</Version>` dotted version number of the problem type.

The name and version of the problemtype is used to compare the version of the problemtype used for a old model and do an automatictransform if necessary to try to map the old and new data fields.

The 'Internet retrieve' tool also uses Name and Version to compare a local problemtype with the remote copy of the Internet repository.

Optional nodes:

- `<MinimumGiDVersion>12.1.11d</MinimumGiDVersion>` to state the minimum **GiD** version

required. If the problemtype is loaded in a **GiD** version lower than the one required a warning message will

be raised.

- `<ImageFileBrowser>images/ImageFileBrowser.gif</ImageFileBrowser>` icon image to be used in the file browser to show a project corresponding to this problem type. The recommended dimensions for this image are 17x12 pixels.
- `<MsgcatRoot>scripts/msgs</MsgcatRoot>` a path, relative or absolute, indicating where the folder with the name msgs is located. The folder msgs contains the messages catalog for translation.
- `<PasswordPath>../PasswordPath</PasswordPath>` a path, relative or absolute, indicating where to write the password information see [ValidatePassword node](#)).
- `<ValidatePassword></ValidatePassword>` provides a custom validation script in order to override the default GiD validation (see [ValidatePassword node](#)).
- `<CustomLibAutomatic>1</CustomLibAutomatic>` This node must be defined only for 'customLib like' problemtypes, with values 0 (default) or 1. If true it allows to do automatic tasks to use the library (otherwise the problemtype developer must write extra code to use the library, like load packages, initialize the library, etc.)
- `<CustomLibNativeGroups>1</CustomLibNativeGroups>` This node must be defined only for 'customLib like' problemtypes, with values 0 (default) or 1, to specify that the library uses 'native GiD groups' instead of 'pseudo-groups GiD conditions'.

It is possible to set other non-standard nodes, to use

ValidatePassword node

The default action taken by **GiD** when validating a problem type password is verifying that it is not empty. When a password is considered as valid, this information is written in the file 'password.txt' which is located in the problem type directory. In order to override this behaviour, two nodes are provided in the .xml file

- PasswordPath**: The value of this node specifies a relative or absolute path describing where to locate/create the file password.txt. If the value is a relative path it is taken with respect to the problem type path.

Example:

```
<PasswordPath>../PasswordPath>
```

- ValidatePassword**: The value of this node is a Tcl script which will be executed when a password for this problem type needs to be validated. The script receives the parameters for validation in the following variables:

key with the contents of the password typed,
dir with the path of the problem type, and
computer_name with the name of host machine.

Note: It's like this Tcl procedure prototype: `proc PasswordPath { key dir computer_name } { ... body... }`

The script should return one of three possible codes:

0 in case of failure.

1 in case of success.

2 in case of success; the difference here is that the problem type has just saved the password information so **GiD** should not do it.

Furthermore, we can provide a description of the status returned for **GiD** to show to the user. If another status is returned, it is assumed to be 1 by default.

Below is an example of a <ValidatePassword> node.

```
<ValidatePassword>
#validation.exe simulates an external program to validate the key for this computername
#instead an external program can be used a tcl procedure
if { [catch {set res [exec [file join $dir validation.exe] $key $computername]} msgerr] } {
    return [list 0 "Error $msgerr"]
}
switch -regexp -- $res {
    failRB {
        return [list 0 "you ask me to fail!"]
    }
    okandsaveRB {
        proc save_pass {dir id pass} {
            set date [clock format [clock second] -format "%Y %m %d"]
            set fd [open [file join $dir .. "password.txt"] "a"]
            puts $fd "$id $pass    # $date Password for Problem type '$dir'"
            close $fd
        }
        save_pass $dir $computername $key
        rename save_pass ""
        return [list 2 "password $key saved by me"]
    }
    okRB {
        return [list 1 "password $key will be saved by gid"]
    }
    default {
        return [list 0 "Error: unexpected return value $res"]
    }
}
</ValidatePassword>
```


PROBLEMTYPE SYSTEM

A problem type is a collection of utilities, which allows the user to interact easily with them by means of a Graphical User Interface (GUI), and facilitates the definition and introduction of all the data necessary for carrying out a particular calculation. In order for GiD to prepare data for a specific analysis program, it is necessary to customize it. The customization is defined in GiD by means of a problem type.

The new system of problem types creation adds some additional capabilities compared with the classic one:

- It takes advantage of the XML (Extensible Markup Language) format features and its hierarchical structure. It stores data more efficiently. The elements in a XML document form a tree-structure that starts at “the root” and branches to “the leaves” with different relationships between the nested elements.
- It permits to process automatically XML documents on a physical data tree view on the GiD window for interfaces creation.
- It facilitates the automatic creation of standard windows in the data tree to enter input data. It couples geometry or mesh entities with identical properties into the called groups using these standard windows.
- It permits to couple entities with identical properties into groups. In this way, it couples geometry or mesh entities with identical properties into the called groups using these standard windows.
- It allows to apply efficiently geometry properties and boundary conditions (i.e. constraints, loads, materials...) into groups and to edit their properties easily.
- In order to configure GiD for a specific type of analysis, it is possible to set the data tree hiding the required parts automatically. It allows to fix the data tree hiding concrete parts if this is convenient, for a specific type of analysis.
- It couples all the common features of the different problem types.
- It facilitates the introduction of all the data to transfer to an analysis program.

Structure of the problem type

The problem type is defined using a directory with the its name and a set of files. The directory with the problem type will be located in the **problemtypes** directory in the **GiD** distribution, or in a subdirectory of it.

Note: now it is also possible to have a /problemtypes extra folder located at the <GiD user preferences folder>. This is interesting in case that the user doesn't has privileges to copy a problemtype inside <GiD folder>/problemtypes

This set of files define the problemtype and contain the full functionality for customizing the pre-process. The main files that configure the problem type are shown in the table below.

File extension	Description
<name>.spd	Main configuration file of the data tree, XML-based
<name>.bas	Information for data input file. It should be an empty file
<name>.tcl	Main TCL file, initialization
<name>.cnd	Conditions definition. It should not be modified by the user
scripts/*.tcl	Output description to the file of analysis

- Main configuration file of the data tree, XML-based.

The main configuration file in XML format contains the definition of all the data (except the geometry) necessary to perform an analysis. It is defined in XML format with the extension .spd (specific problem type data) and contains all the definition of all the data that defines the analysis like boundary conditions, loads, materials, loadcases, etc.

The syntax rules of the .spd file are very simple, logical, concise, easy to learn and to use. The file is human-legible, clear and easy to create. Moreover, the information is stored in plain text format. It can be viewed in all major of browsers, and it is designed to be self-descriptive.

The elements in a XML document form a tree-structure that starts at "the root" and branches to "the leaves" with different relationships between the nested elements. It allows to aggregate efficiently elements. CustomLib takes advantage of this hierarchical structure to convert automatically the main XML file to a physical tree on the GiD window. The XML elements can have attributes, which provide additional information about elements.

It is necessary to modify this XML document in order to add conditions, or general data to the problem type.

- Main Tcl file, initialization

- A Tcl initialization file is used to create complex windows or menus. Contains the initialization routines

Output description to the file of analysis

A Tcl file located in the <scripts> folder determines the way in which the final information has to be written inside the input files that will be read by the solver.

- Conditions file

File with extension .cnd is used but should not be modified by the problemtype creator. It is only required if local axes are used.

- Information for data input file

File .bas is not used it should be existing but void.

- Several other Tcl files

Definitions

CustomLib defines its own XML tags, which clearly describes its content. For more information about this attribute, see the section [Data tree fields](#).

- *TDOM library*

The tDOM library is used by the Toolkit due to it combines high performance XML data processing with easy and powerful Tcl scripting functionality. tDOM is one of the fastest ways to manipulate XML and it uses very little memory in the process of creating a DOM tree from a XML document. In TDOM terminology we call 'field' to the 'Element name' and 'parameter' to the 'attribute'. All data is stored in fields and parameters, where the parameters can contain a value, a xpath expression or a [Tcl command].

- *xpath*

XPath is a language for addressing parts of an XML document using path notations.

It is used for navigating through the hierarchical structure of a XML document to extract information. It is based on a tree of nodes representing the XML file.

It provides the ability to navigate around the tree, selecting nodes from it and computing string-values.

xpath expression -> A search is performed and the result is substituted in the parameter when necessary.

[Tcl command] -> The command between brackets is executed when necessary and the return value is replaced inside the parameter.

Data tree fields

The fields and parameters of the main configuration file (.spd) of the data tree, are described below.

PT_data

<PT_data>

Main root field of the .spd file. It contains the version number and the name of the problem type.

PT must be replaced by the problem type name.

version - Internal version number.

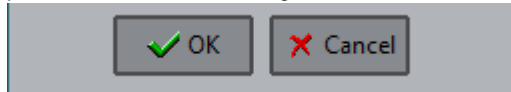
e.g. if the problemtype is named 'cmas2d_customlib' the main node will be

```
<cmas2d_customlib_data version='1.0'>
...
</cmas2d_customlib_data>
```

container

<container>

This is the simplest form to group the data for improving its visualization.
On the resulting window, in addition to the inputs there will be the following set of buttons:



It can contain the following fields: <value>, <container>, <condition>, <function>, <dependencies>

The parameters are as follows,

n - Name used to reference the field, especially when writing the .dat file.

pn - Label that will be visualized by the user. It can be translated.

icon - It allows to put an image in .png format in the data tree. The image should be stored inside the images folder of the problem type.

help - It displays a pop-up window of help information related to the task the user is performing.

help_image - declare an image filename with the image to be shown below the help text. The file must be located inside a problemtype folder named 'images' and the help attribute is required to use help_image

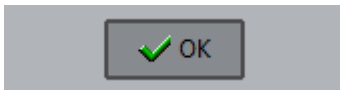
state - Specifies one of two states for the field: **normal**, or **hidden**. Note that hidden <container> field can be used for storing hidden values, that you do not want to show in the user interface. It also permits to handle a Tcl function, by means of square brackets.

update_proc - It calls a Tcl procedure, when clicking on the 'Ok' button in the window. (must not add square brackets in this case). The Tcl procedure must be defined in a <proc> node of the spd

actualize_tree - It updates the information in the whole data tree, and automatically refresh data shown in the user interface. It is a boolean value as a 1 or 0 that indicates if it is activated or deactivated. If the data source is changed, such as new fields have been added or data values and field have been modified, all the user interface will reflect those changes. Furthermore, all the TCL procedures defined in the data tree will be called and the whole data tree will be refreshed. Therefore, this instruction must be carried out only when necessary.

actualize - This only updates a specified field in data tree. Note that only this specified field will be refreshed in the user interface, and not the whole data tree. It is a boolean value as a 1 or 0 that indicates if it is activated or deactivated.

del_cancel_button - It is a boolean value as a 1 or 0 that indicates if the cancel button is removed (1) or not (0). On the resulting window there will be only the 'OK' button, as follows,



Fields to allow customize tree contextual menu:

addcontextualmenu -To add a new option to the menu. Expects a Tcl list of items, each item with subitems "icon text tcl_command"

replacecontextualmenu -To replace a menu option. Expects a Tcl list of items, each item with subitems "old_text icon text tcl_command"

removecontextualmenu -To remove a menu options. Expects a Tcl list of items, each item with subitems "old_text"

title - 0 or 1

icon_end - an icon name

tree_state "open" or "close"

Example:

```
<container n='general_data' pn='General data' icon='general'>
  <value n='max_iterations' pn='Maximum iterations' value='1e3' help='limit of iterations for iterative solver' />
  <value n='stop_tolerance' pn='Stop tolerance' value='1e-8' />
</container>
```

Example: (customize contextual menu)

```
<blockdata n="Simulation_type" pn="Simulation data" icon="[icon_simtype]" addcontextualmenu="{advanced-16 {Add option 1} CompassFEM::Test} {advanced-16 {Add option 2} CompassFEM::Test}"
replacecontextualmenu="{ {Edit} advanced-16 {Edit mod.} CompassFEM::Test} { {View this} advanced-16 {View this mod.} CompassFEM::Test_mod}" removecontextualmenu="{View this} {Edit}"></blockdata>
```

value

<value>

It is the main field to store data. This field allows to define an entry, or a combobox in the window.

It can contain the following fields: <function>, <dependencies>, <edit_command>

The parameters are as follows,

n - Name used to reference the field, especially when writing the .dat file.

pn - Label that will be visualized by the user. It can be translated.

v - The value or default value for a 'value' field

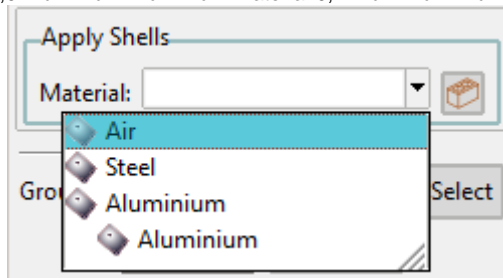
icon - An optional icon name

state - Specifies one of the four states for the entry: normal, disabled, hidden or readonly. If the entry is readonly, then the value may not be changed using widget commands and no insertion cursor will be displayed, even if the input focus is in the widget; the contents of the widget may still be selected. If the entry is disabled, the entry may not be changed, no insertion cursor will be displayed and the contents will not be selectable. Note that hidden entry can be used for storing hidden values. It also permits to define a Tcl function, by means of square brackets.

values - Comma-separated list of strings to fill a combobox. For instance, values="mech,therm". Can call a Tcl proc including arguments, with [] like values=[GetMyValues].

The Tcl procedure must be defined in a <proc> node of the spd or now directly as a normal Tcl procedure, in this case it is possible to add as argument %W that will be replaced with the dom node (this become simpler that a double step of define an extra <proc> node that finally call to a Tcl proc, e.g. values=[GetMyValues %W])

values_tree - A drop-down tree to fill a combobox, defined by a comma-separated list of strings. It can call a Tcl proc, by means of square brackets [], that must returns a list of comma-separated strings, each one represented as the format: "level_in_the_tree name_in_dropdown_tree name_in_combobox enable_boolean". For instance, a drop-down tree defined as (see image below): values_tree="0 Air Air material 1,0 Steel Steel material 1,0 Aluminium Aluminium material 0,1 Aluminium Aluminium material 1"



dict - Comma-separated list of key,value. This operation places a mapping from the given key to the given value, which is shown in the GUI. Values can be translated.

For instance, dict="mech,Mechanical,therm,Thermal" shows Mechanical and Thermal.

string_is - Tests the validity of various interpretations of a string, as follows:

integer - To test if a string is an integer value.

integer_or_void - To test if a string is an integer value, or empty (not-filled).

double - To test if a string is a double value.

double_or_void - To test if a string is a double value, or empty (not-filled).

% - To test if a string is a percentage (%).

list_of_double - To test if a string is a list of doubles, representing a data structure of doubles in Tcl.

entier - To test if a string is an integer (of any size), written in one of the forms Tcl can parse, or an integer in scientific notation (for instance 1e30).

entier_or_void - To test if a string is an integer (of any size), written in one of the forms Tcl can parse, or an integer in scientific notation (for instance 1e30), or empty (not-filled).

help - It displays a pop-up window of help information related to the task the user is performing.

actualize_tree - It updates the information in the whole data tree, and automatically refresh data shown in the user interface. It is a boolean value as a 1 or 0 that indicates if it is activated or deactivated. If the data source is changed, such as new fields have been added or data values and field have been modified, all the user interface will reflect those changes. Furthermore, all the TCL procedures defined in the data tree will be called and the whole data tree will be refreshed. Therefore, this instruction must be carried out only when necessary.

actualize - This only updates a specified field in data tree. Note that only this specified field will be refreshed in the user interface, and not the whole data tree. It is a boolean value as a 1 or 0 that indicates if it is activated or deactivated.

update_proc - It calls a TCL procedure, when clicking on the 'Ok' button in the window or when changing the value of the entry. Square brackets are not supported.

menu_update - It allows to update the menus. Values can be yes or no. It is necessary to define a <dependencies> field, as follows,

<dependencies node="/*blockdata[@n='General_Data']/value[@n='analysis_type']" att1="menu_update" v1="[TCL_proc]" actualize="1"/>

fieldtype - It creates a box in the user interface in order to introduce a text. It can be:

longtext - A text box is created.

editable - It is a boolean value as a 1 or 0 that indicates if the entry could be changed or not. If it is activated (1) the entry may not be changed, no insertion cursor will be displayed and the contents will not be selectable.

unit_magnitude - Physical quantity (i.e. L, for Length). For more information about this attribute, see the section [Description of the units](#)

units - Unit of the physical quantity (i.e. m). For more information about this attribute, see the section [Description of the units](#)

function - Contains a Tcl command, which is executed when is called. It permits to create or edit a function for a determined entry.

function_func - Permits to define a TCL function.

values_check - It allows to handle a Tcl function, by means of square brackets.

min_two_pnts - It is a boolean value as a 1 or 0 and indicates that two points or more are required in a linear interpolation.

unit_definition - The fields <value/> used to choose the default units in the GUI are special. They contain the attribute called unit_definition="magnitude" being magnitude the name 'n' to be used in that field. It is important to note that these kind of fields does not contain dependencies.

show_in_window - Can be 1 or 0 (1 by default). It indicates if the value must be shown in the conditions window. If set to 0, the value will be shown in the tree, but will be hidden in the window.

Example:

```
<value n='units_length' pn='Length' unit_definition='L' />
<value n='units_mass' pn='Mass' unit_definition='M' />
<value n='units_force' pn='Force' unit_definition='F' />
```

unit_mesh_definition - The field <value/> used to choose the mesh unit is special. It has the attribute unit_mesh_definition="1", and it does not contain any "v" attribute or dependencies.

Example:

```
<value n="units_mesh" pn="Geometry units" unit_mesh_definition="1" />
```

units_system_definition - The field <value/> used to choose the units system is special. It has the attribute units_system_definition="1", it does not contain any "v" attribute, and it contains a unique dependency related to the unit fields.

Example:

```
<value n='units_system' pn='Units system' units_system_definition='1' icon='units-16'>
  <dependencies node="//*[@unit_definition or @unit_mesh_definition='1']" att1='change_units_system'
  v1='{@v}' />
</value>
```

proc

<proc>

This kind of xml node allow to define a Tcl scripting procedure the could be called in other field nodes, like values.

It is compulsory to collect all <proc> nodes inside a <procs> xml node

The parameters are as follows,

n - Name of the proc used in other xml nodes

args - optional, it will be set as a list with all arguments provided in the uses of other xml nodes

This node has some tricky implicit arguments:

The local variable `domNode` is 'magically' filled with the xml dom node of the caller,

and there could be other optional variables that can be set with special options in the arguments

```
{ -tree tree "" }
{ -boundary_conds boundary_conds "" }
{ -item item "" }
{ -dict dict "" }
{ -dict_units dict_units "" }
```

the list with the rest of provided arguments will be in the variable `args`

Example:

```
<procs>
  <proc n='GetMaterialsList' args='args'>
    Cmas2d::GetMaterialsList $domNode
  </proc>

  <proc n='EditDatabaseList' args='args'>
    Cmas2d::EditDatabaseList $domNode $dict $boundary_conds $args
  </proc>
</procs>
```

And in a .tcl file can define the true procs (it is possible write the code in the spd <proc> but it is better for debug to separate in a Tcl file)

```
proc Cmas2d::GetMaterialsList { domNode args } {
  set dom_materials [$domNode selectNodes {//container[@n="materials"]}
  set result [list]
  foreach dom_material [$dom_materials childNodes] {
    lappend result [$dom_material @name]
  }
  return [join $result ,]
}

proc Cmas2d::EditDatabaseList { domNode dict boundary_conds args } {
  ...
}
```

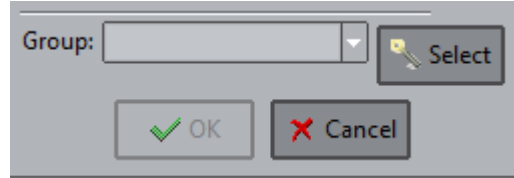
condition

<condition>

It contains some inputs with values and can be applied to groups. For each applied group, a dependent set of values will be created that belong to that group, for this condition.

A group is a category that brings together selected entities (points, lines, surfaces and/or volumes) with identical properties. I should be noted that any entity can belong to more than one group (unlike the concept of layer, where any entity cannot belong to more than one layer). It can contain the following fields: <value>, <dependencies>, <edit_command>

On the resulting window, in addition to the inputs there will be the following set of buttons:



Button 'Select' enables to create a group and choose entities into it.

ov, ovi, i=1,2 - Indicates to which entity types can a 'condition' be applied. Can be one or several of the following values: point, line, surface, volume.

Note: usually a condition is attached to a group, and ov is used, but some special conditions are attached to two groups, then ov1 and ov2 must be used (for example could be used to define master-slave parts)

ovp, ovpi, i=1,2 - Optional, to visually show an string alternative to the ov keyword values

ov_default - Indicates the default entity type selected (used in case of ov with multiple types). Can be one, and only one of the following values: point, line, surface, volume.

ovm, ovmi, i=1,2 - Indicates to which entity can a 'condition' be applied. It can be element, node, face_element or ""

ov_element_types - Optional, to restrict the element types that could be applied. Must be a list of comma separated of the following values: linear, triangle, quadrilateral, tetrahedra, hexahedra, prism, point, pyramid, sphere, circle (by default all element types are allowed)

state - Specifies one of two states for the field: **normal**, or **hidden**. Note that hidden <container> field can be used for storing hidden values, that you do not want to show in the user interface. It also permits to handle a Tcl function, by means of square brackets.

groups_icon -It allows to put a custom image when creating groups in the data tree, with .png format. The image should be stored inside the images folder of the problem type.

allow_group_creation - It is a boolean value as a 1 or 0 that allows to specify that only existing groups can be chosen in the condition window. It is activated by default. A **groups_list** attribute must be added when it is deactivated, as a procedure returning a list of groups (i.e. groups_list="[my_groups_proc]").

Example:

```
<condition n="Point_Weight" pn="Point Weight" icon="constraints" groups_icon="groups-16">
...
</condition>
```


<symbol>

Every condition can have a symbol, that will be drawn when the user selects **Draw symbols** in the contextual menu that appears on user interactions such as right-mouse click operation.

The symbol is defined by a field **<symbol>** inside the condition. The available XML parameters are:

proc: Includes the name of a TCL proc to be defined in the TCL files of the problem types. In that proc, OpenGL is used to make the real drawing.

The procedure must return the id of a OpenGL drawing list (created by GiD_OpenGL draw -genlists 1), or a dict with some required keys depending on the orientation attribute value.

The proc is invoked adding automatically an extra argument: *valuesList* with a list of key value of the condition values stored in the tree.

orientation: can be **global**, **local**, **localLA**, **free** or have some special values **section**, **shell_thickness**, **loads** to invoke internally predefined drawings.

- **global** means that the symbol defined in the proc will be draw with its axes corresponding to that of the global axes of the model.
- **local** means that the symbol will be drawn related to a local axes system dependent on the entity.
- For lines, these local axes system will have x' axe parallel to the line. For surfaces, the z' axe will be parallel to the surface normal (for lines some extra correction of the local axes could be applied)
- **localLA** is similar to local, in this case is compulsory attach local axis to entities, implicit automatic local axis of lines and surfaces are not used (and without the extra correction in case of lines)

- **section** used for lines to draw bar section profiles in its local axes. (some extra correction of the local axes could be applied) In this case the proc must return a dict Tcl object with keys named:

- **obj** the integer representing the opengl list to be drawn.
- **shell_thickness** to draw the surface with a thickness.

In this case the proc must return a dict Tcl object with keys named:

- **thickness** with the value of the thickness associated to the surface.
- **cdg_pos** (optional) with the list of the 3 components of the center position
- **loads** to represent loads.

In this case the proc must return a dict Tcl object with keys named:

- **load_type** with possible values **global**, **global projected**, **localload_vector** with a list of 3 components of the vector

e.g. return [dict create load_type \$load_type load_vector \$load_vector]

- **free** is another special value, in this case the Tcl procedure will be called once by entity with the condition to be draw, instead of only once by group.

The proc is invoked adding automatically some extra arguments: *valuesList geom_mesh ov num pnts points ent_type center scale* this allow to know the information of the entity.

geom_mesh: GEOMETRYUSE or MESHUSE

ov: point line surface volume node element

num: <entity id>

pnts: in case of lines integer?, in case of surfaces its boundary lines and orientations <{line_1 SAME1ST|DIFF1ST} ... line_n SAME1ST|DIFF1ST>

points: in case of lines <{x1 y1 z1} {x2 y2 z2}> start and end coordinates, in case of lines the boundary point coordinates

ent_type: STLINE, ...

center: <x y z>

scale: <scale_to_draw>

The local axes are defined by a special 'classical condition' (point_Local_axes, line_Local_axes, ...) assigned to the entities, or if there is not assigned this 'local axis' then the implicit local axis for lines and surfaces is used (based in its tangent and normal respectively) There are Tcl predefined procedures to facilitate drawing with OpenGL:

- **gid_groups_conds::import_gid_mesh_as_openGL** <filename> <color_lines> <color_surfaces>

To automatically import a GiD mesh file to be drawn with OpenGL.

The filename to be read must be a GiD ASCII mesh of lines, triangles and quadrilaterals (with as few elements as as possible). This mesh could be exported from the menu Files->Export->GiD mesh. The procedure read the mesh and invoke GiD_OpenGL draw commands.

The mesh must represent a normalized shape, centered at the origin and contained in a box of size 1 for every of its dimensions (a 2x2x2 cube)

- gid_groups_conds::draw_symbol_image <image> <values_list>

To automatically import in image file to be drawn with OpenGL.

The image must use some valid image format (png, gif, jpg,...), and is expected to be inside a folder named /images of the problemtype

- The values_list argument is currently not used.

gid_groups_conds::draw_symbol_text <txt> <values_list>

To automatically print a text with OpenGL.

The values_list argument is currently not used.

Note: The package gid_draw_opengl also contains some predefined interesting procedures, and some .msh files with common symbols to be used by gid_groups_conds::import_gid_mesh_as_opengl

Example:

```
<condition n="Point_Weight" pn="Point Weight" ov="point" ovm="node" icon="constraints" help="
Concentrated mass">
  <value n="Weight" pn="Weight" v="0.0" unit_magnitude="M" units="kg" help="Specify the weight that you
want to apply"/>
  <symbol proc="gid_groups_conds::draw_symbol_image darkorange-weight-18.png" orientation="global"/>
</condition>
```

Example:

```
<condition n="Point_Weight" pn="Point Weight" ov="point" ovm="node" icon="constraints" help="
Concentrated mass">
  <value n="Weight" pn="Weight" v="0.0" unit_magnitude="M" units="kg" help="Specify the weight that you
want to apply"/>
  <symbol proc="Cmas2d::DrawSymbolWeight" orientation="global"/>
</condition>
```

And its Tcl drawing procedure, assuming that the mesh file named weight_2d.msh is located in the symbols folder or the problemtype:

```
proc Cmas2d::DrawSymbolWeight { values_list } {
  variable _opengl_draw_list
  if { ![info exists _opengl_draw_list(weight)] } {
    set _opengl_draw_list(weight) [GiD_OpenGL draw -genlists 1]
    GiD_OpenGL draw -newlist $_opengl_draw_list(weight) compile
    set filename_mesh [file join [Cmas2d::GetDir] symbols weight_2d.msh]
    gid_groups_conds::import_gid_mesh_as_opengl $filename_mesh black blue
    GiD_OpenGL draw -endlist
  }
  set weight_and_unit [lrange [lindex $values_list [lsearch -index 0 $values_list Weight]] 1 2]
  set weight [lindex $weight_and_unit 0]
  set scale [expr {$weight*0.1}]
  set transform_matrix [list $scale 0 0 0 0 $scale 0 0 0 0 $scale 0 0 0 0 1]
  set list_id [GiD_OpenGL draw -genlists 1]
  GiD_OpenGL draw -newlist $list_id compile
  GiD_OpenGL draw -pushmatrix -multmatrix $transform_matrix
  GiD_OpenGL draw -call $_opengl_draw_list(weight)
  GiD_OpenGL draw -popmatrix
  GiD_OpenGL draw -endlist
  return $list_id
}
```

blockdata

<blockdata>

Represents a set of properties with some kind of relationship. A 'blockdata' field can copy itself, to duplicate and create several sets. It can contain the following fields: <value>, <container>, <condition>, <function>, <dependencies>, and other <blockdata>

n - Name used to reference the field, especially when writing the .dat file.

name - Label that will be visualized by the user. It can be translated.

sequence - It allows a 'blockdata' field to be duplicated and copied by the user in order to create several sets. Like several load cases with its 'value' and 'condition' included. If it has the 'sequence' parameter activated, it is possible for the user to create consecutive repetitions of the full block data in order to represent, for example, loadcases with all its conditions inside.

sequence_type - It can be:

any - The list can be void (this is the default)

non_void_disabled - At least there needs to be one element. It can be disabled.

non_void_deactivated - At least there needs to be one element. It can be deactivated.

editable_name - can be void "" or 'unique'. The 'unique' means that it is not possible to use the same name ('pn' field), for two different 'sequence' 'blockdata'.

morebutton - It is a boolean value as a 1 or 0 to show a 'More...' button in fields of type 'blockdata'. It is activated by default. For more information about this attribute, see the section [Import export materials](#)

can_delete_last_item - A single blockdata could be deleted.

before_update_proc - It allows to call a TCL procedure, when choosing this field in data tree. Square brackets are not supported.

update_proc - It calls a TCL procedure, when clicking on the 'Ok' button in the window. Square brackets are not supported.

icon - It allows to put an image in .png format in the data tree. The image should be stored inside the images folder of the problem type. A Tcl procedure can be used to return the name at runtime.

help - It displays a pop-up window of help information related to the task the user is performing.

state - Specifies one of two states for the field: **normal**, or **hidden**. Note that hidden <container> field can be used for storing hidden values, that you do not want to show in the user interface. It also permits to handle a Tcl function, by means of square brackets.

allow_import - It is a boolean value as a 1 or 0 that allows to add the 'Import/export materials' item in the contextual menu for a specific 'blockdata' field. It is deactivated by default. For more information about this attribute, see the section [Import export materials](#)

edit_command

<edit_command>

It adds a button in the window that allows to call a TCL procedure when necessary. The parameters are as follows,

n - Name used to reference the field, especially when writing the .dat file.

pn - Label that will be visualized by the user. It can be translated.

icon - It allows to put an image in .png format in the data tree. The image should be stored inside the images folder of the problem type.

proc - Permits to define a TCL proc. The code will receive an implicit argument with name 'domNode' that represents the TDOM node in the calling field context. Square brackets are not necessary in edit_command field.

help - It displays a pop-up window of help information related to the task the user is performing.

dependencies

<dependencies>

When a field of type 'value' changes its value 'v', the <dependencies> field allows to force a change in other values.

The parameters are as follows,

node - It is the xpath expression to the node that should be updated.

value - Field <value> allows to define a condition to execute a dependence.

att, atti, i=1,2 - Indicates to which attributes of a node affect a change in one value.

v,vi, i=1,2 - Indicates the new value for atti, i=1,2. It can be normal, hidden or disabled.

default - Default value for the condition. It permits to execute a dependence.

actualize - This only updates a specified field in data tree. Note that only this specified field will be refreshed in the user interface, and not the whole data tree. It is a boolean value as a 1 or 0 that indicates if it is activated or deactivated.

actualize_tree - It updates the information in the whole data tree, and automatically refresh data shown in the user interface. It is a boolean value as a 1 or 0 that indicates if it is activated or deactivated. If the data source is changed, such as new fields have been added or data values and field have been modified, all the user interface will reflect those changes. Furthermore, all the TCL procedures defined in the data tree will be called and the whole data tree will be refreshed. Therefore, this instruction must be carried out only when necessary.

Example: when the user select 'No' in the combo 'show_weight' the item value with name='weight' will be hidden, and when select 'Yes' is showed.

Note the use of a relative xpath node="..value[@n='weight']" to specify the xml node to be changed by the dependency:

```

<value n="show_weight" pn="Show the weight" values="Yes,No" actualize_tree="1">
  <dependencies value='Yes' node="../value[@n='weight']" att1="state" v1='normal' />
  <dependencies value='No' node="../value[@n='weight']" att1="state" v1='hidden' />
</value>

<value n="weight" pn="Weight" v="0.0" unit_magnitude="M" units="kg" help="Specify the weight that you
want to apply"/>

```

groups

<groups>

Initial groups field. It is always an empty field in the .spd file.

groups_types

<groups_types>

Represents the types of groups. It can contain the following fields: <group_type>

The parameters are as follows,

editable - It is a boolean value as a 1 or 0 that indicates if the entry could be changed or not. If it is activated (1) the entry may not be changed.

group_type

<group_type>

Main configuration field of the group types.

pn - Label that will be visualized by the user. It can be translated.

default - It is a boolean value as a 1 or 0 that indicates the default group type.

auto_from_bc - It is a boolean value as a 1 or 0 that indicates that the group type is a boundary condition (bc).

Example:

```

<group_type pn="normal" default="1" />
<group_type pn="BC" auto_from_bc="1" />

```

units

<units>

Main unit field. It can contain the following fields:

<unit_magnitude>

<unit_mesh>

<units_system>

unit_mesh

<unit_mesh>

Attributes

n Length unit name to be used for the mesh

units_system**<units_system>****Unit system definition**

The field used to choose the unit system is special. It has the attribute `units_system_definition="1"`, it does not contain any "v" attribute, and it contains a unique dependency related to the unit fields.

n unit name

pn printed name

unit_mesh_definition The field used to choose the mesh unit is also special. It has the attribute `unit_mesh_definition="1"`, and it does not contain any "v" attribute or dependencies.

unit_definition The fields used to choose the default units in the GUI are also special. They contain the attribute called `unit_definition="magnitude"` being magnitude the name 'n' to be used in that field. It is important to note that these kind of fields does not contain dependencies.

Here's an example of node type "container" in the .spd file, which allows to choose the geometry units and the general units, as follows,

```
<container n="units" pn="Units" icon="units-16" help="Units definition">
  <value n="units_mesh" pn="Geometry units" unit_mesh_definition="1" icon="units-16"/>
  <value n="units_system" pn="Unit system" units_system_definition="1" icon="units-16" state="
hidden">
    <dependencies node="/*[@unit_definition or @unit_mesh_definition='1']" att1="
change_units_system" vl="{@v}"/>
  </value>
  <container n="basic_units" pn="General units" icon="units-16" state="normal">
    <value n="units_length" pn="Length" unit_definition="L"/>
    <value n="units_mass" pn="Mass" unit_definition="M"/>
    <value n="units_force" pn="Force" unit_definition="F"/>
    <value n="units_pressure" pn="Pressure" unit_definition="P"/>
    <value n="units_temperature" pn="Temperature" unit_definition="Temp"/>
    <value n="units_time" pn="Time" unit_definition="T"/>
  </container>
</container>
```

International and Imperial systems

There are two primary systems used to define units: the international and imperial systems, as seen below. The international system of units is the modern standardized form of the metric system. It sets standard measurements and conversions, and is the most commonly and universally accepted system of units. The imperial system, also known as British Imperial, is the system of units first defined in the British Weights and Measures Act of 1824, which was later refined and reduced.

Unit system	n
Int. system (SI)	SI
Imperial system	imperial

- The fields of type <unit> can contain the attribute called units_system, as follows,

units_system - There are two systems used to define units, the possible values are:

SI - International system

imperial - Imperial system

unit_magnitude

<unit_magnitude>

It can contain the following fields: <unit>

The parameters are as follows,

n - Name used to reference the field, especially when writing the .dat file.

pn - Label that will be visualized by the user. It can be translated.

default - Default unit for a specific magnitude..

SI_base -Unit based in the international system units for a specific magnitude.

active - It is the default unit shown in the user interface, for a specific magnitude.

Defining magnitudes and units

The attributes involved in any field of the .spd file are 'unit_magnitude' and 'units'.

- unit_magnitude: Its value relates to the name 'n' used to reference the unit field. Please see the table below for a complete list of all the names available.
- units: Its value is the default unit shown in the GUI, which could be changed, if desired.

Some functions are useful for writing data with units defined into the calculation file. For more information about this issue, see the section called [Writing the input file for calculation](#).

Example:

```
<value n="ini_temp" pn="Initial temperature" v="20.0" unit_magnitude="Temp" units="°C" />
```

Note: For instance, it is convenient to change magnitudes like unit_magnitude="F/L^2" by unit_magnitude="P".

The table below gives a summary of the names used for all unit magnitudes available.

pn	n	Unit by default
Length	L	m
Mass	M	kg
Time	T	s
Temperature	Temp	K
Frequency	Frequency	Hz
Force	F	N

Pressure	P	Pa
Energy	Energy	J
Power	Power	W
Angle	Angle	rad
Solid_angle	Solid_angle	sr
Velocity	Velocity	m/s
Acceleration	Acceleration	m/s^2
Area	Area	m^2
Volume	Volume	m^3
Density	Density	kg/m^3
Electric current	Electric_current	A
Amount of substance	Amount_of_substance	mol
Luminous intensity	Luminous_intensity	cd
Electric charge	Electric_charge	C
Electric potential	Electric_potential	V
Capacitance	Capacitance	F
Electric resistance	Electric_resistance	?
Electric conductance	Electric_conductance	S
Magnetic flux	Magnetic_flux	Wb
Magnetic flux density	Magnetic_flux_density	T
Inductance	Inductance	H
Delta phi	DeltaPhi	$\Delta\phi$
Delta temperature	DeltaTemp	ΔK
Luminous flux	Luminous_flux	lm
Illuminance	Illuminance	lx
KinematicViscosity	KinematicViscosity	m^2/s
Viscosity	Viscosity	Pa*s
Permeability	Permeability	m^2

The default magnitudes and conversion factors could be seen at the file scripts\customLib\customLib\units.xml

*unit***<unit>**

A unit of measurement is a definite magnitude of a physical quantity, defined and adopted by convention or by law, that is used as a standard for measurement of the same physical quantity. Any other value of the physical quantity can be expressed as a simple multiple of the unit of measurement. For example, length is a physical quantity, and metre is a unit of length that represents a definite predetermined length.

The node <units> in the *.spd file can be used to change some defaults or to add specialized units, which in principle are not supported. The structure and contents of this subtree <units> requires to have the format as further detailed in this document. The following attributes are available:

n - name

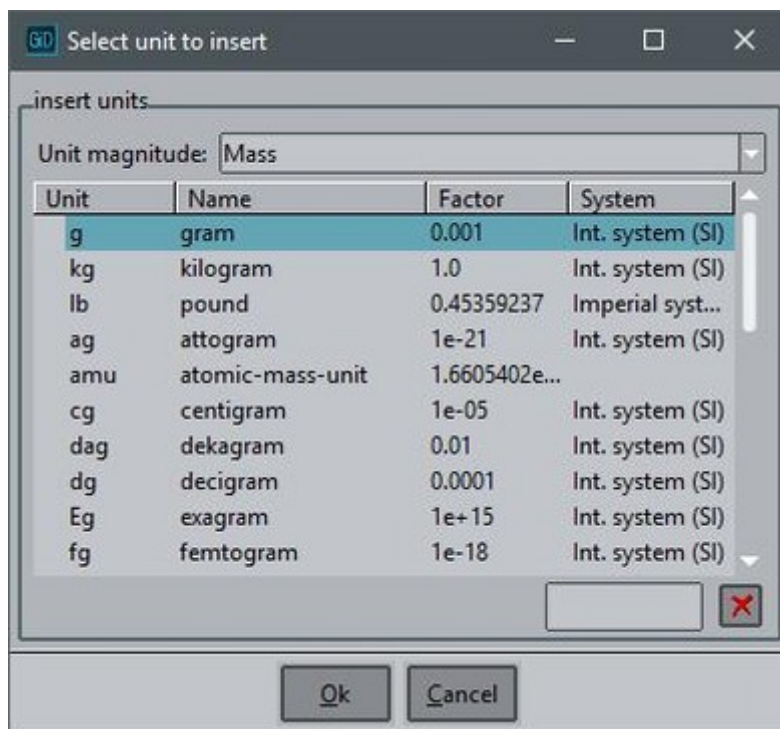
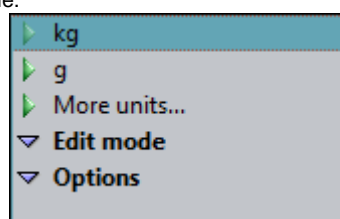
pn - printed name (decorated and translated)

p - Priority of the unit, it is regarded as more important than others units. The possible values for 'p' are 1, 2 and 3, as follows,

1 - It gives maximum priority, and therefore a unit with p="1" is the default unit in the GUI.

2 - A unit with p="2" is always shown in the combobox of below to facilitate the selection of the most used units.

3 - It gives lowest priority, and therefore a unit with p="3" could only be chosen when clicking on 'More units...' option in the combobox of below, which enables the user to choose the unit from a table.



factor - The conversion factor used to multiply a quantity when converting from one system of units to another. It is the mathematical tool for converting between units of measurement [$1 \text{ unit} = \text{factor} * \text{unit}(\text{SI})$]. Example: $1 \text{ mm} = 10 \times 10^{-3} \text{ m}$.

help - To make easier to the user to identify the unit.

Example of <units> field: Species concentration and reference variable are not supported and therefore, both magnitudes are defined in the *.spd file, as follows,


```

<units>
<unit_magnitude n="Reference_variable_unit" pn="Reference variable unit" default="U" SI_base="U"
active="1">
  <unit n="U" pn="ReferenceUnit" p="2" factor="1.0"/>
  <unit n="ppm" pn="parts-per-million" p="2" factor="1.0e-6" help="parts-per-million"/>
  <unit n="ppb" pn="parts-per-billion" p="2" factor="1.0e-9" help="parts-per-billion"/>
  <unit n="%" pn="per-one-hundred" p="2" factor="0.01" help="per-one-hundred"/>
  <unit n="%0" pn="per-one-thousand" p="2" factor="0.001" help="per-one-thousand"/>
  <unit n="Np" pn="Nepper" p="2" factor="1.0" help="Nepper"/>
</unit_magnitude>
<unit_magnitude n="Species_concentration" pn="Species concentration" default="C" SI_base="C" active="1">
  <unit n="C" pn="ReferenceUnit" p="2" factor="1"/>
  <unit n="ppm" pn="parts-per-million" p="2" factor="1.0e-6" help="parts-per-million"/>
  <unit n="ppb" pn="parts-per-billion" p="2" factor="1.0e-9" help="parts-per-billion"/>
  <unit n="%" pn="per-one-hundred" p="2" factor="0.01" help="per-one-hundred"/>
  <unit n="%0" pn="per-one-thousand" p="2" factor="0.001" help="per-one-thousand"/>
  <unit n="mol" pn="mole" p="2" factor="1.0"/>
</unit_magnitude>
</units>

```

Style

<style>

Optional item, for some GUI options
The parameters are as follows,

show_menubutton_search - 0 or 1: to show or hide the search button on the top of the customLib tree widget

showlines - 0 or 1 : to show or hide the lines of the tree

show_menubutton_about - 0 or 1: to show or hide an about customLib button.

function

<function>

Main function field, which contains a function variable field called <functionVariable>.
It permits to create or edit a function defined by points for a determined field.
e.g. for a property variable with the temperature.

It can contain the following fields: <functionVariable>

functionVariable

<functionVariable>

It allows to define the default values of a function defined by xy points. The parameters are as follows,

n - Name used to reference the field, especially when writing the .dat file.

pn - Label that will be visualized by the user. It can be translated.

variable - Name of the variable shown in the GUI.

units - Its value is the default unit shown in the GUI, which could be changed, if desired.

Example:

```

<value n="dens" pn="Density" min_two_pnts="1" help="Density of Steel" unit_magnitude="M/L^3" units="kg
/m^3" function="[density_function]" function_func=" v="1.0">
  <function>
    <functionVariable n="interpolator_func" pn="Interpolation function" variable="x" units="°C">
      <value n="point" pn="Point" v="20.0,7830.0"/>
      <value n="point" pn="Point" v="600.0,7644.0"/>
    </functionVariable>
  </function>
</value>

<procs>
  <proc n='density_function' args='args'>
    <![CDATA[
      MyDensityFunction $domNode $args
    ]]>
  </proc>
</procs>

```

In this example the proc density_function referenced by the 'function' attribute is implemented in the xml .spd file but it is very simple and only invokes another procedure named MyDensityFunction adding some arguments. The body of the tcl procedure could be implemented separately in a .tcl file (sourced in the problemtype), this facilitate its edition and debug.

```

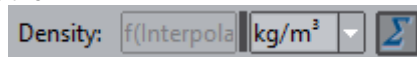
proc MyDensityFunction { domNode args } {
  set result [join [list scalar [list interpolator_func x x Temp]] ,]
  return $result
}

```

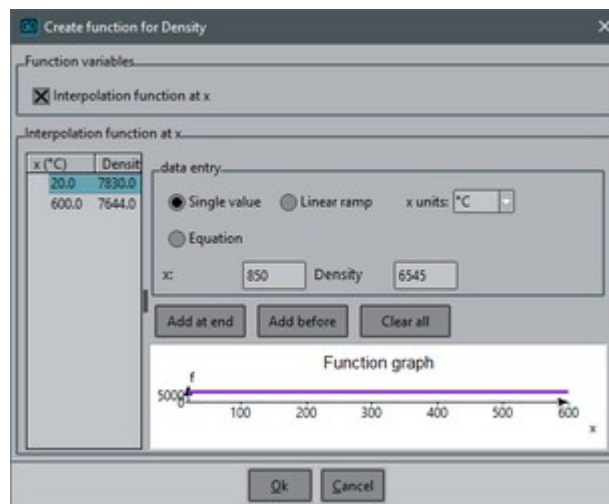
The proc referenced by the 'function' must return as value something like this: scalar, interpolator_func x x Temp

where scalar is a keyword,
 interpolator_func is the same name used in <functionVariable n="interpolator_func" ...
 x is the same variable name used in <functionVariable ... variable="x" ...
 Temp is the magnitude according to the units used in <functionVariable ... units="°C">

The graphical interface will show something like this



and pressing the right button will open a window to edit the xy graph:



Annex I: Using functions

The following section is of purely practical nature. It contains examples of the most common functions from the areas CustomLib deals with.

Example no. 1: Using functions based on Interpolated Data

The linear interpolation uses two or more pairs of input data points that approximate a function. To define functions based on interpolated data, use the function dialog box, which can be opened from the data tree automatically clicking on the button ? as follows,

Discrete load: 

An example of the <value/> node field in the .spd file is the following:

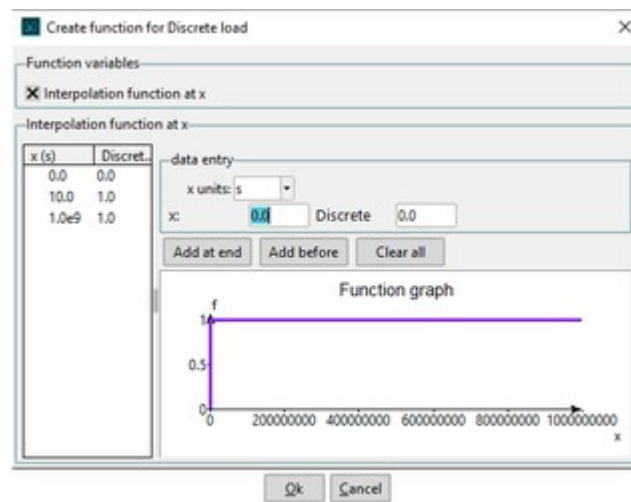
```
<value n="disc_load" pn="Discrete load" min_two_pnts="1" single_value="1" function="
[loads_function_time]" function_func="" state="normal" v="1.0">
  <function>
    <functionVariable n="interpolator_func" pn="Interpolation function" variable="x" units="s"
  >
    <value n="point" pn="Point" v="0.0,0.0"/>
    <value n="point" pn="Point" v="10.0,1.0"/>
    <value n="point" pn="Point" v="1.0e9,1.0"/>
  </functionVariable>
</function>
</value>

<proc n="loads_function_time" args="">
  return [cmas2d_CustomLIB::chk_loads_function_time $domNode]
</proc>

proc cmass2d_CustomLIB::chk_loads_function_time { domNode } {
  set loads [list [list scalar]]
  lappend loads [list interpolator_func x x T]
  return [join $loads ,]
}
```

Notice that both <function/> and <functionVariable/> nodes are required to define an interpolation function.

Note: The nodes of type <value n="point"/> can be omitted to start with an empty table of input data points.

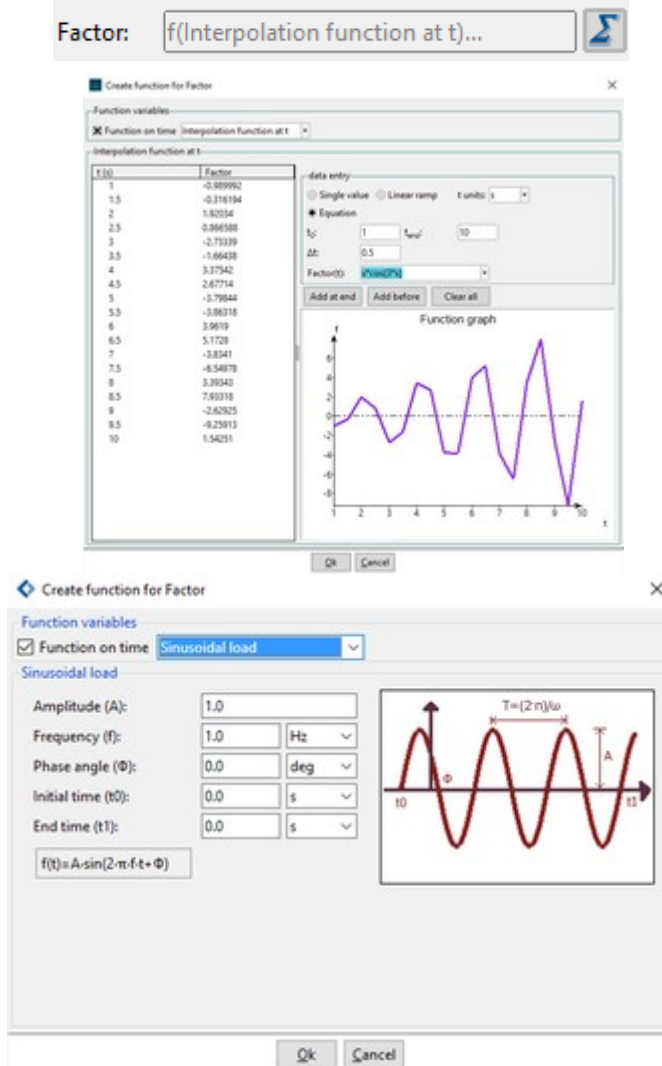


Example no. 2: Using functions based on Interpolated Data, methods for entering data

It is possible to select automatically the method for entering data (Single value, Linear ramp or Equation). For functions of one variable, you can select between the following interpolation methods by default:

- Single value: The linear interpolation uses two or more pairs of input data points that approximate a function.
- Linear ramp: Calculates two or more 'Linear ramp' pairs of input data points that approximate a function.
- Equation: Calculates two or more pairs of input data points that satisfy a given equation.

Use the function dialog box, which can be opened from the data tree clicking on the button ? as follows,



```
<value n="Factor" pn="Factor" v="1.0" help="This factor, that can be a number or a formula,
multiplies the vector load" function="[loads_function Punctual_Load]"
function_func="CompassFEM::function_loads"/>

<proc n="loads_function" args="load_name">
    return [chk_loads_function $domNode $load_name]
</proc>

proc function_loads { ftype what n pn frame domNode funcNode \
    units_var ov_var } {
    switch $ftype {
        sinusoidal_load {
            return [function_loads_sinusoidal $what $n $pn $frame $domNode \
                $funcNode $units_var $ov_var]
        }
        custom_editor {
            return [function_custom_editor $what $n $pn $frame $domNode \
                $funcNode $units_var $ov_var]
        }
    }
}

proc chk_loads_function {domNode load_name} {
    set loads [list [list scalar]]
    lappend loads [list sinusoidal_load t]
    return [join $loads ,]
}

proc function_custom_editor { what n pn f domNode funcNode \
    units_var ov_var } {
```

```

    # Create your own child window implemented in Tcl/Tk..
}

proc function_loads_sinusoidal { what n pn f domNode funcNode \
    units_var ov_var } {

    global ProblemTypePriv

    switch $what {
        create {
            set help [= "This is a a sinusoidal load that depends on the time"]

            set f1 [ttk::frame $f.f1]

            set idx 0
            foreach i [list amplitude circular_frequency phase_angle initial_time end_time] \
                n [list [= "Amplitude (%s)" A] \
                    [= "Frequency (%s)" f] [= "Phase angle (%s)" \u3a6] \
                    [= "Initial time (%s)" t0] [= "End time (%s)" t1]] \
                m [list - Frequency Rotation T T] \
                u [list - 1/s rad s s] \
                v [list 1.0 1.0 0.0 0.0 0.0] {

                ttk::label $f1.l$idx -text $n:
                gid_groups_conds::register_popup_help $f1.l$idx $help

                if { $m ne "-" } {
                    set ProblemTypePriv($i) $v
                    set ProblemTypePriv($i)_units $u

                    gid_groups_conds::entry_units $f1.sw$idx \
                        -unit_magnitude $m \
                        -value_variable ProblemTypePriv($i) \
                        -units_variable ProblemTypePriv($i)_units
                } else {
                    ttk::entry $f1.sw$idx -textvariable ProblemTypePriv($i)
                    set ProblemTypePriv($i) $v
                }
                grid $f1.l$idx $f1.sw$idx -sticky w -padx 2 -pady 2
                grid configure $f1.sw$idx -sticky ew
                incr idx
            }

            if { ![info exists ProblemTypePriv(l_sin_img)] } {
                set ProblemTypePriv(l_sin_img) [image create photo \
                    -file [file join $ProblemTypePriv(problemtypedir) images \
                        sinusoidal_load.gif]]
            }

            ttk::label $f1.l -text "f(t)=A\u00b7sin(2\u00b7\u03C0\u00b7f\u00b7t+\u03a6)" \
                -relief solid -padding 3 -width 20 \
                -anchor w
            grid $f1.l -sticky w -padx 6 -pady 6

            label $f.image -image $ProblemTypePriv(l_sin_img) -bd 1 -relief solid

            grid $f1 $f.image -sticky nw -padx 8 -pady 2
            grid configure $f.image -sticky new

            grid columnconfigure $f 1 -weight 1
            grid rowconfigure $f 0 -weight 1

            if { $funcNode ne "" } {
                set xp {functionVariable[@variable="t" and
                    @n="sinusoidal_load"]}
                set fvarNode [$funcNode selectNodes $xp]
            } else { set fvarNode "" }
            if { $fvarNode ne "" } {
                foreach i [list amplitude circular_frequency phase_angle initial_time
                    end_time] \
                    pn [list [= "Amplitude (%s)" A] \
                        [= "Frequency (%s)" f] [= "Phase angle (%s)" \u3a6] \
                        [= "Initial time (%s)" t0] [= "End time (%s)" t1]] \
                    m [list - Frequency Rotation T T] {

                    set xp [format_xpath {string(value[@n=%s]/@v)} $i]
                    set ProblemTypePriv($i) [$fvarNode selectNodes $xp]
                    if { $m ne "-" } {
                        set xp [format_xpath {string(value[@n=%s]/@units)} $i]
                        set ProblemTypePriv($i)_units \
                            [gid_groups_conds::units_to_nice_units [$fvarNode selectNodes
                                $xp]]
                    }
                }
            }
            return [= "Sinusoidal load"]
        }
    }
    apply {
        set idx 0
        set f1 $f.f1
        foreach i [list amplitude circular_frequency phase_angle initial_time end_time] \
            pn [list [= "Amplitude (%s)" A] \
                [= "Frequency (%s)" f] [= "Phase angle (%s)" \u3a6] \
                [= "Initial time (%s)" t0] [= "End time (%s)" t1]] \

```

```

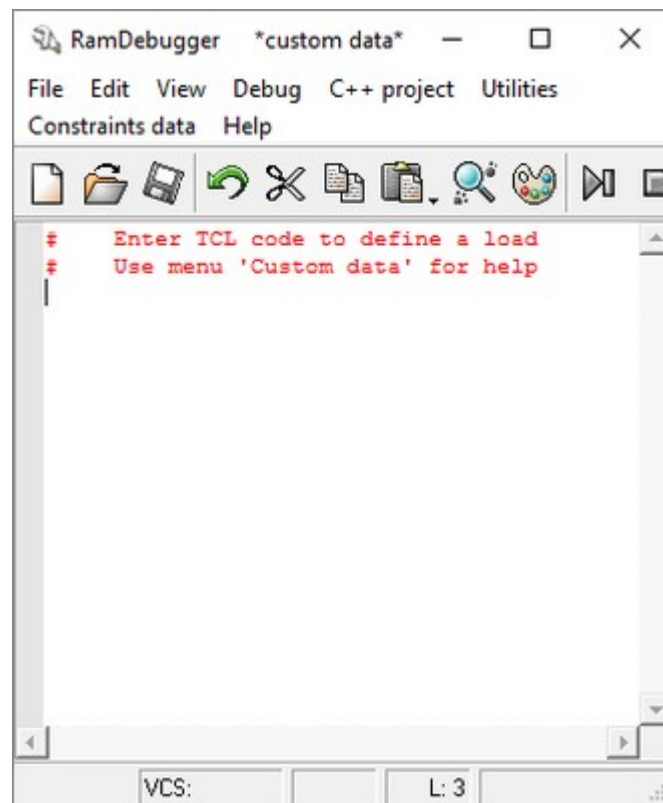
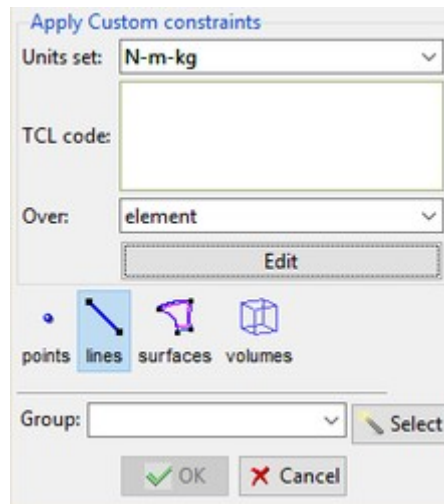
m [list - Rotation Rotation T T] {
  if { ![string is double -strict $ProblemTypePriv($i)] } {
    if { $m ne "-" } { set e $fl.sw$idx.e } else { set e $fl.sw$idx }
    tk::TabToWindow $e
    error [= "Value %s is not OK" $pn]
  }
  incr idx
}
set xp {functionVariable[@variable="t"]}
if { [$funcNode selectNodes $xp] eq "" } {
  set fvarNode [$funcNode appendChildTag functionVariable]
} else {
  set fvarNode [$funcNode selectNodes $xp]
  foreach i [$fvarNode childNodes] { $i delete }
}
$fvarNode setAttribute n sinusoidal_load pn [= "Sinusoidal load"] \
  variable "t"
foreach i [list amplitude circular_frequency phase_angle initial_time end_time] \
  pn [list [= "Amplitude (%s)" A] \
    [= "Frequency (%s)" f] [= "Phase angle (%s)" \u3a6] \
    [= "Initial time (%s)" t0] [= "End time (%s)" t1]] \
  m [list - Frequency Rotation T T] {

  set v [$fvarNode appendChildTag value [list attributes() \
    n $i pn $pn v $ProblemTypePriv($i)]]
  if { $m ne "-" } {
    set newunits [gid_groups_conds::nice_units_to_units \
      $ProblemTypePriv($i)_units]
    $v setAttribute unit_magnitude $m \
      units $newunits
  }
}
}
}
}
}

```

Example no. 3: Editable multiline text for entering mathematical expressions

The following example allows to enter a numerical or math expression, such as a formula, into an editable multiline text box. CustomLib will display automatically the resulting math expression in the data tree. Moreover, the user will be able to overwrite this expression with a different formula using Ramdebugger editor. The 'Edit' button opens Ramdebugger, the graphical debugger for the scripting language Tcl-Tk.



It is possible to create and modify the function inside Ramdebugger editor. The TCL-TK source code is colorized and supports automatic

The <condition/> node field in the .spd file is as follows,

```
<condition n="custom_constraints" pn="Custom constraints" ov="point,line,surface,volume"
state="normal" ov_default="surface" ovm="" help="A constraint defined by using the TCL
extension language">
  <dependencies node="value[@n='over_what']" att1="v" v1="node" att2="state"
v2="disabled" condition="@ov='point'"/>
  <dependencies node="value[@n='over_what']" att1="state" v1="normal"
condition="@ov='line' or @ov='surface' or @ov='volume'"/>
  <value n="units" pn="Units set" v="N-m-kg" values="N-m-kg,N-cm-kg,N-mm-kg"/>
  <value n="tcl_code" pn="TCL code" v="" fieldtype="long text"/>
  <value n="over_what" pn="Over" v="node" values="node,element"
dict="node,node,element,element" editable="0" help="Condition will be applied to either over
every selected element or every selected node"/>
  <edit_command n="edit_custom_data" pn="Edit" proc="edit_custom_data constraints"
edit_type="callback"/>
</condition>

<proc n="edit_custom_data" args="custom_type callback">
  custom_data_edit $domNode $dict $custom_type $callback
</proc>

proc custom_data_edit { domNode dict custom_type callback } {
```

```

if { ![interp exists ramdebugger] } { interp create ramdebugger }
ramdebugger eval [list set argv [list -noprefs -rgeometry 600x500+300+300 \
-onlytext]]
package require RamDebugger

if { [dict exists $dict tcl_code] } {
    set data [string trim [dict get $dict tcl_code]]
} else {
    set xp {string(value[@n="tcl_code"]/@v)}
    set data [string trim [$domNode selectNodes $xp]]
}
if { $data eq "" } {
    set data [format "#      %s\n#      %s\n" [= "Enter TCL code to define a load"] \
[= "Use menu 'Custom data' for help"]]
}
interp alias ramdebugger EditEndInRamDebugger "" CompassFEM::custom_data_edit_update \
$callback
ramdebugger eval [list RamDebugger::OpenFileSaveHandler "*custom data*" \
$data EditEndInRamDebugger]

set menu ""

set file [file join $::ProblemTypePriv(problemtypedir) scripts custom_bc_examples.txt]
set fin [open $file r]
set data [read $fin]
close $fin

set elms ""
set rex {\s*\s*type:\s*(\S.*\S)\s+name:\s*(\S.*\S)\s*$}
foreach "idxsL idxsType idxsName" [regexp -inline -all -line -indices $rex $data] {
    if { $elms ne "" } {
        lset elms end end [expr {[lindex $idxsL 0]-1}]
    }
    set type [eval [list string range $data] $idxsType]
    set name [eval [list string range $data] $idxsName]
    lappend elms [list $type $name [lindex $idxsL 0] ""]
}
if { $elms ne "" } {
    lset elms end end [expr {[string length $data]-1}]
}
set subMenus ""
foreach i $elms {
    foreach "type name idx1 idx2" $i break
    lappend subMenus [list command "$name ($type)" {} \
[= "View example '%s' (%s)" $name $type] "" \
-command [list open_new_window_show $idx1 $idx2]]
}
lappend subMenus [list separator]
set idx2 [expr {[string length $data]-1}]
lappend subMenus [list command [= "All examples"] {} \
[= "View all examples"] "" \
-command [list open_new_window_show 0 $idx2]]
lappend menu [list cascaded [= Examples] {} examples 0 $subMenus]

set cmd {
    set ip [RamDebugger::OpenFileInNewWindow -ask_for_file 0]
    set fin [open %FILE r]
    set data [read $fin]
    close $fin
    set d [string range $data $idx1 $idx2]
    $ip eval [list $::RamDebugger::text insert insert $d]
    $ip eval [list RamDebugger::MarkAsNotModified]
}
set cmd [string map [list %FILE [list $file]] $cmd]
ramdebugger eval [list proc open_new_window_show [list idx1 idx2] $cmd]

lappend menu [list separator]

foreach "n nargs" [list coords 2 conec 1 nnode 0 elem_num 0 elem_normal 1 \
epsilon 0 facLoad 0] {
    set name [= "Insert command '%s'" $n]
    set cmd {
        set t $::RamDebugger::text
        $t insert insert [list %N%]
        $t insert insert "("
        $t insert insert [string repeat , %NARGS%]
        $t insert insert ")"
        $t mark set insert "insert-[expr {%NARGS%+1}]c"
    }
    set cmd [string map [list %N% $n %NARGS% $nargs] \
$cmd]
    lappend menu [list command $name "" "" "" -command $cmd]
}
lappend menu [list separator]

foreach "n nfull" [list \
add_to_load_vector "add_to_load_vector ?-substitute? ?-local? nodenum loadvector" \
addload "addload ?-local? pressure|triangular_pressure|punctual_load args"] {
    set name [= "Insert command '%s'" $n]
    set cmd {
        $::RamDebugger::text insert insert [list %NFULL%]
    }
}

```

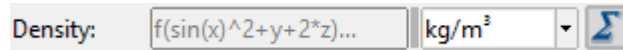


```
    }
    set cmd [string map [list %NFULL% $nfull] $cmd]
    lappend menu [list command $name " " " " -command $cmd]
}
switch $custom_type {
  constraints { set title [= "Constraints data"] }
  properties { set title [= "Properties data"] }
  loads { set title [= "Loads data"] }
  default { set title [= "Custom data"] }
}
ramdebugger eval [list RamDebugger::AddCustomFileTypeMenu $title $menu]
}
```

Example no. 4: Create your own custom editor window for entering mathematical expressions

The user can create any number of custom windows in the user interface. A child window is opened automatically to display information to the user and/or get information from the user. This is a great way to add custom windows for your own purposes.

The Function Editor window can be opened by locating in the data tree the field to modify, and choosing the edit button (e.g. ?).



This will open the Function Editor window and load the current expression into it, which can be empty.

The <value/> node field in the .spd file is as follows,

```
<value n="Density" pn="Density" v="" unit_magnitude="M/L^3" units="kg/m^3" help="Density of the fluid"
function="[loads_function editor]" pn_function="Density" function_func="function_loads"/>

proc function_loads { ftype what n pn frame domNode funcNode \
    units_var ov_var } {
    switch $ftype {
        editor {
            return [function_editor $what $n $pn $frame $domNode \
                $funcNode $units_var $ov_var]
        }
    }
}

proc function_editor { what n pn f domNode funcNode \
    units_var ov_var } {
    # Create your own child window implemented in Tcl/Tk.
}

<proc n="loads_function" args="load_name">
    return [chk_loads_function $domNode $load_name]
</proc>

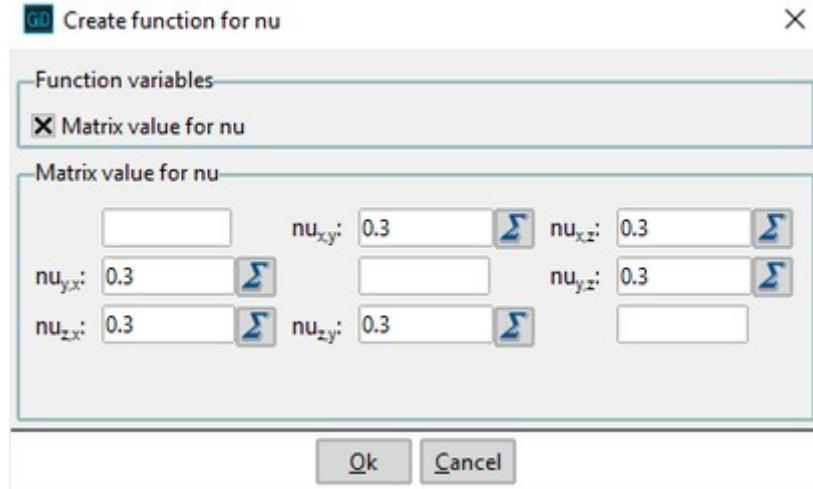
proc chk_loads_function { domNode load_name } {
    set loads [list [list scalar]]
    if { [lsearch "editor" $load_name] != -1 } {
        lappend loads [list editor ""]
    }
    return [join $loads ,]
}
return [join $loads ,]
```

Annex II: Using matrices

CustomLib also allows to enter square matrices of order n automatically. The Matrix Editor window is opened by locating in the data tree the field to modify, and choosing the edit button (e.g. [x]) as follows,

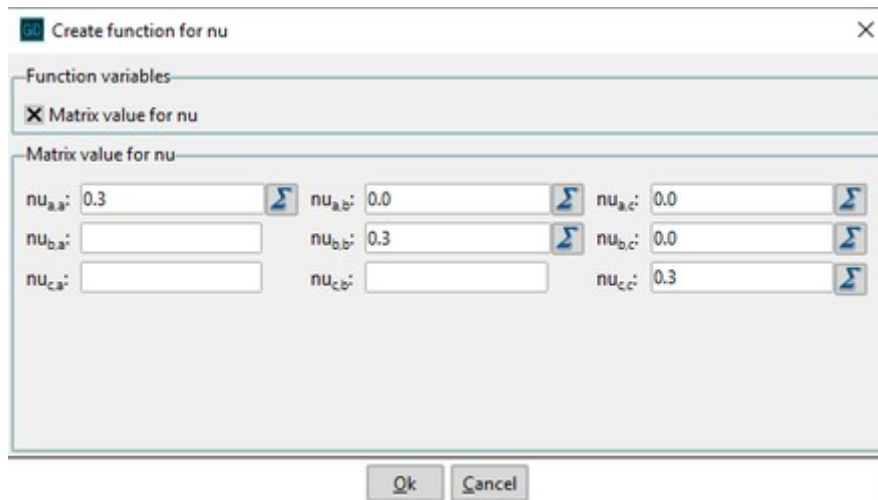


This will open the Matrix Editor window and load the current expression into it, which can be empty.



The <value/> node field in the .spd file is the following:

```
<value n="nu" pn="nu" function="matrix_func,scalar" dimension_function="3" state="normal"
symmetric_function="0" has_diag="0" components_function="x,y,z" v="0.3" function_func="loads_function"
help="Poisson coefficient"/>
```



The <value/> node field in the .spd file is the following:

```
<value n="nu_s" pn="nu" function="matrix_func,scalar" dimension_function="3" state="normal"
symmetric_function="1" has_diag="1" components_function="a,b,c" v="0.3" help="Poisson coefficient"/>
```

The parameters are as follows,

- **dimension_function**: Determine the dimensions of the given square matrix.
- **symmetric_function**: It is a boolean value as a 1 or 0. It allows to indicate that it is given only the upper triangular part of a symmetric matrix.
- **components_function**: List of numbers or names. The natural way to refer to rows and columns in a matrix is via the row and column numbers. However, the user can also give names to these entities.
- **has_diag**: It is a boolean value as a 1 or 0, that allows to indicate if it is a diagonal matrix.

```
<proc n="loads_function" args="">
  return [cmas2d_CustomLIB::chk_loads_function $domNode]
</proc>

proc cmass2d_CustomLIB::chk_loads_function { domNode } {
  set loads [list [list scalar]]
  lappend loads [list interpolator_func x x T]
  return [join $loads ,]
}
```

Access to tree data information

This document shows an example about the access to the data stored in the customlib tree. We are using the cmas2d_customlib tree as example.

The customlib tree is stored in a xml document. To manage an xml object, we use the tdom functions, available in <https://docs.activestate.com/activetcl/8.6/tcl/tdom>



To get the value of the weight applied to the group Point Weight Auto1, we need to define the xpath to the value, and ask it to the document.

```
set document [${::gid_groups_conds::doc} documentElement]
set xpath {/cmas2d_customlib_data/condition[@n='Point_Weight']/group[@name='Point Weight Auto1']/value[@n='Weight']}
set xml_node [$document selectNodes $xpath]
set value [get_domnode_attribute $xml_node v]
```

To get all the groups assigned to that condition:

```
set document [${::gid_groups_conds::doc} documentElement]
set xpath {/cmas2d_customlib_data/condition[@n='Point_Weight']/group}
foreach group_node [$document selectNodes $xpath] {
    set group_id [get_domnode_attribute $group_node name]
}
```

Note: To see the whole xml document, execute

```
W [[$::gid_groups_conds::doc] documentElement] asXML]
```

Xpath

XPath short description

(see https://www.w3schools.com/xml/xpath_intro.asp)

•Selecting XML nodes

XPath uses path expressions to select nodes in an XML document. The node is selected by following a path or steps. The most useful path expressions are listed below:

Expression	Description
<i>nodename</i>	Selects all nodes with the name " <i>nodename</i> "
/	Selects from the root node
//	Selects nodes in the document from the current node that match the selection no matter where they are
.	Selects the current node
..	Selects the parent of the current node
@	Selects attributes

•Predicates

Predicates are used to find a specific node or a node that contains a specific value. Predicates are always embedded in square brackets. In the table below we have listed some path expressions with predicates and the result of the expressions:

/bookstore/book[1]	Selects the first book element that is the child of the bookstore element. Note: In IE 5,6,7,8,9 first node is[0], but according to W3C, it is [1]. To solve this problem in IE, set the SelectionLanguage to XPath: <i>In JavaScript: xml.setProperty("SelectionLanguage","XPath");</i>
/bookstore/book[last()]	Selects the last book element that is the child of the bookstore element
/bookstore/book[last()-1]	Selects the last but one book element that is the child of the bookstore element
/bookstore/book[position()<3]	Selects the first two book elements that are children of the bookstore element
//title[@lang]	Selects all the title elements that have an attribute named lang
//title[@lang='en']	Selects all the title elements that have a "lang" attribute with a value of "en"
/bookstore/book[price>35.00]	Selects all the book elements of the bookstore element that have a price element with a value greater than 35.00
/bookstore/book[price>35.00]/title	Selects all the title elements of the book elements of the bookstore element that have a price element with a value greater than 35.00

•Selecting Unknown Nodes

XPath wildcards can be used to select unknown XML nodes.

*	Matches any element node
@*	Matches any attribute node
node()	Matches any node of any kind

In the table below we have listed some path expressions and the result of the expressions:

/bookstore/*	Selects all the child element nodes of the bookstore element
//*	Selects all elements in the document
//title[@*]	Selects all title elements which have at least one attribute of any kind

•Selecting Several Paths

By using the | operator in an XPath expression you can select several paths.

In the table below we have listed some path expressions and the result of the expressions:

//book/title book/price	Selects all the title AND price elements of all book elements
//title //price	Selects all the title AND price elements in the document
/bookstore/book/title //price	Selects all the title elements of the book element of the bookstore element AND all the price elements in the document

•XPath Operators

Below is a list of the operators that can be used in XPath expressions:

	Computes two node-sets	//book //cd
+	Addition	6 + 4
-	Subtraction	6 - 4
*	Multiplication	6 * 4
div	Division	8 div 4
=	Equal	price=9.80
!=	Not equal	price!=9.80
<	Less than	price<9.80
<=	Less than or equal to	price<=9.80
>	Greater than	price>9.80
>=	Greater than or equal to	price>=9.80
or	or	price=9.80 or price=9.70
and	and	price>9.00 and price<9.90
mod	Modulus (division remainder)	5 mod 2

Main procedures

- The main procedures available to be used in the TCL files, are listed below.

`gid_groups_conds::actualize_conditions_window`

This procedure updates the information of the whole data tree, and automatically refresh data shown in the user interface. If the data source is changed, such as new fields have been added or data values and field have been modified, all the user interface will reflect those changes. Furthermore, all the TCL procedures defined in the data tree will be called and the whole data tree will be refreshed. Note that this instruction must be carried out only when necessary. It has no arguments.

• `gid_groups_conds::begin_problemtypespd_file defaults_file ""`

This procedure allows to load the problem type and should be defined in the `InitGIDProject` procedure. The arguments are the following:

`spd_file` - The directory of the main configuration file (.spd).

- `defaults_file` - The directory of the preferences file. If the directory of the preferences file does not already exist, it is

`created.gid_groups_conds::SetProgramName program_name`

This procedure stores the program name in the preferences file and should be defined in the `InitGIDProject` procedure. The argument is the following:

- `program_name` - Name of the program.

`gid_groups_conds::end_problemtypespd_file`

This procedure will be called when the project is about to be closed, in the `EndGIDProject` procedure. It receives as argument:

`defaults_file` - The directory of the preferences file. If the directory of the preferences file does not already exist, it is created.

• `gid_groups_conds::give_data_version`

- This function returns the version number of the problem type.

`gid_groups_conds::save_spd_file spd_file`

This procedure saves the .spd file and should be defined in the `SaveGIDProject` procedure. Therefore, it will be called when the currently opened file is saved to disk. It receives as argument:

- `spd_file`: path of the file being saved

`gid_groups_conds::import_export_materials widget xpath`

This procedure allows to open the 'Import/export materials' window. It receives as arguments:

`widget` - Parent widget.

`xpath` - It is the xpath expression to the field 'container' of materials.

Description of the local axes

Local axes definition

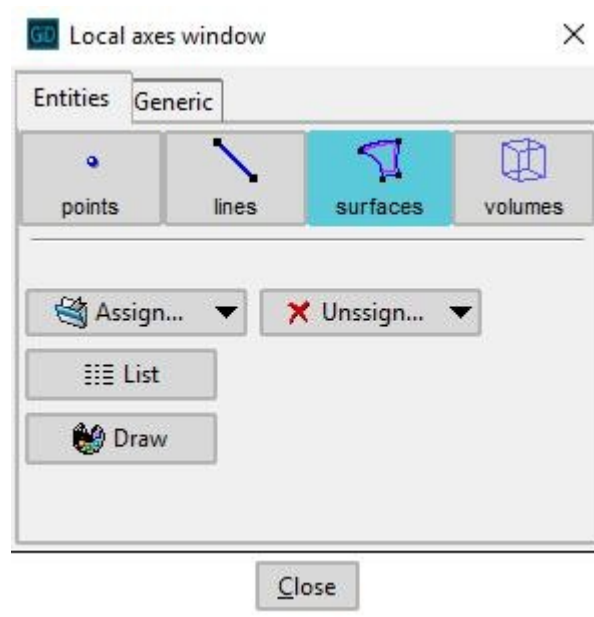
CustomLib lets the user to define new coordinates reference systems. Local axes can be assigned to entities using the *Local axes* window.

Local coordinates systems of lines are defined such that the local x-axis will correspond to the axial direction of the line. Both other axes will be defined perpendicular to this local x-axis.

Local coordinate systems of surfaces are defined such that the local z-axis is defined perpendicular to the surface and both x- and y- axes are defined perpendicular to each other in the plane of the surface.

Local coordinates can be defined by selecting the option called *Assign Automatic* in the Local axes window. Selecting the option *Assign Automatic alt* will define alternative local coordinate systems which are normally rotated 90 degrees around the third axis compared to the first ones.

The procedure `gid_groups_conds::local_axes_window` creates the *Local axes* window automatically, which allows to define the local axes to be used.

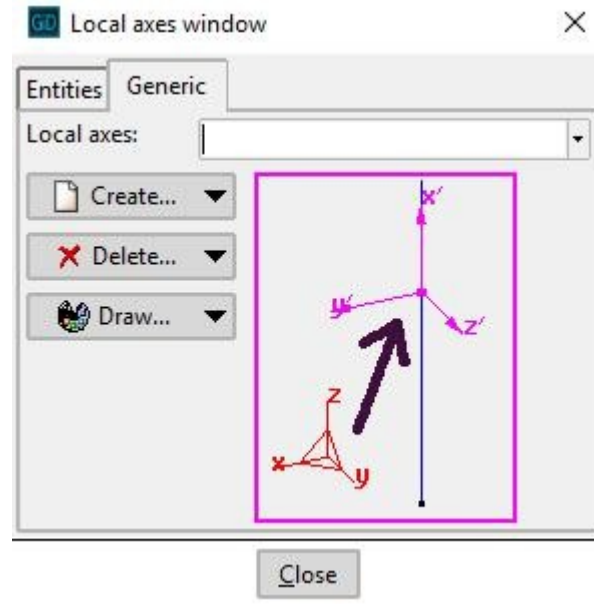


Local coordinates will be shown by selecting the Draw button.

It should be emphasized that it is possible to list the local axes clicking on the 'List' button. This way a childwindow displays both the Euler angles and the transformation matrix.

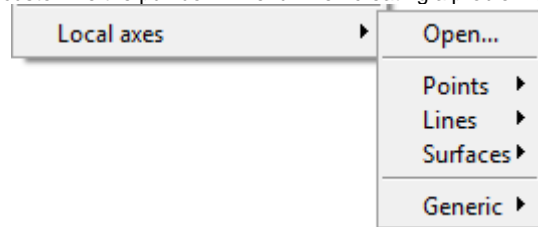
Generic local axes window allows to create new local axes in two ways:

- **Three points XZ:** Enter three points that corresponds to the origin, the X-direction and the Z-direction. The origin and the last introduced point define the Z-axis, whereas the second point indicates the side of the x-z plane where the point lies.
- **X and angle:** Enter two points and one angle. The first point is the center, the second point indicates the x-axis and the angle indicates the position of the Y and Z axes. In the graphical window it is possible to set this angle by moving the mouse on the screen. It also indicates where the origin of the angle is. The angle can be entered either by clicking the mouse or by entering the exact value in degrees.



User defined local axes menu

CustomLib also offers the opportunity to customize the pull-down menu when creating a problem type in the following way:



User can create different named local axes systems and with the different methods that can be chosen there. The names of the defined local axes will be added to the menu, where local axes are chosen.

The code to make this change in the menu would be as follows,

```
GidAddUserDataOptionsMenu [= "Local axes"] "gid_groups_conds::local_axes_menu %W" $ipos
```

The 'ipos' is the index where the new menu will be inserted.

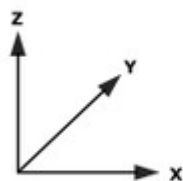
For more details about managing menus reference is made to the 'Managing menus' chapter in the GiD manual.

It is important to note that all user-defined systems are automatically calculated in the problem type. After generation of the mesh, the local coordinates of entities will be automatically defined at each entity element of the entities local coordinates have been defined on.

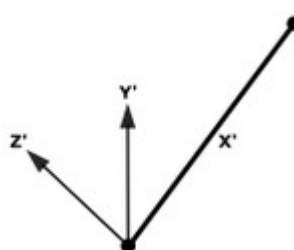
Local axes assigned on lines

The model has been created related to a global axis system XYZ that is unique for the entire problem. The main property of this system is that the local X' axe must have the same direction than the line.

Global axes system



Local axes system



The ways for defining local axes systems are:

-Default: The program assigns a different local axes system to every line with the following criteria:

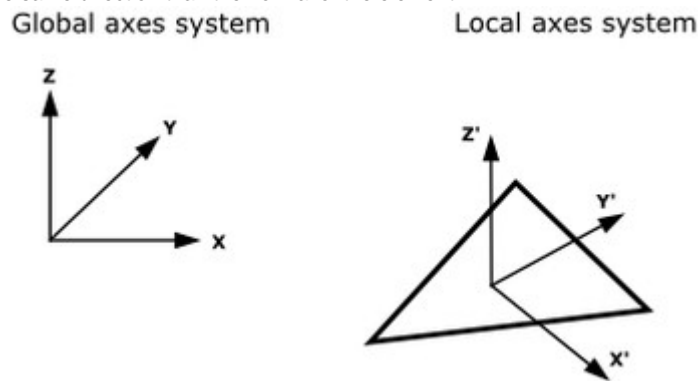
X' axis has the direction of the line. If X' axis has the same direction than global Z axis, Y' axis has the same direction than global X . If not, Y' axis is calculated so as to be horizontal (orthogonal to X' and Z). Z' axis is the cross product of X' axis and Y' axis. It will try to point to the same sense than global Z (dot product of Z and Z' axes will be positive or zero). The intuitive idea is that vertical lines have the Y' axis in the direction of global X . All the other lines have the Y' axis horizontal and with the Z' axis pointing up.

-Automatic: Similar to the previous one but the local axes system is assigned automatically to the line by GiD. The final orientation can be checked with the *Draw Local Axes* option in the GiD Conditions window.

-Automatic alt: Similar to the previous one but an alternative proposal of local axes is given. Typically, user should assign Automatic local axes and check them, after assigning, with the *Draw local axes* option. If a different local axes system is desired, normally rotated 90 degrees from the first one, then it is only necessary to assign again the same condition to the entities with the **Automatic alt** option selected.

Local axes assigned on surfaces

The model is created related to a global axis system XYZ that is unique for the entire problem. The main property of this local axis system is that the local Z' axis must have the same direction than the normal of the element.



The ways for defining local axes systems are:

-Default: The program assigns a different local axes system to the surface with the following criteria:

Be \mathbf{N} the unitary normal of the surface element, and \mathbf{U} the vector $(0,1,0)$ and \mathbf{V} the vector $(0,0,1)$. Then: Z' axis has the direction and sense of \mathbf{N} .

If $N_x < 1/64$ and $N_y < 1/64$, then X' axis will be in the direction of the cross product of \mathbf{U} and \mathbf{N} ($\mathbf{U} \times \mathbf{N}$).

••• If not, X' axis will be in the direction of the cross product of \mathbf{V} and \mathbf{N} ($\mathbf{V} \times \mathbf{N}$). Y' axis will be the cross product of Z' axis and X' axis.

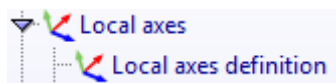
Intuitively, this local axis system is calculated so as if element is approximately contained in the plane XY , local X' axis will point towards global X axis. If not, this X' axis is obtained as orthogonal to global Z axis and local Z' axis.

-Automatic: Similar to the previous one but the local axes system is assigned automatically to the surface. The final orientation can be checked with the *Draw Local Axes* option in the GiD Conditions window.

-Automatic alt: Similar to the previous one but an alternative proposal of local axes is given. Typically, user should assign Automatic local axes and check them, after assigning, with the *Draw local axes* option. If a different local axes system is desired, normally rotated 90 degrees from the first one, then it is only necessary to assign again the same condition to the entities with the **Automatic alt** option selected.

Example no. 1

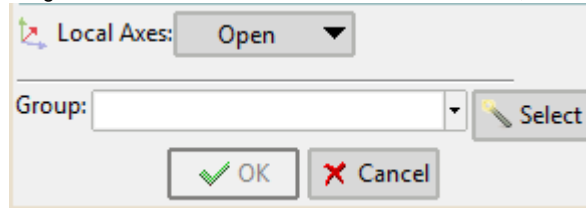
This is an example of the `<container/>` node field in the .spd file to call the *Local axes* window from the data tree:



```
<container n="local_axes" pn="Local axes">
  <container n="local_axes_window" pn="Local axes definition">
    <edit_command n="local_axes_win" proc="gid_groups_conds::local_axes_window" edit_type="
exclusive"/>
  </container>
</container>
```

Example no. 2

Here is an example of application for opening the Local axes window from a conditions window, as follows:



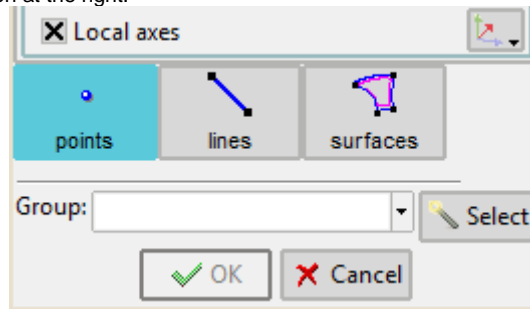
The <condition> node field in the .spd file is:

```
<condition n="local_axes" pn="Local axes condition" ov="surface" local_axes="normal" ovm="element"/>
```

The local_axes attribute specifies one of the three states for the button: normal, disabled or hidden.

Example no. 3

This is an example of Local axes check button definition inside a window of the data tree. If the check button is activated, then the *Local axes* window can be opened pressing the button at the right:



The <condition> node field in the .spd file is the following:

```
<condition n="pressure_load" pn="Pressure load" ov="surface" ovm="element">
<value n="load_type" pn="Load type" v="global" values="global,local" local_axes="disabled" editable="0"
help="The load can be applied in global axes, or in the Local Axes defined for the entity">
  <dependencies node="." attl="local_axes" vl="normal" value="local"/>
  <dependencies node="." attl="local_axes" vl="disabled" not_value="local"/>
</value>
</condition>
```

Example no. 4

Here is an example of Local axes definition, inside for instance, a conditions window of the data tree.



The non-editable combobox In the combobox options The <condition/> node field in the .spd file could be the following:

```

<condition n="pressure_load" pn="Pressure load" ov="surface" ovm="element">
<value n="load_type" pn="Load type" v="global" values="global,local" local_axes="disabled" editable="0"
help="The load can be applied in global axes, or in the Local Axes defined for the entity">
  <dependencies node="." att1="local_axes" v1="normal" value="local"/>
  <dependencies node="." att1="local_axes" v1="disabled" not_value="local"/>
</value>
</condition>

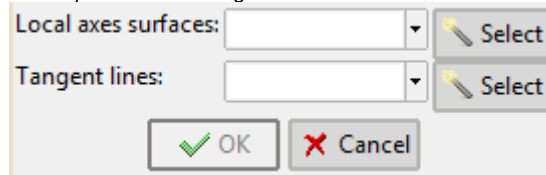
```

Example no. 5

The following is an example for creating a specific window for assigning local axes on surfaces in the data tree. The user selects a group of lines and axe X' will be tangent to the lines:



The item 'Local axes definition' in the data tree opens the following window:



The <condition> node field in the .spd file is as follows:

```

<condition n="local_axes_directions" pn="Local axes directions" ovl="surface" ov2="line" ovlp="Local
axes surfaces" ov2p="Tangent lines" ovml="element" ovml2="none" help="Select surfaces in order to define
its local axes tangents based in selected lines" />

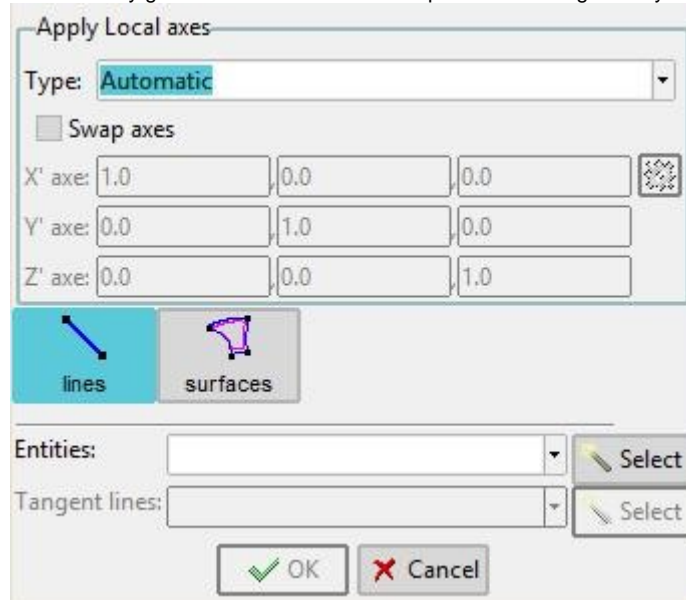
```

Future developments

In the future it will be available a new *Local axes* window, totally integrated in the data tree, that will allow to define groups composed by local axes.

There will be different styles of local axes definitions:

- Automatic: The program automatically generates one local axes compatible with the geometry.



Note: If swap axes check button is selected, axes X' and Y' are swapped.

- Tangent lines: User selects a group of lines and axe X' will be tangent to these lines:

Apply Local axes


Type: **Tangent lines**


☐ Swap axes


X' axe: 1.0 0.0 0.0



Y' axe: 0.0 1.0 0.0

Z' axe: 0.0 0.0 1.0

 surfaces

Entities:  Select

Tangent lines:  Select

 OK  Cancel

- User defined: User defines the three local axes X', Y' and Z':

Apply Local axes





Type: **User defined**


☐ Swap axes


X' axe: 1.0 0.0 0.0



Y' axe: 0.0 1.0 0.0

Z' axe: 0.0 0.0 1.0

 points  lines  surfaces  volumes

Entities:  Select

Tangent lines:  Select

 OK  Cancel

Writing the input file for calculation

The command called `GiD_WriteCalculationFile` facilitate the creation of the output calculation file.
See its syntax at [WriteCalculationFile](#)

•Cheatsheet of writing functions

This functions are based on the implementation of a problemtype for the [Frame3DD](#) solver, but can be useful to anyone.

•Basic and advanced tcl commands for writing the input file - example

How to **write a string**.

See how we print the units, we are asking gid for the current active units for the different magnitudes: F for Force, L for Lenght, M for Mass

```
customlib::WriteString "Input Data file for Frame3DD - 3D structural frame analysis ([gid_groups_conds::
give_active_unit F],[gid_groups_conds::give_active_unit L],[gid_groups_conds::give_active_unit M])"
```

Nodes in the mesh

•See how we get the number of nodes of the whole mesh. Also notice the way we are passing the writing format for the coordinates.

```
customlib::WriteString ""
customlib::WriteString "[GiD_Info Mesh NumNodes] # number of nodes"
customlib::WriteString "#.node x y z r"
customlib::WriteString "# [gid_groups_conds::give_active_unit L] [gid_groups_conds::give_active_unit L]
[gid_groups_conds::give_active_unit L] [gid_groups_conds::give_active_unit L]"
customlib::WriteString ""
customlib::WriteCoordinates "%5d %14.5e %14.5e %14.5e 0.0\n"
```

Elements assigned to the **condition** that assigns a **material**

```
set condition_name "frameData"
set condition_formats [list {"%1d" "element" "id"} {"%13.5e" "property" "Ax"} {"%13.5e" "property"
"Asy"} ... {"%13.5e" "material" "Density"}]
set formats [customlib::GetElementsFormats $condition_name $condition_formats]
set number_of_elements [GiD_WriteCalculationFile elements -count -elemtype Linear $formats]
customlib::WriteConnectivities $condition_name $formats "" active
```

•Contraints

This is the way we get the number of nodes assigned to a condition called 'constraints' in the spd.

```
set constraints_list [list "constraints"]
set number_of_constraints [customlib::GetNumberOfNodes $constraints_list]
```

•Basic data

•This is the way we access to the data in the tree. Notice that we are building what we call xpath, that is a kind of path in the spd, from the root to the item we want.

Then we get the xml node from the document using `selectNodes`, and finally we get the current value with `get_domnode_attribute`

```
set document [$::gid_groups_conds::doc documentElement]
set xpath "/frame3dd_data/container\[@n = 'extraOptions' \]/value\[@n = 'shear_deformation' \]"
set xml_node [$document selectNodes $xpath]
set shear_deformation [get_domnode_attribute $xml_node v]
customlib::WriteString "$shear_deformation # 1=Do, 0=Don't include shear deformation effects"
```

Single load case

This is the way we get the number of elements assigned to a condition, and then print the element id and the properties assigned to it.
set condition_name "line_Uniform_load"

```
set condition_name "line_Uniform_load"
set condition_formats [list {"%ld" "element" "id"} {"%13.5e" "property" "Load_x"} {"%13.5e"
"property" "Load_y"} {"%13.5e" "property" "Load_z"}]
set formats [customlib::GetElementsFormats $condition_name $condition_formats]
set number_of_elements [GiD_WriteCalculationFile elements -count -elemtype Linear $formats]
customlib::WriteConnectivities $condition_name $formats "" active
```

•Several load cases

For the several load case, we will need to iterate over the xml nodes corresponding to the blockdatas that define each load case.
For each load case, we need to print all the loads information, so we need to know if a load has any group assigned.
If so, we'll need to know how many elements are involved, and print them with the properties of the load.

```
set xpath "/frame3dd_data/container\[@n = 'staticCases' \]/blockdata"
set xml_nodes [document selectNodes $xpath]
foreach load_case $xml_nodes {
  set xpath "./condition\[@n = 'uniformLoad' \]/group"
  set groups [$load_case selectNodes $xpath]
  set number_of_elements 0
  set formats_dict [dict create ]
  foreach group $groups {
    set group_name [get_domnode_attribute $group n]
    set Ux_node [$group selectNodes "./value\[@n = 'Ux'\]" ]
    set Ux_value [get_domnode_attribute $Ux_node v]
    set Uy_node [$group selectNodes "./value\[@n = 'Uy'\]" ]
    set Uy_value [get_domnode_attribute $Uy_node v]
    set Uz_node [$group selectNodes "./value\[@n = 'Uz'\]" ]
    set Uz_value [get_domnode_attribute $Uz_node v]
    set format "%5d $Ux_value $Uy_value $Uz_value"
    set formats_dict [dict merge $formats_dict [dict create $group_name $format]]
  }
  set number_of_elements [GiD_WriteCalculationFile elements -count -elemtype Linear $formats_dict]
  if {$number_of_elements > 0} {
    GiD_WriteCalculationFile elements -elemtype Linear $formats_dict
  }
}
```


Alternatives

In this section, we will write the coordinates, connectivities, and conditions assuming that the input can be different, such as the one in [Matfem](#). In some codes, like matfem, the input file into the solver is not written in a text file, but in a "code like file". The coordinates section in the matfem solver is a matlab array, with a single comma splitting x and y (x, y) and a semicolon splitting a node from another (x1, y1; x2, y2; ...)

•Write
coordinates:

Example:

```
%
% Coordinates
%
global coordinates
coordinates = [
    0.00 , 0.00;
    0.50 , 0.00;
    2.50 , 1.00 ];
```

Code:

```
customlib::WriteString "%"
customlib::WriteString "% Coordinates"
customlib::WriteString "%"
customlib::WriteString "global coordinates"
set nodes [GiD_Mesh list node]
customlib::WriteString "coordinates = \["
for {set i 0} {$i < [llength $nodes]} {incr i} {
    set node [lindex $nodes $i]
    lassign [GiD_Mesh get node $node coordinates] x y z
    if {$i < [expr [llength $nodes] -1]} {set end ";"} {set end ""}
    customlib::WriteString "$x , $y $end"
}
customlib::WriteString "\] ; "
```

Write elements:

Example:

```
%
% Elements
%
global elements
elements = [
    1, 2, 7 ;
    2, 3, 8 ;
    18, 17, 12 ];
```

Code:

```

customlib::WriteString "%"
customlib::WriteString "% Elements"
customlib::WriteString "%"
customlib::WriteString "global elements"
set elements_t [GiD_Mesh list -element_type Triangle element]
set elements_q [GiD_Mesh list -element_type Quadrilateral element]
customlib::WriteString "elements = \["
for {set i 0} {$i < [llength $elements_t]} {incr i} {
    set element [lindex $elements_t $i]
    lassign [GiD_Mesh get element $element connectivities] c1 c2 c3
    if {$i < [expr [llength $elements_t] -1]} {set end ";"} {set end ""}
    customlib::WriteString "$c1 , $c2 , $c3 $end"
}
for {set i 0} {$i < [llength $elements_q]} {incr i} {
    set element [lindex $elements_q $i]
    lassign [GiD_Mesh get element $element connectivities] c1 c2 c3 c4
    if {$i < [expr [llength $elements_q] -1]} {set end ";"} {set end ""}
    customlib::WriteString "$c1 , $c2 , $c3 , $c4 $end"
}
customlib::WriteString "\] ; "

```

Conditions

Example:

```
%
% Point loads
%
pointload = [
6, 2, -1.0 ;
12, 2, -1.0 ;
18, 2, -1.0 ];
```

Code:

```
<condition n="PuntualLoads" pn="Load" ov="point" ovm="node" icon="moad">
  <value n="x-force" pn="Force X" v="0.0" units="N" unit_magnitude="F" />
  <value n="y-force" pn="Force Y" v="0.0" units="N" unit_magnitude="F" />
</condition>
```

spd

Tcl

```
# Point load
set root [customlib::GetBaseRoot]
customlib::WriteString ""
customlib::WriteString "%"
customlib::WriteString "% Point loads"
customlib::WriteString "%"
customlib::WriteString "pointload = \"[\"
set displacement_fix_nodes [$root selectNodes "*/condition\[@n='PuntualLoads'\]/group"]
foreach node $displacement_fix_nodes {
  set group [$node @n]
  set val_x [get_domnode_attribute [$node selectNodes ".value\[@n='x-force'\]" ] v]
  set val_y [get_domnode_attribute [$node selectNodes ".value\[@n='y-force'\]" ] v]
  set fix_x [expr $val_x == 0.0 ? "false" : "true"]
  set fix_y [expr $val_y == 0.0 ? "false" : "true"]
  set nodes [GiD_EntitiesGroups get $group nodes]
  set num_nodes [objarray length $nodes]

  for {set i 0} {$i < $num_nodes} {incr i} {
    set node_id [objarray get $nodes $i]
    if {$i < [expr $num_nodes -1]} {set end ";"} {set end ""}
    if {$fix_x eq "true" && $fix_y eq "true"} {set end ";"}
    if {$fix_x eq "true"} { customlib::WriteString "$node_id , 1 , $val_x $end" }
    if {$i < [expr $num_nodes -1]} {set end ";"} {set end ""}
    if {$fix_y eq "true"} { customlib::WriteString "$node_id , 2 , $val_y $end" }
  }
}
customlib::WriteString "\] ; "
```

Units conversion

Auxiliary procedures to convert between different unit types

There are a couple of functions that are essential when writing the input file for the calculation. These functions permit the treatment of the units, and facilitates the conversion from one measurement to another. They are the following:

-

`gid_groups_conds::give_mesh_unit` This

function provides the mesh unit.

- `gid_groups_conds::set_mesh_unit` unit

This function imposes a mesh unit, where the argument is as follows:

- unit - A text string, denoting the unit that you want to apply as mesh

`unit.gid_groups_conds::convert_value_to` nodeObject to_unit

This function converts the value of a nodeObject from the current unit for the original number to another unit, where the arguments are as follows:

nodeObject - DOM node object chosen.

- to_unit - A text string, denoting the unit that you want to convert the original number

`to.gid_groups_conds::convert_value_to_active` nodeObject

This function converts the value of a nodeObject from the current unit type to the general active unit selected in the GUI. The argument is as follows:

nodeObject - DOM node object chosen.

- `gid_groups_conds::convert_unit_value` magnitude value unit_from to_unit

This function converts a number from one unit type to another unit type, for the same magnitude. It receives as arguments:

magnitude - Unit definition, it is the name 'n' of the physical quantity (eg. "L" for Length)

value - The number to be converted.

unit_from - A text string, denoting the unit for the original number.

- to_unit - A text string, denoting the unit that you want to convert the original number

`to.gid_groups_conds::convert_value_to_printable_unit` nodeObject

This function returns a text string, denoting the current unit of the node. It receives as argument:

nodeObject - DOM node object chosen.

- `gid_groups_conds::give_unit_factor` magnitude unit

This function returns the conversion factor for the unit given. The arguments are as follows:

magnitude - Unit definition, it is the name 'n' of the physical quantity (eg. "L" for Length)

unit - A text string, denoting the unit chosen to obtain the conversion factor.

- `gid_groups_conds::convert_v_to_default` value magnitude unit_from

This function converts a number from one unit type to the unit type by default, for the same magnitude. It receives as arguments:

value - The number to be converted.

magnitude - Unit definition, it is the name 'n' of the physical quantity (eg. "L" for Length)

unit_from - A text string, denoting the unit for the original number.

- `gid_groups_conds::give_active_units_system`

- This function returns a string list, where the first item is the units system.

`gid_groups_conds::give_active_unit` magnitude

This function returns the active unit for a given magnitude. It receives as argument:

magnitude - Unit definition, it is the name 'n' of the physical quantity (eg. "L" for Length)

User preferences

User preferences of the problemtype will be automatically saved/read to/from disk in a xml file named `.$problem_type$version.ini`, located in the same user folder as the GiD preferences.

- To set/get user preferences variables these procedures must be used.

`gid_groups_conds::get_preference <name> <default_value>`

This function returns the value of a preference, or the default value if the preference does not already exist. It receives as arguments:

name - Name of the preference

value - Default value

•`gid_groups_conds::set_preference <name> <value>`

This function imposes the value of a preference. It receives as arguments:

name - Name of the preference

value - Default value

Example: get or set a user variable named 'verbosity_level', with default value=0

```
set_level [gid_groups_conds::get_preference verbosity_level 0]
gid_groups_conds::set_preference verbosity_level 2
```

Import export materials

Parameter called **allow_import** in fields of type <blockdata> allows to add the 'Import/export materials' item in the contextual menu for a specific 'blockdata' field.

A material is composed by a set of material properties, which can be applied to geometry entities. **Import/export** tool allows to handle material properties easily. This tool is located inside the right mouse menu when a particular material or a materials collection is selected in the data tree and **allow_import** parameter is activated.

- In order to activate the Import/export materials tool, it is necessary to call `n="materials"` to the global container in the spd-file. Moreover, although the intermediate containers could be defined using any attribute `n`, each final material blockdata has to be called `n="material"`.

- The "Import/export materials" window contains two different material data trees. At the left side there is the local materials list associated to the current model. At the right side there is a material data tree list, which could be imported or exported depending on the user interests.

It is possible to import/export materials in four different ways:

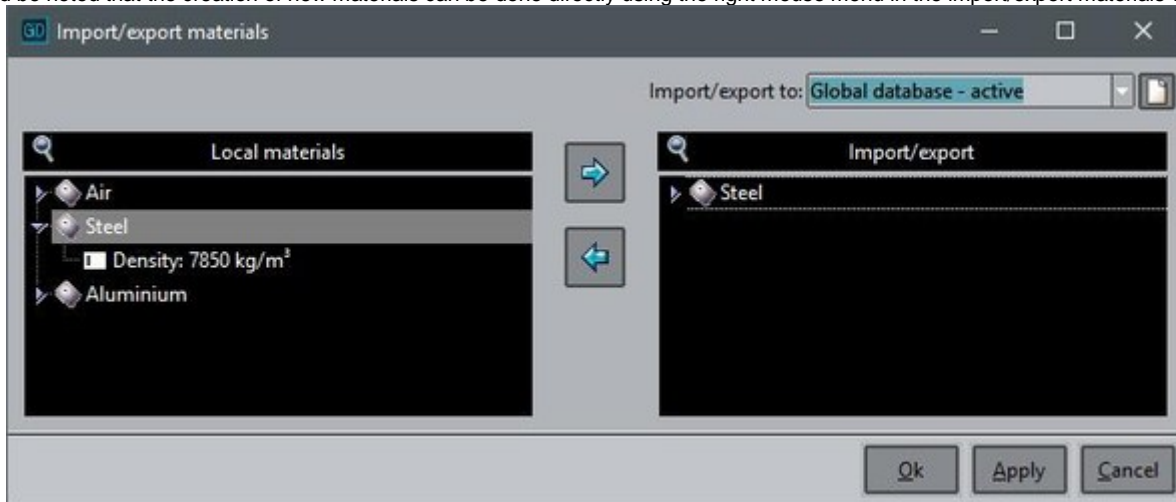
1. Global database active
2. Global database inactive
3. Import from a file
4. Export to a file.

The global database active is the database shown in the data tree located at the left side of the GiD window when a new model is created. The global database inactive is an internal database which does not affect the default data tree for new models. It could be used to store odd materials in order to import them for a particular model.

The global database active also allows to import the original default materials clicking on the button located just at the right side of the "Import /export to" combo-box, which facilitates to recover easily the original input data for materials.

It is also possible to import or export selected materials from particular XML-files without modifying the global database.

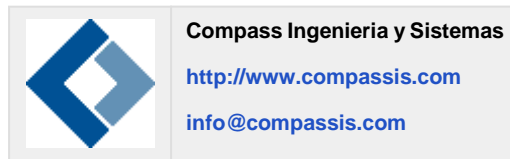
It should be noted that the creation of new materials can be done directly using the right mouse menu in the import/export materials window.



About CustomLib

Cimne's copyrighted CustomLib is developed between CIMNE and [Compass Ingenieria y Sistemas](http://www.compassis.com), and it is the library included in GiD for creating advanced customized problemtypes, i.e. adaptations to third-party simulation codes or external programs. CustomLib terms of use are the same as the GiD ones.

Compass offers the possibility of implementing advanced customizations of GiD, based on customLib, adapted to the specific need of every specific client. For more information about this service, please visit www.compassis.com.



CustomLIB extras

For using this extra functionality, just activate the CustomLibAutomatic field in the [XML file](#) or load it manually by requiring the package customlib_extras

- The extra content on this package is:

- New fields added to the **spd** file.

- New functions to make everything easier.

SPD_fields

include

It's used to join diferent parts of the spd file.

It must contain the following attributes:

- path**: relative path, from the problemtype

folderIt can contain the following attributes:

- active**: Specifies if the part must be added in the final

All attributes will be transfered to the included node, except "n" "active" and "path"

Example:

```
<include path="xml/materials.xml"/>
```

Extra functions

The main procedures available to be used in the TCL files, are listed below. All the usages and examples can be found on the 'cmas2d_customlib' problemtype.

- customlib::ProcessIncludes directory

- This procedure parses the active **include** nodes of the spd file, and replaces them for the node found in the **path**

files.customlib::ProcessIncludesRecurse filename basedir

This procedure parses the active **include** nodes of the specified file, and replaces them for the node found in the **path** files. Basedir is the directory where the file specified by the path field is.

- customlib::InitWriteFile filename

Open the file for writing

- customlib::EndWriteFile

- customlib::InitMaterials list_of_condition_names

The **list_of_condition_names** is the list of these conditions where we can find a material, so we can consider it as 'used', assigning it a MID (Material Identifier) for further queries.

- customlib::WriteString str

Writes the content of str in the opened file

- customlib::WriteConnectivities list_of_condition_names parameters

Utility to print elements and connectivities.

In the spd file, we can define **condition** tags to assign properties to a GiD group. The **list_of_condition_names** is the list of these conditions whose groups' elements must be printed.

parameters is a list of lists of 3 words, and defines the format and the information that we want to print. The first word is always a format. The second word can be "element", "material", "property" or "string". "element" is set to write element information. "material" is set to print any material property. "string" is set to print a string. The third word depends on the second one. If it's "element", the third can be: "id" or "connectivities". If it's "material", the third one can be "MID", "Density" or any material property according to GetMaterials function. Obviously, to use 'material' data, a material must be defined in the condition.

Valid examples:

- {"%10d" "element" "id"}
- {"%10d" "element" "connectivities"}
- {"%10d" "node" "id"}
- {"%10d" "material" "MD"}
- {"%10d" "material" "Density"}
- {"%10d" "property" "Weight"}
- {"%10d" "string" "str..."}
- ...

Example:

```
set list_of_condition_names [list "Shells"]
set parameters [list {"%10d" "element" "id"} {"%10d" "element" "connectivities"} {"%10d" "material" "MID"} ]
customlib::WriteConnectivities $list_of_condition_names $parameters
```

These instructions print the following text

1	54	52	43	1
2	59	61	51	1
3	68	70	67	1
4	53	57	47	1

...

- customlib::WriteNodes list_of_condition_names parameters ?flags?

Utility to print the nodes of the groups of the conditions specified at **list_of_condition_names**, and the information assigned to them.

Example:


```
set list_of_condition_names [list "Point_Weight"]
set parameters [list {"%1d" "element" "id"} {"%13.5e" "property"
"Weight"}]customlib::WriteNodes $list_of_condition_names $parameters
```

Output

```
1 7.80000e+000
83 9.60000e+000
108 7.80000e+000
```

•customlib::GetNumberOfNodes list_of_condition_names ?flags?

Utility to get the number of nodes of the groups of the conditions specified at **list_of_condition_names**.

•customlib::WriteCoordinates formats

Writes the coordinates of the nodes of the model.

2D example:

```
customlib::WriteCoordinates "%5d %14.5e %14.5e%.0s\n"
```

Output:

```
1 6.89301E-002 8.61382E-003
2 7.49755E-002 1.26044E-002
3 7.44487E-002 3.68638E-003
```

3D example:

```
customlib::WriteCoordinates "%5d %14.5e %14.5e %14.5e\n"
```

Output:

```
1 6.89301E-002 8.61382E-003 8.61382E-003
2 7.49755E-002 1.26044E-002 1.26044E-002
3 7.44487E-002 3.68638E-003 3.68638E-003
```

•customlib::GetMaterials ?state?

This procedure returns a nested dict, where the first key is the name of a material, the second key is the name of the property. **state** can be 'used', to return only the used materials, or 'all' to return all the materials.

Example:

```
set mat_dict [customlib::GetMaterials used]
set aluminium [dict get $mat_dict "Aluminium"]
set density [dict get $aluminium "Density"]
```

•customlib::GetMaterials ?state?

This procedure returns a nested dict, where the first key is the name of a material, the second key is the name of the property. **state** can be 'used', to return only the used materials, or 'all' to return all the materials.

Example:

```
set mat_dict [customlib::GetMaterials used]
set aluminium [dict get $mat_dict "Aluminium"]
set density [dict get $aluminium "Density"]
```

- customlib::GetNumberOfMaterials ?state?

state can be 'all' or 'used'.

Returns the number of materials in the database. If state is used, it returns the number of materials used in the model.

- customlib::WriteMaterials parameters ?state?

state can be 'all' or 'used'.

Utility to print the list of materials, and their properties defined in **parameters**.

Example:

```
set parameters [list {"%4d" "material" "MID"} {"%13.5e" "material" "Density"}]
customlib::WriteMaterials $parameters used
```

Output:

```
1 7.85000e+003
2 2.65000e+003
```

Wizards

GiD includes a modified version of the tcl package snitwiz by Steve Cassidy. The package is called **gid_wizard**, and can be found inside the scripts folder.

It is useful to create a step guided GUI for your problemtype or plugin. It provides a window, with a header, a body, and a footer with back, next, cancel and finish buttons, that you can configure.

You can find a basic usage example in the package folder -> GiD > scripts > gid_wizard > test.tcl

We also provide another package to make the implementation easier, called **gid_smart_wizard**, which allows you to define your wizard steps in a xml file.

You can find it's documentation here -> https://github.com/GiDHome/gid_smart_wizard

You can find an example here -> http://github.com/GiDHome/cmas2d_customlib_wizard

EXECUTING AN EXTERNAL PROGRAM

Once all the problem type files are finished (.cnd, .mat, .prb, .sim, .bas files), you can run the solver. You may wish to run it directly from inside GiD.

To do so, it is necessary to create the file `problem_type_name.bat` in the Problem Type directory. This must be a shell script that can contain any type of information and that will be different for every operating system. When you select the Calculate option in GiD Preprocess this shell script is executed (see CALCULATE from Reference Manual).

Because the .bat file will be different depending on the operating system, it is possible to create two files: one for Windows and another for Unix/Linux. The Windows file has to be called: `problem_type_name.win.bat`; the Unix/Linux file has to be called: `problem_type_name.unix.bat`.

If **GiD** finds a .win.bat or .unix.bat file, the file `problem_type_name.bat` will be ignored.

If a .bat file exists in the problem type directory when choosing Start in the calculations window, GiD will automatically write the analysis file inside the example directory assigning the name `project_name.dat` to this file (if there are more files, the names `project_name-1.dat ...` are used). Next, this shell script will be executed. GiD will assign three arguments to this script:

- **1st argument:** `project_name` (name of the current project);
- **2nd argument:** `c:\a\b\c\project_name.gid` (path of the current project);
- **3rd argument:** `c:\a\b\c\problem_type_name.gid` (path of the problem type selected);

Among other utilities, this script can move or rename files and execute the process until it finishes.

Note 1: This file must have the executable flag set (see the UNIX command `chmod +x`) in UNIX systems.

Note 2: GiD sets as the current directory the model directory (example: `c:\examples\test1.gid`) just before executing the .bat file. Therefore, the lines (`cd $directory`) are not necessary in the scripts.

Note 3: In UNIX platforms check you have installed the shell you are using in the .unix.bat script, there are more than one possibilities: `bash`, `csh`, `tcsh`, ...

The first line of the script specify the shell to be used, for example

```
#!/bin/sh
or
#!/bin/bash
```

In Windows platforms, the `command.exe` provided by GiD is used instead the standard `cmd.exe` or `command.com`

Showing feedback when running the solver

The information about what is displayed when Output view: is pressed is also given here. To determine what will be shown, the script must include a comment line in the following form:

For Windows:

```
rem OutputFile: %1.log
```

For Linux/Unix:

```
# OutputFile: "$1.log"
```

where "\$1.log" means to display in that window a file whose name is: `project_name.log`. The name can also be an absolute name like `output.dat`. If this line is omitted, when you press Output view:, nothing will be displayed.

Commands accepted by the GiD command.exe

The keywords are as follows:

```
%
Shift
Rem
Chdir (Cd)
Del (Delete, Erase)
Copy
Rename (Ren, Move)
Mkdir (Md)
Set
Echo
If
Call
Goto
:
Type
```

Unknown instructions will be executed as from an external file.

Not all the possible parameters and modifiers available in the operating system are implemented in the GiD executable command.exe.

The '@' prefix is also handled, as not showing the command being executed.

Note: At the moment, **command.exe** is only used in Windows operating systems as an alternative to command.com or cmd.exe. With the **GiD command.exe** some of the disadvantages of Windows can be avoided (the limited length of parameters, temporary use of letters of virtual units that sometimes cannot be eliminated, fleeting appearance of the console window, etc).

If GiD finds the file **command.exe** located next to **gid.exe**, it will be used to interpret the *.bat file of the problem type; if the file command.exe cannot be found, the *.bat file will be interpreted by the windows command.com.
If conflicts appear by the use of some instruction still not implemented in the **GiD command.exe**, it is possible to rename the command.exe file, so that GiD does not find it, and the operating system command.com is used.

```
%
Returns the value of a variable.
%number
%name%
```

Parameters

```
number
The number is the position (from 0 to 9) of one of the parameters which the *.bat file receives.
name
The name of an environment variable. That variable has to be declared with the instruction "set".
```

Note: GiD sends three parameters to the *.bat file: %1, %2, %3
%1 is the name of the current project (project_name)
%2 is the path of the current project (c:\a\b\c\project_name.gid)
%3 is path of the problem type (c:\a\b\c\problem_type_name.gid)

For example, if GiD is installed in c:\gidwin, the "problemtype" name is cmas2d.gid and the project is test.gid, located in c:\temp (the project is a directory called c:\temp\test.gid with some files inside), parameters will have the following values:

```
%1 test
%2 c:\temp\test.gid
%3 c:\gidwin\problemtypes\cmas2d.gid
```

Note: It is possible that the file and directory names of these parameters are in the short mode Windows format. So, parameter %3 would be: c:\GIDWIN\PROBLE~\CMAS2D.GID.

Examples

```
echo %1 > %2%1.txt
echo %TEMP% >>
%1.txt
```

Predefined dynamic variables:

CD	The current directory (string). <i>Example:</i> c:\GiD\problemtypes
CMDCMDLINE	The original command line that invoked the 'command.exe' <i>Example:</i> C:\GiD\command.exe myProject c:\Users\gid\myProject.gid c:\GiD\problemtypes\myProblemType.gid
DATE	The current date. <i>Example:</i> 2021-12-02
RANDOM	A random integer number, anything from 0 to 32,767 (inclusive). <i>Example:</i> 20211
TIME	The current time, with a resolution of cents of seconds. <i>Example:</i> 20:21:12.02

Shift

The shift command changes the values of parameters %0 to %9 copying each parameter in the previous one. That is to say, value %1 is copied to %0, value %2 is copied to %1, etc.

Parameter

None.

Note: The shift command can be used to create a batch program that accepts more than 10 parameters. If it specifies more than 10 parameters in the command line, those that appear after tenth (%9) will move to parameter %9 one by one.

Rem

Rem is used to include comments in a *.bat file or in a configuration file.

rem[comment]

Parameter

comment

Any character string.

Note: Some comments are GiD commands.

Chdir (Cd)

Changes to a different directory.

chdir[drive:path] [..]

or

cd[drive:path] [..]

Parameters

[drive:path]

Disk and path of the new directory.

[..]

Goes back one directory. For example if you are within the C:\WINDOWS\COMMAND> directory this would take you to C:\WINDOWS>.

Note: When GiD calls the *.bat file, the path of the project is the current path, so it is not necessary to use cd %2 at the beginning of the *.bat file.

Examples

```
chdir e:\tmp cd ..
```

Delete (Del, Erase) Command used to delete files permanently from the computer.
delete[drive:][path] fileName [fileName]

Parameters

[drive:][path] fileName [fileName] Parameters that specify the location and the name of the file that has to be erased from disk. Several file names can be given.

Note: Files will be eliminated although they have the hidden or read only flag. Use of wildcards is not allowed. For example del . is not valid. File names must be separated by spaces and if the path contains blank spaces, the path should be inside inverted commas (the short path without spaces can also be used).

Examples

```
delete %2%1\file.cal
del C:\tmp\fa.dat C:\tmp\fb.dat
del "C:\Program files\test 4.txt"
```

Copy

Copies one or more files to another location.

copy source [+ source [+ ...]] destination

Parameters

source Specifies the file or files to be copied.

destination Specifies the filename for the new file(s).

To append files, specify a single file for destination, but multiple files for source (using the file1 + file2 + file3 format).

Note: If the destination file already exists, it will be overwritten without prompting whether or not you wish to overwrite it. File names must be separated by spaces. If the destination only contains the path but not the filename, the new name will be the same as the source filename.

Examples

```
copy f1.txt f2.txt
copy f1.txt c:\tmp
rem if directory c:\tmp exists, c:\tmp\f1.txt will be created, if it does not exist, file c:\tmp will
be created.
copy a.txt + b.txt + c.txt abc.txt
```

Rename (Ren, Move)

Used to rename files and directories from the original name to a new name.

rename[drive:][path] fileName1 fileName2

Parameter [drive:][path] fileName1 Specifies the path and the name of the file which is to be renamed.

fileName2 Specifies the new name file.

Note: If the destination file already exists, it will be overwritten without prompting whether or not you wish to overwrite it. Wildcards are not accepted (*,?), so only one file can be renamed every time. Note that you cannot specify a new drive for your destination. A directory can be renamed in the same way as if it was a file.

Examples

```
Rename fa.txt fa.dat
Rename "c:\Program Files\fa.txt" c:\tmp\fa.txt
Rename c:\test.gid c:\test2.gid
```

Mkdir (md)

Allows you to create your own directories.

mkdir[drive:]pathmd [drive:]path

Parameter

drive: Specifies the drive where the new directory has to be created.

path Specifies the name and location of the new directory. The maximum length of the path is limited by the file system.

Note: mkdir can be used to create a new path with many new directories.

Examples

```
mkdir e:\tmp2
mkdir d1\d2\d3
```

Set

Displays, sets, or removes Windows environment variables.

set *variable=[string]*

Parameters

variable Specifies the environment-variable name.

string Specifies a series of characters to assign to the variable.

Note: The set command creates variables which can be used in the same way as the variables %0 to %9. Variables %0 to %9 can be assigned to new variables using the set command.

To get the value of a variable, the variable has to be written inside two % symbols. For example, if the environment-variable name is V1, its value is %V1%. Variable names are not case-sensitive.

Examples

```
set basename = %1
set v1 = my text
```

Echo

Displays messages.

echo *[message]*

Parameters

message Specifies the text that will be displayed in the screen.

Note: The message will not be visible because the console is not visible, since GiD hides it. Therefore, this command is only useful if the output is redirected to a file (using > or >>). The symbol > sends the text to a new file, and the symbol >> sends the text to a file if it already exists. The if and echo commands can be used in the same command line.

Examples

```
Echo %1 > out.txt
Echo %path% >> out.txt
If Not Exist %2%1.post.res Echo "Program failed" >> %2%1.err
```

If

Executes a conditional sentence. If the specified condition is true, the command which follows the condition will be executed; if the condition is false, the next line is ignored.

- **if[not] exist fileName command**
- **if [not] string1==string2 command**
- **if[not] errorlevel number command**

Parameters

not Specifies that the command has to be executed only if the condition is false.

exist file Returns true if file exists.

command Is the command that has to be executed if the condition returns true.

string1==string2 Returns true if string1 and string2 are equal. It is possible to compare two strings, or variables (for example, %1).

errorlevel number Returns true if the last program executed returned a code equal to or bigger than the number specified.

Note: Exist can also be used to check if a directory exists.

Examples

```
if exist sphere.igs echo File exists >> out.txt
if not exist test.gid echo Dir does not exist >> out.txt
if %1 == test echo Equal %1 >> out.txt
```

Call

Executes a new program.

call[drive:][path] file [parameters]

Parameter

[drive:][path] file Specifies the location and the name of the program that has to be executed.

parameters Parameters required by the program executed.

Note: The program can be *.bat file, a *.exe file or a *.com file. If the program does a recursive call, some condition has to be imposed to avoid an endless curl.

Examples

```
call test.bat %1
call gid.exe -n -PostResultsToBinary %1.post.res %1.post.bin
```

Goto

The execution jumps to a line identified by a label.

goto label

Parameter

label It specifies a line of the *.bat file where the execution will continue. That label must exist when the Goto command is executed. A label is a line of the *.bat file and starts with ":". Goto is often used with the command if, in order to execute conditional operations. The Goto command can be used with the label :EOF to make the execution jump to the end of the *.bat file and finish.

Note: The label can have more than eight characters and there cannot be spaces between them. The label name is not case-sensitive.

Example

```
goto :EOF
if exist %1.err goto end
...
:end
```


:

Declares a label.

:labelName

Parameter

labelName A string which identifies a line of the file, so that the Goto command can jump there. The label line will not be executed.

Note: The label can have more than eight characters and there cannot be spaces between them. The label name is not case-sensitive.

Examples

```
:my_label  
:end
```

Type

Displays the contents of text files.

type*[drive:][[path] fileName*

Parameters

[drive:][[path] fileName Specifies the location and the name of the file to be displayed. If the file name contains blank spaces it should be inside inverted commas ("file name").

Note: The text will not be visible because the console is not visible, since GiD hides it. Therefore, this command is only useful if the output is redirected to a file (using > or >>). The symbol > sends the text to a new file, and the symbol >> sends the text to a file if it already exists. It is recommended to use the copy command instead of type.

In general, the **type** command should not be used with binary files.

Examples

```
type %2%1.dat > %2%1.txt
```

Managing errors

A line of code like

For Windows

```
rem ErrorFile: %1.err
```

For Linux/UNIX

```
# ErrorFile: "$1.err"
```

included in the .bat file means that the given filename is the error file. At the end of the execution of the .bat file, if the errorfile does not exist or is zero, execution is considered to be successful. If not, an error window appears and the contents of the error file are considered to be the error message. If this line exists, GiD will delete this file just before calculating to avoid errors with previous calculations.

A comment line like

```
rem WarningFile: %1.wrn  
or
```

```
WarningFile: "$1.wrn"
```

included in the .bat file means that the given filename is the warning file. This file stores the warnings that may appear during the execution of the .bat file.

•Examples

Here are two examples of easy scripts to do. One of them is for Unix/Linux machines and the other one is for MS-Dos or Windows.

•UNIX/Linux example:

```
#!/bin/sh  
basename = $1  
directory = $2  
ProblemDirectory = $3  
#   OutputFile: "$1.log" #  
#   ErrorFile: "$1.err" rm -  
f "$basename.post.res"  
"$ProblemDirectory/myprogram" "$basename" mv  
"$basename.results" "$basename.post.res"
```

MS-DOS/Windows example:

```
rem basename=%1  
rem directory=%2  
rem ProblemDirectory=%3  
rem   OutputFile: %1.log  
rem   ErrorFile: %1.err  
del %1.post.res  
%3\myprogram %1  
move %1.post %1.post.res
```

PREPROCESS DATA FILES

Geometry format: Modelname.geo

DESCRIPTION OF THE FORMAT

There are two versions of the <modelname>.geo GiD format: Binary (used by GiD by default) and ASCII. This chapter describes the format of the geometry ASCII file.

During the development of the program, the backward compatibility has been tried to be guaranteed as much as possible, so that, in general, any GiD version is able to read it (some very old version of GiD can ignore some new entities).

- The file ".geo", is written using Export->SaveAsciiProject, but by default the geometry will be saved in binary format. In order to GiD to read the file, it should be placed inside a directory named ".gid"

Document notation:

- By default all the variables are of type integer, the variables of type float will be written underlined, and those of type double with double underlined. (data types of the "C" language)
- A carriage return is denoted for <CR>
- The commas written to separate the variables should not be written in the file, they only appear in this document to facilitate their reading, and the same applies for the white lines.

The file should contain the following fields, and in the described order:

```
Header
Problem type
Must Repair
An entry for each layer
Null entity (denotes end of layers)
An entry for each meshing data (avoid if not
exists)
Null entity (denotes end of mesh data)
An entry for each point
An entry for each curve
An entry for each surface
An entry for each volume
Null entity (denotes end of geometric entities)
```

Header

RAMSAN-ASCII-gid-v7.6 <CR> (it is used to identify the file type and its version, in this case 7.6)

Problem Type

Problem type (variable of type string) IsQuadratic (0 for lineal elements)
(problem type name, UNKNOWN type for not loading none)

Must Repair

0 (0 if it is not necessary to apply the "Repair" function after the reading, "Repair" corrects the value of some fields like the counter of the number of higher entities to those an entity is subordinated to)

Layer

Number of layer, Name (variable of type string), isfrozen (0 or 1), ison (0 or 1), RGB_R, RGB_G, RGB_B (RGB color, values from 0 to 255)
<CR>

Null entity

Code of null entity=0<CR> (used as "flag" for end of entities)

NoStructuredMeshData

Entity code= -1, number ID, NumOfEntities (number of entities than point to it), elementtype, MustBeMeshed (0 or 1, default=0), size (edge element size), <CR>

StructuredMeshData

Entity Code= -2, number ID, NumOfEntities, elementtype, MustBeMeshed, Size, <CR>

StructuredWeightedMeshData

Entity Code= -3, number ID, NumOfEntities, elementtype, MustBeMeshed, Size (number of divisions), weight (positive or negative value to a non uniform concentration of the elements)<CR>

Where ElemType={NoNe=0, Linear=1, Triangle=2, Quadrilateral=3, Tetrahedra=4, Hexahedra=5, Prism=6, OnlyPoints=7}

Point

Entity Code=1, number ID, label, **selection**, number of higher entities, conditions=0, material=0, number layer, mesh data=0<CR>

x, y, z <CR>

*The flag value label is only used by old GiD versions (to set on/off the visualization of the entity label), current versions use only selection for both flags (first bit for selection on/off and second bit for label on/off)

Straight segment

Entity Code=2, number GOES, labels, selection, number of higher entities, conditions=0, material=0, number layer, mesh data=0 < CR >

Number of initial point, number of final point <CR>

Arc

Entity Code=3, number ID, label, selection, number of higher entities, conditions=0, material=0, number layer, mesh data=0,<CR>

Number of initial point, number of final point, x center, y center, radius, initial angle, final angle <CR>

TransformationMatrix [0] [0], ..., TransformationMatrix [0] [3] ,<CR>

...

TransformationMatrix [3] [0], ..., TransformationMatrix [3] [3] ,<CR>

The transformation matrix maps the points from 2D to the final 3D location.

Polyline

Entity Code=4, number of ID, label, selection, number of higher entities, conditions=0, material=0, number of layer, mesh data=0,<CR >

Number of initial point, number of final point, number of parts, length, UsePointsInMeshing=0, checknum=0 < CR >

sense[1] ,..., sense[number parts] <CR > (gives the orientation of the sub line, possible values 0 or 1)

length[1], ...,length[number parts] <CR>

here the description for each one of the n sub-curves is written, with the particularity that the number is set = -1 to all of them, to mark that they are not independent entities.

NURBS curve

Entity Code=11, number ID, label, selection, number of higher entities, conditions=0, material=0, number layer, mesh data=0,<CR>

Number of initial point, number of final point, number de polygon control points, degree, length <CR>

x point[1], y point[1], z point[1] <CR>

...

x point [number points], y point [number points], z point [number points] <CR>

knots[1], ... , knots [number points+degree+1], <CR>

IsRational (0 or 1)

Weight[1] ,..., Weight [number points] (only if rational)

The length is ignored when reading, it is recalculated (it could be set to 0.0)

Planar Surface

Entity Code=5, number ID, label, selection, number of higher entities, conditions=0, material=0, number layer, mesh data=0 <CR>

Number of boundary lines, <CR>

number of curve[1], ... , number of curve[number of lines] <CR>

(Note: if the curve is a part of a polyline it should be substituted "number of curve[i]" for "- number curve[i], relative position inside the polyline". Note the negative sign in the polyline number, and the position in the polyline will be 0,1,...)

sense[1], ..., sense[number of lines],<CR> (set the orientation of a segment, must be 0 or 1)

x center, y center, z center, <CR>

x normal, y normal, z normal, <CR> (vector normal to the plane)

Coons Surfaces (defined by 4 sides, their interior is interpolated from the contour)

Entity Code=6, number of ID, label, selection, number of higher entities, conditions=0, material=0, number of layer, mesh data=0,<CR>

Number of boundary lines=4,<CR>

Number of curve[1], ... , number of curve[4], <CR>

(Note: if the curve is part of a polyline it should be substituted "number of curve[i]" for "- number of curve[i], relative location inside the polyline", note the negative sign in the polyline number, and the position in the polyline will be 0,1,...)

sense[1], ..., sense[4] <CR> (set the segment orientation, must be 0 or 1)

x center, y center, z center, <CR>

x normal, y normal, z normal, <CR> (vector normal to the plane)

NURBS Surface

Entity Code=14, number of ID, label, selection, number of higher entities, conditions=0, material=0, number of layer, mesh data=0,<CR >

Number of boundary lines=4, <CR>

Number of curve[1], ... , number of curve[4], <CR>

sense[1], ..., sense[4] <CR> (set the segment orientation, must be 0 or 1)

x center, y center, z center, <CR>

x normal, y normal, z normal, <CR> (vector normal to the plane)

IsTrimmed, NU (number of control points in the U direction), NV, degreeU (polynomial degree), degreeV <CR>

x point[1], y point[1], z point[1], <CR>

...

x point[NU*NV], y point[NU*NV], z point[NU*NV], <CR>

knotsU[1], ... , knotsU [NU+degreeU+1], <CR>

knotsV[1], ... , knotsV [NV+degreeV+1], <CR>

IsRational (0 or 1)

Weight[1] ,..., Weight [NU*NV] (only if rational)

Volume

Entity Code=9, number of ID, label, selection, number of higher entities, conditions=0, material=0, number of layer, mesh data=0 <CR>

Number of boundary surfaces <CR>

Number of surface[1], ... , number of surface[number of surfaces], <CR>

sense[1], ..., sense [number of surfaces], <CR> (segment orientation, 0 or 1)

x center, y center, z center, <CR>

Rules to follow:

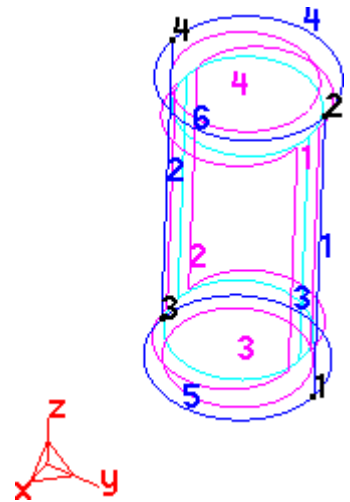
There are four level types of geometrical entities: Point, Curves (straight segment, arc, polyline, Nurbs), Surface (planar, Coon, Nurbs) and Volume.

- Geometrical entities of the same type cannot have the same ID number associated (is not valid to have two curves with same ID) The numeration begins with 1 (not by 0 like in the "C" style), and it could be jumps in the numeration (e.g. when a entity is deleted). The entities of a level are listed with increasing ID.
- The center of the entities doesn't refer to the geometric center, it is simply an approximate center where its label will be drawn. The higher entities number can be initialized to zero, and it will be corrected automatically if "Must Be repaired" flag was set to 1. In any case is recommended to set the right value and avoid the reparation.
- The length of NURBS curves could be set to zero, current versions ignore this value and recalculate it. Parametric curves are normalized to parameter space [0.0,1.0]
- Parametric curve must have its initial point at parameter 0.0 and end point at parameter 1.0. A closed curve must share the same start and end point.
- Parametric surfaces are normalized to parameter space [0.0,1.0]x[0.0,1.0]
- The boundary curves of a surface define a outer loop and some possible inner loops. The outer loop is before outer loops.
- The curves of a loop are ordered consecutively. A loop finish when the starting point of the first curve (taking into account its sense for the surface) is equal to the last point of other curve.
- The ordering and orientation of the boundary curves must agree with the surface normal ($X_u \wedge X_v$) (right-hand rule). The outer loop must point to the same sense as the surface normal, and inner loops is any in the opposite sense.
- Volumes are defined by a closed shell of surfaces: first surfaces must define the outer shell, and then the inner shells.
- The order of surfaces on a 'volume shell' is not relevant, but the orientation must be 0 (SAME1ST) is the surface normal points inside the volume.

Geometry example

This example consists on a simple cylinder, like the one shown on the right. It contains points, curves of type straight lines, circumference arcs and curved NURBS with circumference shape, and surfaces of the types planar, Coon and NURBS with cylindrical form, and there is a single volume.

Note: This model could be found at: Examples\Cylinder_ASCII.gid



```
RAMSAN-ASCII-gid-v7.6
UNKNOWN 0
0
1 Tops 0 1 0 0 255
2 Lateral 0 1 0 255 255
0
0
1 1 1 2 3 0 0 2 0
-1.65134 -1.60324 0
1 2 1 2 3 0 0 2 0
-1.65134 -1.60324 3.76945
1 3 1 2 3 0 0 2 0
-1.80449 -3.49553 0
1 4 1 2 3 0 0 2 0
-1.80449 -3.49553 3.76945
2 1 1 2 2 0 0 2 0
1 2
2 2 1 2 2 0 0 2 0
3 4
11 3 1 2 2 0 0 2 0
1 3 5 2 2.98214
-1.65134 -1.60324 0
-2.59749 -1.52666 0
-2.67407 -2.47281 0
-2.75064 -3.41896 0
-1.80449 -3.49553 0
0 0 0 0.5 0.5 1 1 1
1 1 0.707107 1 0.707107 1
11 4 1 2 2 0 0 2 0
2 4 5 2 2.98214
-1.65134 -1.60324 3.76945
-2.59749 -1.52666 3.76945
-2.67407 -2.47281 3.76945
-2.75064 -3.41896 3.76945
-1.80449 -3.49553 3.76945
0 0 0 0.5 0.5 1 1 1
1 1 0.707107 1 0.707107 1
11 5 1 2 2 0 0 2 0
3 1 5 2 2.98213
-1.80449 -3.49553 0
-0.858344 -3.57211 0
-0.781767 -2.62596 0
-0.70519 -1.67981 0
-1.65134 -1.60324 0
0 0 0 0.5 0.5 1 1 1
1 1 0.707107 1 0.707107 1
3 6 1 2 2 0 0 2 0
4 2 -0.315383 0.025526 0.949244 4.63163 7.77322
1 0 0 0
0 1 0 0
0 0 1 0
-1.41253 -2.57491 3.76945 1
14 1 1 2 1 0 0 2 0
4
1 4 2 3
0 0 1 1
-2.67407 -2.47281 1.88473
0.996741 -0.080672 0
0 2 5 1 2
-1.65134 -1.60324 0
```

```

-1.65134 -1.60324 3.76945
-2.59749 -1.52666 0
-2.59749 -1.52666 3.76945
-2.67407 -2.47281 0
-2.67407 -2.47281 3.76945
-2.75064 -3.41896 0
-2.75064 -3.41896 3.76945
-1.80449 -3.49553 0
-1.80449 -3.49553 3.76945
0 0 1 1
0 0 0 0.5 0.5 1 1 1
1 1 1 0.707107 0.707107 1 1 0.707107 0.707107 1 1
6 2 1 2 1 0 0 2 0
4
1 5 2 6
1 1 0 0
-0.781767 -2.62596 1.88473
-10.6459 0.861634 -0
14 3 1 2 1 0 0 1 0
2
5 3
0 0
-1.72792 -2.54939 0
0 0 1
1 2 2 1 1
-2.7699 -1.5074 0
-2.7699 -3.59137 0
-0.685932 -1.5074 0
-0.685932 -3.59137 0
0 0 1 1
0 0 1 1
0
5 4 1 2 1 0 0 1 0
2
6 4
1 1
-1.41253 -2.57491 3.76945
0 0 -1
9 1 1 2 0 0 0 2 0
4
1 2 4 3
0 0 0 0
-1.72792 -2.54939 1.88473
0

```

This is the explanation of its content:

RAMSAN-ASCII-gid-v7.6
UNKNOWN 0
0

The GiD geometry ASCII file is wrote with rules of version 7.6, and without any problemtype (UNKNOWN)
The model has two layers, created with:

1 Tops 0 1 0 0 255
2 Lateral 0 1 0 255 255
0

The layer number 1 is named 'Tops', is not frozen, visible, and with RGB color R=0, G=0, B=255 (blue)
The layer number 2 is named 'Lateral', is not frozen, visible, and with RGB color R=0, G=255, B=255 (cyan)
Last 0 denotes the end of layers block

0
There is no meshing information attached to entities.

Then four points (code=1) are defined:

1 1 1 2 3 0 0 2 0
-1.65134 -1.60324 0
1 2 1 2 3 0 0 2 0
-1.65134 -1.60324 3.76945
1 3 1 2 3 0 0 2 0

```
-1.80449 -3.49553 0
1 4 1 2 3 0 0 2 0
-1.80449 -3.49553 3.76945
```

```
p1=(-1.65134,-1.60324,0)
p2=(-1.65134,-1.60324,3.76945)
p3=(-1.80449,-3.49553,0)
p4=(-1.80449,-3.49553,3.76945)
```

```
the meaning of
1 1 1 2 3 0 0 2 0
is
1==type_point 1=point_id 1=label_on 2=label_on_selection_off 3=higherentities 0=num_conditions 0=id_material 2=layer 'Lateral'
0=has_mesh_data
the point 1 belong to 3 curves (1, 3 and 5) then higherentities must be 3
```

```
2 1 1 2 2 0 0 2 0
1 2
2 2 1 2 2 0 0 2 0
3 4
```

This define curves 1 and 2 that are straight lines (type==2). In this model that close a volume all curves belong to two surfaces, then its higherentity counter is 2
The curve 1 starts in the point 1 and end in the point 2

```
11 3 1 2 2 0 0 2 0
1 3 5 2 2.98214
-1.65134 -1.60324 0
-2.59749 -1.52666 0
-2.67407 -2.47281 0
-2.75064 -3.41896 0
-1.80449 -3.49553 0
0 0 0 0.5 0.5 1 1 1
1 1 0.707107 1 0.707107 1
11 4 1 2 2 0 0 2 0
2 4 5 2 2.98214
-1.65134 -1.60324 3.76945
-2.59749 -1.52666 3.76945
-2.67407 -2.47281 3.76945
-2.75064 -3.41896 3.76945
-1.80449 -3.49553 3.76945
0 0 0 0.5 0.5 1 1 1
1 1 0.707107 1 0.707107 1
11 5 1 2 2 0 0 2 0
3 1 5 2 2.98213
-1.80449 -3.49553 0
-0.858344 -3.57211 0
-0.781767 -2.62596 0
-0.70519 -1.67981 0
-1.65134 -1.60324 0
0 0 0 0.5 0.5 1 1 1
1 1 0.707107 1 0.707107 1
```

Previous text define curves 3, 4 and 5 that are NURBS (type==11)
1=start point 3=end point 5=num control points 2=degree 2.98214=length
the 5 control points coordinates are:
-1.65134 -1.60324 0
-2.59749 -1.52666 0
-2.67407 -2.47281 0
-2.75064 -3.41896 0
-1.80449 -3.49553 0
and the knots vector is:
0 0 0 0.5 0.5 1 1 1
1=is rational, weights= 1 0.707107 1 0.707107 1

```
3 6 1 2 2 0 0 2 0
4 2 -0.315383 0.025526 0.949244 4.63163 7.77322
1 0 0 0
0 1 0 0
0 0 1 0
-1.41253 -2.57491 3.76945 1
```


And curve 6 is a circumference arc (type==11)

4=start point 2=end point (-0.315383 0.025526)=2D center 0.949244=radius 4.63163=start angle 7.77322=end angle (rad)

and

1 0 0 0

0 1 0 0

0 0 1 0

-1.41253 -2.57491 3.76945 1

is a 4x4 transformation matrix that moves the 2D arc to the final 3D location

14 1 1 2 1 0 0 2 0

4

1 4 2 3

0 0 1 1

-2.67407 -2.47281 1.88473

0.996741 -0.080672 0

0 2 5 1 2

-1.65134 -1.60324 0

-1.65134 -1.60324 3.76945

-2.59749 -1.52666 0

-2.59749 -1.52666 3.76945

-2.67407 -2.47281 0

-2.67407 -2.47281 3.76945

-2.75064 -3.41896 0

-2.75064 -3.41896 3.76945

-1.80449 -3.49553 0

-1.80449 -3.49553 3.76945

0 0 1 1

0 0 0 0.5 0.5 1 1 1

1 1 1 0.707107 0.707107 1 1 0.707107 0.707107 1 1

Surface 1 is a NURBS surface (type==14), it has 4 boundary lines: 1, 4, 2, 3, with orientations same, same, diff, diff respectively

Its approximated center is (-2.67407 -2.47281 1.88473)

normal=(0.996741 -0.080672 0)

0=untrimmed 2=number control points u 5=number control points v 1=degree u 2=degree v

then are listed the control points, the knots u=(0 0 1 1) and knots v=(0 0 0 0.5 0.5 1 1 1)

and the weights=(1 1 0.707107 0.707107 1 1 0.707107 0.707107 1 1)

6 2 1 2 1 0 0 2 0

4

1 5 2 6

1 1 0 0

-0.781767 -2.62596 1.88473

-10.6459 0.861634 -0

Surface 1 is a Coons surface (type==6), it has 4 boundary lines: 1, 5, 2, 6, with orientations diff, diff, same, same. Then is printed its approximated center and normal.

The shape of kind of surface is defined only by its boundary.

14 3 1 2 1 0 0 1 0

2

5 3

0 0

-1.72792 -2.54939 0

0 0 1

1 2 2 1 1

-2.7699 -1.5074 0

-2.7699 -3.59137 0

-0.685932 -1.5074 0

-0.685932 -3.59137 0

0 0 1 1

0 0 1 1

0

Surface 3 is like the 1 a NURBS surface (type==14), but in this case is trimming a planar squared shape.

5 4 1 2 1 0 0 1 0

2

6 4

1 1

-1.41253 -2.57491 3.76945
0 0 -1

Surface 4 is a planar surface (type==5), that is defined by a center and normal, and the trimming boundary

lines.9 1 1 2 0 0 0 2 0
4
1 2 4 3
0 0 0 0
-1.72792 -2.54939 1.88473
0

This define the volume 1 (type==9) that is defined by 4 boundary surfaces: 1, 2, 4, 3 with orientations same, same, same, same.
and approximated center=(-1.72792 -2.54939 1.88473)

Last 0 is a NULL entity (type==0) that finish the definition of geometrical entities.

POSTPROCESS DATA FILES

In GiD Postprocess you can study the results obtained from a solver program. The solver and GiD Postprocess communicate through the transfer of files. The solver program has to write the results to a file that must have the extension *.post.res* and its name must be the project name. Other allowed extensions are *.post.lst*, *.post.bin* and *.post.h5*.

The solver program can also send the postprocess mesh to GiD (though this is not mandatory), in the same *.post.res*, *.post.bin* or *.post.h5* file or in a separate *.post.msh* acii file. If this mesh is not provided by the solver program, GiD uses the preprocess mesh in Postprocess.

Other files with the extensions *.msh*, *.res*, *.bin*, *.lst*, *.h5* can be read into postprocess but only those with the *.post* suffix are automatically read into GiD when entering the post-process model

Postprocessing data files can be:

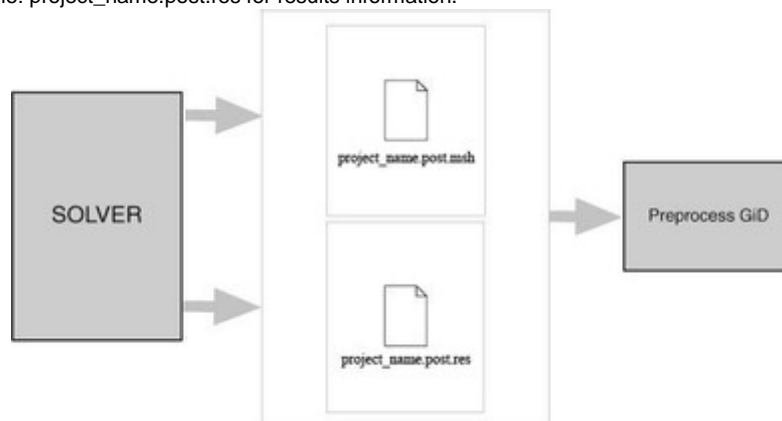
- ASCII files, where the mesh and results should be provided in two separated files;
- binary files, a binary version of the ASCII files but mesh and results can be stored in one file; hdf5 files, mesh and results are stored in one hdf5 file.

All three formats can be written using the freely available GiDPost library, whose source code, and some pre-built binaries, can be downloaded from <https://www.gidsimulation.com> --> GiDPlus --> GiDPost.

From now on the explanation will be focused in the ASCII format. The details about the binary and hdf5 format are explained afterwards in a separated section.

The ASCII format consists of two files:

- **Mesh Data File:** *project_name.post.msh* for volume and surface (3D or 2D) mesh information,
- and **Results Data File:** *project_name.post.res* for results information.



Note: *ProjectName.post.msh* handles meshes of different element types: points, lines, triangles, quadrilaterals, tetrahedra and hexahedra.

If a project is loaded into GiD, when changing to GiD Postprocess it will look for *ProjectName.post.res*. If a mesh information file with the name *ProjectName.post.msh* is present, it will also be read, regardless of the information available from GiD Preprocess.

• **ProjectName.post.msh:** This file should contain nodal coordinates of the mesh and its nodal connectivities as well as the material of each element. At the moment, only one set of nodal coordinates can be entered. Different kinds of elements can be used but separated into different sets. If no material is supplied, GiD takes the material number to be equal to zero.

• **ProjectName.post.res:** This second file must contain the nodal or gaussian variables. GiD lets you define as many nodal variables as desired, as well as several steps and analysis cases (limited only by the memory of the machine). The definitions of the Gauss points and the results defined on these points should also be written in this file.

The files are created and read in the order that corresponds to the natural way of solving a finite element problem: mesh, surface definition and conditions and finally, evaluation of the results. The format of the read statements is normally free, i.e. it is necessary only to separate them by spaces.

Thus, files can be modified with any format, leaving spaces between each field, and the results can also be written with as many decimal places as desired. Should there be an error, the program warns the user about the type of mistake found.

GiD reads all the information directly from the preprocessing files whenever possible in order to gain efficiency.

Results format: ModelName.post.res

Note: Code developers can download the [GiDpost](https://www.gidsimulation.com/downloads/gidpost) tool from the GiD web page (<https://www.gidsimulation.com/downloads/gidpost>); this is a C/C++/Fortran library for creating postprocess files for GiD in both ASCII and compressed binary format.

This is the ASCII format description:

The first line of the files with results written in this new postprocess format should be:

```
GiD Post Results File <version>
```

where <version> must be:

1.0 in general

>=1.1 in case of binary compressed format

>=1.2 in case of contain some OnNurbsSurface result

Comment lines are allowed and should begin with a '#'. Blank lines are also allowed.

Results files can also be included with the keyword **include**, for instance:

```
include "My Other Results File"
```

This is useful, for instance, for sharing several GaussPoints definitions and ResultRangeTable among different analyses.

This 'include' should be outside the **Blocks** of information.

There are several types of **Blocks** of information, all of them identified by a keyword:

- GaussPoints: Information about gauss points: name, number of gauss points, natural coordinates, etc.;
- ResultRangesTable: Information for the result visualization type **Contour Ranges**: name, range limits and range names;
- Result: Information about a result: name, analysis, analysis/time step, type of result, location, values;
- ResultGroup: several results grouped in one block. These results share the same analysis, time step, and location (nodes or gausspoints).

Gauss Points

If Gauss points are to be included, they must be defined before the Result which uses them. Each Gauss points block is defined between the lines GaussPoints and End GaussPoints.

The structure is as follows, and should:

- Begin with a header that follows this model:

GaussPoints "gauss_points_name" **Elemtype** my_type "mesh_name"

where

- GaussPoints, elemtype: are not case-sensitive;
- "gauss_points_name": is a name for the gauss points set, which will be used as reference by the results that are located on these gauss points;
- my_type: describes which element type these gauss points are for, i.e. Line, Triangle, Quadrilateral, Tetrahedra, Prism, Pyramid or Hexahedra ;
- "mesh_name": is an optional field. If this field is missing, the gauss points are defined for all the elements of type my_type. If a meshname is given, the gauss points are only defined for this mesh.

- Be followed by the gauss points properties:

Number of Gauss Points: number_gauss_points_per_element

Nodes included

Nodes not included

Natural Coordinates: Internal

Natural Coordinates: Given

natural_coordinates_for_gauss_point_1 . . .

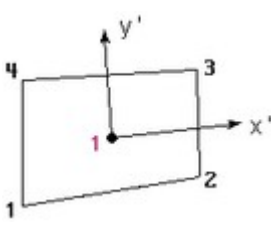
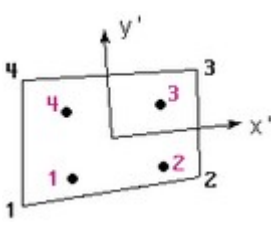
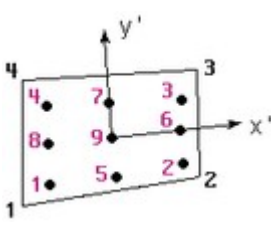
natural_coordinates_for_gauss_point_n

where

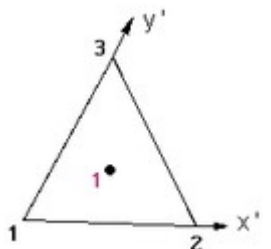
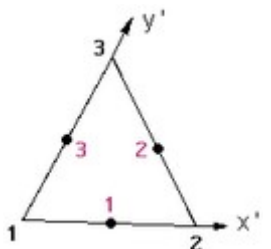
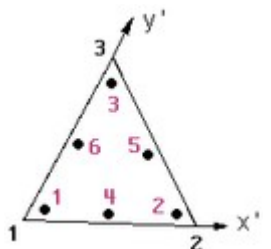
- Number of Gauss Points: number_gauss_points_per_element: is not case-sensitive and is followed by the number of gausspoints per element that defines this set. If Natural Coordinates: is set to Internal, number_gauss_points_per_element should be one of:
 - 1, 3, 6 for Triangles;
 - 1, 4, 9 for Quadrilaterals;
 - 1, 4, 10 for Tetrahedra;
 - 1, 8, 27 for Hexahedra;
 - 1, 6 for Prisms;
 - 1, 5 for Pyramids; and
 - 1, ... n points equally spaced over lines.

For triangles and quadrilaterals the order of the gauss points with **Internal** natural coordinates will be this:

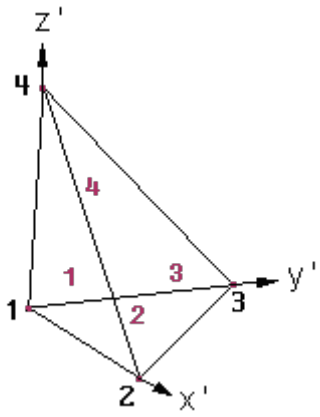
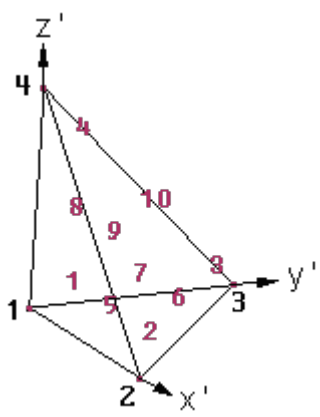
Gauss Points positions of the quadrature of Gauss-Legendre Quadrilaterals

		
Internal coordinates: (0, 0)	Internal coordinates: $a=0.57735027$ (-a,-a) (a,-a) (a, a) (-a, a)	Internal coordinates: $a=0.77459667$ (-a,-a) (a,-a) (a, a) (-a, a) (0,-a) (a, 0) (0, a) (-a, 0) (0, 0)

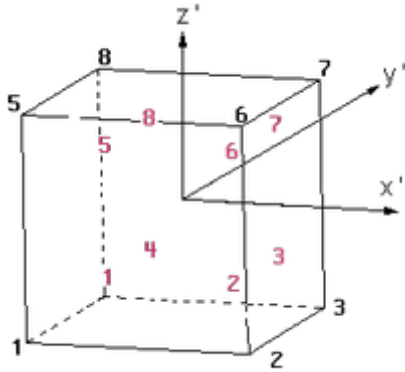
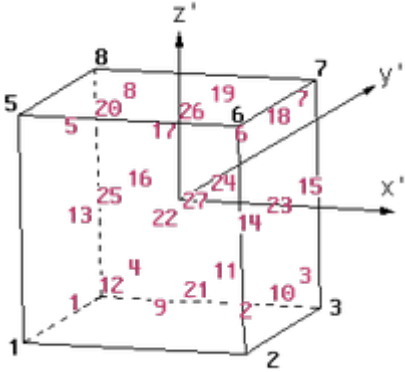
Gauss Points positions of the quadrature of Gauss for Triangles

		
Internal coordinates: $a=1/3$ (a, a)	Internal coordinates: $a=1/2$ (a, 0) (a, a) (0, a)	Internal coordinates: $a=0.09157621$ $b=0.81684757$ $c=0.44594849$ $d=0.10810301$ (a, a) (b, a) (a, b) (c, d) (c, c) (d, c)

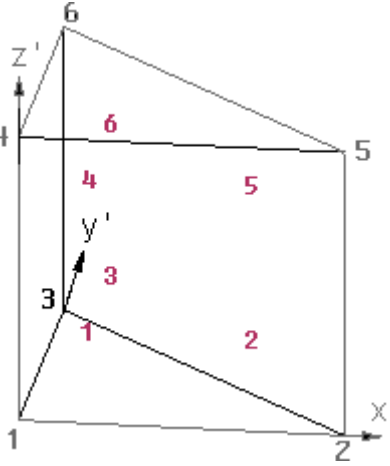
For tetrahedra the order of the **Internal** Gauss Points is this:

	
<p>Internal coordinates: $a = (5 + 3\sqrt{5})/20 = 0.585410196624968$ $b = (5 - \sqrt{5})/20 = 0.138196601125010$</p> <p>(b, b, b) (a, b, b) (b, a, b) (b, b, a)</p>	<p>Internal coordinates: $a = 0.108103018168070$ $b = 0.445948490915965$ $c = 0.816847572980459$</p> <p>(a, a, a) (c, a, a) (a, c, a) (a, a, c) (b, a, a) (b, b, a) (a, b, a) (a, a, b) (b, a, b) (a, b, b)</p>

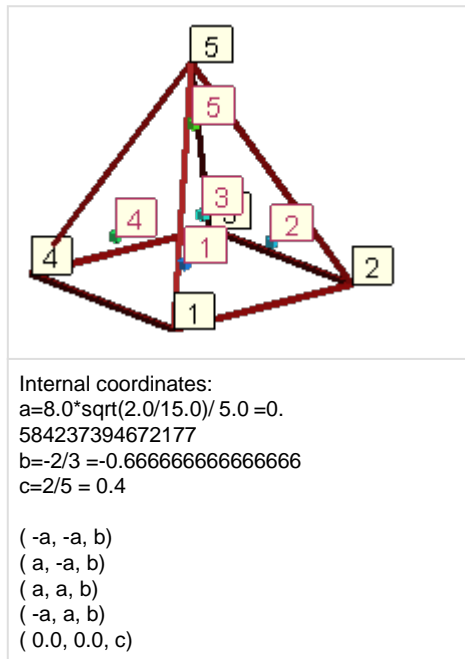
For hexahedra the order of the **Internal** Gauss Points is this:

	
<p>Internal coordinates: $a=0.577350269189626$</p> <p>$(-a, -a, -a)$ $(a, -a, -a)$ $(a, a, -a)$ $(-a, a, -a)$ $(-a, -a, a)$ $(a, -a, a)$ (a, a, a) $(-a, a, a)$ $(0, -a, -a)$ $(a, 0, -a)$ $(0, a, -a)$ $(-a, 0, -a)$ $(-a, -a, 0)$ $(a, -a, 0)$ $(a, a, 0)$ $(-a, a, 0)$ $(0, -a, a)$ $(a, 0, a)$ $(0, a, a)$ $(-a, 0, a)$</p> <p>$(0, 0, -a)$ $(0, -a, 0)$ $(a, 0, 0)$ $(0, a, 0)$ $(-a, 0, 0)$ $(0, 0, a)$ $(0, 0, 0)$</p>	<p>Internal coordinates: $a = 0.774596669241483$</p> <p>$(-a, -a, -a)$ $(a, -a, -a)$ $(a, a, -a)$ $(-a, a, -a)$ $(-a, -a, a)$ $(a, -a, a)$ (a, a, a) $(-a, a, a)$ $(0, -a, -a)$ $(a, 0, -a)$ $(0, a, -a)$ $(-a, 0, -a)$ $(-a, -a, 0)$ $(a, -a, 0)$ $(a, a, 0)$ $(-a, a, 0)$ $(0, -a, a)$ $(a, 0, a)$ $(0, a, a)$ $(-a, 0, a)$</p> <p>$(0, 0, -a)$ $(0, -a, 0)$ $(a, 0, 0)$ $(0, a, 0)$ $(-a, 0, 0)$ $(0, 0, a)$ $(0, 0, 0)$</p>

For prisms the order of the **Internal** Gauss Points is this:


<p>Internal coordinates: $a=1/6=0.1666666666666666$ $b=4/6=0.6666666666666666$ $c=1/2-1/(2\sqrt{3})=0.211324865405187$ $d=1/2+1/(2\sqrt{3})=0.788675134594812$</p> <p>$(a, a, c)$ (b, a, c) (a, b, c) (a, a, d) (b, a, d) (a, b, d)</p>

For pyramids the order of the **Internal** Gauss Points will be this:

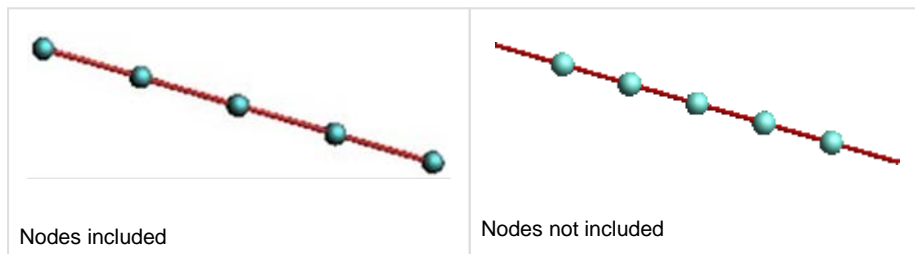


The **given** natural coordinates for Gauss Points should range:

between **0.0** and **1.0** for Triangles, Tetrahedra and Prisms, and
 between **-1.0** and **1.0** for Quadrilaterals, Hexahedra and Pyramids.

Note: If the natural coordinates used are the internal ones, almost all the Results visualization possibilities will have some limitations for tetrahedra and hexahedra with more than one gauss points. If the natural coordinates are given, these limitations are extended to those elements with number_gauss_points_per_element not included in the list written above.

- Nodes Included / Nodes not Included: are not case-sensitive, and are only necessary for gauss points on Line elements which indicate whether or not the end nodes of the Line element are included in the number_gauss_points_per_element count.



The default value is nodes not included

Note: By now, Natural Coordinates for line elements cannot be "Given"

- Natural Coordinates: Internal / Natural Coordinates: Given: are not case-sensitive, and indicate whether the natural coordinates are calculated internally by GiD, or are given in the following lines. The natural coordinates should be written for each line and gauss point.

- End with this tail:

End GaussPoints

where End GaussPoints: is not case-sensitive.

Here is an example of results on Gauss Points:

```
GaussPoints "Board gauss internal" ElemType Triangle "board"
  Number Of Gauss Points: 3
  Natural Coordinates: internal
end gausspoints
```

Internal Gauss points

The following Internal gauss points are automatically defined.

Results can use these names without explicitly define them with a GaussPoints / End GaussPoints statement.

```
GP_POINT_1
GP_LINE_1
GP_TRIANGLE_1 GP_TRIANGLE_3 GP_TRIANGLE_6
GP_QUADRILATERAL_1 GP_QUADRILATERAL_4 GP_QUADRILATERAL_9
GP_TETRAHEDRA_1 GP_TETRAHEDRA_4 GP_TETRAHEDRA_10
GP_HEXAHEDRA_1 GP_HEXAHEDRA_8 GP_HEXAHEDRA_27
GP_PRISM_1 GP_PRISM_6
GP_PIRAMID_1 GP_PIRAMID_5
GP_SPHERE_1
GP_CIRCLE_1
```

Is is possible to use also the generic name GP_ELEMENT_1 to mean all kind of elements with 1 gauss point (instead of the specific element GP_LINE_1, GP_TRIANGLE_1, etc.

Result Range Table

A *Result Range Table* is a customized legends for the *Contour Ranges* visualization and referenced by the *Results*. Each *ResultRangesTable* consist of a list that associate a string or keyword to a range of result values.

If a *Result Range Table* is to be included, it must be defined before the *Result* which uses it.

Each *Result Range Table* is defined between the lines *ResultRangesTable* and *End ResultRangesTable*.

The structure is as follows and should:

- Begin with a header that follows this model:

ResultRangesTable "ResultsRangeTableName"

where ResultRangesTable: is not case-sensitive; "ResultsRangeTableName": is a name for the Result Ranges Table, which will be used as a reference by the results that use this Result Ranges Table.

- Be followed by a list of Ranges, each of them defined as

follows:Min_Value - Max_Value: "Range Name"
where

Min_value : is the minimum value of the range, and may be void if the Max_value is given. If void, the minimum value of the result will be used;

Max_value : is the maximum value of the range, and may be void if the Min_value is given. If void, the maximum value of the result will be used;

"Range Name" : is the name of the range which will appear on legends and labels.

- End with this tail:

End ResultRangesTable

where

End ResultRangesTable: is not case-sensitive.

- Here are several examples of results range tables:

Ranges defined for the whole result:

```
ResultRangesTable "My table"
# all the ranges are min <= res < max except
# the last range is min <= res <= max
  - 0.3: "Less"
  0.3 - 0.7: "Normal"
  0.7 -      : "Too much"
End ResultRangesTable
```

Just a couple of ranges:

```
ResultRangesTable "My table"
  0.3 - 0.7: "Normal"
  0.7 - 0.9: "Too much"
End ResultRangesTable
```

Or using the maximum of the result:

```
ResultRangesTable "My table"
  0.3 - 0.7: "Normal"
  0.7 -      : "Too much"
End ResultRangesTable
```

Result

Each Result block is identified by a Result header, followed by several optional properties: component names, ranges table, and the result values, defined by the lines Values and End Values.

The structure is as follows and should:

- Begin with a result header that follows this model:

Result "result name" "analysis name" step_value result_type result_location "location name"

where

Result: is not case-sensitive;

"result name": is a name for the Result, which will be used for menus; if the result name contains spaces it should be written between "" or between {}.

"analysis name": is the name of the analysis of this Result, which will be used for menus; if the analysis name contains spaces it should be written between "" or between {}.

step_value: is the value of the step inside the analysis "analysis name";

result_type: describes the type of the Result. It should be one of the following:

- Scalar: one component per result
- Vector: two, three or four components for result: x, y, z and (signed) modulusMatrix: three components for 2D matrices, six components for 3D matrices PlainDeformationMatrix: four components: Sxx, Syy, Sxy, Szz
- MainMatrix: the three main unitary eigen vectors (three components each) and three eigen values of the matrix LocalAxes: three euler angles to specify the local axis
- ComplexScalar: two components to specify $a + b \cdot i$
- ComplexVector: four components for 2D complex vectors, six or nine components for 3D vectors: rX iX rY iY rZ iZ |r| |i| |vector| -->to specify the vector (rX + iX, rY + iY, rZ + iZ)
- ComplexMatrix: six components for 2D matrices (Sxx_real, Syy_real, Sxy_real, Sxx_imag, Syy_imag, Sxy_imag), and twelve components for 3D matrices (Sxx_real, Syy_real, Szz_real, Sxy_real, Syz_real, Sxz_real, Sxx_imag, Syy_imag, Szz_imag, Sxy_imag, Syz_imag, Sxz_imag)

- result_location: is where the Result is located. It should be one of the following:

OnNodes

••

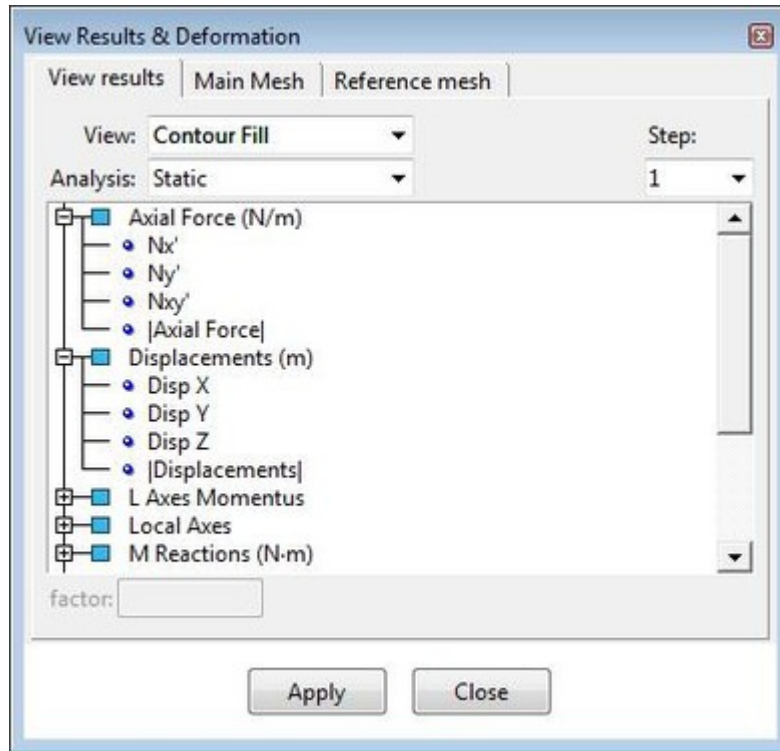
OnGaussPoints

OnNurbsSurface

If the Result is OnGaussPoints, a "location name" should be entered;

"location name": is the name of the Gauss Points on which the Result is defined.

Note: Results can be visually grouped into 'folders' like in the following picture



by just grouping of results using double slashes in the result names:
 Result "Mechanical//Pressures//Water pressure" "Time analysis" 60
 Result "Physical//Saturation" "Time analysis" 60 Scalar OnNodes
 and so on...

- Be followed (optionally) by result properties:

ResultRangesTable "Name of a result ranges table"

ComponentNames "Name of Component 1", "Name of Component 2"

Unit "result unit"

where

ResultRangesTable "Name of a result ranges table": (optional) is not case-sensitive, followed by the name of the previously defined Result Ranges Table, which will be used if the Contour Ranges result visualization is chosen (see [Result Range Table](#));

ComponentNames "Name of Component 1", "Name of Component 2": (optional) is not case-sensitive, followed by the names of the components of the results which will be used in GiD. Missing components names will be automatically generated. The number of Component Names are:

One for a Scalar Result
 Three for a Vector Result
 Six for a Matrix Result
 Four for a PlainDeformationMatrix Result
 Six for a MainMatrix Result
 Three for a LocalAxes Result
 Two for a ComplexScalar Result
 Six or nine for ComplexScalar

Unit: the unit of the result.

End with the result values:

Values

node_or_elem_number component_1_value ... component_n_value
 ...
 node_or_elem_number component_1_value ... component_n_value

End Values

where

Values: is not case-sensitive, and indicates the beginning of the results values section;

The lines

```
node_or_elem_number component_1_value component_2_value
```

```
...
```

```
node_or_elem_number component_1_value component_2_value
```

are the values of the result at the related 'node_or_elem_number'.

The number of results values are limited thus:

If the Result is located OnNodes, they are limited to the number of nodes defined in ProjectName.post.msh.

If the Result is located OnGaussPoints "My GP", and if the Gauss Points "My GP" are defined for the mesh "My mesh", the limit is the number of gauss points in "My GP" multiplied by the number of elements of the mesh "My mesh".

For results in gauss points, each element must have 'ngauss' lines of results.

For example, if the number of gauss points is 3, then for an element, 3 lines of gauss point scalar result must appear.

```
Values
1 1.155
   2.9
   3.955
End Values
```

Holes are allowed in any result. The element nodes with no result defined will not be drawn, i.e. they will appear transparent.

Result location:

OnNodes: results are defined on the nodes of the mesh. If nodes are shared between elements, as usual, then the result field is continuous.

OnGaussPoints: results are defined in some locations of the mesh elements. Usually solvers calculate values in some special locations named 'gauss points' for its numerical integration, then is natural to write the results on these locations. The results field is discontinuous between elements, then some options like calculate streamlines are not allowed for this kind of result. Sometimes results are extrapolated and averaged on nodes providing a continuous and smoothed result.

OnNurbsSurface: results are defined on the 'control points' of the NURBS surfaces.

These surfaces by default are the ones of preprocess, but could be overridden by an optional file <modelname.post.geo> with the same format as the .geo preprocess file. [Geometry format: Modelname.geo](#). In this case the .post.geo file must contain only surfaces of NURBS type, and any volume.

Note: in case of have some result OnNurbsSurface the file header must be declared with version greater of equal to 1.2

GiD Post Results File 1.2

The values block of a result OnNurbsSurface must have for each surface with results something like this:

```
surface_number
value(1)_component_1 ...value(1)_component_n
...
value(num_control_points_u)_component_1 ... value(num_control_points_u)_component_n
...
value(num_control_points_u_x_num_control_points_v)_component_1... value(num_control_points_u_x_num_control_points_v)
_component_n
```

where:

surface_number is the id of the surface (the same integer number that identify the surface in preprocess)

value is the real number of the result. It is compulsory to write the values for all control points of the surface, if some control point doesn't has result it must have a NR value to represent 'no result'. This is typical of trimmed NURBS surfaces, some control points far of trimmed part are not relevant and doesn't have any value.

If the whole surface doesn't has OnNurbsSurface result, it is not necessary to be written.

There is a model with isogeometric results OnNurbsSurface at: Examples\IGA_shell.gid\IGA_shell.post.res

The number of components for each Result Value are:

for Scalar results: one component result_number_i scalar_value

for Vector results: three components, with an optional fourth component for signed modules result_number_i x_value y_value z_value result_number_i x_value y_value z_value signed_module_value

for Matrix results: three components (2D models) or six components (3D models)

2D: result_number_i Sxx_value Syy_value Sxy_value

3D: result_number_i Sxx_value Syy_value Szz_value Sxy_value Syz_value Sxz_value

for PlainDeformationMatrix results: four components result_number_i Sxx_value Syy_value Sxy_value Szz_value

for MainMatrix results: twelve components result_number_i Si_value Sii_value Siii_value Vix_value Viy_value Viz_value Viix_value Viiy_value Viiz_value Viii_x_value Viii_y_value Viii_z_value

for LocalAxes results: three components describing the Euler angles result_number_i euler_ang_1_value euler_ang_2_value euler_ang_3_value.

Look for LocalAxesDef(EulerAngles) at [Multiple values return commands](#) for a more detailed explanation to calculate axis from euler angles and vice-versa.

for ComplexScalar results: two components to specify $a + b \cdot i$

for ComplexVector results: four components for 2D complex vectors, six or nine components for 3D vectors: rX iX rY iY rZ iZ |r| |i| |vector| --> to specify the vector (rX + iX, rY + iY, rZ + iZ)

•End Values: is not case-sensitive, and indicates the end of the results values section.

Note: there is a special real value ((float)-3.40282346638528860e+38) that mean NO_RESULT, must not be operated like the rest of real values (e.g. must be ignored to calculate an averaged value)

There is a Tcl proc IsResultNotDefined to check if a value is 'NO_RESULT' , and other proc GetResultNotDefined that return this special value.

Note: For Matrix and PlainDeformationMatrix results, the Si, Sii and Siii components are calculated by GiD, which represents the eigen values & vectors of the matrix results, and which are ordered according to the eigen value.

Results example

Here is an example of results for the table in the previous example (see [Mesh example](#)):

```

GiD Post Results File 1.0

GaussPoints "Board gauss internal" ElemType Triangle "board"
  Number Of Gauss Points: 3
  Natural Coordinates: internal
end gausspoints

GaussPoints "Board gauss given" ElemType Triangle "board"
  Number Of Gauss Points: 3
  Natural Coordinates: Given
    0.2 0.2
    0.6 0.2
    0.2 0.6
End gausspoints

GaussPoints "Board elements" ElemType Triangle "board"
  Number Of Gauss Points: 1
  Natural Coordinates: internal
end gausspoints

GaussPoints "Legs gauss points" ElemType Line
  Number Of Gauss Points: 5
  Nodes included
  Natural Coordinates: Internal
End Gausspoints

ResultRangesTable "My table"
# el ultimo rango es min <= res <= max
  0.3 - 0.9: "Normal"
  0.9 - 1.2: "Too much"
End ResultRangesTable

Result "Gauss element" "Load Analysis" 1 Scalar OnGaussPoints "Board elements"
Values
  5      0.00000E+00
  6      0.20855E-04
  7      0.35517E-04
  8      0.46098E-04
  9      0.54377E-04
 10      0.60728E-04
 11      0.65328E-04
 12      0.68332E-04
 13      0.69931E-04
 14      0.70425E-04
 15      0.70452E-04
 16      0.51224E-04
 17      0.32917E-04
 18      0.15190E-04
 19     -0.32415E-05
 20     -0.22903E-04
 21     -0.22919E-04
 22     -0.22283E-04
End Values

Result "Displacements" "Load Analysis" 1 Vector OnNodes
ResultRangesTable "My table"
ComponentNames "X-Displ", "Y-Displ", "Z-Displ"
Values
  1      0.0      0.0      0.0
  2     -0.1      0.1      0.5
  3      0.0      0.0      0.8
  4     -0.04     0.04     1.0
  5     -0.05     0.05     0.7
  6      0.0      0.0      0.0
  7     -0.04     -0.04     1.0
  8      0.0      0.0      1.2
  9     -0.1     -0.1      0.5
 10      0.05     0.05     0.7
 11     -0.05     -0.05     0.7
 12      0.04     0.04     1.0
 13      0.04     -0.04     1.0
 14      0.05     -0.05     0.7
 15      0.0      0.0      0.0
 16      0.1      0.1      0.5
 17      0.0      0.0      0.8
 18      0.0      0.0      0.0
 19      0.1     -0.1      0.5
End Values

Result "Gauss displacements" "Load Analysis" 1 Vector OnGaussPoints "Board gauss given"
Values
  5      0.1     -0.1      0.5

```



```

        0.0    0.0    0.8
        0.04 -0.04    1.0
6       0.0    0.0    0.8
       -0.1   -0.1    0.5
       -0.04 -0.04    1.0
7       -0.1    0.1    0.5
        0.0    0.0    0.8
       -0.04  0.04    1.0
8        0.0    0.0    0.8
        0.1    0.1    0.5
        0.04  0.04    1.0
9        0.04  0.04    1.0
        0.1    0.1    0.5
       0.05  0.05    0.7
10       0.04  0.04    1.0
       0.05  0.05    0.7
      -0.04  0.04    1.0
11      -0.04 -0.04    1.0
      -0.1   -0.1    0.5
      -0.05 -0.05    0.7
12      -0.04 -0.04    1.0
      -0.05 -0.05    0.7
        0.04 -0.04    1.0
13      -0.1    0.1    0.5
      -0.04  0.04    1.0
      -0.05  0.05    0.7
14      -0.05  0.05    0.7
      -0.04  0.04    1.0
       0.05  0.05    0.7
15       0.1   -0.1    0.5
       0.04 -0.04    1.0
       0.05 -0.05    0.7
16       0.05 -0.05    0.7
       0.04 -0.04    1.0
      -0.05 -0.05    0.7
17       0.0    0.0    0.8
      -0.04 -0.04    1.0
      -0.04  0.04    1.0
18       0.0    0.0    0.8
       0.04  0.04    1.0
       0.04 -0.04    1.0
19       0.04 -0.04    1.0
       0.04  0.04    1.0
       0.0    0.0    1.2
20       0.04 -0.04    1.0
       0.0    0.0    1.2
      -0.04 -0.04    1.0
21      -0.04 -0.04    1.0
       0.0    0.0    1.2
      -0.04  0.04    1.0
22      -0.04  0.04    1.0
       0.0    0.0    1.2
       0.04  0.04    1.0
End Values

Result "Legs gauss displacements" "Load Analysis" 1 Vector OnGaussPoints "Legs gauss points"
Values
  1   -0.1  -0.1   0.5
      -0.2  -0.2  0.375
      -0.05 -0.05  0.25
       0.2   0.2  0.125
       0.0   0.0   0.0
  2   0.1   -0.1   0.5
       0.2  -0.2  0.375
       0.05 -0.05  0.25
      -0.2   0.2  0.125
       0.0   0.0   0.0
  3   0.1   0.1   0.5
       0.2   0.2  0.375
       0.05  0.05  0.25
      -0.2  -0.2  0.125
       0.0   0.0   0.0
  4  -0.1   0.1   0.5
      -0.2   0.2  0.375
      -0.05  0.05  0.25
       0.2  -0.2  0.125
       0.0   0.0   0.0
End Values

```

Result group

Results can be grouped into one block. These results belong to the same time step of the same analysis and are located in the same place. So all the results in the group are nodal results or are defined over the same gauss points set.

Each Result group is identified by a ResultGroup header, followed by the results descriptions and its optional properties - such as components names and ranges tables, and the results values - all between the lines Values and End values.

The structure is as follows and should:

- Begin with a header that follows this model

ResultGroup "analysis name" step_value my_location "location name"
where

ResultGroup: is not case-sensitive;

"analysis name": is the name of the analysis of this ResultGroup, which will be used for menus; if the analysis name contains spaces it should be written between "" or between {}.

step_value: is the value of the step inside the analysis "analysis name";

my_location: is where the ResultGroup is located. It should be one of the following: OnNodes, OnGaussPoints. If the ResultGroup is OnGaussPoints, a "location name" should be entered.

"location name": is the name of the Gauss Points on which the ResultGroup is defined.

- Be followed by at least one of the results descriptions of the group

ResultDescription "result name" my_result_type[:components_number]

ResultRangesTable "Name of a result ranges table"

ComponentNames "Name of Component 1", "Name of Component 2"

where

ResultDescription: is not case-sensitive;

"result name": is a name for the Result, which will be used for menus; if the result name contains spaces it should be written between "" or between {}.

my_type: describes the type of the Result. It should be one of the following: Scalar, Vector, Matrix, PlainDeformationMatrix, MainMatrix, or LocalAxes. The number of components for each type is as follows:

One for a Scalar: the_scalar_value

Three for a Vector: X, Y and Z

Six for a Matrix: Sxx, Syy, Szz, Sxy, Syz and Sxz

Four for a PlainDeformationMatrix: Sxx_value, Syy, Sxy and Szz

Twelve for a MainMatrix: Si, Sii, Siii, ViX, ViY, ViZ, ViiX, ViiY, ViiZ, ViiiX, ViiiY and ViiiZ

Three for a LocalAxes: euler_ang_1, euler_ang_2 and euler_ang_3

Two for ComplexScalar: real and imag

Six for ComplexVector: x_real, x_imag, y_real, y_imag, z_real, z_imag

Twelve for ComplexMatrix: Sxx_real, Syy_real, Szz_real, Sxy_real, Syz_real, Sxz_real, Sxx_imag, Syy_imag, Szz_imag, Sxy_imag, Syz_imag, Sxz_imag

Following the description of the type of the result, an optional modifier can be appended to specify the number of components separated by a colon. It only makes sense to indicate the number of components on vectors and matrices:

Vector:2, Vector:3 or Vector:4: which specify:

Vector:2: X and Y

Vector:3: X, Y and Z

Vector:4: X, Y, Z and |Vector| (module of the vector, with sign for some tricks)

The default (Vector) is 3 components per vector.

Matrix:3 or Matrix:6: which specify:

Matrix:3: Sxx, Syy and Sxy

Matrix:6: Sxx, Syy, Szz, Sxy, Syz and Sxz

The default (Matrix) is 6 components for matrices.

ComplexVector:4 or ComplexVector:6 which specify

ComplexVector:4: x_real, x_imag, y_real, y_imag

ComplexVector:6: x_real, x_imag, y_real, y_imag, z_real, z_imag

ComplexMatrix:3 or ComplexMatrix:6 which specify

ComplexMatrix:3: Sxx_real, Syy_real, Sxy_real, Sxx_imag, Syy_imag, Sxy_imag

ComplexMatrix:6: Sxx_real, Syy_real, Szz_real, Sxy_real, Syz_real, Sxz_real, Sxx_imag, Syy_imag, Szz_imag, Sxy_imag, Syz_imag, Sxz_imag

Here are some examples:

```
ResultDescription "Displacements" Vector:2
ResultDescription "2D matrix" Matrix:3
ResultDescription "LineDiagramVector" Vector:4
```

and where (optional properties)

- ResultRangesTable "Name of a result ranges table": (optional) is not case-sensitive, and is followed by the name of the previously defined Result Ranges Table which will be used if the Contour Ranges result visualization is chosen (see [Result Range Table](#)); ComponentNames "Name of Component 1", "Name of Component 2": (optional) is not case-sensitive, and is followed by the names of the components of the results which will be used in GiD. The number of Component Names are:

One for a Scalar Result

Three for a Vector Result

Six for a Matrix Result

Four for a PlainDeformationMatrix Result

Six for a MainMatrix Result

Three for a LocalAxes Result

- End with the results values:

Values

location_1 result_1_component_1_value result_1_component_2_value result_1_component_3_value result_2_component_2_value
result_2_component_2_value result_2_component_3_value

...

location_n result_1_component_1_value result_1_component_2_value result_1_component_3_value result_2_component_2_value
result_2_component_2_value result_2_component_3_value

End Values

where

Values: is not case-sensitive, and indicates the beginning of the results values section;

The lines

```
location_1 result_1_component_1_value result_1_component_2_value result_1_component_3_value
result_2_component_2_value result_2_component_2_value result_2_component_3_value
. . .
location_n result_1_component_1_value result_1_component_2_value result_1_component_3_value
result_2_component_2_value result_2_component_2_value result_2_component_3_value
```

are the values of the various results described with ResultDescription for each location. All the results values for the location 'i' should be written in the same line 'i'.

The number of results values are limited thus:

If the Result is located OnNodes, they are limited to the number of nodes defined in ProjectName.post.msh.

If the Result is located OnGaussPoints "My GP", and if the Gauss Points "My GP" are defined for the mesh "My mesh", the limit is the number of gauss points in "My GP" multiplied by the number of elements of the mesh "My mesh".

Holes are allowed. The element nodes with no result defined will not be drawn, i.e. they will appear transparent.

The number of components for each ResultDescription are:

for Scalar results: one component result_number_i scalar_value

for Vector results: three components result_number_i x_value y_value z_value

for Matrix results: six components (3D models)3D: result_number_i Sxx_value Syy_value Szz_value Sxy_value Syz_value Sxz_value

for PlainDeformationMatrix results: four components result_number_i Sxx_value Syy_value Sxy_value Szz_value

for MainMatrix results: twelve components result_number_i Si_value Sii_value Siii_value Vix_value Viy_value Viz_value Viix_value Viiy_value Viiz_value Viii_value Viiy_value Viii_value

for LocalAxes results: three components describing the Euler angles result_number_i euler_ang_1_value euler_ang_2_value euler_ang_3_value

End Values: is not case-sensitive, and indicates the end of the results group values section.

Note: Vectors in a ResultGroup always have three components.

Note: Matrices in a ResultGroup always have six components.

Note: All the results of one node or gauss point should be written on the same line.

Note: For Matrix and PlainDeformationMatrix results, the Si, Sii and Siii components are calculated by GiD, which represents the eigen values & vectors of the matrix results, and which are ordered according to the eigen value.

Nodal ResultGroup example:

```

ResultGroup "Load Analysis" 1 OnNodes
ResultDescription "Ranges test" Scalar
ResultRangesTable "My table"
ResultDescription "Scalar test" Scalar
ResultRangesTable "Pressure"
ResultDescription "Displacements"
Vector
ComponentNames "X-Displ", "Y-Displ" "Z-Displ"
ResultDescription "Nodal Stresses" Matrix
ComponentNames "Sx", "Sy", "Sz", "Sxy", "Syz",
"Sxz"Values
  1 0.0      0.000E+00 0.000E+00  0.000E+00 0.0 0.550E+00  0.972E-01 -0.154E+00 0.0 0.0 0.0
  2 6.4e-01 0.208E-04 0.208E-04 -0.191E-04 0.0 0.506E+00  0.338E-01 -0.105E+00 0.0 0.0 0.0
  3 0.0      0.355E-04 0.355E-04 -0.376E-04 0.0 0.377E+00  0.441E-02 -0.547E-01 0.0 0.0 0.0
...

 115 7.8e-01 0.427E-04 0.427E-04 -0.175E-03 0.0 0.156E-01 -0.158E-01 -0.300E-01 0.0 0.0 0.0
 116 7.4e-01 0.243E-04 0.243E-04 -0.189E-03 0.0 0.216E-02 -0.968E-02 -0.231E-01 0.0 0.0 0.0
End Values

```

Gauss Points ResultGroup example:

```

GaussPoints "My Gauss" ElemType Triangle "2D
Beam"Number Of Gauss Points: 3
Natural Coordinates:
InternalEnd gausspoints

ResultGroup "Load Analysis" 1 OnGaussPoints "My
Gauss"ResultDescription "Gauss test" Scalar
ResultDescription "Vector Gauss" Vector
ResultDescription "Gauss Points Stresses"
PlainDeformationMatrixValues
  1 1.05 1 0 0.0 -19.4607 -1.15932 -1.43171 -6.18601
    2.1 0 1 0.0 -19.4607 -1.15932 -1.43171 -6.18601
    3.15 1 1 0.0 -19.4607 -1.15932 -1.43171 -6.18601
  2 1.2 0 0 0.0 -20.6207 0.596461 5.04752 -6.00727
    2.25 0 0 0.0 -20.6207 0.596461 5.04752 -6.00727
    3.3 2.0855e-05 -1.9174e-05 0.0 -20.6207 0.596461 5.04752 -6.00727
  3 1.35 2.0855e-05 -1.9174e-05 0.0 -16.0982 -1.25991 2.15101 -5.20742
    2.4 2.0855e-05 -1.9174e-05 0.0 -16.0982 -1.25991 2.15101 -5.20742
    3.45 2.0855e-05 -1.9174e-05 0.0 -16.0982 -1.25991 2.15101 -5.20742
  ...
191 29.55 4.2781e-05 -0.00017594 0.0 -0.468376 12.1979 0.610867 3.51885
    30.6 4.2781e-05 -0.00017594 0.0 -0.468376 12.1979 0.610867 3.51885
    31.65 4.2781e-05 -0.00017594 0.0 -0.468376 12.1979 0.610867 3.51885
192 29.7 4.2781e-05 -0.00017594 0.0 0.747727 11.0624 1.13201 3.54303
    30.75 4.2781e-05 -0.00017594 0.0 0.747727 11.0624 1.13201 3.54303
    31.8 2.4357e-05 -0.00018974 0.0 0.747727 11.0624 1.13201 3.54303
End Values

```

Mesh format: ModelName.post.msh

Note: This postprocess mesh format requires GiD version 6.0 or higher.

Comments are allowed and should begin with a '#'. Blank lines are also allowed.

To enter the mesh names and result names in another encoding, just write # encoding your_encoding

for example:

encoding utf-8

• Inside this file one or more MESHes can be defined, each of them should:

Begin with a header that follows this model:

MESH "mesh_name" **dimension** mesh_dimension **Elmtype** element_type **Nnode** element_num_nodes
where

- MESH, dimension, elmtype, nnode: are keywords that are not case-sensitive;
- "mesh_name": is an optional name for the mesh;
- mesh_dimension: is 2 or 3 according to the geometric dimension of the mesh;
- element_type: describes the element type of this MESH. It should be one of the following; Point, Line, Triangle, Quadrilateral, Tetrahedra, Hexahedra, Prism, Pyramid, Sphere, Circle ;
- element_num_nodes: the number of nodes of the element:

Point: 1 node,

Point connectivity:



• Line: 2 or 3 nodes,

Line connectivities:



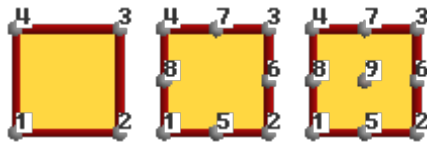
- Triangle: 3 or 6 nodes,

Triangle connectivities:



- Quadrilateral: 4, 8 or 9 nodes,

Quadrilateral connectivities:



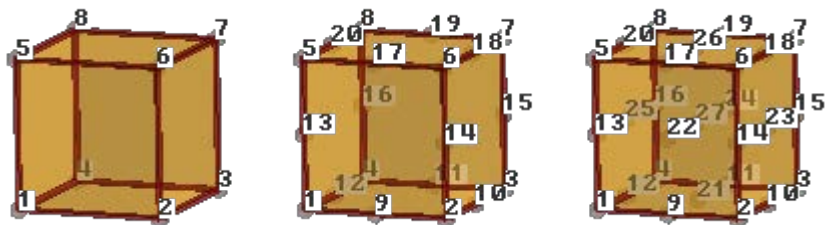
- Tetrahedra, 4 or 10 nodes,

Tetrahedra, connectivities:



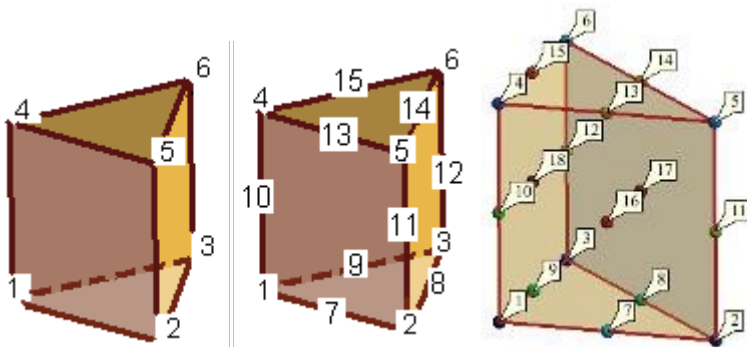
- Hexahedra, 8, 20 or 27 nodes.

Hexahedra, connectivities:



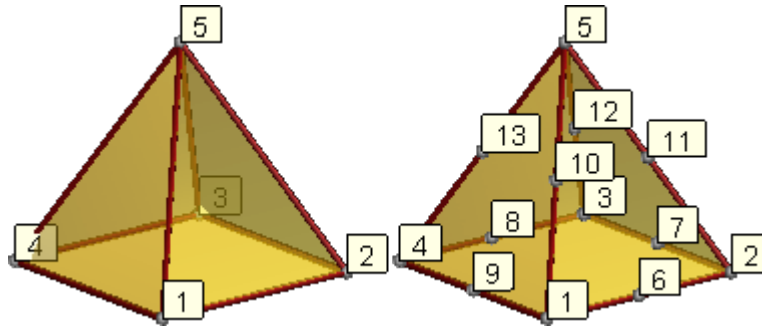
- Prism: 6, 15 or 18 nodes,

Prism connectivities:



- Pyramid: 5 or 13 nodes,

Pyramid connectivities:



- Sphere: 1 node and a radius
- Circle: 1 node, a radius and a normal (x, y, z)

Note: For elements of order greater than linear, the connectivities must be written in hierarchical order, i.e. the vertex nodes first, then the middle ones.

- An optional line declaring the units of the mesh coordinates

Unit "mesh unit"

- An optional line describing its colour with # color R G B A, where R, G, B and A are the Red, Green, Blue and Alpha components of the color written in integer format between 0 and 255, or in floating (real) format between 0.0 and 1.0. (Note that if 1 is found in the line it will be understood as integer, and so 1 above 255, rather than floating, and so 1 above 1.0). Alpha values represent the transparency of the mesh when this visualization options is active, being 0.0 totally opaque and 1.0 totally transparent.

color* 127 127 0

In this way different colours can be specified for several meshes, taking into account that the # color line must be between the MESH line and the Coordinates line.

- Be followed by the coordinates:

coordinates

1 0.0 1.0
3.0 ...1000
-2.5 9.3 21.8

end coordinates

where

- the pair of keywords coordinates and end coordinates are not case-sensitive;
- between these keywords there should be the nodal coordinates of all the MESHes or the current one.

Note: If each MESH specifies its own coordinates, the node number should be unique, for instance, if MESH "mesh one" uses nodes 1..100, and MESH "other mesh" uses 50 nodes, they should be numbered from 101 up.

- Be followed by the elements connectivity

elements

#el_num node_1 node_2 node_3 optional_material_number
1 1 2 3 215

...

1000 32 48 23 215

end elements

where

- the pair of keywords elements and end elements are not case-sensitive;
- between these keywords there should be the nodal connectivities for the my_type elements.

Note: On elements of order greater than linear, the connectivities must be written in hierarchical order, i.e. the vertex nodes first, then the middle ones;

- There is optionally a material number.
- For **Sphere elements** : Element_number Node_number Radius [optional_material_number]
- For **Circle elements** : Element_number Node_number Radius [optional_normal_x optional_normal_y optional_normal_z] [optional_material_number]

If the normal is not written for circles, normal (0.0, 0.0, 1.0) will be used.

Mesh example

This example clarifies the description:

```
#mesh of a table
MESH "board" dimension 3 ElemType Triangle
Nnode 3# color 127 127 0
Coordinates
# node number   coordinate_x coordinate_y coordinate_z
  1             -5             3             -3
  2             -5             3             0
  3             -5             0             0
  4             -2             2             0
  5             -1.66667        3             0
  6             -5             -3            -3
  7             -2             -2             0
  8             0              0             0
  9             -5             -3             0
 10             1.66667         3             0
 11             -1.66667        -3            0
 12             2              2             0
 13             2              -2            0
 14             1.66667         -3            0
 15             5              3            -3
 16             5              3             0
 17             5              0             0
 18             5              -3            -3
 19             5              -3             0
end coordinates

#we put both material in the same
MESH, #but they could be separated
into two MESH

Elements
# element  node_1  node_2  node_3  material_number
  5         19     17     13      3
  6          3      9      7      3
  7          2      3      4      3
  8         17     16     12      3
  9         12     16     10      3
 10         12     10      4      3
 11          7      9     11      3
 12          7     11     13      3
 13          2      4      5      3
 14          5      4     10      3
 15         19     13     14      3
 16         14     13     11      3
 17          3      7      4      3
 18         17     12     13      3
 19         13     12      8      4
 20         13      8      7      4
 21          7      8      4      4
 22          4      8     12      4
end elements

MESH dimension 3 ElemType Line
Nnode 2Coordinates
#no coordinates then they are already in the first
MESHend coordinates

Elements
# element  node_1  node_2
material_number1  9      6
  5
  2         19     18      5
  3         16     15      5
  4          2      1      5
end elements
```

Group of meshes

If the same meshes are used for all the analyses, the following section can be skipped.

A new concept has been introduced in Postprocess: Group, which allows the postprocessing of problems which require re-meshing or adaptive meshes, where the mesh change depending on the time step.

Normal operations, such as animation, displaying results and cuts, can be done over these meshes, and they will be actualized when the selected analysis/step is changed, for example by means of View results -> Default analysis/step

There are two ways to enter in GiD the different meshes defined for different steps or analysis:

1. define separate files for each mesh, for instance:

• binary format: mesh+result_1.post.bin, mesh+result_2.post.bin, mesh+result_3.post.bin, ...
 ascii format: mesh_1.post.msh + mesh_1.post.res, mesh_2.post.msh + mesh_2.post.res, ...

Note: the steps values (or analysis) should be different for each pair mesh+result.

To read them you can use File-->Open Multiple (see POSTPROCESS > Files menu from Reference Manual)

2. define on binary file or two ascii files (msh+res):

Meshes that belong to a group should be defined between the following highlighted commands

```
Group "group name"
MESH "mesh_name" dimension ...
...
end elements
MESH "another_mesh" ...
...
end elements
end group
```

Results which refer to one of the groups should be written between these highlighted commands

```
OnGroup "group name"
Result "result name"
...
end values
...
end ongroup
```

Note: GiD versions 7.7.3b and later only allow one group at a time, i.e. only one group can be defined across several steps of the analysis. Care should be taken so that groups do not overlap inside the same step/analysis.

For instance, an analysis which is 10 steps long:

For steps 1, 2, 3 and 4: an 'environment' mesh of 10000 elements and a 'body' mesh of 10000 elements are used

```
MESH "environment"
... Coordinates
...
10000 ...
end elements
MESH "body" ...
...
20000 ...
end elements
```

and its results

```
GiD Post Results File 1.0
...
Results "result 1" "time" 1.0 ...
...
Results "result 1" "time" 2.0 ...
...
```

```
Results "result 1" "time" 3.0 ...
...
Results "result 1" "time" 4.0 ...
...
end values
```

For steps 5, 6, 7 and 8: with some refinement, the 'environment' mesh now being used has 15000 elements and the 'body' mesh needs 20000 elements

```
MESH "environment"
...
Coordinates
...
15000 ...
end elements
MESH "body" ...
...
35000 ...
end elements
```

and its results are

```
GiD Post Results File 1.0
...
Results "result 1" "time" 5.0 ...
...
Results "result 1" "time" 6.0 ...
...
Results "result 1" "time" 7.0 ...
...
Results "result 1" "time" 8.0 ...
...
end values
```

For steps 9 and 10: the last meshes to be used are of 20000 and 40000 elements, respectively

```
MESH "environment" ...
Coordinates
...
20000 ...
end elements
MESH "body" ...
...
60000 ...
end elements
```

and its results are

```
GiD Post Results File 1.0
...
Results "result 1" "time" 9.0 ...
...
Results "result 1" "time" 10.0 ...
...
end values
```

There are two ways to postprocess this:

- store the information in three pairs (or three binary files), thus:
 - steps_1_2_3_4.post.msh and steps_1_2_3_4.post.msh (or steps_1_2_3_4.post.bin) steps_5_6_7_8.post.msh and steps_5_6_7_8.post.msh (or steps_5_6_7_8.post.bin) steps_9_10.post.msh and steps_9_10.post.msh (or steps_9_10.post.bin)

- and use the 'Open multiple' option (see POSTPROCESS > Files menu from Reference Manual) to selected the six (or three) files;
 - orwrite them in only two files (one in binary) by using the **Group** concept:
- all_analysis.post.msh (note the group - end group pairs)

```

Group "steps 1, 2, 3 and 4"
MESH "environment" ...
...
MESH "body" ...
...
end group
Group "steps 5, 6, 7 and 8"
MESH "environment" ...
...
MESH "body" ...
...
end group
Group "steps 9 and 10"
MESH "environment" ...
...
MESH "body" ...
...
end group

```

and

all_analysis.post.res (note the ongroup - end ongroup pairs)

```

GiD Post Results File 1.0
OnGroup "steps 1, 2, 3 and 4"
...
Results "result 1" "time" 1.0 ...
...
Results "result 1" "time" 2.0 ...
...
Results "result 1" "time" 3.0 ...
...
Results "result 1" "time" 4.0 ...
...
end ongroup
OnGroup "steps 5, 6, 7 and 8"
...
Results "result 1" "time" 5.0 ...
...
Results "result 1" "time" 6.0 ...
...
Results "result 1" "time" 7.0 ...
...
Results "result 1" "time" 8.0 ...
...
end ongroup
OnGroup "steps 9 and 10"
...
Results "result 1" "time" 9.0 ...
...
Results "result 1" "time" 10.0 ...
...
end ongroup

```

and use the normal Open option.

List file format: ModelName.post.lst

New file *.post.lst can be read into GiD, postprocess. This file is automatically read when the user works in a GiD project and changes from pre to postprocess.

This file can also be read with FileOpen

- The file contains a list of the files to be read by the postprocess:

The first line should contain one of these words:

- **Single:** just a single file (two for ascii files: one for mesh and another for results) is provided to be read (see [Open](#));
- **Merge:** several filenames are provided, one filename per line, which are to be merged inside GiD, for instance when the files come from a domain decomposed simulation (see [Merge... \(only Postprocessing\)](#)); Example : Domain partition with a static mesh along time.
- **Multiple:** several filenames are provided, one filename per line, which are read in GiD, for instance if they belong to a DEMsimulation, or FEM simulation with mesh refinement (see [Open multiple... \(only Postprocessing\)](#)); Example : mesh refinement, where each step of the analysis has its own mesh.
- next lines: the mesh and results files to be read, with one filename per line;
- following the postprocess mesh and result files, a list of graphs filenames can be provided so that GiD reads them too; graphs files have the extension **.grf**;
- comment lines begin with the **#** character and blank lines are also admitted; file names may have absolute path names or relative to the list file location;

Example of a list file:

```
Multiple
# postprocess files
cilinder-3D-3-sim2_001.10.post.bin
cilinder-3D-3-sim2_001.100.post.bin
cilinder-3D-3-sim2_001.101.post.bin
cilinder-3D-3-sim2_001.102.post.bin
cilinder-3D-3-sim2_001.11.post.bin
cilinder-3D-3-sim2_001.12.post.bin
cilinder-3D-3-sim2_001.13.post.bin
cilinder-3D-3-sim2_001.14.post.bin
cilinder-3D-3-sim2_001.15.post.bin
cilinder-3D-3-sim2_001.16.post.bin
cilinder-3D-3-sim2_001.17.post.bin
...
cilinder-3D-3-sim2_001.99.post.bin
# graph files
cilinder-3D-3-sim2_001.c2.1.grf
cilinder-3D-3-sim2_001.c2.2.grf
cilinder-3D-3-sim2_001.dem-branch.grf
cilinder-3D-3-sim2_001.dem-contacts.grf
cilinder-3D-3-sim2_001.dem-energy.grf
cilinder-3D-3-sim2_001.dem-stress.grf
```

File names may have absolute path names or relative to the list file location.

Graphs file format: ModelName.post.grf

The graph file that GiD uses is a standard ASCII file.

Every line of the file is a point on the graph with X and Y coordinates separated by a space.

Comment lines are also allowed and should begin with a '#'.
 #

The title of the graph and the labels for the X and Y axes can also be configured with some 'special' comments:

Title: If a comment line contains the Keyword 'Graph:' the string between quotes that follows this keyword will be used as the title of the graph.

Axes labels: The string between quotes that follows the keywords 'X:' and 'Y:' will be used as labels for the X- and Y-axes respectively. The same is true for the Y axis, but with the Keyword 'Y:'.

Axes units:

Example:

```
# Graph: "Stresses"
#
# X: "Szz-Nodal_Streess" Y: "Sxz-Nodal_Stress"
# Units: Pa Pa
-3055.444 1672.365
-2837.013 5892.115
-2371.195 666.9543
-2030.643 3390.457
-1588.883 -4042.649
-1011.5 1236.958
# End
```

The file *.grf (which contains graphs) is read when changing from pre to post process and projectName.gid/projectName.post.grf exists, or the postprocess files are read through File-->Open, then example.msh/res/bin and example.grf are read.

The post list file (.post.lst) can also contain a list of graphs (.grf).

Binary format

The postprocess binary format is strongly based in the ascii format.

Each line of the Mesh and Result headers are stored as a 4-byte integer with the length of the string followed by the 0-terminated string-line.

Mesh coordinates, connectivities and result values are stored as binary data, 4-byte integers and 4-byte floating point values.

Finally the whole binary data file is compressed with z-lib.

You can debug the file following these steps:

1. rename the file for instance to .post.res.gz
2. un-compress it with your favourite compressor
3. you can 'edit' the file with emacs, notepad++ or any other editor
4. you can also look for the 'Result' header with CLI tools like:

```
grep -a Result ./test.post.bin | awk ' BEGIN { RS = "\0" } ; { printf "\n%s\n", $0 }' | grep -a Result | less
```

Output example:

```
Result "PRESSURE" "Kratos" 1 Scalar OnNodes^Y
Result "VELOCITY" "Kratos" 1 Vector OnNodesE
Result "PRESSURE" "Kratos" 2 Scalar OnNodes^Y
Result "VELOCITY" "Kratos" 2 Vector OnNodesE
Result "PRESSURE" "Kratos" 3 Scalar OnNodes^Y
Result "VELOCITY" "Kratos" 3 Vector OnNodesE
...
```

Use the freely available GiDPost library to write post-process data files in ascii, binary or hdf5 format. Switching between these formats only requires a couple of modifications. The source code and some pre-built binaries can be downloaded from www.gidsimulation.com --> GiDPlus --> GidPost.

HDF5 format

In HDF5 the postprocess information is stored in several groups.

You can always look at the structure of GiDPost hdf5 format with the freely available HDFView utility of the HDFGroup and which can be found at <https://support.hdfgroup.org/products/java/>.

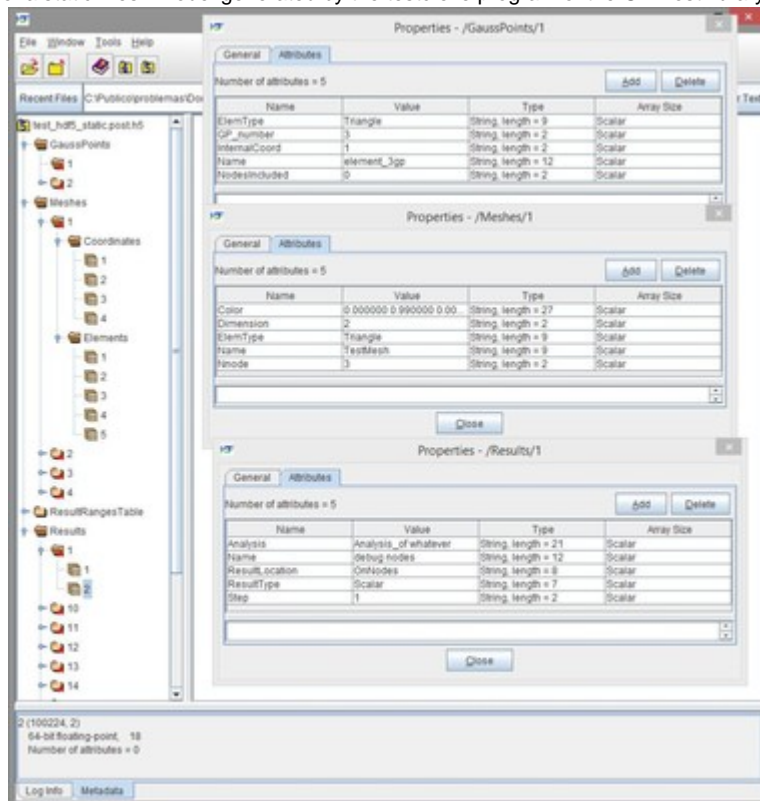
Use the freely available GiDPost library to write post-process data files in ascii, binary or hdf5 format. Switching between these formats only requires a couple of modifications. The source code and some pre-built binaries can be downloaded from www.gidsimulation.com --> GiDPlus --> GiDPost.

HDF5 format for static meshes are supported since version 2.5 and for dynamic meshes since version 2.7.

For a **static mesh model**, i.e. a model whose mesh does not change along all time-steps of all analyses:

- Meshes**: for each mesh of the model a numerated subfolder should be present. The properties of the mesh, like element type, name, etc. (i.e. mesh header in ascii or binary format), are stored as attributes of this numerated folder. Inside each numerated folder, the coordinates and elements connectivities. Numeration of the coordinates are global and it's recommended that the numeration of the elements should be too.
- Results**: for each result of the model a numerated subfolder should be present. The properties of the results, like analysis name, step value, result type, etc. (i.e. result header in ascii or binary format), are stored as attributes of this numerated folder. Inside each numerated folder, the result id's and result values are stored.
- GaussPoints**: for each gauss point referenced by the *Results* a numerated subfolder should be present with its definition. The properties of the gauss point definition, like name, number of integration points inside the element, etc. (i.e. gauss point header in ascii or binary format), are stored as attributes of this numerated folder. Eventually, the numerated folder also contains the natural coordinates of the gauss points if GiD's internal aren't used, i.e. they are defined by the simulation program.
- ResultRangesTable**: list of customized legends for the *Contour Ranges* visualization and referenced by the *Results*. Each ResultRangesTable is stored in separated numerated folders and consist of a list that associate a string or keyword to a range of result values.

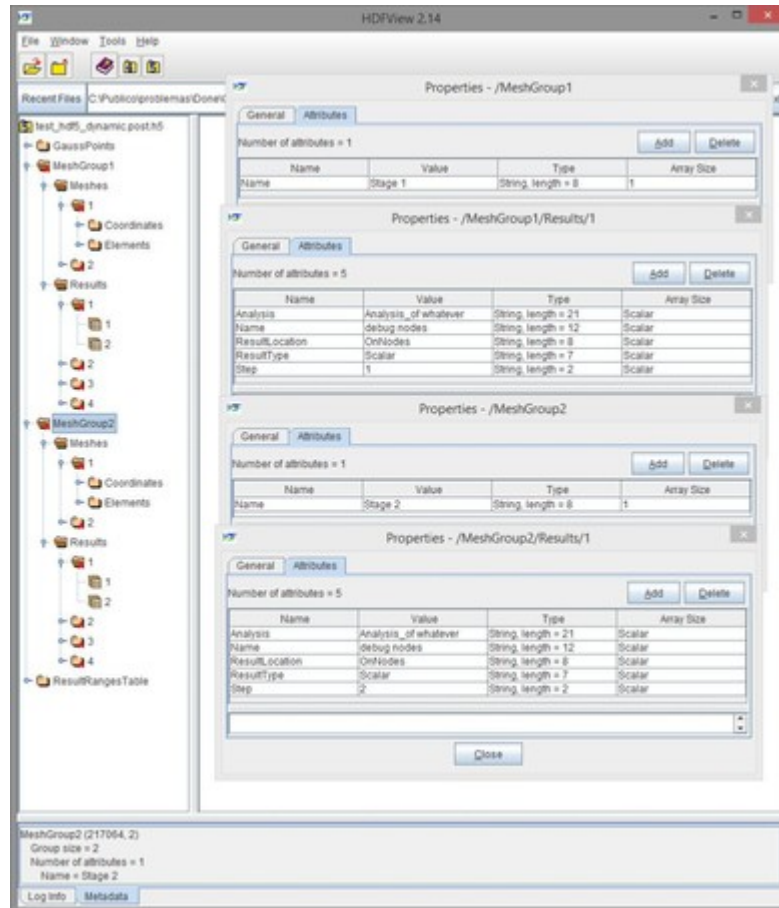
This figure shows an example of a static mesh model generated by the testc.exe program of the GiDPost library:



For a dynamic **mesh model**, i.e. a model whose mesh changes at each time-step, group of time-steps or at each analysis step, a new group should be used:

- MeshGroup**: for each mesh change a new enumerated MeshGroup group should be used. This group contains the Meshes and the Results that belong together in each time-step, group of time-steps or at each analysis step. **Note** that the *Results* on each *MeshGroup* should be defined in different time-steps/analyses so that GiD can show them to the user. Inside each MeshGroup the coordinates i

Following figure shows an example of a dynamic mesh model with two time-steps, each one of them with its own mesh + results definition:



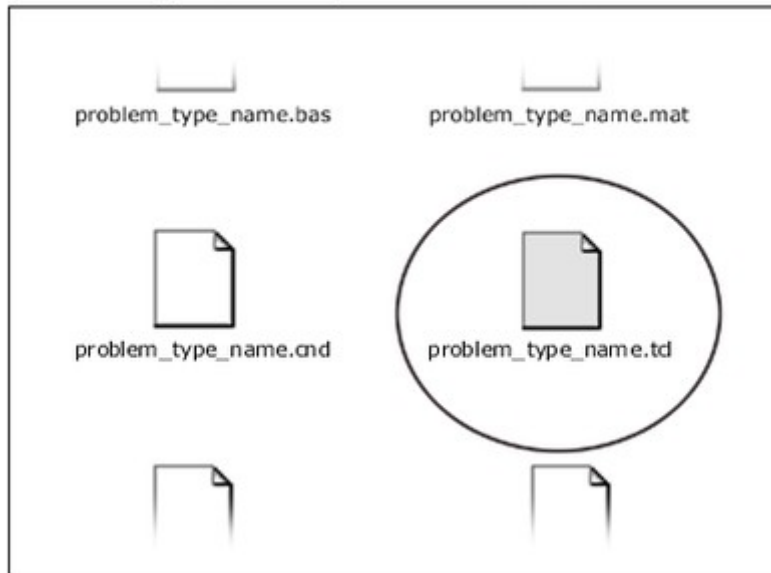
TCL AND TK EXTENSION

This chapter looks at the advanced features of GiD in terms of expandability and total control. Using the Tcl/Tk extension you can create script files to automatize any process created with GiD. With this language new windows and functionalities can be added to the program.

For more information about the Tcl/Tk programming language itself, look at www.scriptics.com <http://www.scriptics.com>.

If you are going to use a Tcl file, it must be located in the Problem Type directory and be called `problem_type_name.tcl`.

Problem Type directory



Event procedures

A 'Event procedure' is a Tcl procedure that is invoked from GiD when doing some actions. It allows developers to interleave its code with the GiD flow.

The structure of `problem_type_name.tcl` can optionally implement some of these Tcl prototype procedures (and other user-defined procedures). The procedures listed below are automatically called by GiD. Their syntax corresponds to standard Tcl/Tk language:

Note: it is possible to get the ordered list of Tcl events with the procedure *GiD_Info events*

Note: default values are `category==GENERAL` and `proprietary==gid`

To be called a Tcl procedure must be previously registered, using

GiD_RegisterEvent <event_name> <procedure> ?<category>? ?<proprietary>?

<event_name> must be a valid event, like `GiD_Event_InitProblemtype`

<procedure> is the name of our procedure, with appropriated prototype of arguments

Can know the expected arguments with the command *GiD_Info events -args <event_name>*, or reading this documentation.

Avoid to use as procedure name the event_name, to avoid conflicts. It is recommended for example use a prefix or namespace that suggest that is defined by our own problemtype, etc.)

<category>: GENERAL, PROBLEMTYPE, PLUGIN (default==GENERAL)

used for a better classification.

<proprietary>: the name of our problemtype, etc. (according with category, default == gid)

GiD defined events are of category=GENERAL and proprietary=gid, these are the default values in case of missing parameters

The category and proprietary allow automatically unregister events when unloading a problemtype or plugin.

GiD_UnRegisterEvent <event_name> <procedure> ?<category>? ?<proprietary>?

To unregister an event, previously registered with *GiD_RegisterEvent*

GiD_UnRegisterEvents ?<category>? ?<proprietary>?

To unregister all events of a category and proprietary

GiD_GetRegisteredEventProcs <event_name> ?<category>? ?<proprietary>?

To get the list of procedures registered for the event, category and proprietary

GiD_GetRegisteredEventProcsAll <event_name>

To get all procedures registered for an event, without take into account the category and proprietary.

GiD_GetIsRegisteredEvent <event_name>

Return 1 if there is some procedure registered for the event, 0 else

GiD_GetIsRegisteredEventProc <event_name> <procedure> ?<category>? ?<proprietary>?

Return 1 if this procedure is registered for the event, 0 else

RaiseEvent_Registered <event_name> <args>

To raise an event from Tcl scripting (providing the appropriated arguments)

Events preprocess**•New***GiD_Event_AfterNewGIDProject***•Read***GiD_Event_BeforeReadGIDProject**GiD_Event_AfterReadGIDProject**GiD_Event_AfterInsertGIDProject**GiD_Event_AfterSetProjectName**GiD_Event_BeforeOpenFile**GiD_Event_AfterOpenFile***•Write***GiD_Event_BeforeSaveGIDProject**GiD_Event_AfterSaveGIDProject**GiD_Event_AfterSaveAsGIDProject**GiD_Event_AfterSaveFile**GiD_Event_AfterSaveImage***•Geometry***GiD_Event_AfterCreatePoint**GiD_Event_AfterCreateLine**GiD_Event_AfterCreateSurface**GiD_Event_AfterCreateVolume**GiD_Event_BeforeDeletePoint**GiD_Event_BeforeDeleteLine**GiD_Event_BeforeDeleteSurface**GiD_Event_BeforeDeleteVolume**GiD_Event_AfterRenumber***•Copy / Move***GiD_Event_BeforeCopy**GiD_Event_AfterCopy**GiD_Event_BeforeMove**GiD_Event_AfterMove***•Mesh***GiD_Event_BeforeMeshGeneration**GiD_Event_AfterMeshGeneration**GiD_Event_BeforeMeshErrors**GiD_Event_BeforeMeshProgress**GiD_Event_MeshProgress**GiD_Event_AfterMeshProgress**GiD_Event_AfterChangeMesh**GiD_Event_AfterRenumber***•Dimensions***GiD_Event_BeforeDeleteDimension**GiD_Event_AfterCreateDimension***•Layers***GiD_Event_AfterCreateLayer**GiD_Event_AfterRenameLayer**GiD_Event_BeforeDeleteLayer**GiD_Event_AfterDeleteLayer**GiD_Event_AfterChangeLayer**GiD_Event_AfterChangeParentLayer**GiD_Event_AfterSetLayerToUse**GiD_Event_AfterSendToLayer**GiD_Event_AfterChangeLayersOrSets***•Groups***GiD_Event_AfterCreateGroup**GiD_Event_AfterRenameGroup**GiD_Event_BeforeDeleteGroup**GiD_Event_AfterDeleteGroup**GiD_Event_AfterChangeGroup**GiD_Event_AfterChangeParentGroup*

New

GiD_Event_AfterNewGIDProject: will be called just after start a new GiD project.

```
proc GiD_Event_AfterNewGIDProject {} {
}
```

Read

GiD_Event_BeforeReadGIDProject: will be called just before read a GiD project. It receives the argument filename, which is the path of the model folder, without the .gid extension.
If it returns -cancel- then the reading is cancelled.

```
proc GiD_Event_BeforeReadGIDProject { filename } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_AfterReadGIDProject: will be called just after read a GiD project. It receives the argument filename, which is the path of the model folder, without the .gid extension.

```
proc GiD_Event_AfterReadGIDProject { filename } {
}
```

GiD_Event_AfterInsertGIDProject: will be called just after insert a GiD project into the current one. It receives the argument filename, which is the path of the model folder, without the .gid extension.

```
proc GiD_Event_AfterInsertGIDProject { filename } {
}
```

GiD_Event_AfterSetProjectName:

```
proc GiD_Event_AfterSetProjectName { name } {
}
```

GiD_Event_BeforeOpenFile:

```
proc GiD_Event_BeforeOpenFile { filename format } {
  ...body...
  set value ...
  return $value
}
```

filename: the full name of the file to be read;
format: ACIS_FORMAT, CGNS_FORMAT, DXF_FORMAT, GID_BATCH_FORMAT, GID_GEOMETRY_FORMAT, GID_MESH_FORMAT, IGES_FORMAT, NASTRAN_FORMAT, PARASOLID_FORMAT, RHINO_FORMAT, SHAPEFILE_FORMAT, STL_FORMAT, VDA_FORMAT, VRML_FORMAT or 3DSTUDIO_FORMAT.

If it returns -cancel- then the reading is cancelled.

GiD_Event_AfterOpenFile: will be called after a geometry or mesh file is read inside GiD. It receives as arguments:

filename: the full name of the file that has been read;

- format: ACIS_FORMAT, CGNS_FORMAT, DXF_FORMAT, GID_BATCH_FORMAT, GID_GEOMETRY_FORMAT, GID_MESH_FORMAT, IGES_FORMAT, NASTRAN_FORMAT, PARASOLID_FORMAT, RHINO_FORMAT, SHAPEFILE_FORMAT, STL_FORMAT, VDA_FORMAT, VRML_FORMAT or 3DSTUDIO_FORMAT.
- error: boolean 0 or 1 to indicate an error when reading.

```
proc GiD_Event_AfterOpenFile { filename format error } {
}
```

Write

GiD_Event_BeforeSaveGIDProject / **GiD_Event_AfterSaveGIDProject** will be called just before save a GiD project. It receives the argument `modelname` which is the path of the model folder, without the `.gid` extension. If **GiD_Event_BeforeSaveGIDProject** returns `-cancel-` then the writing is cancelled.

```
proc GiD_Event_BeforeSaveGIDProject { modelname } {
    ... body ...
    set value ...
    return $value
}

proc GiD_Event_AfterSaveGIDProject { modelname } {
}
```

GiD_Event_AfterSaveAsGIDProject: will be called after GiD save its information in the new `_filename` location, when this folder and the model files exists, and provide the `old_filename` argument for example to allow to copy extra data, like result files handled by the problemtype.

```
proc GiD_Event_AfterSaveAsGIDProject { old_modelname new_modelname } {
}
```

•**GiD_Event_AfterSaveFile**: will be called after a geometry or mesh file is exported to a file. It receives as

arguments: `filename`: the full name of the file that has been written;
`format`: ACIS_FORMAT, DXF_FORMAT, GID_GEOMETRY_FORMAT, GID_MESH_FORMAT, IGES_FORMAT, RHINO_FORMAT, AMELET_FORMAT, KML_FORMAT.
`error`: boolean 0 or 1 to indicate an error when writing.

```
proc GiD_Event_AfterSaveFile { filename format error } {
}
```

GiD_Event_AfterSaveImage: will be called after a picture or model is saved to disk. It receives as arguments:

`filename`: the full name of the file that has been saved;
`format`: eps, ps, tif, bmp, ppm, gif, png, jpg, tga, wrl

```
proc GiD_Event_AfterSaveImage { filename format } {
}
```

Geometry

GiD_Event_AfterCreatePoint/Line/Surface/Volume: will be called just after create the entity, providing its number

```
proc GiD_Event_AfterCreatePoint { num } {
}

proc GiD_Event_AfterCreateLine { num } {
}

proc GiD_Event_AfterCreateSurface { num } {
}

proc GiD_Event_AfterCreateVolume { num } {
}
```

GiD_Event_BeforeDeletePoint/Line/Surface/Volume: will be called just before delete the entity, providing its number

```
proc GiD_Event_BeforeDeletePoint { num } {
}

proc GiD_Event_BeforeDeleteLine { num } {
}

proc GiD_Event_BeforeDeleteSurface { num } {
}

proc GiD_Event_BeforeDeleteVolume { num } {
}
```

GiD_Event_AfterRenumber: will be called after renumber the geometry or the mesh (to update for example fields storing entity identifiers)

- useof : could be GEOMETRYUSE or MESHUSE
- leveltype: the kind of entity that was renumbered.

Geometry: must be ALL_LT.

- Mesh: could be NODE_LT or

ELEM_LT.renumeration:

Geometry: four sublists with the old and new identifiers for points, lines, surfaces and volumes.

Mesh: a sublist with the old and new identifiers for nodes or elements.

```
proc GiD_Event_AfterRenumber { useof leveltype renumeration } {
}
```

GiD_Event_AfterRepair: will be called after repair (the geometry and mesh), to do extra tasks at scripting level
It must return a list of the items: num_repaired and message

- num_repaired: integer, the number of repaired things
- message: a translated message to be shown to the user after the repair

```
proc GiD_Event_AfterRepair { } {
    return [list $num_repaired $message]
}
```


Copy / Move

GiD_Event_BeforeCopy/Move GiD_Event_AfterCopy/Move: will be called just before or after use copy or move ools.

useof : could be GEOMETRYUSE or MESHUSE

transformation : could be ROTATION, TRANSLATION, MIRROR, SCALE, OFFSET, SWEEP or ALIGN

```
proc GiD_Event_BeforeCopy { useof transformation error } {  
}  
  
proc GiD_Event_AfterCopy { useof transformation error } {  
}  
  
proc GiD_Event_BeforeMove { useof transformation error } {  
}  
  
proc GiD_Event_AfterMove { useof transformation error } {  
}
```

Mesh

GiD_Event_BeforeMeshGeneration: will be called before the mesh generation. It receives the mesh size desired by the user as the `element_size` argument. This event can typically be used to assign some condition automatically. If it returns `-cancel-` the mesh generation is cancelled.

```
proc GiD_Event_BeforeMeshGeneration { element_size } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_AfterMeshGeneration: will be called after the mesh generation. It receives as its `fail` argument a true value if the mesh is not created.

```
proc GiD_Event_AfterMeshGeneration { fail } {
}
```

GiD_Event_BeforeMeshErrors: `filename` is the full path to the file that has information about the meshing errors. Returning `-cancel-` the standard 'Mesh error window' won't be opened

```
proc GiD_Event_BeforeMeshErrors { filename } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_BeforeMeshProgress: to start some progressbar, provide an approximated information of the amount of entities to be meshed

```
proc GiD_Event_BeforeMeshProgress { num_other num_lines num_surfaces num_volumes } {
}
```

GiD_Event_MeshProgress: to update some progressbar, provide approximated percents of the progress

```
proc GiD_Event_MeshProgress { percent_total percent_other percent_lines percent_surfaces
percent_volumes num_nodes num_elements } {
}
```

GiD_Event_AfterMeshProgress: to end some progressbar

```
proc GiD_Event_AfterMeshProgress { } {
}
```

GiD_Event_AfterChangeMesh:

```
proc GiD_Event_AfterChangeMesh { num_nodes num_elements } {
}
```

useof : could be GEOMETRYUSE or MESHUSE

leveltype: the kind of entity that was renumbered.

Geometry: must be ALL_LT.

Mesh: could be NODE_LT or ELEM_LT.

renumeration:

Geometry: four sublists with the old and new identifiers for points, lines, surfaces and volumes.

Mesh: a sublist with the old and new identifiers for nodes or elements.

```
proc GiD_Event_AfterRenumber { useof leveltype renumeration } {  
}
```

Dimensions

GiD_Event_BeforeDeleteDimension: will be called just before delete the entity, providing its number

```
proc GiD_Event_BeforeDeleteDimension { num } {  
}
```

GiD_Event_AfterCreateDimension: will be called just after create the entity, providing its number

```
proc GiD_Event_AfterCreateDimension { num } {  
}
```

Layers

GiD_Event_AfterCreateLayer: will be called just after create the layer 'name'

```
proc GiD_Event_AfterCreateLayer { name } {
}
```

GiD_Event_AfterRenameLayer: will be called just after the layer 'oldname' has been renamed to 'newname'

```
proc GiD_Event_AfterRenameLayer { oldname newname } {
}
```

GiD_Event_BeforeDeleteLayer / GiD_Event_AfterDeleteLayer: will be called just before /after delete the layer 'name'
If GiD_Event_BeforeDeleteLayer returns -cancel- the layer deletion is cancelled.

```
proc GiD_Event_BeforeDeleteLayer { name } {
    ...body...
    set value ...
    return $value
}
```

GiD_Event_AfterChangeLayer: will be called just after change some property of the layer 'name'
property' could be ON, OFF, FROZEN, UNFROZEN, ALPHA <AAA>, COLOR <RRRGGBBB?AAA?>
with RRR, GGG, BBB, AAA from 0 to 255

```
proc GiD_Event_AfterChangeLayer { name property } {
}
```

GiD_Event_AfterChangeParentLayer: will be called when moving a layer to another parent of the tree.

```
proc GiD_Event_AfterChangeParentLayer { oldname newname } {
}
```

GiD_Event_AfterSetLayerToUse: will be called when setting 'name' as current layer to use

```
proc GiD_Event_AfterSetLayerToUse { name } {
}
```

GiD_Event_AfterSendToLayer: will be called when changing entities to the layer 'name'

```
proc GiD_Event_AfterSendToLayer { name } {
}
```

GiD_Event_AfterChangeLayersOrSets:

```
proc GiD_Event_AfterChangeLayersOrSets { num_sets num_off num_back num_transparent } {
}
```

Groups

GiD_Event_AfterCreateGroup: similar to layer commands

```
proc GiD_Event_AfterCreateGroup { name } {
}
```

GiD_Event_AfterRenameGroup:

```
proc GiD_Event_AfterRenameGroup { oldname newname } {
}
```

GiD_Event_BeforeDeleteGroup: will be called just before delete the group 'name'
If it returns -cancel- the deletion is cancelled.

```
proc GiD_Event_BeforeDeleteGroup { name } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_AfterDeleteGroup:

```
proc GiD_Event_AfterDeleteGroup { name } {
}
```

GiD_Event_AfterChangeGroup: property could be color, visible, state, allowed_types

```
proc GiD_Event_AfterChangeGroup { name property } {
}
```

GiD_Event_AfterGroupParentGroup: will be called when moving a group to another parent of the tree.

```
proc GiD_Event_AfterChangeParentGroup { oldname newname } {
}
```

Events problemtype**•Start / End**

GiD_Event_InitProblemtype
GiD_Event_BeforeInitProblemtype
GiD_Event_EndProblemtype
GiD_Event_AfterChangeProblemtype
GiD_Event_AfterSetProblemtypeName

•Read / Write

GiD_Event_LoadModelSPD
GiD_Event_LoadProblemtypeSPD
GiD_Event_SaveModelSPD

•Transform

GiD_Event_BeforeTransformProblemType
GiD_Event_AfterTransformProblemType

•Materials

GiD_Event_AfterCreateMaterial
GiD_Event_AfterRenameMaterial
GiD_Event_BeforeDeleteMaterial
GiD_Event_AfterChangeMaterial
GiD_Event_AfterAssignMaterial

•Conditions

GiD_Event_AfterCreateCondition
GiD_Event_BeforeDeleteCondition
GiD_Event_AfterChangeCondition

•Intervals

GiD_Event_AfterCreateInterval
GiD_Event_BeforeDeleteInterval
GiD_Event_AfterDeleteInterval:
GiD_Event_AfterSetIntervalToUse

•Units

GiD_Event_AfterChangeModelUnitSystem

•Calculation file

GiD_Event_BeforeWriteCalculationFile
GiD_Event_AfterWriteCalculationFile
GiD_Event_SelectOutputFileNames

•Run

GiD_Event_SelectGIDBatFile
GiD_Event_BeforeRunCalculation
GiD_Event_AfterRunCalculation
 Event procedures>problemtype>Start / End

Start / End

GiD_Event_InitProblemtype / GiD_Event_BeforeInitProblemtype: will be called when the problem type is selected. It receives the `dir` argument, which is the absolute path to the `problem_type_name.gid` directory, which can be useful inside the routine to locate some alternative files.

Note: *InitGIDProject* is a deprecated alias of *GiD_Event_InitProblemtype*

```
proc GiD_Event_InitProblemtype { dir } {
}
proc GiD_Event_BeforeInitProblemtype { dir } {
}
```

GiD_Event_EndProblemtype: will be called when this project is about to be closed. It has no arguments.

Note: *EndGIDProject* is a deprecated alias of *GiD_Event_EndProblemtype*

```
proc GiD_Event_EndProblemtype {} {
}
```

GiD_Event_AfterChangeProblemtype

```
proc GiD_Event_AfterChangeProblemtype { oldproblemtype newproblemtype } {
}
```

GiD_Event_AfterSetProblemtypeName

```
proc GiD_Event_AfterSetProblemtypeName { name } {
}
```


Read / Write

GiD_Event_LoadModelSPD: will be called when a GiD project is loaded. It receives the argument `filesdpd`, which is the path of the file which is being opened, but with the extension `.spd` (specific problemtype data). This path is typically the file of the model where the problemtype store its own data.

Note: `GiD_Event_AfterLoadGIDProject` Will be called when a GiD project is loaded, but not when a problem type is loaded, then could be used instead of `GiD_Event_LoadModelSPD` as an opportunity to load the problemtype data of the model.

```
proc GiD_Event_LoadModelSPD { filesdpd } {
}
```

GiD_Event_LoadProblemtypeSPD: will be called when a problem type is loaded. It receives the argument `filesdpd`, which is the path of the file which is being opened, but with the extension `.spd` (specific problemtype data). This path is typically the file of the problemtype where the problemtype define its own data.

Note: `LoadGIDProject` is a deprecated confusing event, that is called in both cases: `GiD_Event_LoadModelSPD` and `GiD_Event_LoadProblemtypeSPD`

```
proc GiD_Event_LoadProblemtypeSPD { filesdpd } {
}
```

GiD_Event_SaveModelSPD: will be called when the currently open file is saved to disk. It receives the argument `filesdpd`, which is the path of the file being saved, but with the extension `.spd` (specific problemtype data). This path can be useful if you want to write specific information about the problem type in a new file.

Note: `SaveGIDProject` is a deprecated event alias of `GiD_Event_SaveModelSPD`

```
proc GiD_Event_SaveModelSPD { filesdpd } {
}
```

Transform

GiD_Event_BeforeTransformProblemType: will be called just before transforming a model from a problem type to a new problem type version.

If it returns -cancel- as a value then the transformation will not be invoked.

file: the name of the model to be transformed;

oldproblemtype: the name of the previous problem type;

newproblemtype: the name of the problem type to be transformed.

```
proc GiD_Event_BeforeTransformProblemType { file oldproblemtype newproblemtype } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_AfterTransformProblemType: will be called just after transforming a model from a problem type to a new problem type version.

It must return a list of the items: value and messages

value 1 if there were model changes done in this procedure, 0 else.

If it returns -cancel- as a special value, then the transformation messages won't be shown.

file: the name of the model to be transformed;

oldproblemtype: the name of the previous problem type;

newproblemtype: the name of the problem type to be transformed.

messages: explains the transforming operations done.

```
proc GiD_Event_AfterTransformProblemType { file oldproblemtype newproblemtype messages } {
  ...body...
  set value ...
  return [list $value $messages]
}
```

Materials

GiD_Event_AfterCreateMaterial: will be called just after create the material 'name'

```
proc GiD_Event_AfterCreateMaterial { name } {
}
```

GiD_Event_AfterRenameMaterial: will be called just after the material 'oldname' has been renamed to 'newname'

```
proc GiD_Event_AfterRenameMaterial { oldname newname } {
}
```

GiD_Event_BeforeDeleteMaterial: will be called just before delete the material 'name'
If it returns -cancel- the material deletion is cancelled.

```
proc GiD_Event_BeforeDeleteMaterial { name } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_AfterChangeMaterial: will be called just after change some field value of the material 'name'.
changedfields is a list with the index of the changed fields (index starting from 1)

```
proc GiD_Event_AfterChangeMaterial { name changedfields } {
}
```

GiD_Event_AfterAssignMaterial: will be called just after assign or unassign the material of some entities.

name is the name of the new material. If it is "" then the material has been unassigned.

leveltype: is the kind of entity, it could be:

For geometry: POINT_LT, LINE_LT, SURFACE_LT, VOLUME_LT

For mesh: ELEM_LT.

```
proc GiD_Event_AfterAssignMaterial { name leveltype } {
}
```

Conditions

GiD_Event_AfterCreateCondition: will be called just after create the condition 'name'

```
proc GiD_Event_AfterCreateCondition { name } {  
}
```

GiD_Event_BeforeDeleteCondition: will be called just before delete the condition 'name'
If it returns -cancel- the material condition is cancelled.

```
proc GiD_Event_BeforeDeleteCondition { name } {  
  ...body...  
  set value ...  
  return $value  
}
```

GiD_Event_AfterChangeCondition: will be called just after change some field value of the condition 'name'.
changedfields is a list with the index of the changed fields (index starting from 1)

```
proc GiD_Event_AfterChangeCondition { name changedfields } {  
}
```

Intervals

GiD_Event_AfterCreateInterval: will be called just after a new interval is created, providing its integer id

```
proc GiD_Event_AfterCreateInterval { interval_id } {
}
```

GiD_Event_BeforeDeleteInterval: will be called just before a interval is deleted, providing its integer id
If it returns -cancel- the interval deletion is cancelled.

```
proc GiD_Event_BeforeDeleteInterval { interval_id } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_AfterDeleteInterval: will be called just after a interval is deleted, providing its integer id

```
proc GiD_Event_AfterDeleteInterval { interval_id } {
}
```

GiD_Event_AfterSetIntervalToUse: will be called just after a the current interval is changed, providing its integer id

```
proc GiD_Event_AfterSetIntervalToUse { interval_id } {
}
```

Units

GiD_Event_AfterChangeModelUnitSystem: will be raised when user change the current unit system or the declared model length units. `old_system` and `new_system` are the names of the units systems before and after the change respectively. They could be empty "" or the same (e.g. changing the model length units)

```
proc GiD_Event_AfterChangeModelUnitSystem { old_unit_system new_unit_system } {  
}
```

Calculation file

GiD_Event_BeforeWriteCalculationFile: will be called just before writing the calculation file. It is useful for validating some parameters.
 If it returns -cancel- as a value then nothing will be written.
 file: the name of the output calculation file.

```
proc GiD_Event_BeforeWriteCalculationFile { file } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_AfterWriteCalculationFile: will be called just after writing the calculation file and before the calculation process. It is useful for renaming files, or cancelling the analysis.

If it returns -cancel- as a value then the calculation is not invoked.

file: the name of the output calculation file error: an error code if there is some problem writing the output calculation file.

```
proc GiD_Event_AfterWriteCalculationFile { file error } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_SelectOutputFileNames: allow to select a custom list of output files to be shown by "Calculate->View process info" (in case that we want to not use the list of 'OutFiles' declared in the bat file).

```
proc GiD_Event_SelectOutputFileNames { filenames } {
  ...
  return $new_filenames
}
```

Run

GiD_Event_SelectGIDBatFile: must be used to switch the default batch file for special cases.

This procedure must return as a value the alternative pathname of the batch file. For example it is used as a trick to select a different analysis from a list of batch calculation files.

```
proc GiD_Event_SelectGIDBatFile { dir basename } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_BeforeCalculate: will be called a little earlier than `GiD_Event_BeforeRunCalculation`, e.g. to allow renumber the mesh before write the calculation file and calculate

remote is 0 in case of local calculation, 1 in case or remote (sending to procserverd)

```
proc GiD_Event_BeforeCalculate { remote } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_BeforeRunCalculation: will be called before running the analysis. It receives several arguments:

- batfilename: the name of the batch file to be
- runbasename: the short name model
- dir: the full path to the model directory
- problemtypedir: the full path to the Problem Types directory
- gidexe: the full path to gid
- args: an optional list with other arguments

If it returns -cancel- then the calculation is not started.

```
proc GiD_Event_BeforeRunCalculation { batfilename basename dir problemtypedir gidexe args } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_AfterRunCalculation: will be called just after the analysis finishes.

If it returns -cancel-as a value then the window that inform about the finished process will not be opened.

It receives as arguments:

- basename: the short name model;
- dir: the full path to the model directory;
- problemtypedir: the full path to the Problem Types directory;
- where: must be local or remote (remote if it was run in a server machine, using ProcServer);
- error: returns 1 if an calculation error was detected;
- errorfilename: an error filename with some error explanation, or nothing if everything was ok.

```
proc GiD_Event_AfterRunCalculation { basename dir problemtypedir where error errorfilename } {
  ...body...
  set value ...
  return $value
}
```


Events postprocess**•Start/End**

GiD_Event_BeforeInitGIDPostProcess
GiD_Event_InitGIDPostProcess
GiD_Event_AfterSetPostModelName
GiD_Event_EndGIDPostProcess

•GraphsSet

GiD_Event_AfterCreateGraphSet
GiD_Event_BeforeDeleteGraphSet
GiD_Event_AfterDeleteGraphSet
GiD_Event_AfterChangeGraphSet
GiD_Event_AfterRenameGraphSet
GiD_Event_AfterSetGraphSetToUse

•Graphs

GiD_Event_AfterCreateGraph
GiD_Event_BeforeDeleteGraph
GiD_Event_AfterDeleteGraph
GiD_Event_AfterChangeGraph
GiD_Event_AfterRenameGraph

•Sets

GiD_Event_AfterCreateSurfaceSet
GiD_Event_AfterCreateVolumeSet
GiD_Event_AfterRenameSurfaceSet
GiD_Event_AfterRenameVolumeSet
GiD_Event_BeforeDeleteSurfaceSet
GiD_Event_BeforeDeleteVolumeSet
GiD_Event_AfterChangeLayersOrSets

•Cuts

GiD_Event_AfterCreateCutSet
GiD_Event_AfterRenameCutSet
GiD_Event_BeforeDeleteCutSet

•Results

GiD_Event_AfterLoadResults
GiD_Event_BeforeResultReadErrors
GiD_Event_AfterSetAnalysis
GiD_Event_AfterSetTimeStep
GiD_Event_AfterSetResult
GiD_Event_AfterSetResultComponent

Start/End

GiD_Event_BeforeInitGIDPostProcess: will be called just before changing from pre to postprocess, and before read any postprocess file (this event can be used for example to check the results file existence and/or rename files). It has no arguments. If it returns -cancel- as a value then the swapping to postprocess mode will be cancelled.

```
proc GiD_Event_BeforeInitGIDPostProcess {} {
}
```

GiD_Event_InitGIDPostProcess: will be called when postprocessing starts. It has no arguments.

```
proc GiD_Event_InitGIDPostProcess {} {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_AfterSetPostModelName

```
proc GiD_Event_AfterSetPostModelName { name } {
}
```

GiD_Event_EndGIDPostProcess: will be called when you leave Postprocess and open Preprocess. It has no arguments.

```
proc GiD_Event_EndGIDPostProcess {} {
}
```

GraphsSet

GiD_Event_AfterCreateGraphSet: will be called when a new graphset is created

```
proc GiD_Event_AfterCreateGraphSet { name } {
}
```

GiD_Event_BeforeDeleteGraphSet: will be called just before delete the graphset 'name'
If it returns -cancel- the deletion is cancelled.

```
proc GiD_Event_BeforeDeleteGraphSet { name } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_AfterDeleteGraphSet:

```
proc GiD_Event_AfterDeleteGraphSet { name } {
}
```

GiD_Event_AfterChangeGraphSet: property could be: legend_location, title_visible

```
proc GiD_Event_AfterChangeGraphSet { name property } {
}
```

GiD_Event_AfterRenameGraphSet:

```
proc GiD_Event_AfterRenameGraphSet { oldname newname } {
}
```

GiD_Event_AfterSetGraphSetToUse: will be called when setting 'name' as current graphset to use

```
proc GiD_Event_AfterSetGraphSetToUse { name } {
}
```

Graphs

GiD_Event_AfterCreateGraph: will be called when a new graph is created

```
proc GiD_Event_AfterCreateGraph { name } {
}
```

GiD_Event_BeforeDeleteGraph: will be called just before delete the graph 'name'
If it returns -cancel- the deletion is cancelled.

```
proc GiD_Event_BeforeDeleteGraph { name } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_AfterDeleteGraph:

```
proc GiD_Event_AfterDeleteGraph { name } {
}
```

GiD_Event_AfterChangeGraph: property could be: color, contour_fill, line_pattern, line_width, pattern_factor, point_size, style, values, visible

```
proc GiD_Event_AfterChangeGraph { name property } {
}
```

GiD_Event_AfterRenameGraph:

```
proc GiD_Event_AfterRenameGraph { oldname newname } {
}
```

Sets

GiD_Event_AfterCreateSurfaceSet, GiD_Event_AfterCreateVolumeSet: will be called just after a postprocess set of volumes or surfaces is created, providing its name

```
proc GiD_Event_AfterCreateSurfaceSet { name } {
}

proc GiD_Event_AfterCreateVolumeSet { name } {
}
```

GiD_Event_AfterRenameSurfaceSet, GiD_Event_AfterRenameVolumeSet: will be called just after a postprocess set of volumes or surfaces has been renamed providing its old and current name

```
proc GiD_Event_AfterRenameSurfaceSet { oldname newname } {
}

proc GiD_Event_AfterRenameVolumeSet { oldname newname } {
}
```

GiD_Event_BeforeDeleteSurfaceSet, GiD_Event_BeforeDeleteVolumeSet: will be called just before a postprocess set of volumes or surfaces will be deleted created, providing its name.
If the procedure return -cancel- then the set won't be deleted

```
proc GiD_Event_BeforeDeleteSurfaceSet { name } {
    #value -cancel- to avoid deletion
    return $value
}

proc GiD_Event_BeforeDeleteVolumeSet { name } {
    #value -cancel- to avoid deletion
    return $value
}
```

GiD_Event_AfterChangeLayersOrSets

```
proc GiD_Event_AfterChangeLayersOrSets { num_sets num_off num_back num_transparent } {
}
```

Cuts

GiD_Event_AfterCreateCutSet: will be called just after a postprocess cut is created, providing its name

```
proc GiD_Event_AfterCreateCutSet { name } {
}
```

GiD_Event_AfterRenameCutSet: will be called just after a postprocess cut has been renamed providing its old and current name

```
proc GiD_Event_AfterRenameCutSet { oldname newname } {
}
```

GiD_Event_BeforeDeleteCutSet: will be called just before a postprocess cut will be deleted created, providing its name.
If the procedure return -cancel- then the cut won't be deleted

```
proc GiD_Event_BeforeDeleteCutSet { name } {
    #value -cancel- to avoid deletion
    return $value
}
```

Results

GiD_Event_AfterLoadResults: will be called when a results file is opened in GiD Postprocess. It receives one argument, the name of the file being opened without its extension.

Note: LoadResultsGIDPostProcess is a deprecated alias of GiD_Event_AfterLoadResults

```
proc GiD_Event_AfterLoadResults { file } {
}
```

GiD_Event_BeforeResultReadErrors: filename is the results file that was read, msg is the error message, format provide information about the kind of file: can be "GID_RESULTS_FORMAT", "3DSTUDIO_FORMAT", "TECPLOT_FORMAT", "FEMAP_FORMAT", "XYZ_FORMAT"
Retuning -cancel- the standard 'Read results error window' won't be opened

```
proc GiD_Event_BeforeResultReadErrors { filename msg format } {
  ...body...
  set value ...
  return $value
}
```

GiD_Event_AfterSetAnalysis: will be called just after set the current analysis

```
proc GiD_Event_AfterSetAnalysis { analysis_name } {
}
```

GiD_Event_AfterSetTimeStep: will be called just after set the current time step.
Step_index is an integer index, starting from 0. (a value of -1 mean any time step)

Note: it is possible to get the value of the step with something like this:
lindex [GiD_Info postprocess get all_steps \$analysis_name] \$step_index

```
proc GiD_Event_AfterSetTimeStep { analysis_name step_index } {
}
```

GiD_Event_AfterSetResult: will be called just after set the current result.

```
proc GiD_Event_AfterSetResult { analysis_name step_index result_name } {
}
```

GiD_Event_AfterSetResultComponent: will be called just after set the current result component.

```
proc GiD_Event_AfterSetResultComponent { analysis_name step_index result_name component_name } {
}
```

Events general**•GUI**

GiD_Event_AfterChangeBackground
GiD_Event_ChangedLanguage
GiD_Event_ChangeMainWindowTitle
GiD_Event_BeforeUpdateWindow
GiD_Event_BeforeSetCursor
GiD_Event_BeforeSetWarnLine
GiD_Event_GetRecommendedMeshSize
GiD_Event_MessageBoxModeless
GiD_Event_MessageBoxOk
GiD_Event_MessageBoxEntry
GiD_Event_MessageBoxCombobox
GiD_Event_MessageBoxGetGeneralMeshSize
GiD_Event_MessageBoxOptionsButtons
GiD_Event_MessageBoxOptionsButtonsModeless
GiD_Event_MessageBoxGetNormal
GiD_Event_MessageBoxGetFilename
GiD_Event_GetRecommendedMeshSize

•View

GiD_Event_AfterChangeViewMode
GiD_Event_TclCalcModelBoundaries
GiD_Event_AfterDrawModel
GiD_Event_AfterRedraw
GiD_Event_AfterChangeRenderMode

•Preferences

GiD_Event_AfterReadPreferences
GiD_Event_AfterSavePreferences

•Licence

GiD_Event_AfterChangeLicenceStatus

•Login

GiD_Event_AfterLogin
GiD_Event_AfterLogout
GiD_Event_AfterStartSession
GiD_Event_AfterStopSession

•DataManager

GiD_Event_AfterDataManagerConnect
GiD_Event_AfterDataManagerDisconnect

GiD_Event_AfterDataManagerSetCloudFolder

•**Other**

GiD_Event_AfterProcess

GiD_Event_AfterEndCommand

GiD_Event_BeforeSetVariable

GiD_Event_AfterSetVariable

GiD_Event_AfterSaveKernelDefaults

GiD_Event_BeforeQuit

GUI

GiD_Event_AfterCreateTopMenus: will be called just after creating the top menus.

```
proc GiD_Event_AfterCreateTopMenus { } {
}
```

GiD_Event_AfterChangeBackground: will be called just after change some background property, like color, direction or image.

```
proc GiD_Event_AfterChangeBackground { } {
}
```

GiD_Event_ChangedLanguage: will be called when you change the current language. The argument is the new language (en, es, ...). It is used, for example, to update problem type customized menus, etc.

```
proc GiD_Event_ChangedLanguage { language } {
}
```

GiD_Event_ChangeMainWindowTitle: will be called when GiD changes the title of the main graphical window. Two arguments: project_name and problemtype_name.

```
proc GiD_Event_ChangeMainWindowTitle { project_name problemtype_name } {
}
```

GiD_Event_BeforeUpdateWindow

```
proc GiD_Event_BeforeUpdateWindow { window } {
}
```

GiD_Event_BeforeSetCursor

name is the cursor name that will be set, and toggle is the widget name

```
proc GiD_Event_BeforeSetCursor { name toggle } {
}
```

Some events that allow show the appropriated widgets

GiD_Event_BeforeSetWarnLine
GiD_Event_MessageBoxModeless
GiD_Event_MessageBoxOk
GiD_Event_MessageBoxEntry
GiD_Event_MessageBoxCombobox
GiD_Event_MessageBoxGetGeneralMeshSize

Note: The `GiD_Event_GetRecommendedMeshSize` could be used to modify the default mesh size recommended by GiD (and provided to `GiD_Event_MessageBoxGetGeneralMeshSize`). It must return a positive real value.

GiD_Event_GetRecommendedMeshSize
GiD_Event_MessageBoxOptionsButtons
GiD_Event_MessageBoxOptionsButtonsModeless

Note: This command 'Modeless' open a non-modal window (non-locking) and the selected option will be sent to GiD_Process when the user click its button.

GiD_Event_MessageBoxGetNormal
GiD_Event_MessageBoxGetFilename

```

proc GiD_Event_BeforeSetWarnLine { message } {
}

#show the message and continue
proc GiD_Event_MessageBoxModeless { title message } {
}

#show the message and wait the user action
proc GiD_Event_MessageBoxOk { title question message_small image } {
}

proc GiD_Event_MessageBoxEntry { title question entry_win_type default_answer button_assign image } {
}

proc GiD_Event_MessageBoxCombobox { title question options state image } {
}

proc GiD_Event_MessageBoxGetGeneralMeshSize { last_generation_size recommended_size
get_meshing_parameters_from_model } {
}

proc GiD_Event_GetRecommendedMeshSize { recommended_mesh_size } {
    ...
    return $new_recommended_mesh_size
}

proc GiD_Event_MessageBoxOptionsButtons { title question message_small options labels image } {
}

proc GiD_Event_MessageBoxOptionsButtonsModeless { title question message_small options labels image } {
}

proc GiD_Event_MessageBoxGetNormal { title } {
}

proc GiD_Event_MessageBoxGetFilename { category mode title initial_filename file_types
default_extension multiple more_options } {
}

```

View

GiD_Event_AfterChangeViewMode: useof: can be "GEOMETRYUSE", "MESHUSE", "POSTUSE" or "GRAPHUSE".

```
proc GiD_Event_AfterChangeViewMode { useof } {
    body...
}
```

GiD_Event_GetBoundingBox: will be called when recalculating the bounding box, for example when user select "zoom frame"

useof: can be "GEOMETRYUSE", "MESHUSE", "POSTUSE" or "GRAPHUSE".

This procedure must return xmin ymin zmin xmax ymax zmaz of the bounding box of the entities directly managed by the problemtype (these entities must be directly drawn with the drawopengl command).

If "" is returned instead "xmin ymin zmin xmax ymax zmaz" then any additional bounding box is considered.

Note: Must not use GiD commands asking for the model size, like GiD_Info Layers -bbox , because these commands will invoke again GiD_Event_GetBoundingBox entering in a cyclic loop. (these kind of commands are unneeded here, because GiD_Event_GetBoundingBox must inform only about its own managed objects)

Note: TclCalcModelBoundaries is a deprecated alias of GiD_Event_GetBoundingBox.

```
proc GiD_Event_GetBoundingBox { useof } {
    ...body...
    return [list $xmin $ymin $zmin $xmax $ymax $zmaz]
}
```

GiD_Event_AfterDrawModel: to allow some OpenGL custom draw, before swap buffers.

```
proc GiD_Event_AfterDrawModel {} {
}
```

GiD_Event_AfterRedraw:

```
proc GiD_Event_AfterRedraw {} {
}
```

GiD_Event_AfterChangeRenderMode: render_mode could be "flat", "normal", "smooth",...

```
proc GiD_Event_AfterChangeRenderMode { render_mode } {
}
```

Licence

GiD_Event_AfterChangeLicenceStatus: will be called when the licence status of GiD changes. Possible status could be "academic", "professional" or "temporallyprofessional"

```
proc GiD_Event_AfterChangeLicenceStatus { status } {  
}
```

Login

GiD_Event_AfterLogin: will be called when the login status of GiD changes. It provide the logged username

```
proc GiD_Event_AfterLogin { username } {  
}
```

GiD_Event_AfterLogout: will be called when the username logout.

```
proc GiD_Event_AfterLogout { } {  
}
```

GiD_Event_AfterStartSession: will be called when the logged user start the session of a program, usually GiD (use its named-user password) . It provide the program and its main version, and a JWT session token

```
proc GiD_Event_AfterStartSession { program main_version session_token } {  
}
```

GiD_Event_AfterStopSession: will be called when the logged user finish the session of a program. It provide the program and its main version

```
proc GiD_Event_AfterStopSession { program main_version } {  
}
```

DataManager

GiD_Event_AfterDataManagerConnect: will be called when the DataManager's share has been mounted locally

```
proc GiD_Event_AfterDataManagerConnect { path } {
  # path = Unit in windows ( Z: for instance)
  # path = $HOME/GiD Cloud in linux or macOS
}
```

GiD_Event_AfterDataManagerDisconnect: will be called when the DataManager's share has been dismounted. **Note** that if several GiD's are launched, **only the last GiD to quit** will dismount the DataManager's share and will **trigger this event**.

```
proc GiD_Event_AfterDataManagerDisconnect { } {
  # only the last GiD to quit will trigger this event.
}
```

GiD_Event_AfterDataManagerSetCloudFolder: will be called when DataManager's share is mounted, unmounted, or also if before start GiD it was already mounted, once it was detected.

```
proc GiD_Event_AfterDataManagerSetCloudFolder { path } {
  # path = Unit in windows ( Z: for instance)
  # path = $HOME/GiD Cloud in linux or macOS
  # path = "" if is unmounted or already was not detected
}
```

Other

GiD_Event_AfterProcess: will be called just after GiD has finished the process of 'words' (the parameter `is_rotation==1` if the command is a view movement). This event could be interesting for some tricks, like save processed commands in a batch file, or send the commands to be processed by other GiD slave, etc.

```
proc GiD_Event_AfterProcess { words is_rotation } {
}
```

GiD_Event_AfterEndCommand: will be called just after GiD pop a function from its stack. `is_view==1` if the command is some kind of view command, like a rotation, displacement, etc.

```
proc GiD_Event_AfterEndCommand {is_view } {
}
```

GiD_Event_BeforeSetVariable / GiD_Event_AfterSetVariable: will be called just before or after set the value of a GiD variable

```
proc GiD_Event_BeforeSetVariable { variable value } {
    #to avoid change the variable return -cancel-
}

proc GiD_Event_AfterSetVariable { variable value } {
}
```

GiD_Event_AfterSaveKernelDefaults: will be called just after save the C++ variables in the user preferences file

```
proc GiD_Event_AfterSaveKernelDefaults { filename } {
}
```

GiD_Event_BeforeExit: will be called just before exit GiD

```
proc GiD_Event_BeforeExit { } {
}
```


GiD_Process function

GiD_Process *command_1 command_2 ...*

Tcl command used to execute GiD commands.

This is a simple function, but a very powerful one. It is used to enter commands directly inside the central event manager. The commands have the same form as those typed in the command line within GiD.

You have to enter exactly the same sequence as you would do interactively, including the escape sequences (using the word `escape`) and selecting the menus and operations used.

You can obtain the exact commands that GiD needs by checking the **Right buttons** menu (Utilities -> Tools -> Toolbars). It is also possible to save a batch file (Utilities -> Preferences) where the commands used during the GiD session can be checked.

Here is a simple example to create one line:

```
GiD_Process Mescape Geometry Create Line 0,0,0 10,0,0 escape
```

Note: Mescape is a multiple 'escape' command, to go to the top of the commands tree.

GiD_Info function

GiD_Info <option>

Tcl command used to obtain information about the current GiD project.

This function provides any information about GiD, the current data or the state of any task inside the application.

Depending on the arguments introduced after the GiD_Info sentence, GiD will output different information:

GiD_Info automatictolerance**GiD_Info AutomaticTolerance**

This command returns the value of the Import Tolerance used in the project. This value is defined in the Preferences window under Import.

GiD_Info conditions**GiD_Info conditions over_point | over_line | over_surface | over_volume | over_layer | over_group**

This command returns a list of the conditions in the project. One of the arguments over_point, over_line, over_surface, over_volume must be given to indicate the type of condition required, respectively, conditions over points, lines, surfaces or volumes.

Note: it is also possible ask for the list of conditions declared in mesh as **over_node | over_element | over_face**

Instead of over_point, over_line, over_surface, over_volume, over_layer, over_group the following options are also available:

•GiD_Info conditions ?-interval <intv>? ?-array? ?-localaxes | -localaxesmat | -localaxescenter | -localaxesmatcenter? <condition_name> ?geometry|mesh|groups ?<entity_id>|-count?

if a condition name is given, the command returns the properties of that condition.

It is also possible to add the options **geometry|mesh|groups**, and all geometry or mesh entities that have this condition assigned will be returned in a list if its integer identifiers. The word '**groups**' must be used only if the condition was declared as 'over groups' and then the list contain its names.

If **-interval** "intv" is set, then the conditions on this interval ("intv"=1,...n) are returned instead of those on the current interval.

If **-localaxes** is set, then the three numbers that are the three Euler angles that define a local axes system are also returned (only for conditions with local axes, see DATA>Local Axes from Reference Manual).

Selecting **-localaxesmat**, the nine numbers that define the transformation matrix of a vector from the local axes system to the global one are returned.

If **-localaxescenter** is set, then the three Euler angles and the local axis center are also returned.

Selecting **-localaxesmatcenter** returns the nine matrix numbers and the center.

-array must be used only in combination with -localaxes_xx to get data more efficiently, as a list of two arrays, the first are the integer ids of the entities, and the second array the double values representing the localaxes data.

In case of 'face-element' entities the first item is a list of two items, the integer ids of the elements and the integer ids of the faces (local ids from 1 to 6)

Note: if -array is not used then -localaxes_xx return not only the local axis data of the #LA# question, but also the string values of the other questions non #LA# questions.

Adding the number id of an entity (<entity_id>) after the options mesh or geometry, the command returns the value of the condition assigned to that entity. It is possible to specify a range of ids with the syntax <first_id:last_id>

if **-count** is specified then the amount of entities with the condition is returned instead of the list of entities and its values.

Other options available if the condition name is given are

otherfields: to get some special fields of that condition

book: to get the book of the condition.

condmeshtype: to know how the condition was defined to be applied on mesh entities: over nodes, over body elements, over face elements, over face elements multiple

canrepeat: to know how the condition was defined to be applied only once or more to the same entity: values are 0 or 1

groupallow: to know how the condition 'over groups' was defined to allow the group to have entities of one or more kinds of {points lines surfaces volumes nodes elements faces}

•books: If this option is given, a list of the condition books of the project is returned.

Examples:

in: GiD_Info conditions over_point

out: "Point-Weight Point-Load"

in: GiD_Info conditions Point-Weight

out: "ovpnt 1 Weight 0"

in: GiD_Info conditions Point-Weight geometry

out: {E 8 - 2334} {E 20 - 2334} {E 31 - 343}

in: GiD_Info conditions Point-Weight geometry -count

out: "3"

in: GiD_Info Conditions -localaxes Concrete_rec_section mesh 2

out: {E 2 - {4.7123889803846897 1.5707963267948966 0.0}}

GiD_Info coordinates

GiD_Info Coordinates ?-no_model? <point_id>|<node_id> [geometry|mesh]

This command returns the coordinates (x,y,z) of a given point or node.

If **-no_model** flag is specified then entities are stored in a special container, it doesn't belong to the model **GiD_Info check**

GiD_Info check

This command returns some specialized entities check.

The result for each argument is:

- **line | surface | volume | mesh**: Type of entity.
- **<entity_id>**: The number of an entity.
- **isclosed**: For lines: 1 if start point is equal to end point, 0 otherwise. For surfaces: A surface is closed if its coordinate curves (of the full underlying surface) with parameter 0 and 1 are equal. It returns a bit encoded combination of closed directions: 0 if it is not closed, 1 if it is closed in u, 2 if it is closed in v, 3 if it is closed in both u and v directions.
- **isdegeneratedboundary**: A surface is degenerated if some boundary in parameter space (south, east, north or west) becomes a point in 3d space. It returns a bit encoded combination of degenerated boundaries, for example: 0 if it is not degenerated, 9=2^0+2^3 if south and west boundaries are degenerated.
- **selfintersection ?-tolerance <tolerance>?**: Intersections check between surface boundary lines. It returns a list of detected intersections. Each item contains the two line numbers and their parameter values.
- **orientation**: For volumes, it returns a two-item list. The first item is the number of bad oriented volume surfaces, and the second item is a list of these surfaces' numbers.
- **boundaryclose**: For volumes, a boundary is topologically closed if each line is shared by two volume surfaces. It returns 0 if it is not closed and must be corrected, or 1 if it is closed.
- **contact_elements_connectivities**: For mesh of contact surfaces or contact volumes. Verify that the connectivities of the contact elements are not crossed. It returns as a list 0 if ok, 1 if bad and then an error message.

For lines it has the following syntax:

GiD_Info check line <entity_id> isclosed

For surfaces:

GiD_Info check surface <entity_id> isclosed | isdegeneratedboundary | selfintersection

For volumes:

GiD_Info check volume <entity_id> orientation | boundaryclose

For mesh:

GiD_Info check mesh contact_elements_connectivities

Example:

in: *GiD_Info check volume 5 orientation*

out: 2 {4 38}

GiD_Info events

GiD_Info events ?-deprecated? ?-args <event_name>?

GiD_Info events

This command returns a sorted list with the names of the GiD Tcl event procedures, that could be implemented by a problemtype.

These events are automatically undefined when the problemtype is unloaded

GiD_Info events -deprecated

return a list of Tcl events that are mark as deprecated, there is a new similar event that is recommendable to be used instead.

deprecated events are also raised for back compatibility with problemtypes that are already implementing them.

GiD_Info events -args <event_name>

Return the list of arguments of the event

[Event procedures](#)

GiD_Info gendata

GiD_Info gendata

This command returns the information entered in the Problem Data window (see [Problem and intervals data file \(.prb\)](#)).

The following options are available:

- **[otherfields]**: It is possible to add this option to get the additional fields from the Problem Data window.
- **[books]**: If this option is given, a list of the Problem Data books in the project is returned.

Example:

in: *GiD_Info gendata*

out: "2 Unit_System#CB#(SI,CGS,User) SI Title M_title"

GiD_Info geometry

GiD_Info Geometry

This command gives the user information about the geometry in the project. For each entity, there are two possibilities:

- [NumPoints]: Returns the total number of points in the model.
- [NumLines]: Returns the total number of lines.
- [NumSurfaces]: Returns the total number of surfaces.
- [NumVolumes]: Returns the total number of volumes.
- [NumDimensions]: Returns the total number of dimensions.
- [MaxNumPoints]: Returns the maximum point number used (can be higher than NumPoints).
- [MaxNumLines]: Returns the maximum line number used.
- [MaxNumSurfaces]: Returns the maximum surface number used.
- [MaxNumVolumes]: Returns the maximum volume number used.
- [MaxNumDimensions]: Returns the maximum dimension number used.

GiD_Info gidbits

GiD_Info Gidbits

If GiD is a x32 binary executable, then this command returns 32.

If GiD is a x64 binary executable, then this command returns 64.

GiD_Info gidversion

GiD_Info GiDVersion

This command returns the GiD version number. For example 10.0.8

GiD_Info graphcenter

GiD_Info graphcenter

This command returns the coordinates (x,y,z) of the center of rotation.

GiD_Info intvdata

GiD_Info intvdata

This command returns a list of the interval data in the project (see [Problem and intervals data file \(.prb\)](#)).

The following options are available:

- interval <number>: To get data from an interval different from the number 0 (default).
- [otherfields]: It is possible to add this option to get the additional fields from the Interval Data window.
- [books]: If this option is given, a list of the Interval Data books in the project is returned.
- [num]: If this option is given, a list of two natural numbers is returned. The first element of the list is the current interval and the second element is the total number of intervals.

GiD_Info ispointinside

GiD_Info IsPointInside

GiD_Info IsPointInside ?-no_model? ?-tolerance <tol>? Line|Surface|Volume <num> {<x> <y> <z>}

This commands returns 1 if the point {x y z} is inside the specified volume/surface/curve, or 0 if lies outside.

If **-no_model** flag is specified then entities are stored in a special container, it doesn't belong to the model

GiD_Info layers

GiD_Info layers

This command returns a list of the layers in the project. These options are also available:

- <layer_name>: If a layer name is given, the command returns the properties of that layer.
- on: Returns a list of the visible layers.
- off: Returns a list of the hidden layers.
- hasbacklayers: Returns 1 if the project has entities inside back layers. Note: **GiD_Info back_layers** returns a list with the back layers

Example:

in: GiD_Info back_layers

out: Layer2_back

- bbox ?-use geometry|mesh? <layer_name_1> <layer_name_2> ...

Returns an item with a list of six real numbers representing two coordinates (x1,y1,z1,x2,y2,z2) which define the bounding box of the entities that belong to the list of layers.

If the option -use geometry|mesh is used, the command returns the bounding box of the **geometry** or the bounding box of the **mesh**.

If the list of layers is empty, the maximum bounding box is returned.

- ?-count? -entities <type> ?-elementtype <elementtype>? ?-higherentity <num>?: One of the following arguments

must be given for *<type>*: **nodes, elements, points, lines, surfaces or volumes**. A layer name must also be given. The command will return the nodes, elements, points, lines surfaces or volumes of that layer. If *-count* is set then the amount of entities is returned instead of the list of its ids.

For elements it is possible to specify **-elementtype <elementtype>** to get only this kind of elements.

-higherentity <num> Allow to get only entities with this amount of higherentities.

••**-canbedeleted <layer_name>** returns a list with two items: a boolean 0|1 to know if layer or its child layers have some entity or parts in back layer or conditions over the layer. the second item is a message explaining the cause to not delete it directly.

Examples:

in: GiD_Info layers

out: "layer1 layer2 layer_aux"

in: GiD_Info layers -on

out: "layer1 layer2"

in: GiD_Info layers -entities lines layer2

out: "6 7 8 9"

in: GiD_Info layers -count -entities lines layer2

out: 4

GiD_Info library

GiD_Info library

To access to information of some linked libraries (e.g. gid mesh libraries)

GiD_Info library names

This command returns a list of libraries

GiD_Info library version <library_name>

This command returns a string with the version of the code of the library.

<library_name> must be one of the ones returned by *GiD_Info library names*

GiD_Info library format_version <library_name>

This command returns a string with the version of the data format used to transmit the input/output of the library

<library_name> must be one of the ones returned by *GiD_Info library names*

The data format could be transmitted in memory or serialized in a .gidml file with HDF5 syntax.

GiD_Info listmassproperties

GiD_Info ListMassProperties ?-no_model? Points|Lines|Surfaces|Volumes|Nodes|Elements <entity_id>

If **-no_model** flag is specified then entities are stored in a special container, it doesn't belong to the model

This command returns properties of the selected entities.

*It returns the **length** if entities are lines, **area** if surfaces, **volume** if volumes, or the **center of gravity** if entities are nodes or elements. It has the following arguments:*

•**Points/Lines/Surfaces/Volumes/Nodes/Elements**: Type of entity.

•**entity_id**: The number of an entity. It is also possible to enter a list of entities (e.g.: 2 3 6 45) or a range of entities (e.g.: entities from 3 to 45, would be 3:45).

Example:

in: GiD_Info ListMassProperties Lines 13 15

out:

LINES

n. Length

13 9.876855

15 9.913899

Selected 2 figures

Total Length=19.790754

GiD_Info list_entities

GiD_Info list_entities

•**GiD_Info list_entities ?-no_model? Status|PreStatus|PostStatus**

If **-no_model** flag is specified then entities are stored in a special container, it doesn't belong to the model

This command returns a list with general information about the current GiD project.

PreStatus ask for the information of preprocess

PostStatus ask for the information of postprocess

Status return the information of pre or postprocess depending of where are now, in pre or post mode.

Example:

in: GiD_Info list_entities PreStatus

out:

Project name: UNNAMED

Problem type: UNKNOWN

Changes to save(0/1): 1
 Necessary to mesh again (0/1): 1
 Using LAYER: NONE
 Interval 1 of 1 intervals
 Degree of elements is: Normal
 Using now mode(geometry/mesh): geometry
 number of points: 6
 number of points with 2 higher entities: 6
 number of points with 0 conditions: 6
 number of lines: 6
 number of lines with 1 higher entities: 6
 number of lines with 0 conditions: 6
 number of surfaces: 1
 number of surfaces with 0 higher entities: 1
 number of surfaces with 0 conditions: 1
 number of volumes: 0
 number of nodes: 8
 number of nodes with 0 conditions: 8
 number of Triangle elements: 6
 number of elements with 0 conditions: 6
 Total number of elements: 6
 Last size used for meshing: 10
 Internal information:
 Total MeshingData:0 Active: 0 0%

•GiD_Info list_entities

*This command returns information about entities.
 It has the following arguments:*

- Points / Lines / Surfaces / Volumes / Dimensions / Nodes / Elements / Results**: Type of entity or Results. **Note**: If the user is postprocessing in GiD, the information returned by **Nodes/Elements** concerns the nodes and elements in postprocess, including its results information. To access preprocess information about the preprocess mesh, the following entity keywords should be used: **PreNodes/PreElements**.
- entity_id**: The number of an entity. It is also possible to enter a list of entities (e.g.: 2 3 6 45), a range of entities (e.g.: entities from 3 to 45, would be 3:45) or a layer (e.g.: layer:layer_name).

Using "list_entities Results" you must also specify <analysis_name> <step> <result_name> <indexes> With the option **-more**, more information is returned about the entity. The **-more** option used with lines returns the length of the line, its radius (arcs), and the list of surfaces which are higher entities of that line; used with elements it returns the type of element, its number of nodes and its volume.

Example 1:

in: GiD_Info list_entities Points 2 1

out:

POINT

Num: 2 HigherEntity: 1 conditions: 0 material: 0

LAYER: car_lines

Coord: -11.767595 -2.403779 0.000000

END POINT

POINT

Num: 1 HigherEntity: 1 conditions: 0 material: 0

LAYER: car_lines

Coord: -13.514935 2.563781 0.000000

END POINT

Example 2:

in: GiD_Info list_entities lines layer:car_lines

out:

STLINE

Num: 1 HigherEntity: 0 conditions: 0 material: 0

LAYER: car_lines

Points: 1 2

END STLINE

STLINE

Num: 13 HigherEntity: 0 conditions: 0 material: 0

LAYER: car_lines

Points: 13 14

END STLINE

Example 3 (using **-more**):

in: GiD_Info list_entities -more Lines 2

out:

STLINE

Num: 2 HigherEntity: 2 conditions: 0 material: 0

LAYER: Layer0

Points: 2 3

END STLINE

LINE (more)

Length=3.1848 Radius=100000

Higher entities surfaces: 1 3

END LINE

GiD_Info localaxes

GiD_Info localaxes ?<name>? ?-localaxesmat?info localaxes

Returns a list with all the user defined local axes.

info localaxes <name> returns the parameters (three euler angles and the center) that define the local axes called <name>.

info localaxes <name> -localaxesmat instead of returning the three euler angles, it returns the nine numbers that define the transformation matrix of a vector from the local axes system to the global one.

GiD_Info magnitudes

GiD_Info magnitudes 0|1 ?<magnitude_name> units?

Return information about units and the conversion factor between units of a magnitude.

GiD_Info magnitudes 0|1

GiD_Info magnitudes 0

return a list with the available default magnitude names of GiD (the ones defined in the scripts\units.gid file) and

GiD_Info magnitudes 1

return a list with the available magnitude names of the project (the ones defined loading a <problemtyp>.uni file)

e.g.

in: GiD_Info magnitudes 0

out: ACCELERATION ANGLE LENGTH MASS STRENGTH PRESSURE TEMPERATURE

GiD_Info magnitudes 0|1 <magnitude_name> units

return a list of unit names and conversion factors from the reference unit (the first listed)

<magnitude_name> is the name of the magnitude, like LENGTH or other magnitudes that could be defined by the problemtyp

e.g.

in: GiD_Info magnitudes 0 LENGTH units

out: {1.0 m {} m {} m} {100 cm {} cm {} cm} {1.0e+3 mm {} mm {} mm}

with the multiplication factor to convert the length magnitude from the reference unit to another unit.

(e.g to convert a value from cm to m must multiply by 100)

GiD_Info materials

GiD_Info materials

This command returns a list of the materials in the project.

GiD_Info materials(<bookname>) returns only the materials that belong to the book <bookname>

These options are also available:

- <material_name>: If a material name is given, its properties are returned.
- It is also possible to add the option **otherfields** to get the fields of that material, or the option **book** to get the book of that material.
- books**: If this option is given, a list of the material books in the project is returned.

Examples:

in: GiD_Info materials

out: "Air Steel Aluminium Concrete Water Sand"

in: GiD_Info materials Steel

out: "1 Density 7850"

GiD_Info materials Steel otherfields

GiD_Info materials books

GiD_Info materials(profiles)

GiD_Info mesh

GiD_Info Mesh

This command gives the user information about the selected mesh in the project.

Without arguments it returns 1 if there is mesh, followed by a list with all types of element used in the mesh.

?-pre | -post? -step_index <step_index> | -step_value <step_value>? ?-set_name <set_name>?:

-pre | -post: To specify to use the preproces or postprocess mesh (default -pre).

-step_index <step_index> | -step_value <step_value>: In post can specify time step if the mesh changes along the time (by default the current time step is assumed)

Must set only -step_index <step_index> or alternatively -step_value <step_value>, but not both.

the <step_index> is an integer starting from 0 (a special value **all** is valid and mean 'all steps' for some options)

the <step_value> is a double value representing the value of the time step.

-set_name <set_name>: for Elements of post is optional specify -set_name <set_name> to get only the elements of this set, in case that more than one.

•NumElements <Elemtype> ?<nnode>?: returns the number of elements of the mesh.

Elemtype can be: **Line | Triangle | Quadrilateral | Tetrahedra | Hexahedra | Prism | Pyramid | Point | Sphere | Circle | Any.**

nnode is the number of nodes of an element, if this argument is missing the amount does not take into account the number of nodes.

•••NumNodes: Returns the total number of nodes of the mesh. **MaxNumElements:** Returns the maximum element number. **MaxNumNodes:** Returns the maximum node number.

•Elements <Elemtype> ?<first_id>? ?<last_id>? ?-sublist|-array|-array2? ?-avoid_frozen_layers? ?-layer <layername>? ?-group <groupname>?:

Returns a list with the element numbers, the connectivities, radius if it is a sphere, normal if it is a circle, and the material number, from 'first_id' to 'last_id', if they are specified.

Note: in -post case it return a list of items, one item by set, of the required element type, and each item is a sub-list with the data related previously (number connectivities ?radius? ?normal? material)

•Nodes ?<first_id>? ?<last_id>? ?-sublist|-array|-array2? ?-avoid_frozen_layers? ?-layer <layername>? ?-group <groupname>?: Returns a list with the node number and x y z coordinates, from 'first_id' to 'last_id', if they are specified.

Modifiers:

-sublist : Instead of a flat list it returns each result item as a Tcl list (enclosed in braces)

-array : Instead of a flat list it returns the results as a list of objarrays (more efficient).

For 'Nodes' it returns a list with 1 objarray for the NodeIDs and a list with 3 objarrays: the X coordinates, the Y coordinates and the Z coordinates.

For 'Elements' it returns a list with the element type, an objarray with the element id's, a list with an objarray for each node of the connectivity (i.e. for a triangle an objarray for all node1, another for the node2 and another for the node3), and an objarray for the material id of the elements.

-array2 : Instead of a flat list it returns the results as a list of objarrays (more efficient).

For 'Nodes' it returns a list with 2 objarrays: one for the NodeIDs and another for the xyz coordinates.

For 'Elements' it returns a list with the element type, an objarray with the element id's, an objarray for all the connectivities (i.e. for a triangle an objarray with node1-node2-node3-node1-node2-node3), and an objarray for the material id of the elements.

An 'objarray' is a Tcl_Obj object specialized for arrays, implemented as a Tcl package named 'objarray'. (for more information see scripts /objarray/objarray.pdf|md|html)

-avoid_frozen_layers : to ignore nodes or elements of frozen layers

-layer <layername> : to get only nodes of element of this layer

-group <groupname> : to get only nodes of element of this group

•EmbeddedDistances: Returns a list with 2 items, the objarray of ids of the nodes (integers) and the objarray of distances to theboundary (doubles). This information is only available meshing with embedded mesh type

Examples:

in: GiD_Info Mesh

out: "1 Tetrahedra Triangle"

in: GiD_Info Mesh MaxNumNodes

out: "1623"

set data [GiD_Info Mesh EmbeddedDistances]

```
lassign $data nodes distances
```

```
set length [objarray length $nodes_list]
```

```
for {set i 0} {$i < $length} {incr i} {
```

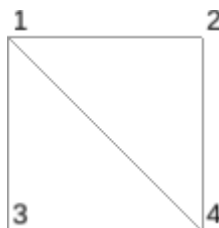
```
set node_id [objarray get $nodes $i]
```

```
set distance [objarray get $distances $i]
```

```
W "$node_id $distance"
```

```
}
```

Example 2:



Mesh with

```
1. node_id x_coord y_coord z_coord
```

```
1 0.0 1.0 0.0
```

```
2 1.0 1.0 0.0
```

```
3 0.0 0.0 0.0
```

```
4 1.0 0.0 0.0
```

```
2. element_id node1 node2 node3 material
```



```

1 3 4 1 0
2 1 4 2 0
GiD_Info Mesh nodes
1 0.0 1.0 0.0 2 1.0 1.0 0.0 3 0.0 0.0 0.0 4 1.0 0.0 0.0
GiD_Info Mesh nodes -sublist
{1 0.0 1.0 0.0} {2 1.0 1.0 0.0} {3 0.0 0.0 0.0} {4 1.0 0.0 0.0}
GiD_Info Mesh nodes -array
{1 2 3 4} 0.0 1.0 0.0 1.0 {1.0 1.0 0.0 0.0} {0.0 0.0 0.0 0.0}
GiD_Info Mesh nodes -array2
{1 2 3 4} {0.0 1.0 0.0 1.0 1.0 0.0 0.0 0.0 1.0 0.0 0.0}
GiD_Info Mesh elements triangle
1 3 4 1 0 2 1 4 2 0
GiD_Info Mesh elements triangle -sublist
{1 3 4 1 0} {2 1 4 2 0}
GiD_Info Mesh elements triangle -array
{Triangle {1 2} 3 1} {4 4} {1 2 {0 0}}
GiD_Info Mesh elements triangle -array2
{Triangle {1 2} {3 4 1 1 4 2} {0 0}}

```

GiD_Info meshquality

GiD_Info MeshQuality

This command returns a list of numbers. These numbers are the Y relative values of the graph shown in the option Meshing -> Mesh quality (see MESH>Mesh Quality from Reference Manual) and two additional real numbers with the minimum and maximum limits.

This command has the following arguments:

- MinAngle / MaxAngle / ElemSize / ElemMinEdge / ElemMaxEdge / ElemShapeQuality / ElemMinJacobian / Radius:** quality criterion.
- Line / Triangle / Tetrahedra / Quadrilateral / Hexahedra / Prism / Pyramid / Point / Sphere / Circle:** type of element.
- <min_value>: e.g. minimum number of degrees accepted.
- <max_value>: e.g. maximum number of degrees accepted.

if min_value and max_value are set to 0 then limits will be automatically set to the minimum and maximum of the mesh

- <num_divisions>: number of divisions.

Example:

```

in: GiD_Info MeshQuality MinAngle Triangle 20 60 4
out: "13 34 23 0 20.0 60.0"

```

GiD_Info opengl

GiD_Info OpenGL

Returns information about the OpenGL/glew version and renderer.

GiD_Info ortholimits

GiD_Info ortholimits

This command returns a list (Left, Right, Bottom, Top, Near, Far, Ext) of the limits of the geometry in the project. In perspective mode near and far have the perspective distance subtracted.

GiD_Info bounding_box

GiD_Info bounding_box ?-geometry|-mesh|-post? ?-layers_off_also? ?point|line|surface|volume <entity_id>?

This command returns a list {x_min x_max y_min y_max z_min z_max} of the bounding box that contains the visualized model with the current settings (on layers) of preproc: depends on the current visualization mode: geometry or mesh.

If the extra flag-geometry, -mesh or -post is set then is used instead of the current visualization.

by default off layers are ignored, to change it can set -layers_off_also

It also could return the bounding box of a geometric entity, specified by its category and <entity_id>.

GiD_Info page and capture settings

GiD_Info postprocess get <option>

The following <option> are available:

pagedimensions: Returns the settings that are applied when an image is sent to the printer or a snapshot is taken. Returns (in inches) pagewidth, leftmargin, topmargin, imgwidth, imgheight, landscape/portrait.

hardcopyoptions: Returns 1 or 0 for the options "show GiD logo in images", "white background for images", "white background for videos", "transparent background for images", "transparent background for animations", "draw background images for images", "draw background images for videos", "show GiD logo in videos".

animationformat: Returns the default animation format.

GiD_Info parametric

GiD_Info parametric

This command returns geometric information (coordinates, derivatives, etc.) about parametric lines or surfaces.

For lines it has the following syntax:

GiD_Info parametric ?-no_model? line <entity_id> coord|deriv_t|deriv_tt|t_fromcoord|t_fromrelativelength|length_to_t <t>|<x> <y> <z> ?<t_seed>?

And for surfaces:

GiD_Info parametric ?-no_model? surface <entity_id>

coord|deriv_u|deriv_v|deriv_uu|deriv_vv|deriv_uv|normal|uv_fromcoord|maincurvatures|uv_projection_z_fromcoord <u> <v> | <x> <y> <z> ?<u_seed> <v_seed>?

If **-no_model** flag is specified then entities are stored in a special container, it doesn't belong to the model

The result for each argument is:

- line|surface:** Type of entity.
- <entity_id>:** The number of an entity.
- coord:** 3D coordinate of the point with parameter t (line) or u, v (surface).
- deriv_t:** First curve derivative at parameter t .
- deriv_tt:** Second curve derivative at parameter t .
- t_fromcoord:** t parameter closest to a 3D point. It is possible to specify also the
- <t_seed>t_fromrelativelength:** parameter corresponding to a relative (from 0 to 1) arc length t **length_to_t:** length of the curve until the parameter t (if $t=1.0$ then it is the total length) **deriv_u,deriv_v:** First partial u or v surface derivatives.
- deriv_uu,deriv_vv,deriv_uv:** Second surface partial derivatives.
- normal:** Unitary surface normal at u, v parameters.
- uv_fromcoord:** u, v space parameters closest to a 3D point. It is possible to specify also the **<u_seed> <v_seed>**
- maincurvatures:** return a list with 8 numbers: $v1x\ v1y\ v1z\ v2x\ v2y\ v2z\ c1\ c2$

$v1x\ v1y\ v1z$: first main curvature vector direction (normalized)

$v2x\ v2y\ v2z$: second main curvature vector direction (normalized)

$c1\ c2$: main curvature values

••**uv_projection_z_fromcoord:** to get the location $u\ v$ of a point $x\ y$ projected in direction z on a surface (the z of the point must not be supplied). It is possible to specify also the **<u_seed> <v_seed>**

e.g: set uv [GiD_Info parametric surface \$id uv_projection_z_fromcoord \$x \$y]

Note: The vector derivatives are not normalized.

Example:

in: GiD_Info parametric line 26 deriv_t 0.25

out: 8.060864 -1.463980 0.000000

GiD_Info perspectivefactor

GiD_Info perspectivefactor

This command returns which perspective factor is currently being used in the project.

GiD_Info postprocess

GiD_Info postprocess get

This command returns information about the GiD postprocess.

Sets

GiD_Info postprocess get <option>

The following <option> are available:

A volumeset is basically a mesh of volume elements, a surfaceset is a mesh of surface (and line) elements. All entities of the set are of the same element type.

A cut store the information to define the cut location, cutting could create true new surfacesets including its interpolated results, or create only visual meshes to be drawn temporarily.

all_volumesets

Returns a list of all sets of volume.

all_surfacesets

Returns a list of all sets of surface (and line).

all_cutsets

Returns a list of all cuts.

cur_volumesets

Returns a list of the visible volume sets.

cur_surfacesets

Returns a list of the visible surface (and line) sets.

cur_cutsets

Returns a list of the visible cut sets.

all_volumes_colors

Returns a list of the volume colors used in the project.

Each item of the list has 4 sub-items: {color_ambient color_diffuse color_specular shininess}

Colors are represented in RGB hexadecimal format #RRGGBB. Example: #000000 would be black, and #FFFFFF would be white.

Shininess is a real value from 0.0 to 1.0

all_surfaces_colors

Returns a list of the surface colors used in the project. Colors are represented in RGB hexadecimal format.

all_cuts_colors

Returns a list of the cut colors used in the project. Colors are represented in RGB hexadecimal format.

property massive|transparent|transparency|displaystyle|edgewidth|visibility|visualizeresults <mesh_name>

Returns value of the specified property for the entered mesh name.

Graphs

GiD_Info postprocess get <option>

See also the more modern commands Graphs and GraphSets

The following <option> are available:

all_graphs

Returns a list of all graphs.

all_line_graphs

Returns a list of the line graphs.

all_graphs_views: Returns all available graphs types.

graphs_option: ?-allowed_values? <graphset_property>| <graph_property> <graph_name>

with -allowed_values flag it is returned a list with the possible values of the property instead of the current property value.

To get graphset properties

•<graphset_property> could be:

CurrentStyle Grids MainTitle TitleVisible LegendLocation CoordType AngleAxis AngleUnit ShowOrigAxes ShowRadMarks ColorOrig ColorRad PatRad OutlineOnModel ShowGraphs X_axis Y_axis ShowModelView LineWidth PointSize

To get graph properties (must specify also the <graph_name>)

•<graph_property> could be:

Style Color ColorAsCFill LineWidth Pattern PatternFactor PointSize Title NumResults ResultsX ResultsY LabelX LabelY Visible

Graph axis options:

axis_options <axis_option>

Returns the current value for the specified property of the drawing axes option.

•<axis_option> could be:

ShowAxes Type Dimensions AxesWidth AxisXColor AxisYColor AxisZColor XYZLabels Grid GridXColor GridYColor GridZColor GridXDivisions GridYDivisions GridZDivisions FactorPatron Patron Label LabelColorAxes LabelType VarFontSize LabelColor Arrow ArrowXColor ArrowYColor ArrowZColor

display style

GiD_Info postprocess get <option>

The following <option> are available:

all_display_styles

Returns a list of all types of display styles available.

cur_display_style

Returns the current display style.

all_display_renders

Returns a list of all types of rendering available.

cur_display_render

Returns the current rendering method.

num_lights

Returns the current number of active lights. The option can be changed in View->Render->Lights menu.

all_display_culling

Returns a list of all types of culling available.

cur_display_culling

Returns the current culling visualization.

cur_display_transparence

Returns Opaque or Transparent depending on the current transparency. Transparency is chosen by the user in the Select & Display Style window.

cur_display_body_type

Returns Massive if the option Massive is selected in the "Select & Display Style" window. It returns Hollow if that option is not activated.

cur_show_conditions

Returns the option selected in the Conditions combo box of the Select & Display Style window. (Possible values: Geometry Mesh None)

all_show_conditions

Returns all the options available in the Conditions combo box of the Select & Display Style window. (Geometry Mesh None)

cur_pre_model_properties

Returns all the options selected in the Draw Model and Model Render combo box of the Select & Display Style window.

all_pre_model_properties

Returns all the options available in the Draw Model and Model Render combo box of the Select & Display Style window.

results

GiD_Info postprocess get <option>

See also the more modern command Results

The following <option> are available:

all_analysis

Returns a list of all analyses in the project.

all_steps <analysis_name>

Returns the list of time step values for all steps of "analysis_name".

cur_analysis

Returns the name of the current analysis.

cur_step

Returns the current time step value, a double.

cur_step_index

Returns the current time step index, an integer (starting from 0).

is_mesh_variable_along_steps

Returns 1 if the postprocess mesh is changing along the time

all_results_views

Returns all available result views.

cur_results_view

Returns the current result view.

cur_results_list <visualization_type>

The available kinds of result visualization are given by the option all_results_views. The command returns a list of all the results that can be represented with that visualization in the current step of the current analysis.

result_unit <result_name>

Returns the unit name, the multiplier factor and the addition factor of the result "result_name".

results_list <visualization_type> <analysis_name> <step_value>

The available kinds of result visualization are given by the option all_results_views. The command returns a list of all the results that can be represented with that visualization in the given step.

cur_result

Returns the current selected result. The kind of result is selected by the user in the View results window.

cur_components_list <result_name>

Returns a list of all the components of the result "result_name".

cur_complex_components_list <result_name>

Returns a list of all the complex components of the result "result_name".

components_list <result_view_type> <result_name> <analysis_name> <step_value>

Returns a list of all the components of the result "result_view_type" "result_name" "analysis_name" "step_value".

cur_component

Returns the current component of the current result.

results_view_list

Returns the current result view, analysis name, step, result name and component name.

main vs reference

GiD_Info postprocess get <option>

The following <option> are available:

It is possible to display two copies of the geometry, the main one (where results are drawn), and a reference one.

Main geometry

main_geom_state

Returns whether the main geometry is Deformed or Original.

main_geom_factor <analysis_name> <step_value> <result_name>

Returns the deformation factor of the main geometry of "analysis_name" "step_value" "result_name".

main_geom_all_deform

Returns a list of all the deformation variables (vectors) of the main geometry.

main_geom_cur_deform

Returns the current deformation variable (vectors) of the main geometry.

main_geom_cur_step

Returns the main geometry current step.

main_geom_cur_anal

Returns the main geometry current analysis.

main_geom_cur_factor

Returns the main geometry current deformation factor.

Reference geometry (auxiliary geometry to be compared with main geometry, usually deforming one of them)

show_geom_state

Returns whether the reference geometry is Deformed or Original.

show_geom_color

Returns the reference geometry current color.

show_geom_factor <analysis_name> <step_value> <result_name>

Returns the deformation factor of the reference geometry of "analysis_name" "step_value" "result_name".

show_geom_cur_deform

Returns the current deformation variable (vectors) of the reference geometry.

show_geom_cur_analysis

Returns the reference geometry current analysis.

show_geom_cur_step

Returns the reference geometry current step.

show_geom_deformation_type

Returns whether the reference geometry deformation is relative or absolute

mesh elements

GiD_Info postprocess get <option>

The following <option> are available:

border_criteria

Returns the value of border angle option. Select the angle criteria between faces to consider the shared edge a boundary edge, i.e. sharp edge, or not. Angles between normals of adjacent faces smaller than the criteria set will be considered a sharp edge and visualized when the mesh style 'boundaries' is selected.

edge_colour

Returns the edge color for drawing the elements.

Points:

info_point_size

Returns the current values for the Point element options in Postprocess->Mesh elements preferences window.

info_point_size_factor "analysis_name" "step_value" "result_name" "component_name"

Returns the current point element factor for "analysis_name" "step_value" "result_name" "component_name".

Spheres:

info_sphere_size

Returns the current values for the Sphere element options in Postprocess->Mesh elements preferences window.

info_sphere_size_factor "analysis_name" "step_value" "result_name" "component_name"

Returns the current sphere element factor for "analysis_name" "step_value" "result_name" "component_name".

Lines:

info_line_size

Returns the current values for the Line element options in Postprocess->Mesh elements preferences window.

legends and comments

GiD_Info postprocess get <option>

The following <option> are available:

comments

Returns the comments that appear in different results views.

info_legend

Returns the current values showed in the legend.

legends_state

Returns the current values in Postprocess->Legends and comments preferences window.

contour fill

GiD_Info postprocess get <option>

The following <option> are available:

contour_limits

Returns the minimum and maximum value of the contour limits. Before each value, the word STD appears if the contour limit value is the default value, and USER if it is defined by the user.

cur_contour_limits

Returns the minimum and maximum value of the current value.

cur_contour_color_config

Returns the current values in the More color options window in Postprocess->Contour fill and lines preferences window.

contour_lines_width

Returns the contour lines width value.

current_color_scale

Returns a list of the colors used for the color scale; the first element of the list is the number of colors. Each color is represented in RGB hexadecimal format. #RRGGBB

Example: #000000 would be black, and #FFFFFF would be white.

colour_map_list

Returns the list of available color maps for contour fill visualization

vector

GiD_Info postprocess get <option>

The following <option> are available:

cur_vector_factor <results_view_type> <result_name> ?<component_name>? <analysis_name> <step_value>

Returns the current vector factor of the result "results_view_type" "result_name" "component_name" "analysis_name" "step_value".

vector_detail

Returns the current value for vector detail option.

vector_size

Returns the current value for vector size option.

AllVectors

Returns Yes if the draw interior vectors option is enabled, No if don't.

VectorNumCols

Returns the current number of colors for vectors when the color mode is set to "by modules".

VMonoColor

Returns the current color for vectors when the color mode is set to "monochrome".

VectorTensionColour

Returns the current color for tension vector color option.

VectorCompressionColour

Returns the current color for compression vector color option.

VectorColour

Returns the current color mode for vectors.

VectorOffset

Returns the current value for vector offset option.

VectorFilterFactor

Returns the current value for vector filter factor option.

iso surfaces

GiD_Info postprocess get <option>

The following <option> are available:

iso_all_display_styles

Returns a list of all available display styles for isosurfaces.

iso_cur_display_style

Returns the current display style for isosurfaces.

iso_all_display_renders

Returns a list of all types of rendering available for isosurfaces.

iso_cur_display_render

Returns the current rendering method for isosurfaces.

iso_cur_display_transparency

Returns Opaque or Transparent depending on the current transparency of isosurfaces.

iso_cur_result_values

Returns the current result values of isosurfaces.

isosurface_options

Returns the current values in Postprocess->Iso surfaces preferences window.

stream lines

GiD_Info postprocess get <option>

The following <option> are available:

info_stream_line_size

Returns the stream line size value.

result_stream_lines_options

Returns the current stream lines detail render, the color mode, the color of monochrome mode and the initial rotation. The values can be found in Postprocess->Stream lines preferences window.

stream_draw_arrows Arrows|ArrowsSize|ArrowsFreq|ArrowsColor

Returns the current values for draw arrows?, arrows size, arrows frequency or arrows color option.

stream_label

Returns the current value for stream line label option.

stream_length

Returns the current value for stream line maximum length option.

stream_max_points

Returns the current value for stream line maximum points option.

result surface**GiD_Info postprocess get <option>**

A 'result surface' is basically a surface-graph drawn in the local axis of the surface elements (the result value represented normal to the surface)

The following <option> are available:

cur_result_surface_factor <results_view_type> <result_name> <component_name> <analysis_name> <step_value>

Returns the current surface factor of the result "results_view_type" "result_name" "component_name" "analysis_name" "step_value".

result_surface_options

Returns the current values in Postprocess->Result surface preferences window.

line diagrams**GiD_Info postprocess get <option>**

The following <option> are available:

A 'line diagram' is basically a graph drawn in the local axis of the line elements, used for example to represent in bars graphs of bending moments, shear and axial efforts, etc.

cur_diagram_factor <result_name> <analysis_name> <step_value>

Returns the current diagram factor of "result_name" "analysis_name" "step_value".

diagram_options

Returns the current value in Postprocess->Line diagrams preferences window.

others

The following <option> are available:

changed_analysis_step

Returns 1 if the analysis step has changed from the last query, 0 if has not changed.

changed_results_view

Returns 1 if the results view has changed from the last query, 0 if has not changed.

changed_geom_list

Returns 1 if the geometry has changed from the last query, 0 if has not changed.

changed_graph_list

Returns 1 if the graph list has changed from the last query, 0 if has not changed.

Results_Preference <result_name> <visualization> <property>

Returns the current value of the preference defined by "result_name" "visualization" "property".

scale_result_options

Returns the current values for options in Utilities->Scale result view menu.

view_follows_node_options

Returns the current values for "view centered and following node" in the Window->Animate window.

GiD_Info problemtypepath**GiD_Info problemtypepath**

This command returns the absolute path to the current problem type.

GiD_Info project

GiD_Info Project <item>?

This command returns information about the project. More precisely, it returns a list with:

- Problem type name.
- Current model name.
- 'There are changes' flag.
- Current layer to use.
- Active part (GEOMETRYUSE, MESHUSE, POSTUSE or GRAPHUSE).
- Quadratic problem flag.
- Drawing type (normal, polygons, render, postprocess).NOPOST or YESPOST.
- Debug or nodebug.
- GiD temporary directory.
- Must regenerate the mesh flag (0 or 1).
- Last element size used for meshing (NONE if there is no mesh).
- BackgroundFilename is the name of a background mesh file to assign mesh sizes.
- RequireMeshSize. (1 if all sizes are specified by the number of divisions, then user is not required to specify the mesh size)RecommendedMeshSize. (The value of the mesh size that the program will recommend, based on the model size)
- HelpAboutMoreInfo (extra information)
- GeoVersion (the current version of the .geo file)
- SomeFigureWithItsMesh (0 or 1, to know if mesh is linked to geometry)

It is possible to ask for a single item only rather than the whole list, with <item> equal to:

ProblemType | ModelName | AreChanges | LayerToUse | ViewMode | Quadratic | RenderMode | ExistPost | Debug | TmpDirectory | MustReMesh | LastElementSize | BackgroundFilename | RequireMeshSize | RecommendedMeshSize | HelpAboutMoreInfo | GeoVersion | SomeFigureWithItsMesh

Example:

```
in: GiD_Info Project
out: "cmas2d e:\models\car_model 1 layer3 MESHUSE 0 normal YESPOST nodebug C:\TEMP\gid2 0 1.4 0"
in: GiD_Info Project ModelName
out: "e:\models\car_model"
```

GiD_Info unitssystems

GiD_Info unitssystems ?gid|prj|usersys|modunit|prbsys|udstate|magused?

return information about the systems of units

GiD_Info unitssystems

return 1 if the problemtype is using units, 0 else

•GiD_Info unitssystems

gid|prj|usersys|modunit|prbsys|udstate|magusedgid

It return a list with the names of the units systems defined in the file gid.uni

•prj

It return a list with the names of the units systems of the project defined in the file <problemtype>.uni

•usersys

It return a list with the names of the units systems defined at runtime by the user (in case that it is not disabled by <problemtype>.uni)

•modunit

It return information about the current length units of the model.

Return a list with three items like "LENGTH 1 0" or "LENGTH 2 1"

the first item is LENGTH, the magnitude name of length

the second item is the index in magnitudes of the current length unit

the third item is 0 if length is defined in the 'gid units set' or 1 if it is defined in the 'problemtype units set'

e.g.

in:

```
lassign [GiD_Info unitssystems modunit] magnitude_name unit_index set_index
```

```
set model_length_unit [lindex [GiD_Info magnitudes $set_index $magnitude_name $unit_index] 1]
```

out: m

•prbsys

Is the name of the current unit system to use for the model

•udstate

return the 'user defined systems' state: 0 or 1 (0 disabled, 1 enabled)

•magused

List of names of magnitudes used by the model (some magnitudes defined in gid.uni could be neglected to not show them)

GiD_Info variables**GiD_Info variables ?-expand_array_names? ?-mesh? ?<variable_name>?****GiD_Info variables**

It returns a sorted list of the GiD variables, including the ones related to meshing

Note: For array-like names it only return the base name, the array names can be obtained in a second step with

GiD_Set -array_names <variable_base_name>.

If -expand_array_names is used then instead the base names it replace them by the list of expanded names

GiD_Info variables -mesh

If returns a sorted list of the GiD meshing variables only.

GiD_Info variables <variable_name>

This command returns the value of the variable indicated.

Note: this command is deprecated to get a variable value. The more modern *GiD_Set* command can be used to get or set the value of a GiD variable, look into *Special Tcl commands>Other* for more information.

GiD variables can be found in the Right buttons menu (see UTILITIES>Tools from Reference Manual), under the option Utilities -> Variables.

GiD_Info view**GiD_Info view**

```
{x -13.41030216217041 13.41030216217041} {y 10.724431991577148 -10.724431991577148} {z -30.0 30.0} {e
10.0} {v 0.0 0.0 0.0} {r 1.0}
{m 1.0 0.0 0.0 0.0 0.0 1.0 0.0 0.0 0.0 1.0 0.0 0.0 0.0 1.0} {c 0.0 0.0 0.0} {pd 0.0} {pno 0.0}
{pfo 0.0} {pf 4.0} {pv 0.0} {NowUse 0}
{DrawingType 0} {LightVector 90.0 90.0 150.0 0.0}
```

This command returns the current view parameters. Something like:

See VIEW>View entry>Save/Read View of Reference Manual for a brief explanation of this parameters

Special Tcl commands

GiD add to the standard Tcl/Tk keywords some extra commands, to do specific thinks.

Project

GiD_Project batchfile | set | view | transform_problemtype

Command including a collection of subcommands, most of them replacing deprecated commands attached to the 'togl' object (Tk OpenGL widget)

GiD_Project batchfile

To handle the batch file where processed commands are recorded

GiD_Project set <option> ?<value>?

To set on off or ask the current state of some options, to control the redraw and wait state of GiD:

option could be

disable_graphics|disable_windows|disable_warnline|disable_graphinput|disable_graphinterp|disable_writebatch|waitstate|windows_layout
value 0|1

GiD_Project set windows_layout value <1 | 2LR | 2UD | 3L | 3U | 3R | 3D | 4 | EXT>

GiD_Project view <option> ?<value>?

To handle view parameters.

GiD_Project view clip_planes_x ?{left right}?

GiD_Project view clip_planes_y ?{top bottom}?

GiD_Project view clip_planes_z ?{near far}?

GiD_Project view clip_planes_margin ?value?

GiD_Project view rotation_vector ?{x y z}?

GiD_Project view rotation_factor ?value?

GiD_Project view rotation_matrix ?{v11 ... v44}?

GiD_Project view rotation_center ?{x y z}?

GiD_Project view perspective_distance ?value?

GiD_Project view perspective_ortho_near ?value?

GiD_Project view perspective_ortho_far ?value?

GiD_Project view perspective_factor ?value?

GiD_Project view perspective_view ?value?

GiD_Project view mode ?GEOMETRYUSE|MESHUSE|POSTUSE|GRAPHUSE?

GiD_Project view render_mode ?value?

GiD_Project view lighth_vector ?{x y z ?w?}?

GiD_Project transform_problemtype <new_problemtype>

To update a model of a problemtype version to the new version

backup

GiD_Project backup

To allow save/read the model in a backup fast format

GiD_Project backup save <name>

To save as <name> the model.

GiD_Project backup read <name>

To read the model <name>, saved with backup-fast format

batchfile

GiD_Project batchfile

To handle the batch file where processed commands are recorded

GiD_Project batchfile get name

It returns the file name of the batch, associated to the current model

GiD_Project batchfile flush

To force flush in the file (e.g. to read its updated contents)

set**GiD_Project set <option> ?<0|1>?**

To set on off or ask the current state of some options, to control the redraw and wait state of GiD:
 <option> could be

disable_graphics|disable_windows|disable_warnline|disable_graphinput|disable_writebatch|waitstate|windows_layout

GiD_Project set disable_graphics ?<0|1>?

The value 0/1 Enable/Disable Graphics (GiD does not redraw)

EXAMPLE to disable the redraw:

GiD_Project set disable_graphics 1

GiD_Project set disable_windows ?<0|1>?

The value 0/1 Enable/Disable Windows (GiD displays, or not, windows which require interaction with the user)

EXAMPLE to disable the interaction windows:

GiD_Project set disable_windows 1

GiD_Project set disable_warnline ?<0|1>?

The value 0/1 Enable/Disable printing messages in the lower messages bar.

GiD_Project set disable_graphinput ?<0|1>?

The value 0/1 Enable/Disable GraphInput (enable or disable peripherals: mouse, keyboard, ...)

EXAMPLE to disable the peripherals input:

GiD_Project set disable_graphinput 1

GiD_Project set disable_writebatch ?<0|1>?

The value 0/1 Enable/Disable writing the batch file that records the commands send to be processed.

GiD_Project set waitstate ?<0|1>?

The value 0/1 Enable/Disable the Wait state (GiD displays a hourglass cursor in wait state)

EXAMPLE to set the state to wait:

GiD_Project set waitstate 1

Note: Usually these command are used jointly, to temporary disable redraws to accelerate the process.

It is recommended for a Tcl developer to use the more 'user-friendly' procedures defined inside the file 'dev_kit.tcl' (located in the \scripts directory). For example, to disable and enable redraws, you can use:

::GidUtils::DisableGraphics

::GidUtils::EnableGraphics

GiD_Project set windows_layout ?<1 2LR 2UD 3L 3U 3R 3D 4>?

To set or ask the layout of the drawing windows

1-> 1 window

2LR -> 2 windows placed horizontally (Left-Right)

2UD -> 2 windows placed vertically (Up - Down)

3L -> 3 windows, the biggest on the Left

3U-> 3 windows, the biggest Up

3R -> 3 windows, the biggest on the Right

3D -> 3 windows, the biggest Down

4 -> 4 windows

view

GiD_Project view <option> ?<value>?

To handle view parameters

<option> could be

clip_planes_x|clip_planes_y|clip_planes_z|clip_planes_margin|rotation_vector|rotation_factor|rotation_matrix|rotation_center|perspective_distance|perspective_ortho_near|perspective_ortho_far|perspective_factor|perspective_view|view_mode|render_mode|light_vector

if <value> is provided then the value is set, else the current values are get

GiD_Project view clip_planes_x ?{left right}?

Left and right clip planes, real values

GiD_Project view clip_planes_y ?{top bottom}?

Top and bottom clip planes, real values

GiD_Project view clip_planes_z ?{near far}?

Near and far clip planes

GiD_Project view clip_planes_margin ?value?

Margin between view and model box

GiD_Project view rotation_vector ?{x y z}?

Rotation vector

GiD_Project view rotation_factor ?value?

Rotation factor

GiD_Project view rotation_matrix ?{v11 ... v44}?

Rotation matrix (4x4)

GiD_Project view rotation_center ?{x y z}?

Rotation center

GiD_Project view perspective_distance ?value?

Perspective distance

GiD_Project view perspective_ortho_near ?value?

Perspective near plane

GiD_Project view perspective_ortho_far ?value?

Perspective far plane

GiD_Project view perspective_factor ?value?

Perspective factor

GiD_Project view perspective_view ?0|1?

1 if perspective conical visualization mode is active, 0 if false

GiD_Project view mode ?GEOMETRYUSE|MESHUSE|POSTUSE|GRAPHUSE?

Current view mode

GiD_Project view render_mode ?value?

Current render mode (integer)

GiD_Project view lighth_vector ?{x y z ?w?}

Light direction.

If w is missing the is defaulted to 0.0 and the lighth will be directional

transform_problemtype

GiD_Project transform_problemtype <new_problemtype>

To invoke a transform to update the fields of conditions, materials, etc. of a model saved with a different problemtype version to the current problemtype version.

<new_problemtype> is the name of the new problemtype to be transformed (assumed very similar to the old problemtype to be successful)

Geometry

GiD_Geometry -v2 ?-no_model? create|delete|get|list|edit point|line|surface|volume <num>|append <data>

To create, delete, get data or list the identifiers of geometric entities:

- <num>|append: <num> is the entity identifier (integer > 0). You can use the word 'append' to set a new number automatically.
- <data>: is all the geometric definition data (**create**) or a selection specification (**delete**, **get** or **list**):

-v2 mean version 2 of this command (deprecated version 1 documentation must be seen in help of old versions of the program), the flag really is optional and if missing version 1 is assumed.

If -no_model flag is specified then entities are stored in a special container, it doesn't belong to the model

create: to make new geometric entities

- GiD_Geometry create volume <num>|append volume|contactvolume <layer> {surface1...surfacen} {verso1...version} ? <transformation_matrix>?

for contactvolume is necessary to specify the <transformation_matrix> : a vector of 16 reals representing a 4x4 transformation matrix that maps surface1 into surface2

- GiD_Geometry create surface <num>|append planarsurface|nurbsurface|coonsurface|meshsurface|contactsurface <layer> {line1...linen} {verso1...version} <geometrical_data>

<geometrical data> depends of each entity type (see get command)

- GiD_Geometry create line <num>|append stline|nurbsline|arcline <layer> <inipoint> <endpoint> <geometrical_data>

Instead the NURBS parameters is possible to create a curve that interpolates a list of points (also tangents at start and end can be specified)

- GiD_Geometry -v2 create line <num> | append nurbsline <layer> <inipoint> <endpoint> {-interpolate {p1_x p1_y p1_z ... pn_x pn_y pn_z} ?-tangents {t0_x t0_y t0_z} {t1_x t1_y t1_z}??}
- GiD_Geometry create line <num>|append stline <layer> <inipoint> <endpoint>
- GiD_Geometry create point <num>|append <layer> <point_x> <point_y> <point_z>

delete: to erase model entities

- GiD_Geometry delete point|line|surface|volume <args>

with <args>: num numa:numb numa: layer:layer_name

get: to obtain all the geometrical data to define a single entity

- GiD_Geometry -v2 get point|line|surface|volume <args>

with <args>: num ?line_uv <line_index> | index_boundaries | has_holes | render_mesh | mesh | material?

line_uv <line_index> extra arguments must be only used in case of nurbs surfaces, to get the information of the <line_index> curve (integer from 1 to the number of trimming curves) on the surface, defined in its uv space parameter.

index_boundaries extra argument to get an objarray of integers with the index of the list of curves where each boundary start.

If the surface doesn't has any hole, it return 0 (the start index of the outer loop)

It the surface has holes it return an index by hole (the start index or the hole inner loop)

has_holes: it return 0 if the surface doesn't has any hole, 1 else

-v2 mean version 2 of this command and then return this data (deprecated version 1 documentation must be seen in help of old versions of the program)

- GiD_Geometry get point <num>

will return:

<layer> <geometrical data>

<layer> is the layer name

<geometrical data> the coordinates x y z

- GiD_Geometry get line <num>

will return:

<type> <layer> <p1> <p2> <geometrical data>

<type> can be: stline, nurbsline, arcline, polyline

<layer> is the layer name

<p1> identifier of start point
 <p2> identifier of end point
 <geometrical data> item depends of each entity type
 stline: nothing

nurbline: <d> <n> {x y z ... x y z} {w ... w} {k ... k}
 <d> degree of the polynomials
 <n> number of control points
 <xi yi zi> control points coordinates (objarray of 3*n double values)
 <wi> are the weights associated to each control point (objarray of double values. Empty array if is non-rational)
 <ki> knots (the amount of knots = amount of control points+degree+1)

arcline: <xc> <yc> <r> <sa> <ea> {m11 ... m44}
 <xc> <yc> 2D center
 <r> radius
 <sa> <ea> start and end angle (rad)
 {m11 ... m44} transformation 4x4 matrix (the identity for a 2D case)
 m11 ... m33 is a rotation 3x3 matrix
 m14 ... m34 is a translation 3x1 vector
 m44 is an scale factor
 m41 ... m43 must be 0

polyline: <line ... line> <orientation ... orientation>
 <line ...> is a list with the data of each sub-line of the polyline
 •<orientation ... orientation> is an objarray of values 0 or 1 (0==natural orientation, along tangent, 1== opposite direction)

GiD_Geometry get line <line_id> render_mesh

Will return the information of the render mesh of the line <line_id> as a list: {element_type element_num_nodes coordinates connectivities ts}
 element_type: line
 element_num_nodes: 2
 coordinates: objarray with 3*num_nodes items of float with x y z of the render mesh nodes.
 connectivities: objarray with element_num_nodes*num_elements of int with the connectivities of the elements (zero-based)
 ts: optional objarray with num_nodes items of float with t space parameters (from 0.0 to 1.0) of each node. (it is optional, the array could have zero length)

•GiD_Geometry get surface <num>

will return:
 <type> <layer> <trimmed> {l1 ... ln} {o1 ... on} <geometrical data>
 <type> can be: nurbsurface planarsurface coonsurface meshsurface
 <layer> is the layer name
 <trimmed> 1 if the surface valid part is a trim of a bigger underlying shape, 0 else
 {li...} objarray of integer identifiers of the surface lines (outer and inner boundaries)
 {o1 ... on} orientation of the lines for the surface (0==natural orientation, along tangent, 1== opposite direction)
 Note: turning left of a line with orientation 0 must points inside the surface.
 <geometrical data> depends of each entity type
 planarsurface: nothing

coonsurface: nothing

nurbssurface <du> <dv> <nu> <nv> {x y z ... x y z} {w ... w} {ku ... ku} {kv ... kv}
 <du> <dv> degree in u, v direction
 <nu> <nv> number of control points in each direction
 {xi yi zi} control points coordinates. (objarray of 3*nu*nv double values)
 <wi> are the weights associated to each control point (objarray of double values. Empty array if is non-rational)
 <ku> <kv> knots in each direction

meshsurface: <nnode> {x1 y1 z1 ... xnn ynn znn} {a1 b1 c1 ?d1? ... ane bne cne ?dne?}
 nnode: number of nodes by element: 3 or 4 (triangles or quadrilaterals)
 xi yi zi: coordinates
 ai bi ci di: connectivities (di only for quadrilaterals)

•GiD_Geometry get surface <surface_id> line_uv <line_index>

will return the information of the curve in uv space of the surface <surface_id>, with similar format as GiD_Geometry get line <num> in case of a nurbs curve.

•GiD_Geometry get surface <surface_id> has_holes

Return a boolean 1 or 0 indicating if the surface is trimmed with inner holes.

•GiD_Geometry get surface <surface_id> index_boundaries

Return an objarray of int of num_holes+1 items with the index in the list of boundary curves where each loop starts (the first is the outer loop and then the inner loops)

•GiD_Geometry get surface <surface_id> ?-force? render_mesh

Will return the information of the render mesh of the surface <surface_id> as a list: {element_type element_num_nodes coordinates connectivities normals uvs}

use -force flag to create the render mesh if it was not created.

element_type: triangle or quadrilateral

element_num_nodes: 3 or 4

coordinates: objarray with 3*num_nodes items of float with x y z of the render mesh nodes.

connectivities: objarray with element_num_nodes*num_elements of int with the connectivities of the elements (starting by zero)

normals: optional objarray with 3*num_nodes items of float with x y z of the render mesh normals. (it is optional, the array could have zero length)

uvs: optional objarray with 2*num_nodes items of float with u v space parameters of each node. (it is optional, the array could have zero length)

•GiD_Geometry get volume <num>

will return:

<type> <layer> {s1 s_n} {o1 ... o_n}

<type> can be: volume or contactvolume

<layer> is the layer name

{s_i} identifier of surfaces bounding the volume (including holes. the first must be the outer boundary)

{o_i} are its orientation for the volume (0 along to the surface normal, 1 opposite)

Note: the normal of a surface with orientation 0 points inside the volume

•GiD_Geometry get volume <volume_id> has_holes

Return a boolean 1 or 0 indicating if the volume has inner holes.

•GiD_Geometry get volume <volume_id> index_boundaries

Return an objarray of int of num_holes+1 items with the index in the list of boundary surfaces where each shell starts (the first is the outer shell and then the inner shells)

•GiD_Geometry get point|line|surface|volume <id> mesh

Will return the information of the mesh of the geometric entity <id> as a list: {element_type element_num_nodes node_ids coordinates element_ids connectivities ?radius_and_normals?}

element_type: string

element_num_nodes: integer with the amount of nodes of an element (all mesh elements are of same type)

node_ids: objarray of integers with num_nodes items, where num_nodes is the amount of nodes of the elements of this mesh (node id one-based)

coordinates: objarray with 3*num_nodes items of double with x y z of the mesh nodes.

element_ids: objarray of integers with num_elements items (element id one-based)

connectivities: objarray with element_num_nodes*num_elements of int with the connectivities of the elements (one-based)

radius_and_normals: only for sphere and circle elements. objarray with num_elements items of double with the radius of each element or num_elements*4 items for circles, with the radius and the normal 3D vector (normal to the plane of the circle).

•GiD_Geometry get point|line|surface|volume <id> material

Will return an integer with its material index

•GiD_Geometry get point|line|surface|volume <id> label_on

Will return 1 if the label flag is set to on (to be shown), 0 otherwise

•GiD_Geometry get point|line|surface|volume <id> selected

Will return 1 if the selected flag is set to on (to be drawn in red), 0 otherwise

list: to get a list of entity identifiers of a range or inside some layer

•GiD_Geometry list ?<filter_flags>? point|line|surface|volume ?<args>?

<filter_flags> could be: ?-unrendered? ?-higherentity <num_higher>? ?-material <id_material>? ?-layer <layer_name>? ?-plane {a b c d r}?

-unrendered flag is only valid for surface

-higherentity <num_higher> to filter the selection and list only the entities with the amount of parents equal to <num_higher> (integer >=0)

-*material* *<id_material>* to filter the selection and list only the entities with material id equal to *<id_material>* (integer ≥ 0)
 -*plane* *<a b c d r>* to list only the entities with center that match the plane equation $a*x+b*y+c*z+d \leq r$
 -*entity_type* *<types_allowed>* to list only the entities of a type contained in *<types_allowed>*, that must be a list of allowed types ("STLINE | ARCLINE | POLYLINE | NURBLINE | NURBSURFACE | PLSURFACE | COONSURFACE | MESH | SURFACE | CONTACTSURFACE | VOLUME | CONTACTVOLUME")

<args>: *<num>*|*<num_min>*:*<num_max>*

<num_max> could be 'end' to mean the last index
 if *<args>* is not provided it is considered as 1:end, and then all ids are returned

edit: to modify some option

• **GiD_Geometry edit** *point|line|surface|volume* *<num>* *material|label_on|selected* *<value>*

<num> the entity id

<material> to set the material number (value ≥ 0)

<label_on> to set the label flat (value 0 or 1)

<selected> to set the selection flag (value 0 or 1)

Examples:

Creation of a new NURBS surface:

```
GiD_Geometry create surface 1 nurbsurface Layer0 0 {1 4 3 2} {1 1 1 1} \
{1 1 2 2 {0.17799 6.860841 0.0 -8.43042200 6.86084199 0.0 0.17799400 0.938510 0.0 -8.43042 0.938510
0.0} \
{} {0.0 0.0 1.0 1.0} {0.0 0.0 1.0 1.0}}
```

Get the list of points of the layer named 'Layer0':

```
GiD_Geometry list -layer Layer0 point
```

Get the list of surfaces that not belong to any volume:

```
GiD_Geometry list -higherentity 0 surface
```

Get the list of problematic surfaces that couldn't be rendered:

```
GiD_Geometry list -unrendered surface
```

Get the list of lines of type nurblines or arclines

```
GiD_Geometry list -entity_type {nurblines arclines} line
```

Dimension

GiD_Dimension create|delete|edit|get|list <num>|append <data>

To create, delete, get data or list the identifiers of dimensions:

- **<num>|append**: <num> is the entity identifier (integer > 0). You can use the word 'append' to set a new number automatically.
- **<data>**: is all the dimension definition data, depending on each type (**create**) or a selection specification (**delete**, **get** or **list**):

create: to make new dimension

• **GiD_Dimension create <num>|append {<layer> <type> <text> <show_box> <more_data...>}**

<type>: vertex distance angle radius text

<text>: the text to be showed

<show_box>: boolean 0|1 to hide or show a box around the text

• **GiD_Dimension create <num>|append {<layer> vertex <text> <show_box> <p_x p_y p_z> <text_x text_y text_z>}**

<p_x p_y p_z> real coordinates of the vertex

<text_x text_y text_z> real space coordinates where the text is placed

• **GiD_Dimension create <num>|append {<layer> distance <text> <show_box> <p1_x p1_y p1_z> <p2_x p2_y p2_z> <text_x text_y text_z>}**

<p1_x p1_y p1_z> and <p2_x p2_y p2_z> real coordinates of two vertex for a distance

• **GiD_Dimension create <num>|append {<layer> angle <text> <show_box> <vertex_x vertex_y vertex_z> <p1_x p1_y p1_z> <p2_x p2_y p2_z> <text_x text_y text_z>}**

<vertex_x vertex_y vertex_z> real coordinates of the vertex corner for an angle dimension

• **GiD_Dimension create <num>|append {<layer> radius <text> <show_box> <p_arc_x p_arc_y p_arc_z> <center_x center_y center_z> <text_x text_y text_z>}**

<p_arc_x p_arc_y p_arc_z> real coordinates of a point on the arc curve

<center_x center_y center_z> real coordinates of the arc center

• **GiD_Dimension create <num>|append {<layer> text <text> <show_box> <text_x text_y>}**

<text_relative_x text_relative_y> real 2D coordinates of the text in screen, in a range from -1.0 to 1.0 (0.0, 0.0 is the center of the screen)

delete: to erase dimensions

• **GiD_Dimension delete <num>|<numa>:<numb>|layer:<layer_name>**

<num> the integer id of the dimension to be deleted

<numa>:<numb> (the range from <numa> to <numb>, the word 'end' can be used for numb, e.g. use 1:end to delete all)

layer:<layer_name> : to delete the dimensions of this layer

edit: to modify a dimension

• **GiD_Dimension edit <num> text|show_box <new_value>**

<num> the integer id of the dimension to be modified

text <new_text> to modify the text shown by the dimension

show_box 0|1 to show or not the box drawn around the text

get: to obtain all the dimension data to define a single entity

• **GiD_Dimension get <num> ?type|text|show_box?**

<num> the integer id of the dimension to get its data

the kind of data returned depends on the dimension type (see create command)

if the optional argument type, text or show_box is provided instead of all data it is returned only this information.

list: to get a list of dimension identifiers of a range or inside some layer

•GiD_Dimension list ?<filter_flags>? ?<args>?

<filter_flags> could be: ?-layer <layer_name>?

args>: <num>|<num_min>:<num_max>

<num_max> could be 'end' to mean the last index

if <args> is not provided it is considered as 1:end, and then all ids are returned

Examples:

Creation of a new dimension of text with the text "hello word" in the center-top of the screen, inside a layer named Layer0 that must exists

```
set new_id [GiD_Dimension create append {Layer0 text 1 "hello world" {0.0 1.0}}]
```

Get the list of ids of dimensions that belong to the layer Layer0

```
set ids [GiD_Dimension list -layer Layer0]
```

Get the list of all dimensions

```
set all_ids [GiD_Dimension list]
```

Get the information of the dimension id==2

```
GiD_Dimension get 2
```

Delete all dimensions

```
GiD_Dimension delete 1:end
```

Mesh data

GiD_MeshData

size|size_by_chordal_error|size_background_mesh|size_correct|unstructured|mesher|structured|semi_structured|boundary_layer|element_type|mesh_criteria|reset

To assign mesh data to geometrical entities

GiD_MeshData size points|lines|surfaces|volumes <size> <ids>

To assign desired mesh size to geometrical entities

<size>: double, the desired mesh size.

<ids>: objarray of integers with the ids of the geometrical entities to be meshed with this size

GiD_MeshData size_by_chordal_error <min_size> <max_size> <chordal_error>

To assign sizes to the whole model, based on a chordal error (distance from the approximated mesh to shape of the geometry). The values calculated cannot be outside the range [min_size, max_size]

GiD_MeshData size_background_mesh <filename>

To assign the mesh sizes using an auxiliary file with 'background mesh format' (a mesh covering the domain with desired sizes on nodes or elements)

GiD_MeshData size_correct <max_size>

To modify the current assigned sizes to be 'more feasible' (some assigned sizes can be decreased to avoid strong spatial change of sizes)

GiD_MeshData unstructured lines|surfaces|volumes <ids>

To set entities to be meshed unstructuredly (with assigned or general size)

GiD_MeshData mesher surfaces|volumes default|rfast|rsurf|advancingfront|tetgen|octree <ids>

To set entities with the kind of meshing algorithm to be used to mesh them unstructured

GiD_MeshData structured lines|surfaces|volumes ?<ids_surfaces_or_volumes>? num_divisions|num_divisions_by_size|weights <num_divisions>|<size>|<w1> <w2> <ids_lines>

To set entities to be meshed structuredly, setting the number of divisions in some of its lines.

<ids_surfaces_or_volumes>: objarray of integers (only in case of surfaces or volumes)

num_divisions <num_divisions>: integer amount of mesh divisions to be assigned to the lines

num_divisions_by_size <size>: an alternative double value, it is an approximated size that will be converted to an integer number of divisions base on each line length.

weights {<weight_start> <weight_end>}: to set a not uniform size, assigning weights (double value from -1.0 to 1.0) to start and end of the lines.

<ids_lines>: objarray of integers with the ids of the lines to be assigned the number of divisions.

••••GiD_MeshData structured lines num_divisions <num_divisions> <ids_lines>

GiD_MeshData structured lines num_divisions_by_size <size> <ids_lines>

GiD_MeshData structured lines weights {<weight_start> <weight_end>} <ids_lines>

•GiD_MeshData structured surfaces|volumes <ids_surfaces_or_volumes> num_divisions|num_divisions_by_size <num_divisions>|<size> <ids_lines>

••GiD_MeshData semi_structured <num_divisions>|master_surfaces|structured_directions <ids>

GiD_MeshData semi_structured <num_divisions> <ids_volumes>

To set volumes to be meshed semi-structuredly, setting the amount of divisions in the structured direction.

••GiD_MeshData semi_structured master_surfaces <ids_surfaces>

To explicitly set the surfaces that will be meshed first and then extruded to the opposite tap. Before use this command the volumes must be set as semi-structured

••GiD_MeshData semi_structured structured_directions <ids_lines>

To explicitly set direction that define the extrusion direction. Before use this command the volumes must be set as semi-structured.

GiD_MeshData element_type surfaces|volumes <element_type>|default <ids>

To set the type of element to be generated (default to restore its initial value)

GiD_MeshData mesh_criteria default_all|to_be_meshed|to_be_duplicated|force_points|skip <value|force_points_data> points|lines|surfaces|volumes <ids>

To set some mesh criteria to entities: (0=default, 1=no, 2=yes)

•••default_all points|lines|surfaces|volumes <ids>

to_be_meshed 0|1|2 points|lines|surfaces|volumes <ids>

to_be_duplicated 0|1|2 lines|surfaces <ids>

••force_points surfaces|volumes <ids_to_force> points

<ids>skip 0|1|2 point|lines <ids>

GiD_MeshData reset

To reset all meshing information data assigned to the model

GiD_MeshData boundary_layer assign|unassign lines|surfaces|all <parent_ids> {<num_layers> <size_first_layer>} <ids>

To assign or unassign boundary layer mesh data

••GiD_MeshData boundary_layer assign lines|surfaces <parent_ids> {<num_layers> <size_first_layer>} <ids>

To assign the boundary layer mesh data of a selection of lines (2D) or surfaces (3D).

<parent_ids>: objarray of integer ids of the parent entities (surfaces 2D, volumes 3D)

<num_layers>: integer, the desired amount of boundary layers

<size_first_layer>: double, the depth size of the first layer

<ids>: objarray of integer ids of the entities (lines 2D, surfaces 3D)

••GiD_MeshData boundary_layer unassign lines|surfaces <ids>

To unassign the boundary layer mesh data of a selection of lines (2D) or surfaces (3D)

••GiD_MeshData boundary_layer unassign all

To unassign the boundary layer mesh data of the whole model

GidUtils::GetMeshData

This Tcl procedure allow ask the MeshData information of a geometric entity ,with this syntax

GidUtils::GetMeshData <entity_type> <entity_id> ?<key>?

<entity_type>: point line surface volume

<entity_id>: the integer >0 of that identify the entity

<key>: Elemtype IsStructured Meshing size num_divisions weight tops SkipMesh BLMnlevels1 BLMnlevels2 BLMfirst1 BLMfirst2 Mesher

if key is omitted or "" it return all MeshData items, otherwise only the required item

out codes

Elemtype: 0 None 1 Linear 2 Triangle 3 Quadrilateral 4 Tetrahedra 5 Hexahedra 6 Prism 7 Only points 8 Pyramid 9 Sphere 10 Circle

Meshing: No Default Yes No,Duplicate Duplicate

SkipMesh: -1 No 0 Automatic 1 Yes

Mesher: 1 RFast 2 Rsurf 3 DelaunaySurf3 4 AdvancingFront4 5 MinElem 6 "Advancing front" 7 DelaunayVol7 8 Isosurface 9 Tetgen 10 Octree 11 PVolume11

weight: (w1,w2)

tops: semi-structured volumes codification (num_surface_1,num_surface_2),top surface global ids (if negative mean that is also set as master)

e.g. for the surface number 1 that has been set forced to be meshed

GidUtils::GetMeshData surface 1

IsStructured 0 Elemtype 0 size 0 Meshing Yes

GidUtils::GetMeshData surface 1 Meshing

Yes

Mesh

Preprocess mesh

GiD_Mesh create|delete|edit|get|list

To create, delete, modify, list or know information about mesh nodes or elements of the preprocess:

create: to create a new node or element

GiD_Mesh create node <num>|append <x y z>

- <num>|append: <num> is the identifier (integer > 0) for the node. You can use the word 'append' to set a new number automatically. The number of the created entity is returned as the result.
- <x y z> are the node coordinates. If the z coordinate is missing, it is set to z=0.0.

GiD_Mesh create element <num>|append <elemtype> <nnode> <N1 ... Nnnode> <radius> <nx> <ny> <nz> ?<matname>?

- <num>|append: <num> is the identifier (integer > 0) for the node. You can use the word 'append' to set a new number automatically. The number of the created entity is returned as the result.
- <elemtype>: must be one of "Point | Line | Triangle | Quadrilateral | Tetrahedra | Hexahedra | Prism | Pyramid | Sphere | Circle"
- <nnode> is the number of nodes an element has
- <N1 ... Nnnode> is a Tcl list with the element connectivities
- <radius> is the element radius, only for sphere and circle elements
- <nx> <ny> <nz> is the normal of the plane that contain the circle, must be specified for circle elements only
- <matname> is the optional element material name

delete: to delete one or more nodes or elements

GiD_Mesh delete node|element <num_1 ... num_n>

<num> is the identifier (integer > 0) for the node or element to be deleted. It is possible to use a list of multiple ids.

edit: to modify a node or element

GiD_Mesh edit node <num> <x y z>

GiD_Mesh edit element <num> <elemtype> <nnode> <N1 ... Nnnode> <radius> <nx> <ny> <nz> ?<matname>?

Same syntax as create

or

GiD_Mesh edit node|element <num> material|label_on|selected <value>

<num> the entity id

<material> to set the material number (value>=0)

<label_on> to set the label flat (value 0 or 1)

<selected> to set the selection flag (value 0 or 1)

get: to get the information of a node or element

GiD_Mesh get node <num> ?coordinates|material|label_on|selected?

It return the list: <node_layer> <x> <y> <z>

with the extra word

- <coordinates> only the <x> <y> <z> coordinates are returned
- <material>: the material integer id is returned
- <label_on>: 0 or 1 is returned (label flag)
- <selected>: 0 or 1 is returned (selection flag)

GiD_Mesh get element <num>|from_face|from_edge ?

face|face_linear|num_faces|edge_linear|num_edges|normal|tangent|center|connectivities|geometry_source|material|label_on|selected

?<face_id>|<edge_id>??

- <num> is the identifier (integer > 0) for the element to be asked
- <face> optional, instead of the element nodes it returns the nodes of the face, first the linear corner nodes and then the quadratic nodes
- <face_linear> optional, instead of the element nodes it returns only the linear corner nodes, also is the element is quadratic
- <num_faces> returns the amount of faces of the element (for surface elements its edges act as faces)
- <face_id> is the local face index from 1 to the number of faces of the element. If <face_id> is missing then a list with all faces is returned

- **edge_linear** optional, instead of the element nodes it returns only the linear edge nodes, also is the element is quadratic
- **num_edges** returns the amount of edges of the element
- **<edge_id>** is the local edge index from 1 to the number of edges of the element. If **<edge_id>** is missing then a list with all edges is returned
- **normal** return a 3D vector with the normal direction for surface elements (and for line elements in 2D the normal to the tangent)
- **tangent** return a 3D vector with the tangent direction for line elements
- **center** return a 3D vector with the element center
- **connectivities** return a list of integers with the element's nodes
- **geometry_source** return a list where the first item is the category: POINT_LT|LINE_LT|SURFACE_LT|VOLUME_LT and then the integer ids of the geometric entity source of the mesh element (usually one entity, but could be more than one meshing with [Rjump](#))
- **material**: the material integer id is returned
- **label_on**: 0 or 1 is returned (label flag)
- **selected**: 0 or 1 is returned (selection flag)

get element return the list: <element_layer> <elemtype> <nnode> <N1> ... <Nnnode>

get element face|face_linear: <N1_face> ... <Nnnode_face>

get element edge_linear: <N1_edge> <N2_edge>

If **from_face** is specified then the command has this syntax

GiD_Mesh get element from_face <face_nodes> ?-ordered?

it find and return the list of element ids that have a face with these nodes

- **<face_nodes>** is the list of integer ids of the face nodes {<face_node_1> ... <face_node_n>} (only corner lineal nodes must be specified in the list)
- if **-ordered** is specified then only faces with the same orientation of the nodes will be taken into account (else the order of the facenodes doesn't matter)

If **from_edge** is specified then the command has this syntax

GiD_Mesh get element from_edge <edge_nodes>

it find and return the list of element ids that have an edge with these nodes

- **<edge_nodes>** is the list of integer ids of the edge nodes {<edge_node_1> <edge_node_2>} (only corner lineal nodes must be specified in the list)

Note: get element from_face or from_edge could be an expensive operation because the whole mesh is traversed to find them.

GiD_Mesh get nodesdistance <num1> <num2>

returns the distance between two mesh nodes, specified by its numbers

list: to get a list of entity identifiers of a range, filtered with some conditionals

- **GiD_Mesh list ?<filter_flags>? node|element|face ?<num>|<num_min:num_max>?**

<filter_flags> could be: **?-higherentity <num_higher>? ?-material <id_material>? ?-layer <layer_name>? ?-plane {a b c d r}? ?-element_type <types_allowed>?**

-higherentity <num_higher> to filter the selection and list only the entities with the amount of parents equal to **<num_higher>** (integer >=0)

-material <id_material> to filter the selection and list only the entities with material id equal to **<id_material>** (integer >=0)

-layer <layer_name> to filter the selection and list only the entities with layer equal to **<layer_name>**

-plane {a b c d r} to list only the entities with center that match the plane equation $a*x+b*y+c*z+d \leq r$ ($r \geq 0.0$)

-element_type <types_allowed> to list only the elements of a type contained in **<types_allowed>**, that must be a list of allowed types ("Point | Line | Triangle | Quadrilateral | Tetrahedra | Hexahedra | Prism | Pyramid | Sphere | Circle")

if **<num>** or **<num_min:num_max>** are not provided it is considered as **1:end**, and then all ids are returned

In case of face it is returned a value with two lists: element_ids and face_index

Each face is identified by the element_id and the local face_index (from 1 to the number of faces of the element)

the valid flags are

GiD_Mesh list ?-unique? ?-normals_dotprod_threshold <threshold_value> -higherentity|higherentity_not <higherentity> -element_type <list_of_types> face

-unique is to avoid repeat faces shared between multiple elements, only one is arbitrary retained

-normals_dotprod_threshold <threshold_value> is to get only the faces (edges really) that belong to the surface elements with dot product of its normals below the threshold_value (from 1.0 to -1.0). e.g. -0.9 for problematic faces of folded elements. In this case is implicit **-higherentity 2** and **-element_type {triangle quadrilateral}**

-higherentity <higherentity> is to get only faces that belong exactly to **<higherentity>** elements (an integer number > 0)

-higherentity_not <higherentity> is to get only faces that not belong to **<higherentity>** elements

-element_type <list_of_types> is a list of types of elements (faces of other types are ignored). By now only triangle, quadrilateral

Examples:

```
GiD_Mesh create node append {1.5 3.4e2 6.0}  
GiD_Mesh create element 58 triangle 3 {7 15 2} steel  
GiD_Mesh delete element {58 60}  
set triangles_and_quads [GiD_Mesh list -element_type {triangle quadrilateral} element]
```

Cartesian grid

This command is only valid for preprocess

GiD_Cartesian **get|set** **ngridpoints|boxsize|corner|dimension|coordinates|iscartesian|auto_calculated** <values>

To get and set cartesian grid properties

ngridpoints: the number of values of the grid axis on each direction x, y,z (3 integers)

boxsize: the size of the box of the grid on each direction (3 reals)

corner: the location of the lower-left corner of the grid box (3 reals)

dimension: the dimension of the grid: 2 for 2D or 3 for 3D

coordinates: the list of grid coordinates on each direction (nx+ny+nz reals)

iscartesian: (valid only for get) return 1 if current mesh is cartesian, 0 else.

auto_calculated **GEOMETRY|MESH|GEOMETRY_AND_MESH** <mesh_size>: (valid only for set) to fill in the GiD-calculated automatic values, based on the current geometry and/or mesh and the general mesh size to be used.

Postprocess mesh

GiD_MeshPost create <meshname> <elemtype> <elementnnodes> ?-zero_based_array? <node_ids> <nodes> <element_ids> <elements> ?<radius+normals?>? ?<r g b a>?

To create a postprocess mesh.

This command create all mesh nodes and elements in a single step (unlike GiD_Mesh that create each node or element one by one)

- <meshname>: the name of the mesh
- <elemtype>: must be one of "point | line | triangle | quadrilateral | tetrahedra | hexahedra | prism | pyramid | sphere | circle"
- <elementnnodes>: is the number of nodes an element has. All elements of the mesh must have the same number of nodes.
- zero_based_array: optional flag. By default node and element indexes start from 1, but setting this flag indexes must start from 0.
- <node_ids>: list of node indentifiers. If it is an empty list them numeration is implicitly increasing.
- <nodes>: a list of real numbers with the three coordinates of each node $\{x_0 \ y_0 \ z_0 \ \dots \ x_{nn-1} \ y_{nn-1} \ z_{nn-1}\}$
- <element_ids>: list of element indentifiers. If it is an empty list them numeration is implicitly increasing.
- <elements>: a list of integers with the <elementnnodes> nodes of each element: the id of each node is the location on the vector of nodes, starting from 0
- <radius+normals>:
 - <radius>:only for spheres. Is a list of reals with the radius or each sphere $\{r_0 \ \dots \ r_{ne-1}\}$
 - <normals>:only for circles. Is a list of reals with the radius and normal to the plane or each circle $\{r_0 \ nx_0 \ ny_0 \ nz_0 \ \dots \ r_{ne-1} \ nx_{ne-1} \ ny_{ne-1} \ nz_{ne-1}\}$
- <r g b a>: optional color components, to set the mesh color. r g b a are the red, green, blue and alpha transparency components of the color, must be real numbers from 0.0 to 1.0. If the color is not specified, an automatic color will be set.

Tools

GiD_Tools geometry/mesh

This GiD_Tools command pretends to have subcommands of tools specialized in some geometry or mesh tasks

Geometry

classify_connected_parts

GiD_Tools geometry classify_connected_parts <entity_type> <entity_ids> ?<forced_separator_ids>?

To split the input items of <entity_ids> in one or several parts where the entities of the part are connected

Two entities are assumed as connected only if they share a lower entity boundary figure.

e.g lines sharing points, or surfaces sharing lines

but are considered not connected two surfaces that share only one point

if two figures only share a figure contained in <forced_separator_ids> they will be considered in different parts

<entity_type>: line | surface | volume

<entity_ids> objarray of integers with the entity identifiers (ids start from 1, not 0)

<forced_separator_ids> is an optional list of ids of entities of lower level that act as barrier and prevent consider be connected

entities of lower level means for <entity_type> line -> point, surface -> line, volume -> surface

it returns a list where each item is an objarray of integers with the ids of the entities of each connected part

{<part1_1 ... part1_n1> ... <partn_1 ... partn_nn>}

mass_properties

GiD_Tools geometry mass_properties <volume_id>

To calculate the mass properties of volume, gravity center and inertia tensor of a volume

<volume_id> Is the of integer id of the volume to be computed

It returns a list with 3 items: mass {center_x center_y center_z} {Ixx Iyy Izz Ixy Iyz Ixz}

the 6 inertia values describe the symmetric 3x3 inertia tensor

Ixx Ixy Ixz

Ixy Iyy Iyz

Ixz Iyz Izz

Note: this command is similar to GiD_Info listmassproperties, but this command is not computing inertias

Mesh

mass_properties

GiD_Tools mesh mass_properties <tetrahedra_ids> | -boundary_elements <triangle_ids>

To calculate the mass properties of volume, gravity center and inertia tensor of a volume, defined by a selection of tetrahedra or the selection of the triangles bounding the volume.

<tetrahedra_ids> A list of integer ids of the tetrahedra of the volume to be computed

<triangle_ids> A list of integer ids of the triangles that enclose a volume, with normals pointing inside.

It returns a list with 3 items: mass {center_x center_y center_z} {Ixx Iyy Izz Ixy Iyz Ixz}
the 6 inertia values describe the symmetric 3x3 inertia tensor

Ixx Ixy Ixz

Ixy Iyy Iyz

Ixz Iyz Izz

Note: this command is similar to GiD_Info listmassproperties, but this command is not computing inertias

intersectvolumeslines

GiD_Tools mesh intersectvolumeslines <group_lines>

Specialized command that calculate the intersections of the line elements that belong to the group named <group_lines> with the current mesh of hexahedra.

Layers

Definition

•GiD_Layers

create|delete|edit|get|list|window|exists|is_forbidden_name

GiD_Layers create <layer>

To create a new layer. <layer> must be the full name (e.g. if a layer B has as parent A then must use as fullname A/B)

•GiD_Layers delete <layer>

To delete a layer

•GiD_Layers edit name|color|opaque|visible|frozen|parent|state|to_use <layer> <value>

To modify layer properties:

name: change its name

color: set the color to draw its entities (with format #rrggbbaa)

opaque: opaque or transparent (0 or 1)

visible: set visibility of its entities (0 or 1)

frozen: set frozen to disable select its entities (0 or 1)

parent: to change the parent of a layer

state: to change the layer state (normal, disabled or hidden). hidden layers are not listed or visible in windows.

to_use: in this case <value> must not be provided, and <layer> is the one to be set as current 'layer to use' (where new entities will be created)

•GiD_Layers get color|opaque|visible|frozen|parent|state|num_entities|num_conditions|id|back|to_use|all_properties <layer>

To obtain the current value of some property:

num_entities: the total number of geometric or mesh entities that belong to the layer

num_conditions: the total number of conditions applied to the layer

id: the numeric identifier of the layer

back: return 1 if the layer has entities in its 'back' layer (entities in back are not drawn until they are sent again to front)

to_use: in this case <layer> must not be provided, it is returned the current 'layer to use' (where new entities will be created)

all_properties: return a list of property value with the properties of the layer

•GiD_Layers list ?<parent>? ?descendants?

To get the list of fullnames of the current layers.

If a parent is specified, then only relative names of child layers will be listed. Root parent could be specified with an empty string ""

If descendants is specified return a list of all descendants (childs, childs of childs, ...)

•GiD_Layers window open|close|update

Show or hide the layers window or update its content

•GiD_Layers exists <layer>

Return 1 if layer exists, 0 else

•GiD_Layers is_forbidden_name <layer>

Return 1 if layer name has forbidden syntax

Entities

GiD_EntitiesLayers assign|get|entity_layer

To handle the entities that belong to layers

•GiD_EntitiesLayers assign|assign_back_layer|assign_front_layer|get <layer> ?-also_lower_entities? <over> <selection>

To add or know entities of a layer

•GiD_EntitiesLayers assign <layer> ?-also_lower_entities? ?-also_higher_entities? <over> <selection>

To assign the selection of entities of kind over to the layer

<layer> is the full name of the layer

<-also_lower_entities> is an optional flag, to select also all lower entities of the selected ones (e.g. curves and points of the selected surfaces)

<-also_higher_entities> is an optional flag, to select also all higher entities of the selected ones (e.g. volumes of the selected surfaces)

<over> could be points, lines, surfaces, volumes, nodes, elements, all_geometry, all_mesh

<selection> is a list of integer entity id's starting from 1.

In case of all_geometry is expected a list with 4 items with the list of ids of points, lines, surfaces and volumes.

In case of all_mesh is expected a list with 2 items with the list of ids of nodes and elements respectively

•GiD_EntitiesLayers assign_back_layer ?-also_lower_entities? ?-also_higher_entities? <over> <selection>

To send the selection of entities of kind over to the back (hidden part) of its layer

<-also_lower_entities> is an optional flag, to select also all lower entities of the selected ones (e.g. curves and points of the selected surfaces)

<-also_higher_entities> is an optional flag, to select also all higher entities of the selected ones (e.g. volumes of the selected surfaces)

<over> could be points, lines, surfaces, volumes, nodes, elements, all_geometry, all_mesh

<selection> is a list of integer entity id's starting from 1.

In case of all_geometry is expected a list with 4 items with the list of ids of points, lines, surfaces and volumes.

In case of all_mesh is expected a list with 2 items with the list of ids of nodes and elements respectively

•GiD_EntitiesLayers assign_front_layer ?-also_lower_entities? ?-also_higher_entities? geometry|mesh
all_entities|layer_entities|<over> <layer>|<selection>

To send the entities of <layer> again to the front (visible part) of its layer

<-also_lower_entities> is an optional flag, to select also all lower entities of the selected ones (e.g. curves and points of the selected surfaces)

<-also_higher_entities> is an optional flag, to select also all higher entities of the selected ones (e.g. volumes of the selected surfaces)

geometry|mesh specify wich layer entities must be sent to front: geometry or mesh entities

all_entities: will send all entities of the geometry or mesh. In this case <layer> must not be specified

layer_entities <layer>: will send to front only the geometry or mesh entities of this layer.

<over> <selection>: will send the selection of entities of type <over> (points, lines, surfaces, volumes, nodes or elements)

•GiD_EntitiesLayers get <layer> <over> ?-count? ?-element_type <types_allowed>?

To get the list of entities of kind <over> that belong to <layer>.

If <over> is all_geometry then is obtained a list with 4 sublists: point id's, line id's, surface id's and volume id's

If <over> is all_mesh then is obtained a list with 2 sublists: node id's and element id's

if -count is specified, then only the number of objects is returned instead of its list.

if -element_type <types_allowed> is specified then only the types of elements listed in <types_allowed> will be taken into account.

<types_allowed> must be a list of triangle quadrilateral, etc.

In fact it is returned an 'objarray': a Tcl_Obj object specialized for arrays, implemented as a Tcl package named 'objarray'. (for more information see scripts\objarray\objarray.pdf)

•GiD_EntitiesLayers entity_layer <over> <id>

To get the layer to which the entity <id> of type <over> belongs

<over> could be points, lines, surfaces, volumes, nodes, elements

<id> is the entity number, starting from 1.

Groups

Definition

•**GiD_Groups create|clone|delete|edit|get|list|window|exists|is_forbidden_name|draw|end_draw**

GiD_Groups create <group>

To create a new group. <group> must be the full name (e.g. if a group B has as parent A then must use as fullname A/B)

•**GiD_Groups clone <source_group> <destination_group>**

create a new cloned group, with the same entities that its source group.

<source_group> must be the full name of an existing group to be duplicated.

<destination_group> must be the tail name of a non existing group. The group parent will be the same as the source. In case of require other parent must be changed after with GiD_Groups edit parent

•**GiD_Groups delete <group>**

To delete a group

•**GiD_Groups edit name|color|opaque|visible|allowed_types|allowed_element_types|parent|state <group> <value>**

To modify group properties:

name: change its name

color: set the color to draw its entities (with format #rrggbbaa)

opaque: opaque (1) or transparent (0)

visible: set visibility of its entities. (Visible = 1, Hidden = 0)

allowed_types: set the list type of geometric or mesh entities allowed to be in the group, must be a list with some of {points lines surfaces volumes nodes elements faces}

allowed_element_types: set the list type of mesh elements allowed to be in the group, must be a list with some of {linear triangle quadrilateral, ... }, by default all element types are allowed

parent: to change the parent of a group

state: to change the groups state (normal, disabled or hidden). hidden groups are not listed or visible in windows.

•**GiD_Groups get**

color|opaque|visible|allowed_types|allowed_element_types|parent|state|num_entities|num_conditions|id|all_properties <group>

To obtain the current value of some property:

num_entities: the total number of geometric or mesh entities that belong to the group

num_conditions: the total number of conditions applied to the group

id: the numeric identifier of the group

all_properties: return a list of property value with the properties of the layer

•**GiD_Groups list ?<parent>?**

To get the list of fullnames of the current groups.

If a parent is specified, then only relative names of child groups will be listed. Root parent could be specified with an empty string ""

•**GiD_Groups window open|close|update**

Show or hide the groups window or update its content

•**GiD_Groups exists <group>**

Return 1 if group exists, 0 else

•**GiD_Groups is_forbidden_name <group>**

Return 1 if group name has forbidden syntax

•**GiD_Groups draw {<group_1>...<group_n>}**

Starts drawing the specified groups

•**GiD_Groups end_draw**

Finish drawing groups.

Entities

GiD_EntitiesGroups assign|unassign|get|entity_groups

To handle the entities that belong to groups

Note: GiD_WriteCalculationFile could be interesting to tranverse an print data based on groups without the extra cost of GiD_EntitiesGroups serializing potentially big lists of entities.

•GiD_EntitiesGroups assign|unassign|get <group> ?-also_lower_entities? <over> <selection>

To add, remove or know entities of a group

•GiD_EntitiesGroups assign <group> ?-also_lower_entities? <over> <selection>

To assing the selection of entities of kind over to the group

<group> is the full name of the group

<-also_lower_entities> is an optional flag, to select also all lower entities of the selected ones (e.g. curves and points of the selected surfaces)

<over> could be points, lines, surfaces, volumes, nodes, elements, faces, all_geometry, all_mesh

<selection> is a list of integer entity id's starting from 1.

In case of faces it is a list with 2 items, the first is the list of element id's and the second the list of face id's (the local number of the face on the element: a number from 1 to nfaces of the element)

In case of all_geometry is expected a list with 4 items with the list of ids of points, lines, surfaces and volumes.

In case of all_mesh is expected a list with 3 items with the list of ids of nodes, elements and faces, and for faces there are two subitems {element_ids face_ids}

•GiD_EntitiesGroups unassign <group> ?-also_lower_entities? ?-element_type <types_allowed>? <over> ? <selection>?

To unassign the selection of entities of kind over of the group.

If -element_type <types_allowed> is specified then only the types of elements listed in <types_allowed> will be taken into account.

<types_allowed> must be a list of triangle quadrilateral, etc.

If <selection> is missing, then all entities of kind <over> are unassigned of <group>

•GiD_EntitiesGroups unassign all_geometry|all_mesh|all

all_geometry: To unassign all groups of all geometric entities

all_mesh: To unassign all groups of all mesh entities

all: To unassign all groups of all entities

•GiD_EntitiesGroups get <group> <over> ?-count? ?-element_type <types_allowed>? ?-visible?

To get the list of entities of kind <over> that belong to <group>.

If <over> is faces then is obtained a list with 2 sublists: element id's and face id's

If <over> is all_geometry then is obtained a list with 4 sublists: point id's, line id's, surface id's and volume id's

If <over> is all_mesh then is obtained a list with 3 sublists: node id's, element id's, face id's (and face id's is a list with 2 items: element id's and face id's)

if -count is specified, then only the number of objects is returned instead of its list.

if -element_type <types_allowed> is specified then only the types of elements listed in <types_allowed> will be taken into account.

<types_allowed> must be a list of triangle quadrilateral, etc.

if -visible is specified, then only the entities visible must be taken into account

In fact it is returned an 'objarray': a Tcl_Obj object specialized for arrays, implemented as a Tcl package named 'objarray'. (for more information see scripts\objarray\objarray.pdf)

•GiD_EntitiesGroups entity_groups <over> <id>

To get the list of groups to which the entity <id> of type <over> belongs

<over> could be points, lines, surfaces, volumes, nodes, elements, faces

<id> is the entity number, starting from 1. In case of faces it is a list with tho items: {<element_id> <face_id>}, with <face_id> starting from 1

Data (problemtype classic)

GiD-Tcl special commands to manage books, materials, conditions, intervals, general data or local axes:

Books**GiD_Book material|condition create|set|exists**

To create or know if a book of materials or conditions exists, or to set its current book.

Books are like a container to visually separate materials or conditions in submenus and different windows

•GiD_Book material|condition create <book>

To create a new book named **<book>** in the collection of books of materials or conditions

•GiD_Book material|condition set <book> <name>

•To set as **<book>** as current book of a material or condition named **<name>**

GiD_Book material|condition exists <book>

To check if the book **<book>** exists in the collection of books of materials or conditions

CreateData

GiD-Tcl special commands to create and delete materials and conditions:

GiD_CreateData create|delete material|material_base|condition ...

To create or delete materials or conditions:

GiD_CreateData create material <basename> <name> <values>

To create a material with the same question fields as a base material but different values:

- <basename> this only applies to the **create material** operation, and is the base material from which the new material is derived;
- <name> is the name of material itself;
- <values> is a list of all field values for the new material.

GiD_CreateData delete material|material_base <name>

To delete a material

GiD_CreateData create material_base <name> {{question_1 ... question_n} ?{value_1 value_n}??}

To create a base material: (define the fields of a new material to be used to derive new materials from it)

- <name> is the name of material;
- <questions> and <values> are lists of all questions and values for the new material. If values are missing empty values are used.

GiD_CreateData create condition <name> <over_geometry> <over_mesh> {{question_1 ... question_n} ?{value_1 value_n}??}

To create a condition:

- <over_geometry> must be **over_point|over_line|over_surface|over_volume|over_layer|over_group**
- <over_mesh> must be **over_node|over_element|over_face**

GiD_CreateData delete condition <name>

To delete a condition:

Example:

```
set id_material [GiD_CreateData create material Steel Aluminium {3.5 4 0.2}]
GiD_CreateData delete material Aluminium
set id_condition [GiD_CreateData create condition surface_pressure over_surface over_face {{pressure}
{0.0}}]
GiD_CreateData delete condition surface_pressure
```

AssignData

GiD_AssignData material|condition <name> <over> ?<values>? <entities>

To assign materials or conditions over entities:

- <name> is the name of the material or condition. In case of material it is allowed a numeric index (0 for not assigned material)
- <over> must be: points, lines, surfaces, volumes, layers, groups, nodes, elements, body_elements, or face_elements (elements is equivalent to body_elements). Layers and groups is valid for conditions defined over them, but not for materials.
- <values> is only required for conditions. If it is set to "" then the default values are used;
- <entities> a list of entities (it is valid to use ranges as a:b ,can use "all" to select everything, "end" to specify the last entity, layer: <layername> to select the entities in this layer) ; if <over> is face_elements then you must specify a list of "entity numface" instead just "entity". (numface starting from 1)

Example:

```
GiD_AssignData material Steel Surfaces {2:end}
GiD_AssignData condition Point-Load Nodes {3.5 2.1 8.0} all
GiD_AssignData condition Face-Load face_elements {3.5 2.1 8.0} {15 1 18 1 20 2}
```

UnAssignData

GiD_UnAssignData material|condition <name> <over> <entities> ?wherefield <fieldname> <fieldvalue>?

To unassign materials or conditions of some entities:

- <name> is the name of the material or condition; Can use "*" to match all materials
- <over> must be: points, lines, surfaces, volumes, layers, nodes, elements, body_elements, or face_elements (elements is equivalent to body_elements);

It is possible to use all_geometry|all_mesh||all to unassign of all entities of geometry or mesh or both. Then <entities selection> must not be provided.

- <entities> a list of entities (it is valid to use ranges as a:b ,can use "all" to select everything, "end" to specify the last entity, layer: <layername> to select the entities in this layer) ; if <over> is face_elements then you must specify a list of "entitynumface" instead just "entity".
- wherefield <fieldname> <fieldvalue> To unassign this condition only for the entities where the field named 'fieldname' has the value 'fieldvalue'

Example:

```
GiD_UnAssignData material * surfaces {end-5:end}
GiD_UnAssignData condition Point-Load nodes layer:Layer0
GiD_UnAssignData condition Face-Load face_elements {15 1 18 1 20 2}
```

AccessValueAssignedCondition

GiD_AccessValueAssignedCondition ?-field_index? set|get <condition> <over> {<question>|<field_index> ?<value>? ... <question>|<field_index> ?<value>?} <entities>

To get or set field values of conditions applied over entities:

- <condition> is the name of the condition;
- <over> must be: points, lines, surfaces, volumes, layers, groups, nodes, elements, body_elements, or face_elements (elements is equivalent to body_elements).
- <question> is the field name to be changed
- <field_index> is the field number, starting from 0 (if -field_index was used)
- <value> is the new value to be set
- <entities> a list of entities (it is valid to use ranges as a:b ,can use "all" to select everything, "end" to specify the last entity, layer: <layername> to select the entities in this layer) ; if <over> is face_elements then you must specify a list of "entity numface" instead just "entity". (numface starting from 1)

Example:

```
GiD_AccessValueAssignedCondition -field_index set Point_BC points {0 5sec} {2}
```


ModifyData

GiD_ModifyData ?-book? material|condition|intvdata|gendata|localaxes ?<name>? <values>

To change all field values of materials, interval data or general data:

- <name> is the material name or interval number;
- <values> is a list of all the new field values for the material, interval data or general data.

if **-book** is specified then this value is the new book name, and could be applied only to **material** or **condition**

Example:

GiD_ModifyData material Steel {2.1e6 0.3 7800}

GiD_ModifyData intvdata 1 ...

GiD_ModifyData gendata ...

GiD_ModifyData -book material Steel my_new_book

GiD_ModifyData localaxes <condition_name> ?geometry|mesh? <entity_ids...> <entiy_euler_angles...>

To change the 3 euler angles of the local axis of a condition <condition_name> attached to entities of geometry or mesh.

- <condition_name> is the name of the condition (local axis are associated to a condition with an special #LA# field)
- <entity_ids...> is a list (objarray) of integers with the ids of the entities (of the type as the condition was defined over geometry or mesh)
- <entiy_euler_angles...> is a list (objarray) of reals with the consecutive 3 values to set to each entity of the list.

The amount of angles must 3*amount of ids

AccessValue

GiD_AccessValue ?-index? set|get ?-default? material|condition|intvdata|gendata ?<name>? <question> ?<attribute>? <value>

To get or set some field values of materials, interval data or general data:

if **-index** is specified then the material or condition number (starting from 1) will be expected instead of its name

if **-default** is specified then the get option returns the default value instead of the current value

(the default value is value set in the problemtype file)

- <name> is the material, condition name or interval number (not necessary for gendata);
- <question> is a field name;
- <attribute> is the attribute name to be changed (STATE, HELP, etc.) instead of the field value;
- <value> is the new field or attribute value.

Example:

```
GiD_AccessValue set gendata Solver Direct
set mass [GiD_AccessValue -index get material $material_id mass]
set default_value [GiD_AccessValue get -default gendata $question]
```

IntervalData

GiD_IntervalData <mode> ?<number>? ?copyconditions?

To create, delete or set interval data;

- <mode> must be 'create', 'delete' or 'set';
- <number> is the interval number (integer >=1).

Create returns the number of the newly created interval and can optionally use 'copyconditions' to copy to the new interval the conditions of the current one.

For **create** mode, if <number> is supplied the new interval is inserted in this location, else is append to end.

For **set** mode, if <number> is not supplied, the current interval number is returned.

Example:

```
set current [GiD_IntervalData set]
GiD_IntervalData set 2
set newnum [GiD_IntervalData create]
set newnum [GiD_IntervalData create copyconditions]
set newnum [GiD_IntervalData create $i_insert copyconditions]
```

LocalAxes

GiD_LocalAxes <mode> <name> ?<type>? <Cx Cy Cz> <PAxex PAxey PAxex> <PPlanex PPlaney PPlanez>?

To create, delete or modify local axes:

- <mode>: must be one of "create|delete|edit|exists", which correspond to the operations: create, delete, edit or exists;
- <name>: is the name of local axes to be created or deleted;
- <type>: must be one of "rectangular|cylindrical|spherical C_XZ_Z|C_XY_X". Currently, GiD only supports rectangular axes. C_XZ_Z is an optional word to specify one point over the XZ plane and another over the Z axis (default). C_XY_X is an optional word to specify one point over the XY plane and another over the X axis;
- <Cx Cy Cz> is a Tcl list with the real coordinates of the local axes origin;
- <PAxex PAxey PAxex> is a Tcl list with the coordinates of a point located over the Z' local axis (where Z' is positive). The coordinates must be separated by a space. If the z coordinate is missing, it is set to z=0.0;
- <PPlanex PPlaney PPlanez> is a Tcl list with the coordinates of a point located over the Z'X' half-plane (where X' is positive).

For the 'exists' operation, if only the <name> field is specified, 1 is returned if this name exists, and 0 if it does not. If the other values are also specified, <name> is ignored.

The value returned is:

- 1 if the global axes match;
- 2 if the automatic local axes match;
- 3 if the automatic alternative local axes match;
- 0 if it does not match with any axes;
- <n> if the user-defined number <n> (n>0) local axes match.

Example:

```
GiD_LocalAxes create "axes_1" rectangular C_XY_X {0 0 0} {0 1 0} {1 0 0}
GiD_LocalAxes delete axes_1
GiD_LocalAxes exists axes_1
GiD_LocalAxes exists "" rectangular C_XY_X {0 0 0} {0 1 0} {1 0 0}
```

this last sample returns -1 (equivalent to global axis)

Units

GiD_Units edit|get magnitude_units|magnitudes|model_unit_length|system ?<value>?

To allow get or modify units and magnitudes.

Note: units doesn't exist without load a problemtype that define them

•**GiD_Units get magnitude_units <magnitude>**

Returns the list of allowed units for the <magnitude> (of the current unit system)
<magnitude> must be one of the values returned by [GiD_Units get magnitudes]

•**GiD_Units get magnitudes**

Returns the list of defined magnitudes

•**GiD_Units get system**

To return the current units system string

e.g.

GiD_Units get system

-> SI

•**GiD_Units edit system <value>**

To set the current units system

e.g.

GiD_Units edit system imperial

•**GiD_Units get model_unit_length**

To return the current geometry length unit string.

This unit declare the length unit of the coordinates of the geometry or mesh

e.g.

GiD_Units get model_unit_length

-> m

•**GiD_Units edit model_unit_length <value>**

To set the current geometry length unit.

<value> must be one of the values returned by [GiD_Units get magnitude_units <length_magnitude>]

Where <length_magnitude> must be the length magnitude name. This name depends on the problemtype units definition (L, LENGTH, or others)

e.g.

GiD_Units edit model_unit_length mm

Local axis tools

Special GiD commands related to [local axis](#)

•gid_groups_conds::local_axes_window

This Tcl procedure open a window to handle local axis assigned to entities

•correct_local_axes_with_lines nodes|elements <groupList> <groupLinesList>

To use this command must apply to some surfaces local axis automatic.

When generating the mesh each element will have a local axis with z' pointing to the surface normal at the element center, but the axes x' and y' will be arbitrary set parallels to a direction,

and maybe this automatic x' direction is not appropriated to be used for example to define the fibers direction of a composite material.

It is possible to select some auxiliary curves to generate an approximated interpolated field of directions,

nodes: to correct the current local axis on the nodes of the mesh

elements : to correct the current local axis on the elements of the surface's mesh

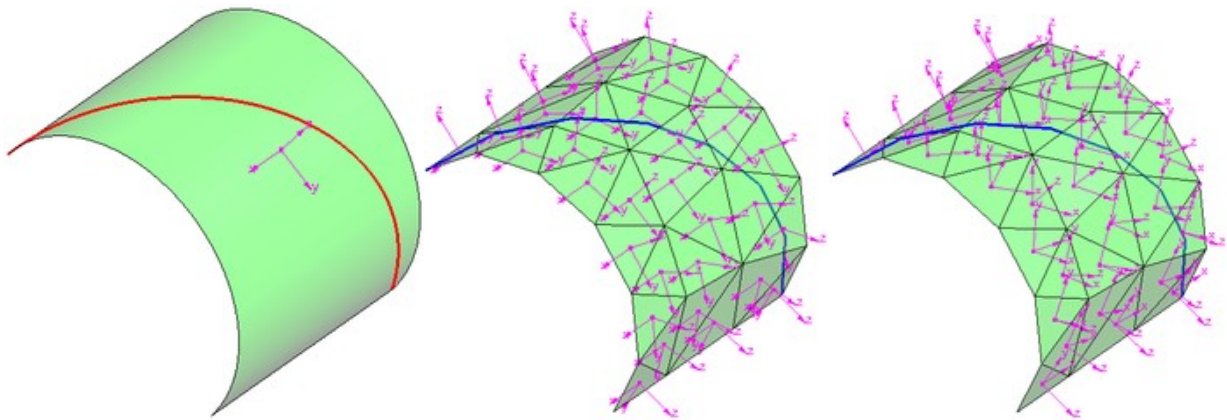
<groupList> a list of group names with the surfaces with applied local axis automatic that want to be enhanced in its mesh (nodes or elements)

<groupLinesList> a list of group names with the auxiliary curves

e.g.

Assuming a group named `my_surfaces_group` with some surface with automatic local axis applied, and a group named `my_lines_group` with some curves approximating the wanted local x' direction

```
set num_enhanced [correct_local_axes_with_lines elements [list my_surfaces_group] [list my_lines_group]]
```



The first image show a surface and a curve to be used to orientate the x' local axis, the second image show the automatic local axis, and the third image after call `correct_local_axes_with_lines` with the x' local axis 'near-parallel' to the auxiliary curve

Note: it is possible to call automatically this command when a mesh is generated, implementing [GiD_Event_AfterMeshGeneration](#)

Print

File

GiD_File fopen|fclose|fprintf|fflush|list

To allow print data from a Tcl procedure with standard fprintf command, specially to write the calculation file from Tcl or a mix of .bas template and Tcl procedures

•GiD_File fopen <filename> ?<access>?

Open a file named <filename> for writing access. By default access is "w", but is possible to use "a" to append to a previous file (and "wb" or "ab" to open the file in binary mode). It returns a long integer <file_id> representing the channel

•GiD_File fclose <file_id>

Close a channel

•GiD_File fprintf -nonewline <file_id> <format> ?<arg>? ... ?<arg>?

Print data from a Tcl procedure in a file opened with GiD_File fopen and returns the number of printed characters.

(a .bas template implicitly open/close a file with this command, and the file_id could be send to a tcl procedure as a parameter with the *FileId template keyword)

<file_id> must be a valid file descriptor, that could be obtained in a .bas template with the *FileId command (or with GiD_File fopen)

<format> must be a valid C/C++ format, according with the arguments

return

•GiD_File fflush <file_id>

Force unwritten buffered data to be written to the file

•GiD_File list

It returns a list of currently open file_ids

Example:

.bas file:

```
Number of points and lines: *tcl(MyMethod *FileId)
```

.tcl file:

```
proc MyMethod { channel } {
    GiD_File fprintf -nonewline $channel {%d %d} [GiD_Info Geometry NumPoints] [GiD_Info Geometry
    NumLines]
}
```

WriteCalculationFile

GiD_WriteCalculationFile

The command called GiD_WriteCalculationFile facilitate the creation of the output calculation file.
See also [Writing the input file for calculation](#)

•

Note: This command is provided to allow efficiency and relative flexibility writing the mesh and tree data related to groups, traversing the data structures without the cost of create a serialized copy in memory.
It is not compulsory to use this command, it is possible, and sometimes necessary, to use other Tcl commands to get mesh, groups and other data and reorder the information in order to be written.

GiD_WriteCalculationFile init|end|puts|coordinates|all_connectivities|connectivities|nodes|elements|has_elements ?-elemtype <etype>? ?-localaxes <formatLADict> ""? ?-elements_faces all|elements|faces? ?-number_ranges <NRDict>? ?-unique? ?-multiple? ?-all_entities? ?-do_subst? ?-sorted? ?-count? ?-return? ?-print_faces_conecs ? -connec_ordering? corner_face_corner_face <groupsDict>

•**GiD_WriteCalculationFile** init ?-mode append? ?-encoding external|utf-8? <filename>

To open for writing the calculation file.

Before to print any information to the file it must be opened with this command. Next prints of GiD_WriteCalculationFile will use implicitly this opened channel.

GiD internal strings are utf-8 codified,

-encoding external : (default) strings to print are converted to the current external encoding.

-encoding utf-8: strings are not converted.

It returns an file identifier that could be used with 'GiD_File fprintf', but must not be used with Tcl standard print commands like 'puts' or 'write'

Example:

```
set file_id [GiD_WriteCalculationFile init -mode append {c:/temp/my output.dat}]
lassign {1.5 2.3} x y
GiD_File fprintf $file_id "x=%15.5f y=%15.5f" $x $y
```

•**GiD_WriteCalculationFile** end

To close the calculation file

Example

```
GiD_WriteCalculationFile end
```

•**GiD_WriteCalculationFile** puts ?-nonewline? <string>

Print the string in the calculation file and a carriage return

-nonewline avoid the carriage return.

Example:

```
GiD_WriteCalculationFile puts "hello world"
```

•**GiD_WriteCalculationFile** coordinates ?-count? ?-return? ?-factor <factor>? <format>

This command must be used to print all nodes of the mesh. It prints <num> <x> <y> <z> for each node.

<format> must be a "C-like" format for an integer and three doubles.

Are only printed the values with supplied format, e.g. if the format is "%d %f" only the node number and its x will be printed.

If a %.0s is specified then the corresponding value is not printed (trick to avoid print some values)

••If -count is specified then only return the number of entities, without print. If -return is specified then return the string, without print.

- **-sorted** can be set to apply a string ordering.
- **-print_faces_conecs** can be set to print face element connectivities. A `-elements_faces` of face elements should be specified when a `-print_faces_conecs` is used.
- **-connec_ordering** can be set to specify a connectivities ordering as corner-middle edge-corner (**corner_face_corner_face**), default is first all corner nodes (**corners_faces**)
- If **-count** is specified then only the number of entities is returned, without print.
- If **-return** is specified then the complete string is returned, without print.
- <groupsDict>** is a dictionary (list of pairs key value), with key=`group_name` and value=`format`

if key is "" then, is considered like a wildcard and match all entities that are not matched by other keys.

Example: to print the id and connectivities of the triangle elements of the groups named `$group_a` or `$group_b`

```

•
•

set format_by_group [dict create $group_a "A: %d - %d %d %d" $group_b "B: %d - %d %d %d"]
GiD_WriteCalculationFile connectivities -elemtype Triangle $format_by_group

```

Example: to print the node id of the nodes belonging to `$group_name` and a text with the group name

```

set format_by_group [dict create $group_name "%d of group $group_name"]
GiD_WriteCalculationFile nodes $format_by_group

#or to print also its x y z coordinates:
set format_by_group [dict create $group_name "%d of group $group_name with coordinates %g %g %g"]
GiD_WriteCalculationFile nodes $format_by_group

#or as a trick to avoid printing the z coordinate could use "%.0s" to jump this extra argument
set format_by_group [dict create $group_name "%d of group $group_name with coordinates %g %g %.0s"]
GiD_WriteCalculationFile nodes $format_by_group

```

Example: to print the element id of the triangle elements belonging to `$group_name`

```

GiD_WriteCalculationFile elements -elemtype Triangle [dict create $group_name "element id:%d node ids:%d %d %d\n"]

```

Example: to print the faces of tetrahedra belonging to `$group_name`, printing its 3 face node ids (the `%.0s` is to not print the tetrahedra id)

```

GiD_WriteCalculationFile elements -elemtype Tetrahedra -elements_faces faces -print_faces_conecs [dict create $group_name "%.0snode ids: %d %d %d\n"]

```

Example: to print the faces of tetrahedra belonging to `$group_name`, printing the tetrahedra id and the local face index (0 to 4)

```

GiD_WriteCalculationFile elements -elemtype Tetrahedra -elements_faces faces [dict create $group_name "element id:%d face index:%d\n"]

```

Results

GiD_Result create|delete|exists|get|get_nodes|gauss_point|result_ranges_table ?-array? <data>

To create, delete or get postprocess results:

••**GiD_Result create** ?-array? {Result header} ?{Unit <unit_name>}? ?{componentNames name1 ...}? {entity_id scalar|vector|matrix_values} {...} {...} : these creation parameters are the same as for the postprocess results format (see [Result of Results format: ModelName.post.res](#)) where each line is passed as Tcl list argument of this command;

Optionally the names of the result's components could be specified, with the componentNames item, and the unit label of the result with the Unit item

if the -array flag is used (recommended for efficiency), then the syntax of the data changes. Instead to multiple items {id1 vx1 vy1 ...} ... {idn vxn vyn} a single item with sublists is required, id1 ... idn} {{vx1...vxn} {vy1...vyn}}, where idi are the integers of the node or element where the result are defined, and vi are the real values. The amount of values depends on the type of result: 1 for Scalar, 2 for ComplexScalar, 3 for Vector (4 if signed modulus is provided), 6 for Matrix.

In fact with -array it is returned an 'objarray': a Tcl_Obj object specialized for arrays, implemented as a Tcl package named 'objarray'. (for more information see scripts\objarray\objarray.pdf)

Examples:

```
GiD_Result create -array {Result "MyVecNodal" "Load analysis" 10 Vector OnNodes} {ComponentNames "Vx" "Vy" "Vz" "|velocity|"} 1 3}
{{2.0e-1 -3.5e-1} {2.0e-1 4.5e-1} {0.4 -2.1}
GiD_Result create {Result "Res Nodal 1" "Load analysis" 1.0 Scalar OnNodes} {1 2} {2 2} {113 2} {3 5} {112 4}
GiD_Result create {Result "Res Nodal 2" "Load analysis" 4 Vector OnNodes} {ComponentNames "x comp" "y comp" "z comp" "modulus"} {1
0.3 0.5 0.1 0.591} {2 2.5 0.8 -0.3 2.641}
GiD_Result create -array {Result "Res Nodal 2" "Load analysis" 4 Vector OnNodes} {ComponentNames "x comp" "y comp" "z comp"
"modulus"} 1 2} {{0.3 2.5} {0.5 0.8} {0.1 -0.3} {0.591 2.641}}
```

••**GiD_Result delete** {Result_name result_analysis step_value} : deletes one result;

Examples:

```
GiD_Result delete {"Res Nodal 1" "Load analysis" 4}
```

••**GiD_Result exists** {Result_name result_analysis step_value} : return 1 if the result exists.

••**GiD_Result get** ?-max|-min|-compmax|-compmin|-info? ?-sets <set_names_list>? ?-selection <sorted_ids>? ?-array? ?-ignore_no_result? {Result_name result_analysis step_value} : retrieves the results value list of the specified result.

-array: optional flag. The values are returned more efficiently grouping the information in arrays, else values are grouped as a list with one item by entity

-ignore_no_result : optional flag. The values unset (with special value -3.40282346638528860e+38 that mean no_result) will be ignored

-sets <set_names_list>: only the results of nodes/elements (depending on the result) of the sets belonging to <set_names_list> are returned

-selection <sorted_ids>: only the results of nodes/elements (depending on the result) with id belonging to <sorted_ids> are returned
<sorted_ids> must be an intarray (list of integer ids) of increasing ids of nodes/elements to be returned.

if one of the **-max**, **-min**, **-compmax**, **-compmin**, or **-info** flags was specified instead of the full results value only the minimum/maximum value of the result, every minimum/maximum of the components of the result, or the header information of the result is retrieved, respectively;

Examples: (case of a scalar result defined on triangles with 3 gauss points)

```
GiD_Result get -selection {169 170} -array [list "Test Gauss" "LOAD ANALYSIS" 10]
-> {Result "Test Gauss" "LOAD ANALYSIS" 10 Scalar OnGaussPoints "Triangles"} {ComponentNames "Test Gauss"} 169 170} {{26.25
27.299999237060547 28.350000381469727 26.399999618530273 27.450000762939453 28.5}
GiD_Result get -selection {169 170} [list "Test Gauss" "LOAD ANALYSIS" 10]
-> {Result "Test Gauss" "LOAD ANALYSIS" 10 Scalar OnGaussPoints "Triangles"} {ComponentNames "Test Gauss"} {169 26.25
27.299999237060547 28.350000381469727} {170 26.399999618530273 27.450000762939453 28.5}
```

••**GiD_Result get_nodes**: returns a list of nodes and their coordinates.

•**GiD_Result gauss_point** create|get|names|delete <name> <elementtype> <npoint> ?-nodes_included? <coordinates> ? <mesh_name>?

•create <name> <elementtype> <npoint> ?-nodes_included? <coordinates> ? <mesh_name>?

Define a new kind of gauss point where element results could be related.

<name> is the gauss point name. Internal Gauss points are implicitly defined, and its key names (GP_LINE_1,GP_TRIANGLE_1,...) are reserved words and can't be used to create new gauss points or be deleted. (see [Gauss Points](#))

<elementtype> must be one of "point | line | triangle | quadrilateral | tetrahedra | hexahedra | prism | pyramid | sphere | circle". (see [Mesh format: ModelName.post.msh](#))

<npoint> number of gauss points of the element

-nodes_included: optional word, only for line elements, to specify that start and end points are considered (by default are not included)

<coordinates>: vector with the local coordinates to place the gauss points: 2 coordinates by node for surface elements, 3 coordinates for volume elements. For line elements now is not possible to specify its coordinates, the n points will be equispaced.

If coordinates are "" then internal coordinates are assumed.

••<mesh_name>: optional mesh name where this definition is applied, by default it is applied to all

meshesget <name>

Return the information of this gauss point

●●●names

Return a list with the names of all gauss points defined

●●●delete <name>

Examples:

GiD_Result gauss_point create GPT1 Quadrilateral 1 {0.5 0.5}

GiD_Result create {Result "Res Gauss 1" "Load analysis" 1.0 Scalar OnGaussPoints GPT1} {165 2} {2} {3} {164 5} {4} {3}

●●GiD_Result result_ranges_table create|get|names|delete <name> {<min1> <max1> <label1> ... <minn> <maxn> <labeln> }

●create <name> {<label1> <min1> <max1> ... <labeln> <minn> <maxn>}

Define a new kind of result ranges table to map ranges of result values to labels.

<name> is the result ranges table name.

<min_{iii}

●●●get <name>

Return the information of this result ranges table

●●●names

Return a list with the names of all result ranges tables defined

●●●delete <name>

-array flag can be specified, for create and get subcommands, to use list of vectors to handle the information in a more efficient way

Sets

Definition

GiD_Sets get

To handle the definition of postprocess sets (similar to preprocess layers)

•**GiD_Sets get color|visible|type|num_entities|id <set_name>**

To obtain the current value of some property:

type: set type. could be 0==unknown, 1==mesh, 2==set, 3==cut

num_entities: the total number of mesh elements that belong to the set

id: the numeric identifier of the set

Entities

GiD_EntitiesSets get|entity_sets

To handle the entities that belong to postprocess sets (similar to preprocess layers)

•GiD_EntitiesSets get|entity_sets

To know the entities of a set or to know the sets of an entity

•GiD_EntitiesSets get <set_name> nodes|elements|all_mesh ?-count?

To get the list of entities of kind <over> that belong to <set_name>

If <over> is all_mesh then is obtained a list with 2 sublists: node id's, element id's

if -count is specified, then only the number of objects is returned instead of its list.

In fact it is returned an 'objarray': a Tcl_Obj object specialized for arrays, implemented as a Tcl package named 'objarray'. (for more information see scripts\objarray\objarray.pdf)

Example:

```
set count_elements_set [GiD_EntitiesSets get Layer0 elements -count]
set nodes_ids_one_set [GiD_EntitiesSets get Layer0 nodes]
```

•GiD_EntitiesSets entity_sets nodes|elements <id>

To get the set that contain the element <id> or the list of set that contain a node <id> (the sets that contain elements with the node as vertex)

Example:

```
set element_21_set [GiD_EntitiesSets entity_sets elements 21]
set node_8_sets [GiD_EntitiesSets entity_sets nodes 8]
```

Graph

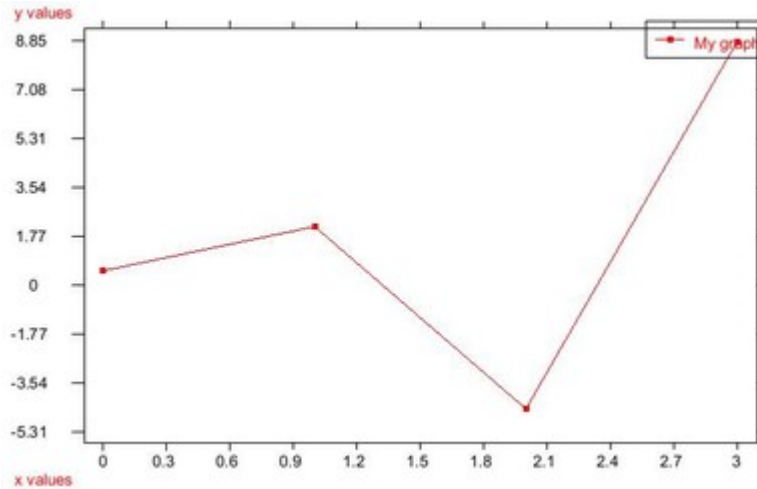
GiD_Graph clear|create|delete|edit|exists|get|hide|list|show

To create, delete or get postprocess graphs:

All commands accept an optional parameter <graphset_name>, else the current graphset is assumed.

- **clear?**<graphset_name>?: delete all graphs in GiD;
- **create** <graph_name> <label_x> <label_y> <x_values> <y_values> <x_unit> <y_unit> ?<graphset_name>?: creates the graph "graph_name" with the provided information, causing an error if the graph already exists: for instance the graph of the picture was created with

GiD_Graph create "My graph" "x values" "y values" {0 1 2 3} {0.5 2.1 -4.5 8.8} "" ""



- **delete** <graph_name>?<graphset_name>?: deletes the graph "graph_name" causing an error if does not exists;
- **edit** <graph_name> <label_x> <label_y> <x_values> <y_values> <x_unit> <y_unit> ?<graphset_name>?: modify a the graphwith the new values, but without lost other settings like line color, etc.
- **exists** <graph_name>?<graphset_name>?: return 1 the graph "graph_name" exists.
- **get** <graph_name> ?<graphset_name>?: gets a list with the values of the graph with name "graph_name", the values are the same used to create a graph: <label_x> <label_y> <x_values> <y_values>
- **get_name** <num> ?<graphset_name>? : get the name of the graph with global identifier <num>. If <graphset_name> is specified then the graph is only find in this graphset instead of all graphsets.
- **hide?**<graphset_name>?: hides the graphs and switches back to mesh view.
- **list** ?<graphset_name>?: gets a list of the existent graphs, an empty list if there is no graph.
- **selection get|swap| set <value> <graph_name>**: set or get the flag of selection of the graph. <value> is must be 0 or 1 and is only needed in case of set. swap change the current selection state to the opposite one. Selected graphs are showed in red color.
- **show?**<graphset_name>?: switches the graphic view and shows the graphs.

GraphSet

GiD_GraphSet create|current|delete|edit|exists|list

To create, delete or get postprocess graphs sets:

A graphset is a container of graphs sharing the same x, y axes

- **create ?<graphset_name>?**: creates a new graph set. If the name is not provided an automatic unused name is assigned. The name of the graph set is returned.
- **current ?<graphset_name>?**: get or set the current graph set. There is always a current graph set.
- **delete <graphset_name>**: deletes the graphset "graphset_name".
- **edit <graphset_name> name|legend_location|title_visible <new_value>** : modify the property of the graph set.

name: is the graphset identifier name

legend_location: 0 - to not show the graph legend

1 - to show the legend on the top-right (default)

2 - to show the legend on the top in a single line, like a title

title_visible: 0 or 1, to print or not in the legend also the graphset name

- **exists <graphset_name>**: return 1 the graph set "graphset_name" exists;
- **list** : gets a list of the existent graph sets;

OpenGL

GiD_OpenGL

register|unregister|registercondition|unregistercondition|draw|drawtext|project|unproject|get|doscrzoffset|drawentity|font|pgfont

This command is a Tcl wrapper of some OpenGL functions. It allows to use OpenGL commands directly from GiD-Tcl.

For example, for C/C++ is used:

```
glBegin(GL_LINES);
glVertex(x1,y1,z1);
glVertex(x2,y2,z2);
glEnd();
```

For GiD-Tcl must use:

```
GiD_OpenGL draw -begin lines
GiD_OpenGL draw -vertex [list $x1 $y1 $z1]
GiD_OpenGL draw -vertex [list $x2 $y2 $z2]
GiD_OpenGL draw -end
```

The standard syntax must be changed according to these rules: - OpenGL constants: "GL" prefix and underscore character '_' must be removed; the command must be written in lowercase.

Example:

GL_COLOR_MATERIAL -> colormaterial

OpenGL functions: "GL" prefix must be removed and the command written in lowercase. Pass parameters as list, without using parentheses ()

Example:

glBegin(GL_LINES) -> glBegin lines

The subcommand "GiD_OpenGL draw" provides access to standard OpenGL commands, but other "GiD_OpenGL" special GiD subcommands also exists:

- register <tclfunc>** Register a Tcl procedure to be invoked automatically when redrawing the scene. It returns a handle to unregister.

Example:

```
proc MyRedrawProcedure { } { ...body... }
set id [GiD_OpenGL register MyRedrawProcedure]
```

- unregister <handle>** Unregister a procedure previously registered with **register**.

Example:

```
GiD_OpenGL unregister $id
```

- registercondition <tclfunc> <condition>** Register a Tcl procedure to be invoked automatically when redrawing the specified condition.

The tcl function must have this prototype:

```
proc xxx { condition use entity_id values } {
    ...
    return 1
}
```

The supplied parameters are:

condition :the condition name, defined in the .cnd file of the problemtype

use: GEOMETRYUSE, MESHUSE or POSTUSE

entity_id: the integer number that identity the entity where the condition is applied. The kind of entity is known because it is declared in the definition of the condition in the .cnd, depending if the current state is geometry or mesh

values: a list of the field's values applied to this entity. The amount of values must match the amount of fields of the condition definition.

The return value of this procedure is important:

return 0: then the standard representation of the condition is also invoked after the procedure

return 1: the standard representation of the condition is avoided.

- unregistercondition <condition>** Unregister a procedure previously registered with **registercondition**.

- draw <-cmd args -cmd args>** This is the most important subcommand, it calls standard OpenGL commands. See the list of supported OpenGL functions.

- drawtext <text> ?-check_non_latin?** Draw a text more easily than using standard OpenGL commands (draw in the current 2D location, see rasterpos OpenGL command).

If *-check_non_latin* flag is provided, then the text is checked to detect non-latin characters, like a Japanese string, to be drawn properly (otherwise it is considered as a latin string). The flag must be provided only if *<text>* could potentially be non-latin, like translated strings. In case of numbers for example it is unneeded (more efficient without the extra check)

Example:

```
GiD_OpenGL draw -rasterpos [list $x $y $z]
GiD_OpenGL drawtext "hello world"
```

- drawentity ?-mode normal|filled? point|line|surface|volume|node|element|dimension <id list>** To draw an internal GiD preprocess entity.

Example:

```
GiD_OpenGL drawentity -mode filled surface {1 5 6}
```

For elements it is possible to draw only a face, specifying items of *element_id* and *face_id*, with *face_id* a number from 1 to the number of faces of the element.

Example:

```
GiD_OpenGL drawentity -mode filled element {{1 1} {5 1} {6 3}}
```

- project <x y z>** Given three world coordinates, this returns the corresponding three window coordinates.

- unproject <x y z>** Given three window coordinates, this returns the corresponding three world coordinates.

- get modelviewmatrix|projectionmatrix|viewport**

return a list of values:

modelviewmatrix: 16 doubles

projectionmatrix: 16 doubles

viewport: 4 integers

- doscrczoffset <boolean>** Special trick to avoid the lines on surfaces hidden by the surfaces.

•font push <font_name font_size>|pop|measure <text>|current|metrics ?-ascent|-descent|-linespace|-fixed?

push sets the current OpenGL font, pop restores the previous one

measure <text> returns the amount of space in pixels to display this <text>

current returns a list with the current font name and size

metrics returns a list with current font metrics information: -ascent -descent and -linespace in pixels, -fixed is 1 if all characters have equal size

Example:

```
GiD_OpenGL font push {"Times New Roman" 18}
set with [GiD_OpenGL measure "hello world"]
GiD_OpenGL drawtext "hello world"
GiD_OpenGL pop
```

pgfont pushfont <font_type>|popfont|print <text>|dimensions <text>|foreground <red> <green> <blue>
•<alpha>|background <red> <green> <blue> <alpha>

Similar to font but using pgfonts

font types: defaultfont, axisfont, legendfont, labelfont, graphfont, asianfont, tkdrawoglfont, pmfont

List of supported OpenGL functions:

```
accum alphafunc begin bindtexture blendfunc call calllist clear clearaccum clearcolor cleardepth
clearstencil clipplane color colormask colormaterial cypixels cullface deletelists deletetextures
depthfunc depthmask dfactorBlendTable disable drawbuffer drawpixels edgeflag enable end endlist
evalcoord1 evalcoord2 evalmesh1 evalmesh2 finish flush fog frontface frustum genlists gentextures
getstring hint hintModeTable initnames light lightmodel linestipple linewidth loadidentity loadmatrix
loadname lookat map1 map2 mapgrid1 mapgrid2 material matrixmode modeColorMatTable multmatrix newlist
newListTable normal opStencilTable opStencilTable ortho perspective pickmatrix pixeltransfer pixelzoom
pointsize polygonmode popattrib popmatrix popname pushattrib pushmatrix pushname rasterpos readbuffer
readpixels rect rendermode rotate scale scissor selectbuffer shademodel stencilfunc stencilmask
stencilop texcoord texenv texgen teximage1d teximage2d texparameter translate vertex viewport
```

List of special non OpenGL standard functions:

```
getselection
```

List of supported OpenGL constants:

```
accum accumbuffer accumbufferbit add alphatest always allattrib allattribbits ambient ambientanddiffuse
autonormal aux0 aux1 aux2 aux3 back backleft backright blend bluebias bluescale ccw clamp clipplane0
clipplane1 clipplane2 clipplane3 clipplane4 clipplane5 colorbuffer colorbufferbit colorindex
colormaterial compile compileandexecute constantattenuation cullface current currentbit cw decal decr
depthbuffer depthbufferbit depthtest diffuse dither dstalpha dstcolor enable enablebit emission equal
eval evalbit exp exp2 extensions eyelinear eyeplane feedback fill flat fog fogbit fogcolor fogdensity
fogend fogmode fogstart front frontandback frontleft frontright gequal greater greenbias greenscale
hint hintbit incr invert keep left lequal less light0 light1 light2 light3 light4 light5 light6 light7
lighting lightingbit lightmodelambient lightmodellocalviewer lightmodeltwoside line linebit linear
linearattenuation lineloop lines linesmooth linestipple linestrip list listbit load map1color4
map1normal map1texturecoord1 map1texturecoord2 map1texturecoord3 map1texturecoord4 map1vertex3
map1vertex4 map2color4 map2normal map2texturecoord1 map2texturecoord2 map2texturecoord3
map2texturecoord4 map2vertex3 map2vertex4 modelview modulate mult nearest never none normalize notequal
objectlinear objectplane one oneminusdstalpha oneminusdstcolor oneminussralpha oneminussrccolor
packalignment packlsbfirst packrowlength packskippixels packskiprows packswapbytes pixelmode
pixelmodebit point pointbit points polygon polygonbit polygonoffsetfill polygonstipple
polygonstipplebit position projection q quadraticattenuation quads quadstrip r redbias redscale render
render repeat replace return right s scissor scissorbit select shininess smooth specular spheremap
spotcutoff spotdirection spotexponent sralpha sralphasaturate srccolor stenciltest stencilbuffer
stencilbufferbit t texture texture1d texture2d texturebit texturebordercolor textureenv textureenvcolor
textureenvmode texturegenmode texturegens texturegent texturemagfilter textureminfilter texturewraps
texturewrapt transform transformbit triangles trianglefan trianglestrip unpackalignment unpacklsbfirst
unpackrowlength unpackskippixels unpackskiprows unpackswapbytes vendor version viewport viewportbit zero
```

You can find more information about standard OpenGL functions in a guide to OpenGL.

Raster

GiD_Raster create|interpolate

To create from the mesh a raster (2D grid) with a value that represents the z, and use a raster to efficiently interpolate values to other points or all points of other raster.

A raster is defined with a Tcl list of the following data:

```
<ncols> <nrows> <xllcenter> <yllcenter> <xcellsize> <ycellsize> <nodata_value> <values>
```

ncols, nrows are the number of columns and rows of values (representing values on center of cells)
xllcenter, yllcenter are the x,y coordinates of the lower-left corner
xcellsize, ycellsize are the x,y sizes of the cells, usually the same value (compulsory to be exported as ArcInfo grid ASCII)
<nodata_value> is the special value to represent that the data doesn't exists (usually -9999.0 for ArcInfo grid ASCII)
<values> is an objarray of (*ncols* x *nrows*) doubles with the scalar value to be represented, usually the z of the x,y node.

```
GiD_Raster create {nodes <xyz_nodes>} ?{<cellsize>}{<ncols> <nrows>}{<ncols> <nrows> <xllcenter> <yllcenter> <xcellsize> <ycellsize> <nodata_value>}?
```

To create a raster from a cloud of 3D nodes

<xyz> must be an objarray of doubles with "x1 y1 z1 ... xn yn zn" values of a cloud of coordinates.
 z data will be extrapolated to grid with and averaged value of the near nodes

```
GiD_Raster create {nodes <xyz_nodes>}
```

The raster will be created with automatic number of rows, columns and limits

```
GiD_Raster create {nodes <xyz_nodes>} {<cellsize>}
```

The raster will be created with the specified cell size, equal for x and y, and automatic number of columns, rows, and limits

```
GiD_Raster create {nodes <xyz_nodes>} <ncols> <nrows>
```

The raster will be created with the specified number of columns and rows, and automatic limits

```
GiD_Raster create {nodes <xyz_nodes>} <ncols> <nrows> <xllcenter> <yllcenter> <xcellsize> <ycellsize> <nodata_value>}
```

The raster will be created with the specified number of columns and rows, lower-left center, cell sizes and no data value.

```
GiD_Raster create {triangles <xyz_nodes> <elements>} ?{<cellsize>}{ncols nrows}{ncols nrows <xllcenter> <yllcenter> <xcellsize> <ycellsize> <nodata_value>}?
```

To create a raster from a irregular mesh of 3D triangles

<xyz> must be an objarray of doubles with "x1 y1 z1 ... xn yn zn" values of a the node's coordinates.

<elements> must be an objarray of integers with the index (starting by 1) or the 3 nodes of each triangle. "t1_1 t1_2 t1_3 .. tn_1 tn_2 tn_3"
 z data will be extrapolated to grid finding the triangle that contain the coordinate in 2D projection, or a near node.

```
GiD_Raster create {triangles <xyz_nodes> <elements>}
```

The raster will be created with automatic number of rows, columns and limits

```
GiD_Raster create {triangles <xyz_nodes> <elements>} {<cellsize>}
```

The raster will be created with the specified cell size, equal for x and y, and automatic number of columns, rows, and limits

```
GiD_Raster create {triangles <xyz_nodes> <elements>} <ncols> <nrows>
```

The raster will be created with the specified number of columns and rows, and automatic limits

```
GiD_Raster create {triangles <xyz_nodes> <elements>} <ncols> <nrows> <xllcenter> <yllcenter> <xcellsize> <ycellsize> <nodata_value>}
```

The raster will be created with the specified number of columns and rows, lower-left center, cell sizes and no data value.

```
GiD_Raster interpolate ?-closest? <raster_interpolation> {nodes <xy_nodes>}{raster <raster_to_interpolate_without_data>}
```

To use *<raster_interpolation>* to calculate interpolated values of a collection of 2D nodes or all grid nodes of another raster

It returns and obj_array of doubles with the interpolated value for each node

<xy_nodes> list of x y coordinates "x1 y1 ... xn yn" of the points to interpolate the value

<raster_to_interpolate_without_data> another raster (its values could be a empty objarray)

If -closest flag is set, then instead interpolate it get the value of the closest grid node (interesting to map non-continous integer values)

Some auxiliary Tcl procedures:

```
GIS::GetRasterFromTriangles { {cellsize 0.0} }
```

It returns a raster from the current mesh of triangles. If cellsize is not specified an automatic value is used

```
GIS::GetRasterFromNodes { {cellsize 0.0} }
```

It returns a raster from the current mesh nodes. If cellsize is not specified an automatic value is used

```
GIS::SubsampleRaster { raster increment show_advance_bar }
```

It returns a new raster subsampling the input raster jumping columns and row by *increment*

show_advance_bar must be 1 to show and advancing bar during the process

```
GIS::ImportRaster_Geometry { raster show_advance_bar }
```

It creates geometrical surfaces (and its lines and points) from the raster

```
GIS::ImportRaster_Mesh { raster show_advance_bar }
```

It creates mesh quadrilaterals (and its nodes) from the raster

```
GIS::SaveRaster_ArcInfoASCII { raster filename }
```

It saves the *raster* in the file named *filename* with ArcInfo grid ASCII format

GDAL::ReadRaster { filename show_advance_bar }

It return a raster with from the file named *filename*.

The format could be some raster geospatial data format allowed by the GDAL (Geospatial Data Abstraction Library).

Some of them are: ArcInfo, geotiff, png, jpg, and much more.

Other

GiD_Set ?-meshing_parameters_model? ?-default|-array_names? <varname> ?<value>?

This command is used to set or get GiD variables. GiD variables can be found through the **Right buttons** menu under the option Utilities -> Variables:

- <varname> is the name of the variable;
- <value> if this is omitted, the current variable value is returned (analogous with 'GiD_Info variables <varname>').
- default return the default value of the variable (<value> its not accepted)
- array_names return a list with the subnames of the array, or an empty list if is not an array
- meshing_parameters_model to use the copy of the variable used in meshing the current model instead of the general preference variable

Example:

```
GiD_Set CreateAlwaysNewPoint
GiD_Set CreateAlwaysNewPoint 1
GiD_Set -default CreateAlwaysNewPoint
GiD_Set -meshing_parameters_model SurfaceMesher
```

GiD_SetModelName <name>

To change the current model name.

If name is not specified then the current model name is returned.

GiD_SetProblemtypeName <name>

To change the current problemtype name.

If name is not specified then the current problemtype name is returned.

GiD_ModifiedFileFlag set|get ?<value>?

There is a GiD internal flag to indicate that the model has changed, and must be saved before exit.

With this command it is possible to set or get this flag value:

- <value> is only required for **set**: must be 0 (false), or 1 (true).

Example:

```
GiD_ModifiedFileFlag set 1
GiD_ModifiedFileFlag get
```

GiD_MustRemeshFlag set|get ?<value>?

There is a GiD internal flag to indicate that the geometry, conditions, etc. have changed, and that the mesh must be re-generated before calculations are performed.

With this command it is possible to set or get this flag value:

- <value> is only required for **set**: must be 0 (false), or 1 (true).

Example:

```
GiD_MustRemeshFlag set 1
GiD_MustRemeshFlag get
```

GiD_Redraw

To force a redraw

GiD_BackgroundImage get|set show|filename|location <values>

This command allow to get and set the background image properties

Valid set values are:

- show: 1 or 0
- filename:

the full filename of some valid GiD image format to be used as background image
or "", to release the current image

•**location:**

'fill' to fill the whole screen,
or a list (objarray) with six floating values for a real size image, to set the origin and x',y' local axes: ox oy ix iy jx jy
They are 3 points (in 2D space, z=0.0) that represent:
o=origin lower-left point
i=end point of the local x' axis
j=end point of the local y' axis (the size ratio of the image could change)
Note: 'GiD_BackgroundImage set location' must be called after 'GiD_BackgroundImage set filename'

GiD_RegisterExtensionProc <extension> PRE|POST|PREPOST <procedure>

To register a Tcl procedure to be automatically called when dropping a file with this extension

Example:

```
GiD_RegisterExtensionProc ".h5" PRE Amelet::ReadPre
```

GiD_RegisterPluginAddedMenuProc <procedure>

To register a Tcl procedure to be automatically called when re-creating all menus (e.g. when doing files new)
this procedure is responsible to add its own options to default menu.

Example:

```
GiD_RegisterPluginAddedMenuProc Amelet::AddToMenu
```

GiD_Thumbnail get | get_pixels | get_vectorial | get_statistics_mean

Returns the image data of an downscaled view of the current graphical window.

•**get ?<width> <height>?**

The image is a downscaled from the current size to width x height. The parameters width and height are optional and by default the view is scaled to 192x144. The result of this command can be directly used by the Tk image command, like this:

Example:

```
label .l -image [image create photo -data [GiD_Thumbnail get]]
```

•**get_pixels ?-quality <quality>? ?RGB|BGR|RGBA|BGRA|GREY|png|jpeg|raw?**

It returns a list {width height data} of the current image in a direct way, binary and without modifications.
It allow to specify the kind and order of the data for Red, Green, Blue colors and for Alpha transparency factor.
If png format is specified that data is get as RGB and with png (portable network graphics format)
quality: integer value from 0 to 100, valid only for jpeg format (that compress losing image quality)

Example:

```
lassign [GiD_Thumbnail get_pixels png] w h pixels
set my_image [image create photo -width $w -height $h -data [encrypter BASE64 encode $pixels]]
label .l -image $my_image
```

•**get_vectorial STL|VRML|OBJ**

It returns the current view in binary STL or ascii VRML format. As STL only accepts triangles, lines are formatted as collapsed triangles and polygons are triangularized.

GiD_Thumbnail get_vectorial obj --> returns a list with 3 elements:

1st - the obj file contents (refers to the 'ObjInMemory.mtl' material library file eventually)

2nd - (optional) the mtl file contents (i.e. the 'ObjInMemory.mtl', may refer to the 'ObjInMemory.png')
 3rd - (optional) the texture data (the 'ObjInMemory.png' binary data)
 The returned data can be written directly in a file

Example:

```
# writing STL output
set fo [ open file.stl wb]
puts $fo [GiD_Thumbnail get_vectorial stl]
close $fo
# writting OBJ output
set obj_data [GiD_Thumbnail get_vectorial OBJ]
lassign $obj_data obj_objects obj_mtl obj_tex
set fo [open file.obj w]
puts $fo $obj_objects
close $fo
if { [llength $obj_mtl] > 0} {
  # inside $obj_objects there is the reference 'usemtl ObjInMemory.mtl'
  set fo [open ObjInMemory.mtl w]
  puts $fo $obj_mtl
  close $fo
}
if { [llength $obj_tex] > 0} {
  # inside $obj_mtl there is the reference 'map_Kd ObjInMemory.png'
  set fo [open ObjInMemory.png wb]
  puts $fo $obj_tex
  close $fo
}
```

•get_statistics_mean

It returns an array of three real numbers with the mean of r g b components of all image pixels. (values from 0.0 to 255.0)
 e.g.

```
GiD_Thumbnail get_statistics_mean
```

->254.5277054398148 254.52159288194446 254.51443142361111

GiD_GetWorldCoord screen_x screen_y

Given the screen coordinates (screen_x, screen_y) returns a list with six coordinates:

{ x y z nx ny nz }

being

(x,y,z) the coordinates mapped into the world (model) of the screen coordinates,

(nx,ny,nz) the normal vector components of the world (model) pointing to the user.

The mapping screen --> world (model) is done by intersecting the line perpendicular to the screen, passing through the coordinates (screen_x,screen_y), with the plane parallel to the screen (in real model world) at the centre of the view / model. The returned normal is the normal of this plane.

GiD_Togl current|list

To get or set the current togl (Tk OpenGL object), and the list of all togl's of all windows

Example:

```
set togl [GiD_Togl current]
GiD_Togl current $togl
set togl [GiD_Togl list]
```


GiD_GetUserSettingsFilename ?-create_folders? ?-ignore_alternative_configuration_file?

To get the file name where the user settings are stored.

If *-create_folders* flag is provided, then all intermediate folders are created if doesn't exists

If *-ignore_alternative_configuration_file* flag is provided, then *alternative_file* provided by *-c* or *-c2* command line argument is ignored.

GiD_GetUserSettingsCommonDirectory ?-create_folders?

Similar to *GiD_GetUserSettingsFilename* but return the common directory, not depend on GiD version or *-c* / *-c2* command line flags

HTML help support

Problem type developers can take advantage of the internal HTML browser if they wish to provide online help. The GiDCustomHelp procedure below is how you can show help using the new format:

GiDCustomHelp ?args?

where args is a list of pairs option value. The valid options are:

- **-title** : specifies the title of the help window. By default it is "Help on <problem_type_name>".
- **-dir** : gives the path for the help content. If **-dir** is missing it defaults to "<ProblemType dir>/html". Multilingual content could be present; in such a case it is assumed that there is a directory for each language provided. If the current language is not found, language 'en' (for English) is tried. Finally, if 'en' is not found the value provided for **-dir** is assumed as the base directory for the help content.
- **-start** : is a path to an html link (and is relative to the value of **-dir**).

HelpDirs

With HelpDirs we can specify which of the subdirectories will be internal nodes of the help tree. Moreover, we can specify labels for the nodes and a link to load when a particular node is clicked. The link is relative the node. For instance:

```
HelpDirs {html-version "GiD Help" "intro/intro.html"} \
        {html-customization "GiD Customization"} \
        {html-faq "Frequently Asked Questions"} \
        {html-tutorials "GiD Tutorials" "tutorials_toc.html"} \
        {html_whatsnew "What's New"}
```

Structure of the help content

Assuming that html has been chosen as the base directory for the multilingual help content, the following structure is possible:

html

___en - English content
___es - Spanish content

Each content will probably have a directory structure to organize the information. By default the help system builds a tree resembling the directory structure of the help content. In this way there will be an internal node for each subdirectory, and the html documents will be the terminal nodes of the tree.

You can also provide a help.conf configuration file in order to provide more information about the structure of the help. In a help file you can specify a table of contents (TocPage), help subdirectories (HelpDirs) and an index of topics (IndexPage).

TocPage

TocPage defines an html page as a table of contents for the current node (current directory). We have considered two ways of specifying a table of contents:

 ... (default)
<DT> <DL> ... </DT>

The first is the one generated by texinfo.
For instance:

```
TocPage gid_toc.html
TocPage contents.ht DT
```

IndexPage

If we specify a topic index by IndexPage, we can take

advantage of the search index. In IndexPage we can provide a set of html index pages along with the structure type of the index. The type of the index could be:

<DIR> ... </DIR> (default)
 ... (only one level of)

The first is the one generated by texinfo.

For instance:

```
IndexPage html-version/gid_18.html html-faq/faq_11.html
```

Managing menus

GiD offers you the opportunity to customize the **pull-down** menus. You can add new menus or to change the existing ones. If you are creating a problem type, these functions should be called from the `InitGIDProject` or `InitGIDPostProcess` functions (see [TCL AND TK EXTENSION](#)).



Note: Menus and option menus are identified by their names.

Note: It is not necessary to restore the menus when leaving the problem type, GiD does this automatically.
The Tcl functions are:

•**GiDMenu::Create { menu_name_untranslated prepost {pos -1} {translationfunc _} }**

••Creates a new menu. New menus are inserted between the **Calculate** and **Help** menus.

menu_name_untranslated: text of the new menu (English).

•prepost can have these values:

"PRE" to create the menu only in GiD Preprocess.

"POST" to create the menu only in GiD Postprocess.

"PREPOST" to create the menu in both Pre- and Postprocess.

••pos: optional, index where the new menu will be inserted (by default it is inserted before the 'Help' menu)
translationfunc: optional, must be _ for GiD strings (default), or = for problemtype strings

•**GiDMenu::Delete { menu_name_untranslated prepost {translationfunc _} }**

Deletes a menu.

••menu_name_untranslated: text of the menu to be deleted (English).
prepost can have these values:

"PRE" to delete the menu only in GiD Preprocess.

"POST" to delete the menu only in GiD Postprocess.

"PREPOST" to delete the menu in both Pre- and Postprocess.

••translationfunc: optional, must be _ for GiD strings (default), or = for problemtype strings

•**GiDMenu::InsertOption { menu_name_untranslated option_name_untranslated position prepost command {acceler ""} {icon ""} {ins_repl "replace"} {translationfunc _} }**

Creates a new option for a given menu in a given position (positions start at 0, the word 'end' can be used for the last one).

••menu_name_untranslated: text of the menu into which you wish to insert the new option (English), e.g "Utilities"
option_name_untranslated: name of the new option (English) you want to insert.

The option name, is a menu sublevels sublevels list, like [list "List" "Points"]

If you wish to insert a separator line in the menu, put "---" as the option_name.

••position: position in the menu where the option is to be inserted. Note that positions start at 0, and separator lines also count.

•prepost: this argument can have the following values:

"PRE" to insert the option into GiD Preprocess menus.

"POST" to insert the option into GiD Postprocess menus.

"PREPOST" to insert the option into both Pre- and Postprocess menus.

••command: is the command called when the menu option is selected.
acceler: optional, key accelerator, like "Control-s"

•icon: optional, name of a 16x16 pixels icon to show in the menu

ins_repl: optional, if the argument is:

•replace: (default) the new option replaces the option in the given position

insert: the new option is inserted before the given position.

insertafter: the new option is inserted after the given position.

•translationfunc: optional, must be _ for GiD strings (default), or = for problemtype strings

•**GiDMenu::RemoveOption {menu_name_untranslated option_name_untranslated prepost {translationfunc _} }**

Removes an option from a given menu.

- menu_name_untranslated**: name of the menu (English) which contains the option you want to remove. e.g "Utilities"
- option_name_untranslated**: name of the option (English) you want to remove. The option name, is a menu sublevels list, like [list "List" "Points"]
- prepost**: this argument can have the following values:

"PRE" to insert the option into GiD Preprocess menus.

"POST" to insert the option into GiD Postprocess menus.

"PREPOST" to insert the option into both Pre- and Postprocess menus.

- translationfunc**: optional, must be `_` for GiD strings (default), or `=` for problemtype strings

To remove separators, the **option_name** is `---`, but you can append an index (starting from 0) to specify wich separator must be removed, if there are more than one.

e.g.

```
GiDMenu::RemoveOption "Geometry" [list "Create" "---2"] PRE
```

•**GiDMenu::ModifyOption** { **menu_name_untranslated** **option_name_untranslated** **prepost** **new_option_name** {**new_command** -default-} {**new_acceler** -default-} {**new_icon** -default-} {**translationfunc** `_`} }

Edit an existent option from a given menu

some parameters can be `'-default-'` to keep the current value for the command, accelerator, etc

•**GiDMenu::UpdateMenus** {}

Updates changes made on menus. This function must be called when all calls to create, delete or modify menus are made.

•**GiD_RegisterPluginAddedMenuProc** and **GiD_UnRegisterPluginAddedMenuProc**

This commands can be used to specify a callback procedure name to be called to do some change to the original menus

```
GiD_RegisterPluginAddedMenuProc <procname>
```

```
GiD_UnRegisterPluginAddedMenuProc<procname>
```

The procedure prototype to be registered must not expect any parameter, something like this.

```
proc <procname> {} {
... do something ...
}
```

e.g. a plugin can modify a menu to add some entry, but this entry will be lost when GiD create again all menus, for example when starting a new model. Registering the procedure will be applied again when recreating menus.

•**GiD_RegisterExtensionProc** and **GiD_UnRegisterExtensionProc**

This tcl command must be used to register a procedure that is able to handle when using 'drag and drop' of a file on the GiD window.

It is possible to specify the extension (or a list of extensions) of the files to be handled, the mode PRE or POST where it will be handled, and the name of the callback procedure to be called.

```
GiD_RegisterExtensionProc <list of extensions> <prepost> <procname>
```

```
GiD_UnRegisterExtensionProc <list of extensions> <prepost>
```

`<extension>` is the file extension, preceded by a dot

`<prepost>` could be PRE or POST

The procedure prototype to be registered must expect a single parameter, the dropped file name, something like this.

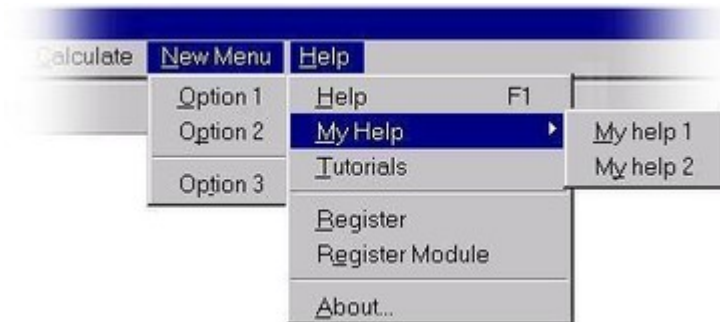
```
proc <procname> { filename } {
... do something ...
}
```

Example:

```
GiD_RegisterExtensionProc ".gif .png" PRE MyImageProcedure
```

EXAMPLE: creating and modifying menus

In this example we create a new menu called "New Menu" and we modify the GiD **Help** menu:



The code to make these changes would be:

```
GiDMenu::Create "New Menu" "PRE" -1 =
GiDMenu::InsertOption "New Menu" [list "Option 1"] 0 PRE "Command_1" "" "" replace =
GiDMenu::InsertOption "New Menu" [list "Option 2"] 1 PRE "Command_2" "" "" replace =
GiDMenu::InsertOption "New Menu" [list "---"] 2 PRE "" "" "" replace =
GiDMenu::InsertOption "New Menu" [list "Option 3"] 3 PRE "Command_3" "" "" replace =
GiDMenu::InsertOption "Help" [list "My Help"] 1 PRE "" "" "" insert _
GiDMenu::InsertOption "Help" [list "My Help" "My help 1"] 0 PRE "Command_help1" "" "" replace _
GiDMenu::InsertOption "Help" [list "My Help" "My help 2"] 1 PRE "Command_help2" "" "" replace _
GiDMenu::RemoveOption "Help" [list "Customization Help"] PRE _
GiDMenu::RemoveOption "Help" [list "What is new"] PRE _
GiDMenu::RemoveOption "Help" [list "FAQ"] PRE _
GiDMenu::UpdateMenus
```

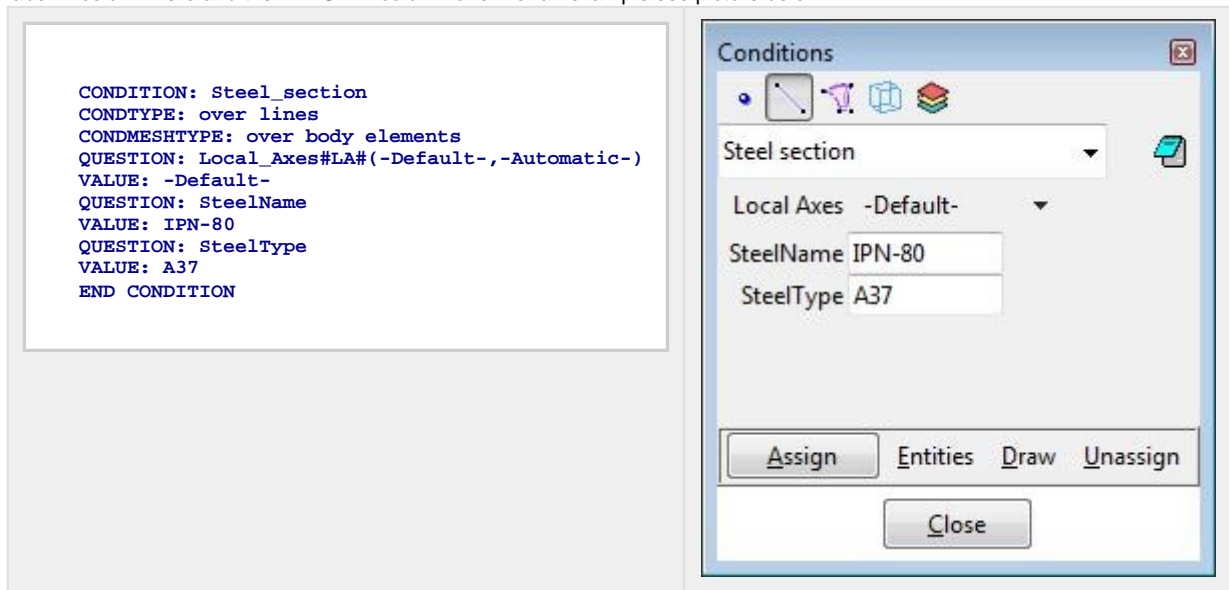
Custom data windows

In this section the **Tcl/Tk** (scripted) customization of the look and feel of the data windows is shown. The layout of the properties drawn in the interior of any of the data windows - either Conditions, Materials, Interval Data or Problem Data - can be customized by a feature called TkWidget; moreover, the common behaviour of two specific data windows, Conditions and Materials, can be modified by a Tcl procedure provided for that purpose. This common behaviour includes, in the case of Materials for example, assigning/unassigning, drawing, geometry types, where to assign materials, creating/deleting materials, etc.

TkWidget

The problem type developer can change the way a **QUESTION** is displayed and if he wishes he can also change the whole contents of a window, while maintaining the basic behavior of the data set, i.e. in the Condition window: assign, unassign, draw; in the Material window: create material, delete material; and so on.

With the default layout for the data windows, the questions are placed one after another in one column inside a container frame, the **QUESTION**'s label in column zero and the **VALUE** in column one. For an example see picture below.



The developer can override this behavior using **TKWIDGET**. **TKWIDGET** is defined as an attribute of a **QUESTION** and the value associated with it must be the name of a Tcl procedure, normally implemented in a Tcl file for the problem type. This procedure will take care of drawing the **QUESTION**. A **TKWIDGET** may also draw the entire contents of the window and deal with some events related to the window and its data.

The prototype of a **TKWIDGET** procedure is as follow:

```
proc TKWidgetProc {event args} {
  switch $event {
    INIT {
      ...
    }
    SYNC {
      ...
    }
    DEPEND {
      ...
    }
    CLOSE {
      ...
    }
  }
}
```

Note: It is also allowed to add extra arguments before the 'event' argument, and provide its values in the **TKWIDGET** field

e.g. declare this tkwidget procedure with a first argument "-width 20"

```
TKWIDGET: GidUtils::TkwidgetEntryConfigure {-width 20}
```

and then define the Tcl procedure ready to get the first extra argument 'configure_options' before 'events':

```
proc GidUtils::TkwidgetEntryConfigure { configure_options event args } {
    ...
}
```

The argument event is the type of event and args is the list of arguments depending on the event type. The possible events are: **INIT**, **SYNC**, **CLOSE** and **DEPEND**. Below is a description of each event.

- **INIT**: this event is triggered when **GiD** needs to display the corresponding **QUESTION** and the list of arguments is {frame row-var GDN STRUCT QUESTION}: frame is the container frame where the widget should be placed; row-var is the name of the variable, used by **GiD**, with the current row in the frame; **GDN** and **STRUCT** are the names of internal variables needed to access the values of the data; **QUESTION** is the **QUESTION**'s name for which the **TKWIDGET** procedure was invoked. Normally the code for this event should initialize some variables and draw the widget.
- **SYNC**: this is triggered when **GiD** requires a synchronization of the data. Normally it involves updating some of the **QUESTIONS** of the data set. The argument list is {GDN STRUCT QUESTION}.
- **CLOSE**: this is triggered before closing the window, as mentioned this can be canceled if an **ERROR** is returned from the procedure. **DEPEND**: this event is triggered when a dependence is executed over the **QUESTION** for which the **TKWIDGET** is defined, ie. that **QUESTION** is an lvalue of the dependence. The list of arguments is {GDN STRUCT QUESTION ACTION value} where **GDN**, **STRUCT** and **QUESTION** are as before, **ACTION** could be **SET**, **HIDE** or **RESTORE** and value is the value assigned in the dependence.

The argument args is a variable amount of arguments, provided here as a list. Its content depends on the 'event' argument.

e.g.

INIT args: PARENT CURRENT_ROW_VARIABLE GDN STRUCT QUESTION

SYNC args: GDN STRUCT QUESTION

DEPEND args: GDN STRUCT QUESTION ACTION VALUE

CLOSE args: GDN STRUCT QUESTION

and its meaning is:

CURRENT_ROW_VARIABLE: store a name of variable that provide the integer row number of the current field

GDN and STRUCT: identify the data (e.g set value [DWLocalGetValue \$GDN \$STRUCT \$QUESTION])

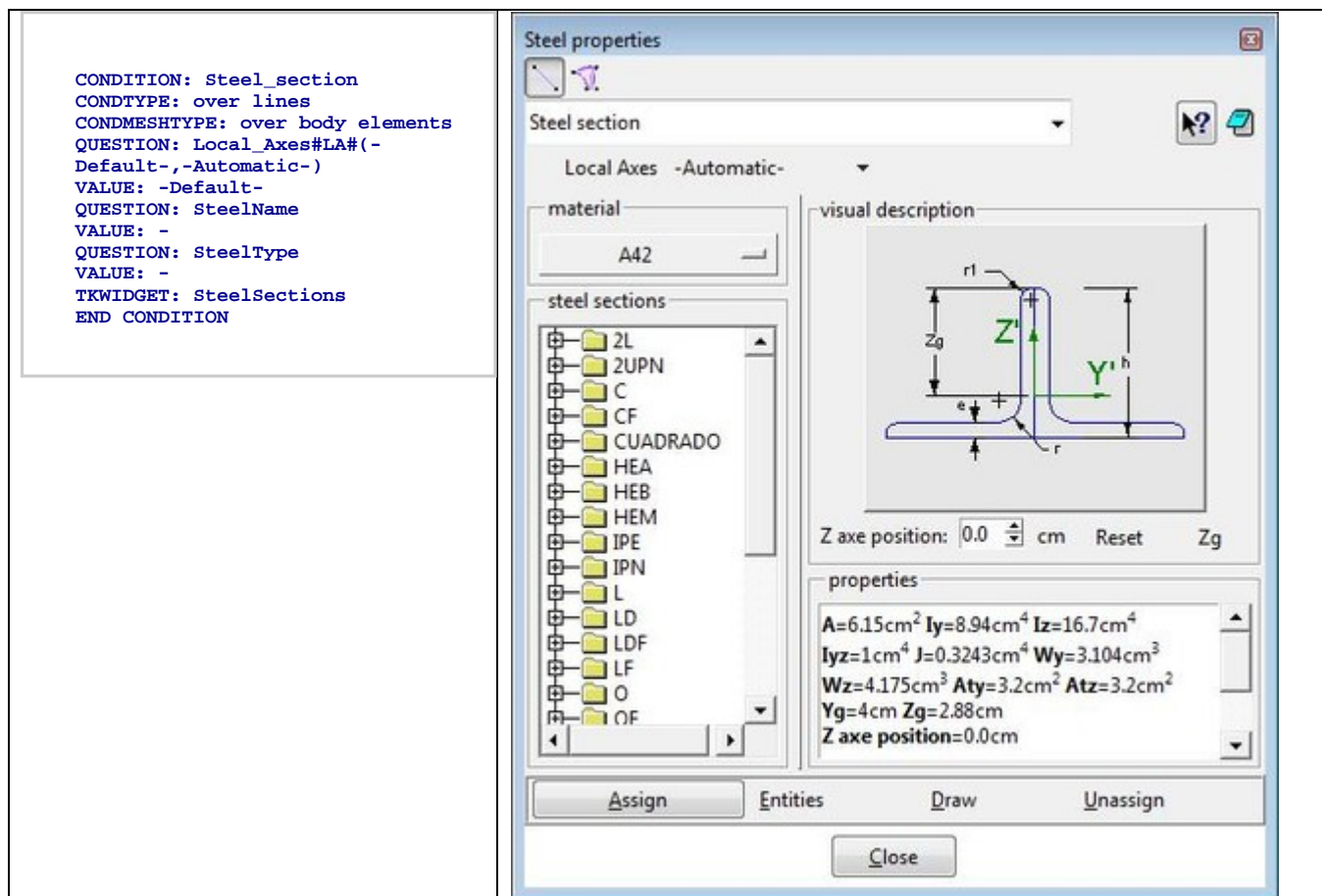
QUESTION is the name of the question that identify the field

ACTION could be "HIDE", "SET" or "RESTORE"

The procedure should return:

- an empty string "" meaning that every thing was OK;
- a two-list element {ERROR-TYPE Description} where ERROR-TYPE could be ERROR or WARNING. ERROR means that something is wrong and the action should be aborted. If ERROR-TYPE is the WARNING then the action is not aborted but Description is shown as a message. In any case, if Description is not empty a message is displayed.

The picture below shows a fragment of the data definition file and the **GUI** obtained. This sample is taken from the problem type RamSeries /rambshell and in this case the **TKWIDGET** is used to create the whole contents of the condition windows. For a full implementation, please download the problem type and check it.

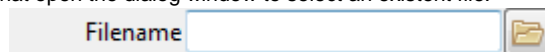


Predefined TKWIDGET procedures:

There are some useful features that have been implemented in tcl procedures provided by default in GiD, inside the dev_kit.tcl file, specially to replace the standard entry of a question by some specialized widget.

•GidUtils::TkwidgetGetFilenameButton

To show an entry and a select file button that open the dialog window to select an existent file.



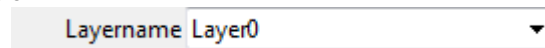
•GidUtils::TkwidgetPickPointOrNode

To show an entry and a button to pick in screen the id number of a point in geometry mode or a node in mesh mode



•GidUtils::TkwidgetGetLayername

To show a combobox with the current layers

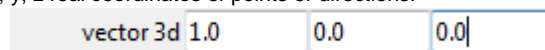


•GidUtils::TkwidgetGetGroupname

To show a combobox with the current groups

•GidUtils::TkwidgetGetVector3D

To show in a single row three entries for x, y, z real coordinates of points or directions.



Data windows behavior

In this subsection we explain a Tcl procedure used to configure the common behaviour of Materials. We are working on providing a similar functionality for Conditions using the same interface.

GiD_DataBehaviour controls properties of data windows for Materials and Conditions (not currently implemented). For Materials we can modify the behaviour of assign, draw, unassign, impexp (import/export), new, modify, delete and rename. We can also specify the entity type list with the assign option through the subcommands geomlist and meshlist.

The syntax of the procedure is as follows:

`GiD_DataBehaviour data_class name ?cmd? proplist`

where

- `data_class` could be "material" if we want to modify the behaviour of a particular material, or "materials" if a whole book is to be modified;
- `name` takes the value of a material's name or a book's name, depending on the value of `data_class`;
- In case that the materials are not classified in books the keyword "Default" means its default implicit
 - book.cmd can take one of the values: show, hide, disable, geomlist and meshlist;
 - `proplist` is a list of options or entity types. When the value of `cmd` is show, hide or disable, then `proplist` can be a subset of {assign draw unassign impexp new modify delete}. If the value of `cmd` is show it makes the option visible, if the value is hide then the option is not visible, and when the value is disable then the option is visible but unavailable. When the value of `cmd` is geomlist then `proplist` can take a subset of {points lines surfaces volumes} defining the entities that can have the material assigned when in geometry mode; if the value of `cmd` is meshlist then `proplist` can take the value elements. Bear in mind that only elements can have a material assigned in mesh mode. If `cmd` is not provided, the corresponding state for each of the items provided in `proplist` is obtained as a result.

Example:

```
GiD_DataBehaviour materials Default geomlist {surfaces volumes}
GiD_DataBehaviour materials Solid hide {delete impexp}
```

GiD_ShowBook is a procedure to hide/show a book from the menus

`GiD_ShowBook class book show`
where

- `class` must be: gendata materials conditions or intvdatatbook is the name of the book to be show or hidden show must be 0 or 1

After change the book properties is necessary to call to `GiDMenu::UpdateMenus`

Example:

```
GiD_ShowBook materials tables 0
GiDMenu::UpdateMenus
```

Interaction with themes

From GiD 11, a themes system has been integrated inside GiD.

In the following chapters, how to manage and use these system is explained in order to get a full integrated look of your module.

A theme contain:

- visual aspect as colors and shape of toolbars and windows. collection of cursors.
- collection of images (images can be classified in 2 categories, images on a root folder: logos, auxiliar images, ... and images representin icons, this ones can be found on subfolders grouped by image size).
- Definitions of which icon size is used in each categories (toolbars, menus).
- Default colors of entities and background (this colors will be applied to user the first time that the theme is charged, after that, user could change colors by going to preferences)

Now there are only two themes inside GiD: GiD_classic and GiD_black.

As this manual is for modules developers, you must know that the most common situation is to use most images provided by GiD and for only the new icons that you want to use in your module, implement the themes structure inside your module, creating the appropriated folder structure and configuration files, and providing images of the new icons for each theme.

Common

See source code of gid_themes package inside the scripts folder.

gid_themes::GetThemes

Return the list of available themes

Example:

```
gid_themes::GetThemes
```

-> GiD_black GiD_classic GiD_classic_renewed

GiD_Set Theme(Current)|Theme(Size)|Theme(MenuType)|Theme(HighResolutionScaleFactor)

Some theme options can be accessed to get/set as GiD variables, with the GiD_Set command

•Theme(Current) ?<theme_name>?

To get or set the current theme in use (to set a theme it is necessary a GiD restart)

Example:

```
GiD_Set Theme(Current)
```

-> GiD_classic

•Theme(Size)

Integer index that represent the icons collection size (small, medium, large,...)

•Theme(MenuType)

could be: *native, generic*

native:

On Mac OS X: use traditional Apple's menu bar instead of embed the menu bar inside the GiD main window

On Windows: use native menus

On Linux: is the same as generic

generic: Use Tk buttons for the menus

•Theme(HighResolutionScaleFactor)

Double value to scale the icons (1.0 by default), to avoid problems with some screen resolutions.

Asking for images

Use in your module the same image as GiD use

In order to use an image that GiD use, you must use the tcl function `gid_themes::GetImage`, to see a complet list of images available you can take a look for example to the folder:

(GiD Folder)\themes\GiD_classic\images\large_size(24andmore)

proc gid_themes::GetImage { filename IconCategory }

IconCategory could be: "small_icons", "medium_icons", "large_icons", "menu", "toolbar"

There is another IconCategory, "generic", that is the category used when the parameter is omitted. Using this category the image is retrieved from root image folder (Example: (GiD Folder)\themes\GiD_classic\images\), but the use of this category its not recommended, since images from root folders are not guaranteed on future versions.

The corresponding folder for each icon category is defined on the configuration of theme (will be one of the subfolders inside image folder)

Example: To get surface image for use on a button.

```
gid_themes::GetImage surface.png toolbar
```

This will return appropriate image depending on current theme, it could be for example:

(GiD Folder)\themes\GiD_classic\images\large_size(24andmore)\surface.png

Use in your module your own images

If your module needs other images from ones supplied by GiD

You can use:

`gid_themes::GetImageModule` to get the appropriate image, from inside the module folder, depending on current theme.

`gid_themes::GetImage { full_path_filename }`, image will be equal regardless of current theme. This is the 'old sytle', with the module images stored as module developer want, without follow the previously recommended folder layout.

Note: the `full_path_filename` points to a 'module' file, but it must be build in a relative way, based on the problemtype location.
e.g.

```
proc InitGIDProject { dir } {
    set full_path_filename [file join $dir my_images_folder my_image.png]
    set img [gid_themes::GetImage $full_path_filename]
    ...
}
```

To use **proc gid_themes::GetImageModule { filename IconCategory }**, you must replicate folder structure of (GiD Folder)\themes\ inside your module folder.

inside each theme must be a configuration.xml file (could be a copy of the one found in GiD) but also can be configured with only the following parameters:

```

<MODULE_THEME name="GiD_classic">
  <version>1.1</version>
  <alternative_theme>GiD_black</alternative_theme>

  <SizeDefinitions>
    <SizeDefinition name='small_size'>
      <label>small_size</label>
      <folder>small_size(16andmore)</folder>
      <alternative_folder>GiD_black/images/size16</alternative_folder>
    </SizeDefinition>

    <SizeDefinition name='large_size'>
      <label>large_size</label>
      <folder>large_size(24andmore)</folder>
      <alternative_folder>GiD_black/images/size24</alternative_folder>
    </SizeDefinition>
  </SizeDefinitions>

  <IconCategories>
    <menu>small_size</menu>
    <toolbar>large_size</toolbar>
    <small_icons>small_size</small_icons>
    <large_icons>large_size</large_icons>

  </IconCategories>

</MODULE_THEME>

```

The option "alternative_theme" its used if some file its not found, for try to find on the alternative theme (example themes still on develop) Also using this redirection, a complet themes that module is not interested on can be redirected to our main theme, in this case we will need just 1 folder for each theme and configuration.xml inside it.

Note: for high resolution displays, with high DPI, check also the **proc gid_themes::getDefaultHighResolutionScaleFactor {}** to scale you images. This functions returns a scale factor, which may be changed by the user in the preferences window, you'll need to scale your custom images and fonts. This factor is initially set to a value so that the medium theme size icons are legible in high DPI screens. The above mentioned functions, gid_themes::GetImage and gid_themes::GetImageModule already take into account this factor.

Forcing themes use

When a module is loaded from inside GiD, module can not control theme configuration.

But if you are installing GiD together with your module, there is a variable inside scripts/ConfigureProgram.tcl to control how GiD manage themes.

The variable is ::GidPriv(ThemeSelection), and its value can be:

- 1 (by default), user can choose the theme selection
- Any existing theme (Example: GiD_classic), user will be forced to use this theme and options on menus and windows about themes will be disabled.

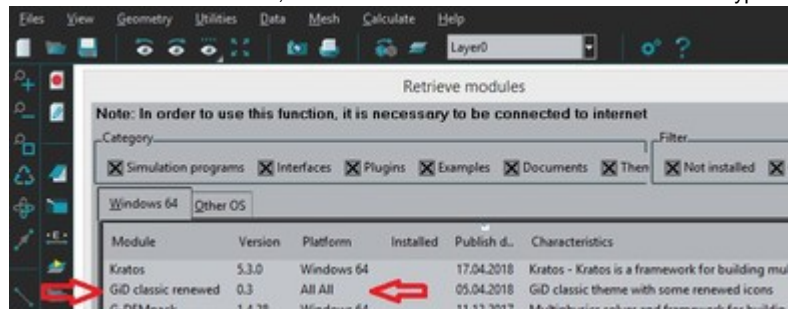
With this option its possible to obtain a package gid+module totally customiced with your prefered visual aspect.

Creating new themes

Create themes are only for modules that distribute they own package including GiD, or for GiD developers.

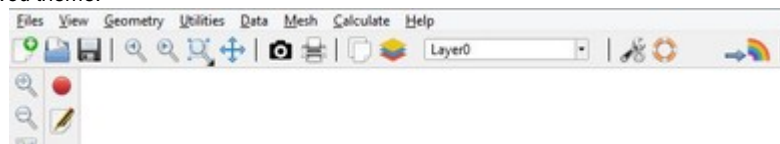
If you have created a theme for GiD and want that the theme will be distributed with GiD, just contact us at: gid@cimne.upc.edu

An example of a customized theme is *GiD classic renewed*, which can be found in GiD's Data --> Problem type --> Internet retrieve:



After downloading it, go to the *Preferences* window, select *Graphical* --> *Appearance* in the left tree and select the *GiD classic renewed* under GiD theme. GiD will ask you to restart the program for the changes to take effect.

Look of the *GiD classic renewed* theme:



This theme is a work in progress and new version will be released with new icons.

For creating a new theme you must know.

Inside (GiD_Folder)/themes:

Each folder represent a theme (Example (GiD_Folder)/themes/GiD_black)

Inside each folder must be a configuration.xml

This file must contain the following information:

```
<GID_THEME name="GiD_black">
  <info>  theme name is the same as its folder name
          images folder is always "GiD/themes/(name of theme)/images/"
          cursors folder is always "GiD/themes/(name of theme)/cursors/"
          if it does not find a folder it will use the alternative definition
          size folder start inside "images" theme folder
          alternative folder start from "GiD/themes/" folder
  </info>
  <label>GiD Black</label>
  <version>1.1</version>
  <alternative_theme>GiD_classic</alternative_theme>

  <SizeDefinitions>
    <SizeDefinition name='small_size'>
      <label>small size</label>
      <folder>size16</folder>
      <alternative_folder>GiD_classic/images/small_size(16andmore)</alternative_folder>
    </SizeDefinition>

    <SizeDefinition name='large_size'>
      <label>large size</label>
      <folder>size20</folder>
      <alternative_folder>GiD_classic/images/large_size(24andmore)</alternative_folder>
    </SizeDefinition>
  </SizeDefinitions>

  <IconCategories>
    <IconCategory name='menu'>size16</IconCategory>
    <IconCategory name='toolbar'>size20</IconCategory>
    <IconCategory name='small_icons'>size16</IconCategory>
    <IconCategory name='large_icons'>size20</IconCategory>
  </IconCategories>

  <OnlyDefineColorsOnMainGiDFrame>false</OnlyDefineColorsOnMainGiDFrame>
  <TtkTheme>newgid</TtkTheme>
  <TkFromTheme>true</TkFromTheme>

  <BackgroundColor> 000#000#000 </BackgroundColor>
  <BackColorType> 1 </BackColorType>
  <BackColorTop> #000000 </BackColorTop>
  <BackColorBottom> #323c4b </BackColorBottom>
```

```

<BackColorSense> d </BackColorSense>
<ColorPoints> 220#220#220 </ColorPoints>
<ColorNodes> 220#220#220 </ColorNodes>

<ColorLines> 091#094#225 </ColorLines>
<ColorPolyLines> 000#143#039 </ColorPolyLines>
<ColorSurfaces> 218#036#220 </ColorSurfaces>
<ColorSurfsIsoparametric> 220#218#036 </ColorSurfsIsoparametric>
<ColorVolumes> 086#217#216 </ColorVolumes>
<ColorElements> 153#153#153 </ColorElements>
</GID_THEME>

```

This file configure colors of entities and background, also which images to display and where to find images, but for total different look of GiD, you must understand this 3 lines:

```

<OnlyDefineColorsOnMainGiDFrame>false</OnlyDefineColorsOnMainGiDFrame>
<TtkTheme>newgid</TtkTheme>
<TkFromTheme>true</TkFromTheme>

```

If option *OnlyDefineColorsOnMainGiDFrame* is true, only GiD's main frame will be modified with the visual aspects, windows and other elements will remain in native look (as GiD_classic does)

The second option associate a ttk theme in this case is *<TtkTheme>newgid</TtkTheme>*, the configuration of ttk themes can be found on: (GiD_Folder)\scripts\gid_themes\ltk_themes\ by changing the ttk configuration, we can achieve any look of GiD.

The last option *TkFromTheme* its important to obtain a complet integrated look, tk colors will be adapted with ttk colors, we recoment oposite value of *OnlyDefineColorsOnMainGiDFrame* (*true=> false, false=>true*)

GiD version

The version of GiD is returned by [GiD_Info gidversion](#)

Normally, a problem type requires a minimum version of GiD to run. Because the problem type can be distributed or sold separately from GiD, it is important to check the GiD version before continuing with the execution of the problem type. GiD offers a function, **GidUtils::VersionCmp**, which compares the version of the GiD currently being run with a given version.

GidUtils::VersionCmp { Version }

This returns a negative integer if Version is greater than the currently executed GiD version; zero if the two versions are identical; and a positive integer if Version is less than the GiD version.

Example:

```
proc InitGIDProject { dir } {  
    set VersionRequired "10.0"  
    if {[GidUtils::VersionCmp $VersionRequired] < 0 } {  
        WarnWin [= "This interface requires GiD %s or later" $VersionRequired]  
    }  
}
```


PLUG-IN EXTENSIONS

This section explains a new way to expand GiD capabilities: the plug-in mechanism. Plug-ins which should be used by GiD should be present inside the **\$GiD/plugins** directory. There are two possible plugin mechanisms:

- Tcl plug-in
- GiD Dynamic library plug-in

Tcl plug-in

If a file with extension .tcl is located inside the GiD 'plugins' folder, with the same name as the folder containing it, then it is automatically sourced when starting GiD.

To avoid source some unwanted tcl file is also required the existence of an XML file with the same name as the folder and .xml extension, and a syntax like this:

```
<InfoPlugin version="1.0">
  <Program>
    <Name>Collada</Name>
    <Version>0.1</Version>
    <MinimumGiDVersion>14.1.8d</MinimumGiDVersion>
    <Description>GiD plugin to import Collada .dae files</Description>
    <NewsInVersion version="0.1" date="2019-11-01">
      * First version import triangles mesh with texture
    </NewsInVersion>
  </Program>
</InfoPlugin>
```

This allow to do what the developer want, with Tcl language, e.g. change menus, source other Tcl files or load a 'Tcl loadable library' that extend the Tcl language with new commands implemented at C level.

To know how to create a 'Tcl loadable library' some Tcl book must be read.

See chapter about 'C Programming for Tcl' of

"Practical Programming in Tcl and Tk" by Brent Welch, Ken Jones, and Jeff Hobbs at <http://www.beedub.com/book>

Tcl Mesh plug-in

This is a particular case of a Tcl plug-in to generate the mesh from geometric entities.

It is a special case, because it requires that GiD invoke our procedures while meshing, and provide the input of each geometric entity, and store the generated mesh, and it requires also to offer this new algorithm as a possible meshing method, and handle its generic parameters with the model and in preferences.

GiD nowadays is ready to implement only plugins of volume mesher of spheres (and circles for surfaces), other cases must be implemented in a similar way in the future...

An example of mesh plug-in is the 'granular' mesher to generate spheres or circles.

It is implemented by the files of the folder <GiD>\plugins\Mesh\Spheres

GiD mesh plug-in mechanism:

1- Register the meshing procedure

To register a new sphere volume mesher this proc must be called

```
proc GiD_RegisterPluginGenerateSphereMesh { value procedure label require_tetrahedra }
```

value is an integer that identifies the value of the mesher (a GiD variable stores the kind of sphere mesher to be used)
the value 0 is reserved for the GiD inner sphere mesher

- the value 1 is used for the "Granular" mesher
- other values must be used for future sphere meshers...
- procedure is a tcl procedure that will be called by GiD when meshing each volume (in case that was marked to be meshed with spheres). This procedure must have this prototype:

```
proc procname { id_entity input_mesh input_boundary_parts size } {  
    ...  
    return [list $nodes_coordinates $element_radius $output_boundary_parts]  
}
```

id_entity is the id of the volume that will be meshed

input_mesh defines the input mesh: nodes coordinates and elements (if require_tetrahedra == 0 the triangles of the boundary that define the volume)

== [list [GiD_Info Mesh Nodes 1 end -array] [GiD_Info Mesh Elements Triangle 1 end -array]]

input_boundary_parts allow define parts of the boundary associated to each surface of the volume, to allow classify the output spheres mesh related to these surfaces, to apply boundary conditions, layers, groups, materials...

size is the general mesh size asked by the user

The procedure inside must do what he wants, in this case write the information in a temporary file, and start sphere_mesher.exe, the is the really sphere mesher, read its results in another auxiliary file, and finally return the data as three items, to define the sphere centers, its radius, and possible extra classification of parts.

return [list \$nodes_coordinates \$element_radius \$output_boundary_parts]

- label is the string that will identify the mesher in GiD windows (Utilities->Preferences... Meshing->Sphere/Circle)
- require_tetrahedra must be 0 or 1 (0 if the input of the mesher are the triangles closing the volume to be meshed with spheres, 1 if it requires the tetrahedra filling the volume)

The file Spheres.tcl do this:

```
#register the procedure Spheres::GenerateSpheres as GiD-Tcl event to generate the mesh when the value  
of SphereMesher is 1 (showed with label $info_plugin(Name))  
GiD_RegisterPluginGenerateSphereMesh 1 Spheres::GenerateSpheres $info_plugin(Name) 0
```

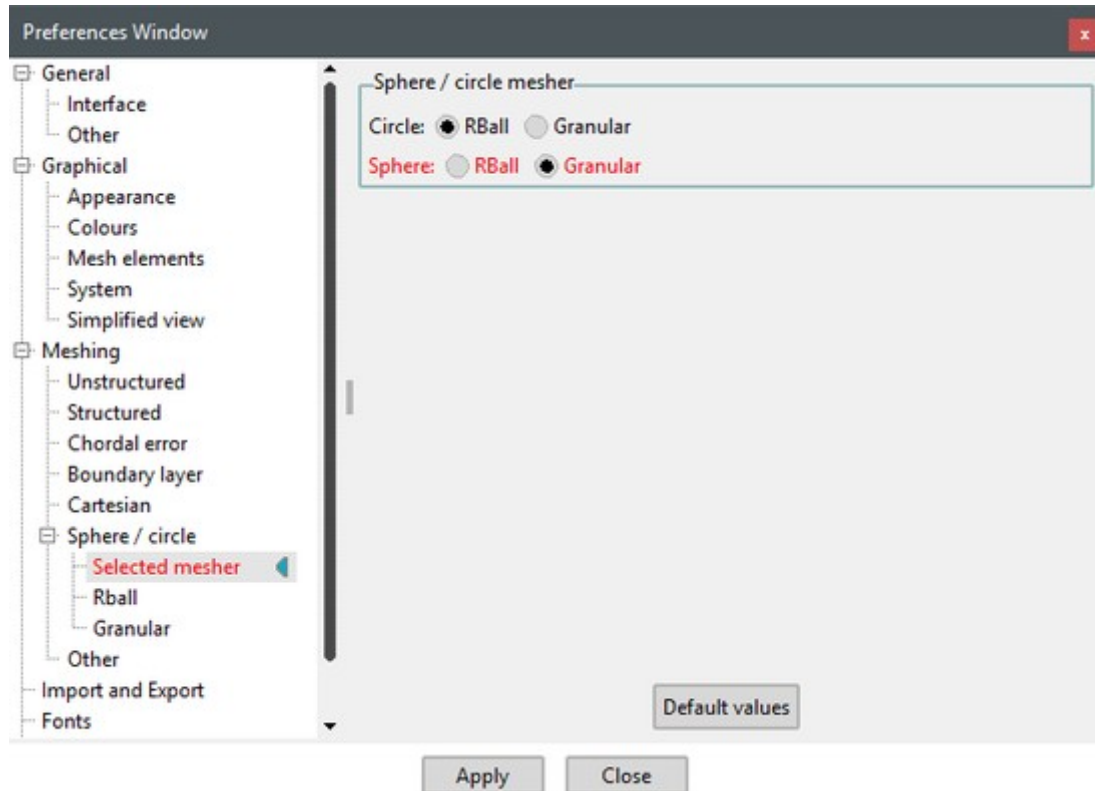
and define the procedure that somehow do the mesh, and return it in the format expected by GiD

```

proc Spheres::GenerateSpheres { id_entity input_mesh input_boundary_parts size } {
    ...
    return [list $nodes_coordinates $element_radius $output_boundary_parts]
}

```

Then the registered name "Granular" appear as a meshing method to generate circles and/or spheres in the preferences window



2- Mesh parameters

Usually the mesher will require some general parameters, and several copies of these parameters must be handled by GiD.

A copy of values is used to fill the Utilities->Preferences... Meshing window

Another copy of values is used the the model, and saved/restored with it in disk

A third copy is used while meshing, with the values of one of them: from current preferences or from the previous values of the model

To handle several copies Granular.tcl uses the packate TclOO to define a class with the variables required for the mesher.

This class is named SphereMeshVariables and must be derived from GiDMeshVariables

```

# class that store the mesher variables derived from GiDMeshVariables
package require TclOO

if { [info commands SphereMeshVariables] != "" } {
    SphereMeshVariables destroy ;#to protect of multiple source
}

oo::class create SphereMeshVariables {
    superclass GiDMeshVariables
    #variables showed in preferences:
    variable MinRadiusFactor MaxRadiusFactor PartSizeDist
    variable DistributionParam,StandarDeviation DistributionParam,Scale DistributionParam,Width
    variable Random BoundaryTolerance
    #variables hidden in preferences:
    variable ParticleType
    variable Seed Advanced Priority
    variable HighPorosity Porosity

    constructor { owner } {
        next $owner ;#to invoke the parent constructor
    }

    method GetDefault { key } {
        if { $key == "ParticleType" } {
            set value 1
        } elseif { $key == "MaxRadiusFactor" } {
            set value 1.2
        } elseif { $key == "MinRadiusFactor" } {
            set value 0.8
        } elseif { $key == "PartSizeDist" } {
            set value 0
        } elseif { $key == "DistributionParam,StandarDeviation" } {
            set value 0.1
        } elseif { $key == "DistributionParam,Scale" } {
            set value 1.0
        } elseif { $key == "DistributionParam,Width" } {
            set value 1.0
        } elseif { $key == "Random" } {
            set value 0
        } elseif { $key == "Seed" } {
            set value 1
        } elseif { $key == "Priority" } {
            set value 0
        } elseif { $key == "BoundaryTolerance" } {
            set value 1.0
        } elseif { $key == "Advanced" } {
            set value 0
        } elseif { $key == "HighPorosity" } {
            set value 0
        } elseif { $key == "Porosity" } {
            set value 0.95
        } else {
            set value 0
            WarnWinText "SphereMeshVariables::GetDefault. Unexpected key $key"
        }
        return $value
    }

    method IsValid {} {
        set valid 1
        if { $ParticleType != 1 } {
            set valid 0
        } elseif { $MinRadius <= 0 } {
            set valid 0
        }
        return $valid
    }
    export GetDefault IsValid
}

```

And this class is declared (registered) to GiD as 'mesh variables'

```

#to register this class as a inner mesher and handle its variables (used copy in the model, in
preferences and the ones to use meshing)
GiD_RegisterPluginMeshVariablesClass SphereMeshVariables

```

The values that are required by the mesher are the ones of 'MESHING'. The object with these values is obtained with

```

set obj [PluginMeshVariablesClass_GetClassObject SphereMeshVariables MESHING]

```

(the values of 'PREFERENCES' and 'MODEL' are handled internally by GiD to save/restore values with the model and with the user preferences)

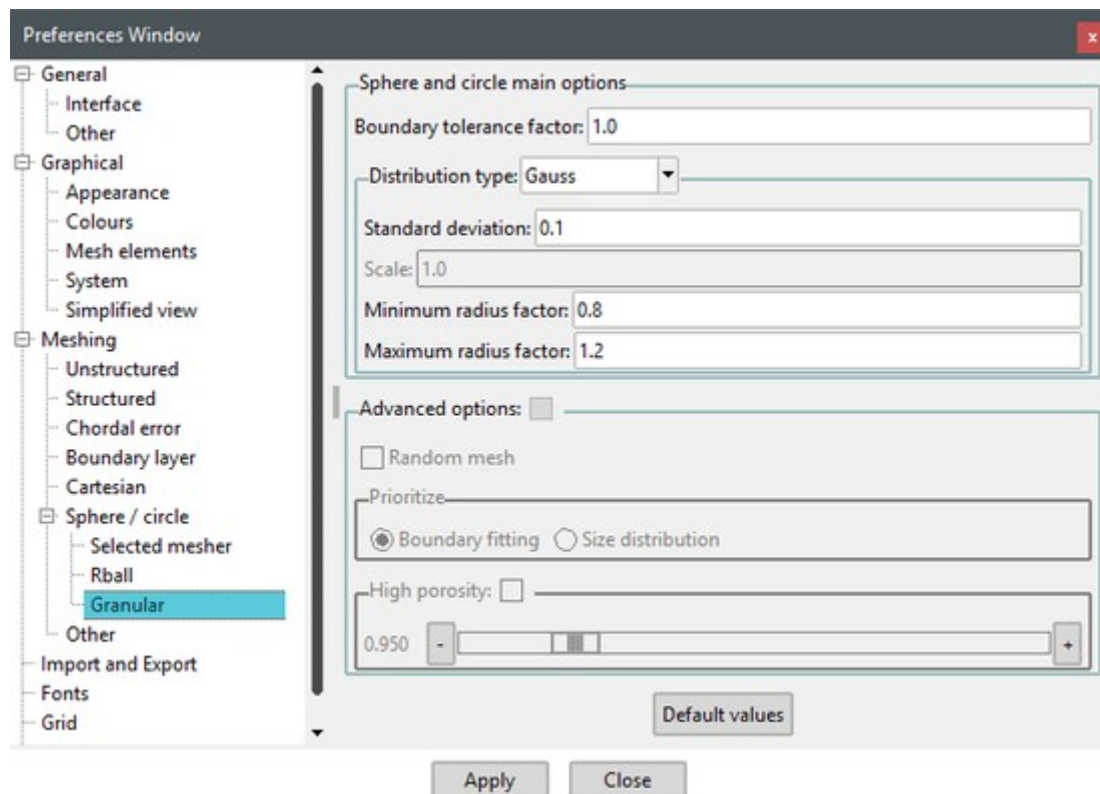
To define how to show in Utilities->Preferences... Meshing these mesh variables a XML file SpheresPreferences.xml was created with this content:

```
<group name='mesher_granular' label='Granular'>
  <labelframe name='sphere_and_circle_main_options' label='Sphere and circle main options'>
    <entry name='boundary_flag' variable='BoundaryTolerance' variablemanager='Spheres::VariableManager'
label='Boundary tolerance factor' help='Factor to select spheres close to the boundary to apply
conditions' validation='IsFloatingPointPositive'>
      <comboboxframe name='distribution_type' variable='PartSizeDist' variablemanager='Spheres::
VariableManager' label='Distribution type' help='Type of distribution'>
        <option value='0' label='Gauss' setactivate='distribution_standarde deviation min_radius_factor
max_radius_factor'>
          <option value='1' label='Exponential' setactivate='distribution_scale min_radius_factor
max_radius_factor'>
            <option value='3' label='Cauchy' setactivate='distribution_scale min_radius_factor
max_radius_factor'>
              <option value='5' label='Flat' setactivate='min_radius_factor max_radius_factor'>
                <option value='6' label='Constant'>
                  <entry name='distribution_standarde deviation' variable='DistributionParam,StandarDeviation'
variablemanager='Spheres::VariableManager' label='Standard deviation' help='Standar deviation of the
gaussian distribution' validation='IsFloatingPointPositive'>
                    <entry name='distribution_scale' variable='DistributionParam,Scale' variablemanager='Spheres::
VariableManager' label='Scale' help='?' validation='IsFloatingPointPositive'>
                      <entry name='min_radius_factor' variable='MinRadiusFactor' variablemanager='Spheres::
VariableManager' label='Minimum radius factor' help='Minimum radius=min_factor*mean_radius'
validation='IsFloatingPointPositive'>
                        <entry name='max_radius_factor' variable='MaxRadiusFactor' variablemanager='Spheres::
VariableManager' label='Maximum radius factor' help='Maximum radius=max_factor*mean_radius,
max_factor>=1' validation='IsFloatingPointPositive'>
                          </comboboxframe>
                        </labelframe>
                      <checkboxbuttonframe variable='Advanced' variablemanager='Spheres::VariableManager' label='Advanced
options'>
                        <checkboxbutton name='random' variable='Random' variablemanager='Spheres::VariableManager'
label='Random mesh' help='Allow Random Generator Seed'>
                          <radiobuttonframe name='prioritize_boundary_or_distribution' label='Prioritize' variable='Priority'
variablemanager='Spheres::VariableManager' help=''>
                            <option value='0' label='Boundary fitting' help='Prioritize boundary fitting'>
                              <option value='1' label='Size distribution' help='Prioritize size distribution'>
                                </radiobuttonframe>
                              <checkboxbuttonframe variable='HighPorosity' variablemanager='Spheres::VariableManager' label='High
porosity'>
                                <scale variable='Porosity' variablemanager='Spheres::VariableManager' whenreadvar='FormatG' from='
0.94' to='0.998' resolution='0.001' showvalue='0' showbuttons='1' needsredraw='1' help='This option
gives the porosity needed to achieve'>
                                  <entry width='6" state='readonly' validation='IsFloatingPoint'>
                                    </scale>
                                  </checkboxbuttonframe>
                                </checkboxbuttonframe>
                              </checkboxbuttonframe>
                            </checkboxbuttonframe>
                          </checkboxbuttonframe>
                        </checkboxbuttonframe>
                      </checkboxbuttonframe>
                    </checkboxbuttonframe>
                  </checkboxbuttonframe>
                </checkboxbuttonframe>
              </checkboxbuttonframe>
            </checkboxbuttonframe>
          </checkboxbuttonframe>
        </checkboxbuttonframe>
      </checkboxbuttonframe>
    </checkboxbuttonframe>
  </checkboxbuttonframe>
</group>
```

Spheres::VariableManager is a procedure used in this xml to allow set/get/reset these variables in a predefined way

This new tab is created from the xml definition to represent the parameters required by this meshing algorithm

```
proc Spheres::VariableManager { operation var {value ""} } {
  return [PluginMeshVariablesClass_VariableManager SphereMeshVariables $operation $var $value]
}
```



3- User feedback

Advance bar:

To update a progress bar while meshing the granular procedure is invoking

```
GiD_ProgressInMeshing $entity_type $id_entity $factor $num_elements $num_elements
```

this update the GiD progress bar, and allow the user stop the meshing (the procedure must stop its work if the user ask it) the percent of mesh done could be passed from sphere_mesher.exe in a simple way using an extra file

Error messages:

In case of detect errors, the procedure registered to mesh, instead of return a list of three items, could return a single item with the error message (provided in an error file written by the mesher).

GiD will finally show the error message in its own standard window.

GiD dynamic library plug-in

This is a particular mechanism of plugin, different of the Tcl-plugin, and it only allows to import mesh and results in postprocess, and require to compile a special dll with our rules.

Note that 'GiD dynamic libraries' are different of 'Tcl loadable libraries'

'GiD dynamic libraries' must do specifically the task that GiD expects: now it is only available an interface for libraries that import mesh and create results for GiD postprocess. In the future new interfaces to do other things could appear, and to be usable must follow the rules explained in this chapter.

Introduction

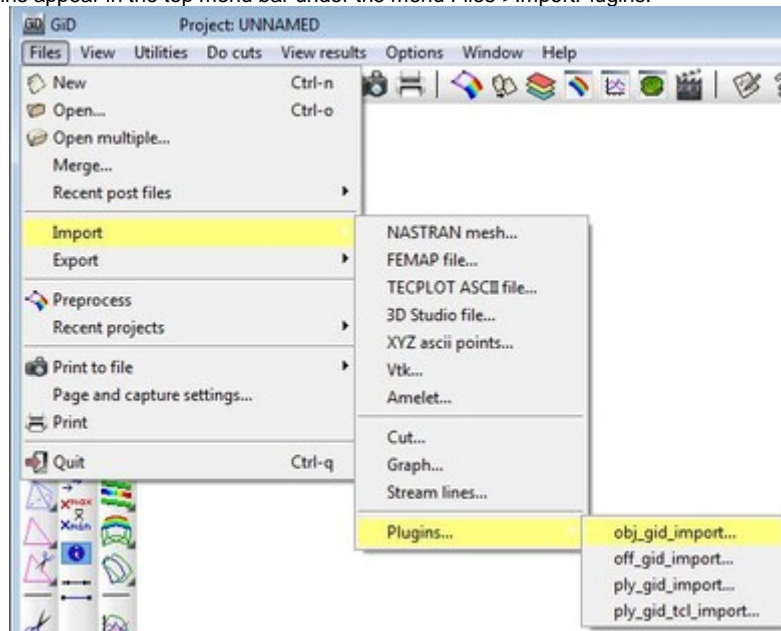
As the variety of existent formats worldwide is too big to be implemented in GiD and, currently, the number of supported formats for mesh and results information in GiD is limited, the GiD team has implemented a new mechanism which enables third party libraries to transfer mesh and results to GiD, so that GiD can be used to visualize simulation data written in whatever format this simulation program is using. This new mechanism is the well know mechanism of plug-ins. Particularly GiD supports the loading of dynamic libraries to read any simulation data and transfer the mesh and results information to GiD.

Viewing GiD as a platform of products, this feature allows a further level of integration of the simulation code in GiD by means of transferring the results of the simulation to GiD in any format specified by this simulation code thus avoiding the use of a foreign format.

The recognized plug-ins are automatically loaded in GiD and appear in the top menu bar in the Files->Import->Plugins submenu.

In GiD

The recognized import plug-ins appear in the top menu bar under the menu Files->ImportPlugins:



These dynamic libraries can be manually loaded and called via TCL scripts, in GiD post-process's command line, or using the post-process's right menu 'Files ImportDynamicLib' and the options LoadDynamicLib, UnloadDynamicLib, CallDynamicLib:

For one plug-in library, named **MyImportPlugin.dll** (or **MyImportPlugin.so** in Linux or **MyImportPlugin.dylib** in mac OS X) to be automatically recognized by GiD and to be loaded and listed in the top's menu Files->Import->Plugins, the library should lie inside a directory of the same name, i.e. MyImportPlugin/MyImportPlugin.dll, under any sub-folder of the %GiD%/plugins/Import directory:

Note that only the GiD 32 bits version can handle 32 bits import plug-in dynamic libraries, and only GiD 64 bits can handle 64 bits import plug-in dynamic libraries. Which version of GiD is currently running can be easily recognized in the title bar of the main window (Title bar of GiD's window showing 'GiD x64', so the current GiD is the 64 bits version)

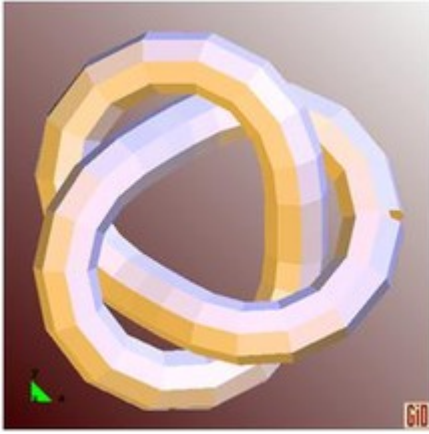
Together with the GiD installation, following import plug-ins are provided:

OBJ: Wavefront Object format from Wavefront Technologies

OFF: Object file format vector graphics file from Geomview

PLY: Polygon file format, aka Stanford Triangle Format, from the Stanford graphics lab.

PLY-tcl: this plug-in is the same as the above PLY one but with a tcl's progress bar showing the tasks done in the library while a ply file is imported. For all of these plug-in examples both the source code, the Microsoft Visual Studio projects, Makefiles for Linux and Mac OS X, and some little models are provided



The 'tref.off' Object File Format example



'bunny_standford.ply' Polygon File Format

Developing the plug-in

GiD is compiled with the Tcl/Tk libraries (the Tcl version could be seen in Help->About - More...).

Remember that if the developed plugin is targeted for 32 bits, only GiD 32 bits can handle it. If the developed plugin is developed for 64 bits systems, then GiD 64 bits is the proper one to load the plugin.

•Header inclusion

In the plug-in code, in one of the .cc/.cpp/.cxx source files of the plug-in, following definition must be made and following file should be included:

```
#define BUILD_GID_PLUGIN
#include "gid_plugin_import.h"
```

In the other .cc/.cpp/.cxx files which also use the provided functions and types, only the gid_plugin_import.h file should be included, without the macro definition.

The macro is needed to declare the provided functions as pointers so that GiD can find them and link with its internal functions.

•Functions to be defined by the plug-in

Following functions should be defined and implemented by the plug-in:

```
extern "C" GID_DLL_EXPORT int GiD_PostImportFile( const char *filename) ) {
    ... ;
    return 0; // 1 - on error
}
extern "C" GID_DLL_EXPORT const char *GiD_PostImportGetLibName( void) {
    return "Wavefront Objects import";
}
extern "C" GID_DLL_EXPORT const char *GiD_PostImportGetFileExtensions( void) {
    return "{{Wavefront Objects} {.obj}} {{All files} {*} }";
}
extern "C" GID_DLL_EXPORT const char *GiD_PostImportGetDescription( void) {
    return "Wavefront OBJ import plugin for GiD";
}
extern "C" GID_DLL_EXPORT const char *GiD_PostImportGetErrorStr( void) {
    return _G_err_str; // if error, returns the error string
}
```

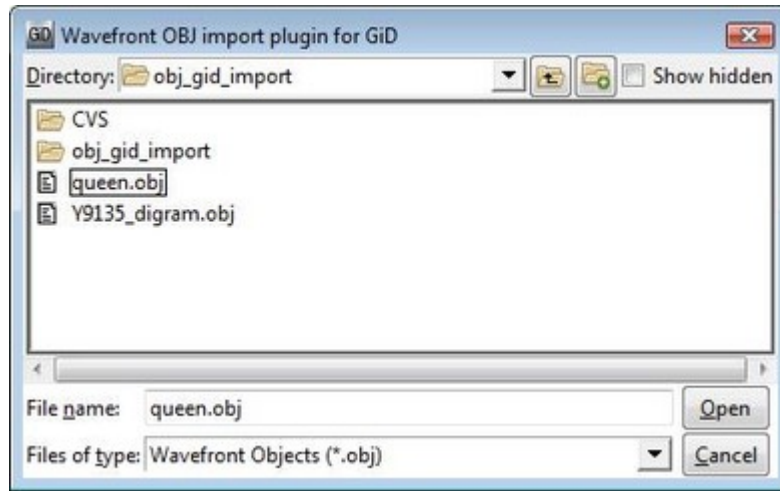
When GiD is told to load the dynamic library, it will look for, and will call these functions:

GiD_PostImportGetLibName : returns the name of the library and should be unique. This name will appear in the File->Import->Plugin menu and in the right menu.

GiD_PostImportGetFileExtensions : which should return a list of extensions handled by the library and will be used as filter in the Open File dialogue window.

GiD_PostImportGetDescription : returns the description of the library and will be displayed in the title bar of the Open File dialogue window.

Once the library is registered, when the user selects the menu entry File->Import->Plugin->NewPlugin the Open File dialogue window will appear showing the registered filters and description of the plug-in.



The file selection window showing the plug-in description as title of the window and filtering the file list with the registered extension

When the user selects a file then following functions are called:

GiD_PostImportFile : this function should read the file, transfer the mesh and results information to GiD and return 0 if no problems appeared while the file was read or 1 in case of error.

GiD_PostImportGetErrorStr : this function will be called if the previous one returns 1, to retrieve the error string and show the message to the user.

Functions provided by GiD

Inside the `GiD_PostImportFile` function, following functions can be called to pass information to GiD:

```
extern "C" int GiD_NewPostProcess( void);

extern "C" int GiD_NewMesh( _t_gidMode gid_mode, _t_gidMeshType mesh_type, const
char *name);

extern "C" int GiD_SetColor( int id, float red, float green, float blue, float alpha);

extern "C" int GiD_SetVertexPointer( int id,
_t_gidBasicType basic_type,
_t_gidVertexListType list_type,
int num_components,
int num_vertices,
unsigned int offset_next_element, const void *pointer);

extern "C" int GiD_SetElementPointer( int id,
_t_gidBasicType basic_type,
_t_gidElementListType list_type,
_t_gidElementType element_type,
int num_elements,
unsigned int offset_next_element,
const void *pointer,
unsigned int offset_float_data,
const void *float_ptr);

extern "C" int GiD_NewResult( const char *analysis_name, double step_value,
const char *result_name, int mesh_id);

extern "C" int GiD_SetResultPointer( int id,
_t_gidBasicType basic_type,
_t_gidResultListType list_type,
_t_gidResultType result_type,
_t_gidResultLocation result_location,
int num_results,
unsigned int offset_next_element,
const void *pointer);

extern "C" int GiD_EndResult( int id);

extern "C" int GiD_EndMesh( int id);

extern "C" Tcl_Interp *GiD_GetTclInterpreter();
```

Here is the description for each provided function:

GiD_NewPostProcess : starts a new post-process session, deleting all mesh and results information inside GiD.

GiD_NewMesh : a new mesh will be transferred to GiD and an identifier will be returned so that more information can be defined for this mesh. Following parameters must be specified:

`_t_gidMode gid_mode` : may be one of `GID_PRE` or `GID_POST`. At the moment only `GID_POST` is supported

`_t_gidMeshType mesh_type` : may be one of `GIDPOST_NEW_MESH`, `GIDPOST_MERGE_MESH` and `GIDPOST_MULTIPLE_MESH`. At the moment only `GIDPOST_NEW_MESH` has been tested

`const char *name` : name of the mesh which will appear in the Display Style window.

GiD_SetColor : to specify a colour for the mesh identified by the given id. The red, green, blue and alpha components should be between 0.0 and 1.0.

GiD_SetVertexPointer : sets the vertices of the mesh identified by the given id. This vertices are the ones to be referred from the element's connectivity. Following parameters may be set:

`_t_gidBasicType basic_type` : data type of the coordinates of the vertices, should be one of `GIDPOST_FLOAT` or `GIDPOST_DOUBLE`;

`_t_gidVertexListType list_type` : herewith the format of the vertices is specified. Should be one of

`GIDPOST_VERTICES`: where all `num_components` coordinates are specified with no label and so they will be numerated between 0 and `num_vertices-1`

`GIDPOST_IDX_VERTICES`: where each set of `num_components` coordinates are preceded by a label indicating its node number (should be a 4-byte integer)

`int num_components` : number of coordinates per vertex

`int num_vertices` : number of vertices in the list

`unsigned int offset_next_element` : distance in bytes between the beginning of vertex `i` and the beginning of vertex `i + 1`. If 0 is entered then the vertices are all consecutive

`const void *pointer` : pointer to the list of vertices.

GiD_SetElementPointer : sets the elements of the mesh identified by the given id. The elements connectivity refers to the previous specified list of vertices. Note that for spheres and circles not only their connectivity should be specified but also their radius and eventually their normal. In this case two separate vectors should be passed: one for the integer data and another one for the floating point data. Following parameters may be set:

`_t_gidBasicType basic_type` : data type of the extra data entered for sphere and circle elements, should be one of `GIDPOST_FLOAT` or `GIDPOST_DOUBLE`.

`_t_gidElementListType list_type` : herewith the format of the elements is specified. Should be one of `GIDPOST_CONNECTIVITIES`: where all the elements are specified without element number, thus being automatically numbered between 0 and `num_elements-1`

`GIDPOST_IDX_CONNECTIVITIES`: where each element is preceded by a label indicating its element number (should be a 4-byte integer)

`_t_gidElementType element_type` : type of element to be defined. May be one of `GIDPOST_TRIANGLE`, `GIDPOST_QUADRILATERAL`, `GIDPOST_LINE`, `GIDPOST_TETRAHEDRON`, `GIDPOST_HEXAHEDRON`, `GIDPOST_POINT`, `GIDPOST_PRISM`, `GIDPOST_PYRAMID`, `GIDPOST_SPHERE`, `GIDPOST_CIRCLE`.

`int num_elements` : number of elements in the list

`unsigned int offset_next_element` : distance in bytes between the beginning of element `i` and the beginning of element `i+1`. If 0 is entered then the elements are all consecutive

`const void *pointer` : pointer to the list of the elements connectivity (integer data)

`unsigned int offset_float_data` : distance in bytes between the beginning of float data of element `i` and the beginning of float data of element `i+1`. If 0 is entered then the element's float data are all consecutive.

`const void *float_ptr` : pointer to the list of the floating point data for the elements. For spheres only the radius should be specified, so just a single value, and for circles four values should be specified: its radius and the three components of the normal.

GiD_NewResult : a new result will be defined for GiD and an identifier will be returned so that more information can be defined for this result. Following parameters must be specified:

`const char *analysis_name` : analysis name of the result

`double step_value` : step value inside the analysis where the result should be defined

`const char *result_name` : result name

`int mesh_id` : mesh identifier where the result is defined. If 0 is entered the result will be defined for all meshes.

GiD_SetResultPointer : specifies the list with the result values for a given result's id. Following parameters may be set:

`_t_gidBasicType basic_type` : data type of the results, should be one of `GIDPOST_FLOAT` or `GIDPOST_DOUBLE`

`_t_gidResultListType list_type` : herewith the format of the results is specified. Should be one of `GIDPOST_RESULTS`: where all results are defined consecutively and will refer to the nodes / elements between 0 and `num_results-1`

`GIDPOST_IDX_RESULTS`: where each result is preceded by a label indicating its node /element number (should be a 4-byte integer)

`_t_gidResultType result_type` : type of result which will be defined. May be one of `GIDPOST_SCALAR`, `GIDPOST_VECTOR_2` (vector result with 2 components), `GIDPOST_VECTOR_3` (vector with 3 components), `GIDPOST_VECTOR_4` (vector with 4 components, including signed modulus), `GIDPOST_MATRIX_3` (matrix with 3 components `Sxx`, `Syy` and `Sxy`), `GIDPOST_MATRIX_4` (`Sxx`, `Syy`, `Sxy` and `Szz`, `GIDPOST_MATRIX_6` (`Sxx`, `Syy`, `Sxy`, `Szz` and `Syz` and `Sxz`), `GIDPOST_EULER` (with 3 euler angles), `GIDPOST_COMPLEX_SCALAR` (real and imaginary part), `GIDPOST_COMPLEX_VECTOR_4` (2d complex vector: `Vxr`, `Vxi`, `Vyr` and `Vyi`), `GIDPOST_COMPLEX_VECTOR_6` (3d complex vector: `Vxr`, `Vxi`, `Vyr`, `Vyi`, `Vzr` and `Vzi`) and `GIDPOST_COMPLEX_VECTOR_9` (3d complex vector: `Vxr`, `Vxi`, `Vyr`, `Vyi`, `Vzr`, `Vzi`, |real part|, |imaginary part| and signed |vector|)

`_t_gidResultLocation result_location` : location of the result. May be one of `GIDPOST_NODAL`, `GIDPOST_ELEMENTAL` or `GIDPOST_GAUSSIAN`. At the moment `GIDPOST_GAUSSIAN` is not supported

`int num_results` : number of results in the list

`unsigned int offset_next_element` : distance in bytes between the beginning of result `i` and the beginning of result `i+1`. If 0 is entered then the results are all consecutive

`const void *pointer` : pointer to the list of results.

GiD_EndResult : indicates GiD that the definition of the result with the give id is finished. GiD will process the result.

GiD_EndMesh : indicates GiD that the definition of the mesh with the give id is finished. GiD will process the mesh.

GiD_GetTclInterpreter : returns a pointer to GiD's global interpreter so that the plug-in can open their windows or execute their tcl scripts using the predefined tcl procedures of GiD.

The developer should keep in mind that all the plug-in code is executed inside GiD's memory space and so, all the memory allocated inside the plug-in should also be freed inside the plug-in to avoid memory accumulation when the dynamic library is called repeatedly. This also includes the arrays passed to GiD, which can be deleted just after passing them to GiD.

List of examples

The plug-in examples provided by GiD also include some little models of the provided import format. These are the import plug-ins provided by GiD so far:

OBJ: Wavefront OBJ format

This is a starter example which includes the `create_demo_triangs` function which creates a very simple mesh. The obj format is a very simple ascii format and this plug-in:
reads the file,
creates a GiD mesh with the read triangles and quadrilaterals,
and, if the information about the vertex normals is present, then this information is passed to GiD as nodal vector results.

OFF: Object file format

This example is very similar to the previous one. The off format is a very simple ascii format but including n-agons and colour on vertices and faces. So, this plug-in:
reads the file,
creates a GiD mesh with the read triangles and quadrilaterals and triangulates the read pentagons and hexagons (and discards bigger n-agons),
if colour information is present in the off file, which can be present on the nodes or on the elements, then this information is passed to GiD as nodal or elemental results.

PLY: Polygon file format

This example is a little bit more complex. Ply files can be ascii or binary, and the code of this plug-in is based in Greg Turk's code, developed in 1998, to read ply files. This format allows the presence of several properties on nodes and faces, too. This plug-in:
reads the file,
creates a GiD mesh with the read lines, triangles and quadrilaterals,
if information about the vertex normals is found, then this information is passed to GiD as nodal vector results,
all the properties defined in the ply file are passed to GiD. This properties can be defined on the nodes or on the faces of the model, and so are they transferred to GiD.
Here the complexity also resides in the liberation of the reserved memory, which is wildly allocated in the ply code.

PLY + Tcl : Polygon file format

This plug-in is the same as the previous PLY plug-in but a tcl script is added inside the code to show a progress bar in tcl to keep the user entertained while big files are read.

APPENDIX A (PRACTICAL EXAMPLES)

- To learn how to configure GiD for a particular type of analysis, you can find some practical examples: By following the tutorial of the chapter **Defining a problem type** of the GiD user manual.
- By studying and modifying some existing Problem Types

Problem types included in GiD by default as example in \$GID/problemtypes/Examples:

- **cmas2d**: This is the problem type created in the tutorial, which finds the distance of each element relative to the center of masses of a two-dimensional surface. It uses the following files: .cnd, .mat, .prb, .bas, .tcl and .bat. There is a help file inside directory **cmas2d.gid** called **cmas2d.html**
- **cmas2d_customlib**: The same problem type, but implemented using the 'CustomLib library'.
- **cmas2d_customlib_wizard**: The same problem type, but implemented using the GiD Smart Wizard package, to generate a wizard GUI
- **complex_example**: uses some basic Tcl/Tk interaction (in complex_example.tcl) with GiD to:
 - add a new menu in GiD's menu bar;
 - create an icon bar for the problem type, with their own images;
 - using conditions to evaluate user defined formulae at the nodes of the domain: look into complex_example.cnd and complex_example.bas .

Other problemtypes can be downloaded from the **Data->Problem type->Internet retrieve** menu:

- **Kratos**: Multiphysics FEM open source C++ code.
- **RamSeries**: This is a problem type which performs the structural analysis of either beams or shells or a combination of both using the Finite Element Method. This problem type uses the latest features offered by GiD . The .exe file for Windows systems is also included in a limited version.
- **Tdyn**: Multiphysics solver (including CFD, heat transfer, species advection, pde solver and free surface problems).

CompassFEM is a suite that includes both Tdyn and RamSeries codes

- **NASTRAN**: Static and dynamic interface for the NASTRAN commercial analysis program (not included)

For the full version without limitations check <http://www.compassis.com>.

APPENDIX B (classic problemtype system)

From version 13 of GiD, a new system of problem types has been developed, which offers several advantages compared with the old (classic) system: it organizes better the data, provides with a more intuitive and user-friendly GUI, enable more sophisticated integration tools, etc... From this version on, this 'classic' system of problemtypes is considered deprecated, however, it is still supported by GiD.

The classic problem type system uses the files .prb, .mat, .cnd, .uni to define general properties, materials, conditions, units. Each category of data is showed in a kind of window, and materials and conditions are associated to entities.

About writing the input file for the solver, the classical approach uses simple .bas GiD templates

PROBLEMTYPE 'CLASSIC'

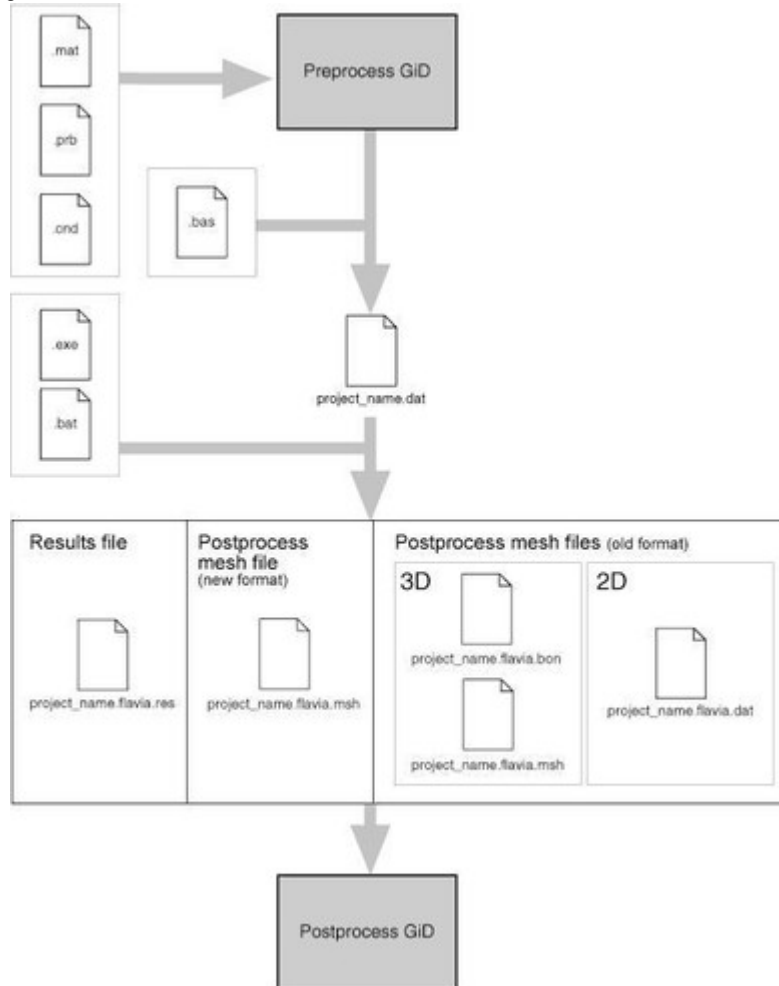
The creation of a **Problem Type** involves the creation of a directory located inside the \problemtypes GiD folder, with the name of the problem type and the extension .gid.

Note: now it is also possible to have a /problemtypes extra folder located at the <GiD user preferences folder>. This is interesting in case that the user doesn't has privileges to copy a problemtype inside <GiD folder>/problemtypes

Problemtypes that exists when GiD start will appear in its menu (see Problem type from Reference Manual).

The problemtype configuration files are a collection of files inside this subfolder. The name for most of them will follow the format problem_type_name.xxx where the extension refers to their particular function.

Considering problem_type_name to be the name of the problem type and project_name the name of the project, file configuration is described by the following diagram:



- **Directory name:** problem_type_name.gid
- **Directory location:** c:\a\b\c\GiD_directory\problemtypes

Configuration files

- problem_type_name.xml XML-based configuration
- problem_type_name.cnd Conditions definitions
- problem_type_name.mat Materials properties
- problem_type_name.prb Problem and intervals
- dataprobem_type_name.uni Units Systems

problem_type_name.sim Conditions symbols

- ***.geo Symbols geometrical definitions
- ***.geo Symbols geometrical definitions ...

•Template files

- problem_type_name.bas Information for the data input file
- ***.bas Information for additional files
- ***.bas Information for additional files ...

•Tcl extension files

- problem_type_name.tcl Extensions to GiD written in the Tcl/Tk programming language

•Command execution files

- problem_type_name.bat Operating system shell that executes the analysis process

The files problem_type_name.sim, ***.geo and ***.bas are not mandatory and can be added to facilitate visualization (both kinds of file) or to prepare the data input for restart in additional files (just ***.bas files). In the same way problem_type_name.xml is not necessary; it can be used to customize features such as: version info, icon identification, password validation, etc.

CONFIGURATION FILES

These files generate the conditions and material properties, as well as the general problem and intervals data to be transferred to the mesh, at the same time giving you the chance to define geometrical drawings or symbols to represent some conditions on the screen.

Conditions file (.cnd)

Files with extension .cnd contain all the information about the conditions that can be applied to different entities. The condition can adopt different field values for every entity. This type of information includes, for instance, all the displacement constraints and applied loads in a structural problem or all the prescribed and initial temperatures in a thermal analysis.

An important characteristic of the conditions is that they must define what kind of entity they are going to be applied over, i.e. over points, over lines, over surfaces, over volumes, over layers or over groups, and what kind of mesh entity they will be transferred over, i.e. over nodes, over face elements or over body elements.

- Over nodes** This means that the condition will be transferred to the nodes contained in the geometrical entity where the condition is assigned.
- Over face elements ?multiple?** If this condition is applied to a line that is the boundary of a surface or to a surface that is the boundary of a volume, this condition is transferred to the higher elements, marking the affected face. If it is declared as **multiple**, it can be transferred to more than one element face (if more than one exists). By default it is considered as **single**, and only one element face will be marked.
- Over body elements** If this condition is applied to lines, it will be transferred to line elements. If assigned to surfaces, it will be transferred to surface elements. Likewise, if applied to volumes, it will be transferred to volume elements.

Note: For backwards compatibility, the command 'over elements' is also accepted; this will transfer the condition either to elements or to faces of higher level elements. Another important feature is that all the conditions can be applied to different entities with different values for all the defined intervals of the problem.

Therefore, a condition can be considered as a list of fields containing the name of the particular condition, the geometric entity over which it is applied, the mesh entity over which it will be transferred, its corresponding properties and their values.

The format of the file is as follows:

```
CONDITION: condition_name
CONDTYPE: 'over points', 'over lines', 'over surfaces', 'over volumes', 'over layers', 'over groups'
CONDMESHTYPE: 'over nodes', 'over face elements', 'over face elements multiple', 'over body elements'
GROUPALLOW: points lines surfaces volumes nodes elements faces
QUESTION: field_name['#CB#'(...,optional_value_i,...)]
VALUE: default_field_value['#WIDTH#'(optional_entry_length)]
...
QUESTION: field_name['#CB#'(...,optional_value_i,...)]
VALUE: default_field_value['#WIDTH#'(optional_entry_length)]
END CONDITION

CONDITION: condition_name
...
END CONDITION
```

Note: CONDTYPE and CONDMESHTYPE are compulsory, and only a kind of type must be set.

Note: GROUPALLOW is only valid for conditions 'over groups', is a special optional field to restrict allowed categories of entities of the group to the ones listed (if this field is missing then all kind of entities are allowed). A list of multiple types could be set.

Note: #CB# means Combo Box.

Note: #WIDTH# means the size of the entry used by the user to enter the value of the condition. Specifies an integer value indicating the desired width of the entry window, in average-size characters of the widget's font.

Local Axes

QUESTION: field_name['#LA#('global','automatic','automatic alternative','automatic main')]

VALUE: default_field_value['#WIDTH#(optional_entry_length)]

This type of field refers to the **local axes** system to be used. The position of the values indicates the kind of **local axes**.

If it only has a single default value, this will be the name of the global axes.

If two values are given, the second one will reference a system that will be computed automatically for every node and will depend on geometric constraints, like whether or not it is tangent, orthogonal, etc.

If a third value is given, it will be the name of the automatic alternative axes, which are the automatic axes rotated 90 degrees.

If a fourth value is given, it will be the name of the automatic main axes, valid only for surfaces using the main curvature directions. (note that sometimes main curvatures are not well defined, e.g. for a planar surface or a sphere all directions are main directions, because the curvature is constant)

All the different user-defined systems will automatically be added to these default possibilities.

To enter only a specific kind of local axes it is possible to use the modifiers #G#,#A#,#L#,#M#.

- #G#: global axes;
- #A#: automatic axes;
- #L#: automatic alternative axes.
- #M#: main curvature axes

When using these modifiers the position of the values does not indicate the kind of local axes.

Example

```
QUESTION: Local_Axes#LA#(Option automatic#A#,Option automatic_alt#L#)
VALUE: -Automatic-
```

Note: All the fields must be filled with words, where a word is considered as a string of characters without any blank spaces. The strings signaled between quotes are literal and the ones inside brackets are optional. The interface is case-sensitive, so any uppercase letters must be maintained. The default_field_value entry and various optional_value_i entries can be alphanumeric, integers or reals. GiD treats them as alphanumeric until the moment they are written to the solver input files.

Global axes:

X=1 0 0

Y=0 1 0

Z=0 0 1

Automatic axes:

For surfaces, this axes are calculated from the unitary normal N:

$z'=N$

if N is coincident with the global Y direction (N_x or $N_z >$ some tolerance) then

$x'=Y \times N / |Y \times N|$

else

$x'=Z \times N / |Z \times N|$

$y'=N \times x'$

$z'=N$

For lines, this axes are calculated from the unitary tangent T:

$x'=T$

if T is coincident with the global Z direction (N_x or $N_y >$ some tolerance) then

$y'=Y \times x' / |Y \times x'|$

else

$y'=Z \times x' / |Z \times x'|$

$z'=x' \times y'$

Automatic alternative axes:

They are calculated like the automatic case and then swap x and y axes:

$x''=y'$

$y''=-x'$

$z''=z'$

For curves

x' =unitary tangent to the curve on the place where the condition is applied

If this tangent is different of the Z global axe=(0,0,1) then

$y'=Y \times x'$

else

$y'=Z \times x'$

$z'=x' \times y'$

Note: the tangent x' is considered different of (0,0,1) if the first or second component is greater than 1/64

Main curvature axes:

They are calculated for surfaces finding on a point the directions where the curvatures are maximum and minimum, but these directions are not always well defined.

e.g. in a planar point the curvature is zero in all directions, all directions could be seen as main directions, and in a sphere the curvature is constant=1/Radius and it happens the same.

Multiple assign:

By default a condition can be applied only once to the same entity and last assignation replace previous one, but this behavior can be changed:

One flag that can optionally be added to a condition is:

CANREPEAT: yes

It is written after CONDMESHTYPE and means that one condition can be assigned to the same entity several times.

Self Calculated #FUNC# fields:

Another type of field that can be included inside a condition is a #FUNC# to do some calculation,

where the key #FUNC#, means that the value of this field will be calculated just when the mesh is generated. It can be considered as a function that evaluates when meshing.

Valid variables for a #FUNC# field are:

- NumEntity: to track the numerical id of the geometric source entity
- x y z : to use the coordinates of the node or entity center where the condition is applied
- Cond(num_field,REAL): to use the value of other fields of this condition (REAL or INT declare that must be considered as a real or integer number)
- Valid mathematical operations are the same as the used for the *Operation template command.

e.g.

```
QUESTION: Surface_number#FUNC#(NumEntity)
VALUE: 0
```

In the above example, NumEntity is one of the possible variables of the function. It will be substituted by the label of the geometrical entity from where the node or element is generated.

In this second example, the x variable is used, which means the x-coordinate of the node or of the center of the element. Others fields of the condition can also be used in the function. Variables y and z give the y- and z-coordinates of this point.

```
QUESTION: X_press#FUNC#(Cond(3,REAL)*(x-Cond(1,REAL))/(Cond(2,REAL)-Cond(1,REAL)))
VALUE: 0
```

Note: There are other options available to expand the capabilities of the Conditions window (see [Special fields](#)).

Example: Creating the conditions file

Here is an example of how to create a conditions file, explained step by step:

First, you have to create the folder or directory where all the problem type files are located, `problem_type_name.gid` in this case. Then create and edit the file (`problem_type_name.cnd` in this example) inside the recently created directory (where all your problem type files are located). As you can see, except for the extension, the names of the file and the directory are the same. Create the first condition, which starts with the line:

```
CONDITION: Point-Constraints
```

The parameter is the name of the condition. A unique condition name is required for this conditions file.

- This first line is followed by the next pair:

```
CONDTYPE: over points
CONDMESHTYPE: over nodes
```

which declare what entity the condition is going to be applied over. The first line, `CONDTYPE:...` refers to the geometry, and may take as parameters the sentences "over points", "over lines", "over surfaces" or "over volumes".

The second line refers to the type of condition applied to the mesh, once generated. GiD does not force you to provide this second parameter, but if it is present, the treatment and evaluation of the problem will be more accurate. The available parameters for this statement are "over nodes" and "over elements".

- Next, you have to declare a set of questions and values applied to this condition.

```
QUESTION: Local-Axes#LA#(-GLOBAL-)
VALUE: -GLOBAL-
QUESTION: X-Force
VALUE: 0.0
QUESTION: X-Constraint:#CB#(1,0)
VALUE: 1
QUESTION: X_axis:#CB#(DEFORMATION_XX,DEFORMATION_XY,DEFORMATION_XZ)
VALUE: DEFORMATION_XX
END CONDITION
```

After the `QUESTION:` prompt, you have the choice of putting the following kinds of word:

- An alphanumeric field name.
- An alphanumeric field name followed by the `#LA#` statement, and then the single or double parameter.
- An alphanumeric field name followed by the `#CB#` statement, and then the optional values between parentheses.

The `VALUE:` prompt must be followed by one of the optional values, if you have declared them in the previous `QUESTION:` line. If you do not observe this format, the program may not work correctly.

In the previous example, the X-Force `QUESTION` takes the value 0.0. Also in the example, the X-Constraint `QUESTION` includes a Combo Box statement (`#CB#`), followed by the declaration of the choices 1 and 0. In the next line, the value takes the parameter 1. The X_axis `QUESTION` declares three items for the combo box: `DEFORMATION_XX,DEFORMATION_XY,DEFORMATION_XZ`, with the value `DEFORMATION_XX` chosen.

Beware of leaving blank spaces between parameters. If in the first question you put the optional values (`-GLOBAL-`, `-AUTO-`) (note the blank space after the comma) there will be an error when reading the file. Take special care in the Combo Box question parameters, so as to avoid unpredictable parameters.

- The conditions defined in the `.cnd` file can be managed in the Conditions window (found in the **Data** menu) in the Preprocessing component of GiD.



Problem and intervals data file (.prb)

Files with the extension .prb contain all the information about general problem and intervals data. The general problem data is all the information required for performing the analysis and it does not concern any particular geometrical entity. This differs from the previous definitions of conditions and materials properties, which are assigned to different entities. An example of general problem data is the type of solution algorithm used by the solver, the value of the various time steps, convergence conditions, etc.

Within this data, one may consider the definition of specific problem data (for the whole process) and intervals data (variable values along the different solution intervals). An interval would be the subdivision of a general problem that contains its own particular data. Typically, one can define a different load case for every interval or, in dynamic problems, not only variable loads, but also variation in time steps, convergence conditions and so on.

The format of the file is as follows:

```

PROBLEM DATA
QUESTION: field_name['#CB#'(...,optional_value_i,...)]
VALUE: default_field_value
...
QUESTION: field_name['#CB#'(...,optional_value_i,...)]
VALUE: default_field_value
END PROBLEM DATA

INTERVAL DATA
QUESTION: field_name['#CB#'(...,optional_value_i,...)]
VALUE: default_field_value
...
QUESTION: field_name['#CB#'(...,optional_value_i,...)]
VALUE: default_field_value
END INTERVAL DATA

```

All the fields must be filled with words, where a word is considered as a string of characters without any blank spaces. The strings signaled between quotes are literal and the ones inside brackets are optional. The interface is case-sensitive, so any uppercase letters must be maintained. The default_field_value entry and various optional_value_i entries can be alphanumeric, integers or real numbers, depending on the type.

Note: There are other options available to expand the capabilities of the Problem Data window (see [Special fields](#)).

Example: Creating the PRB data file

Here is an example of how to create a problem data file, explained step by step:

Create and edit the file (problem_type_name.prb in this example) inside the problem_type_name directory (where all your problem type files are located). Except for the extension, the names of the file and the directory must be the same.
Start the file with the line:

```
PROBLEM DATA
```

Then add the following lines:

```
QUESTION: Unit_System#CB#(SI,CGS,User)
VALUE: SI
QUESTION: Title
VALUE: Default_title
```

The first question defines a combo style menu called Unit_System, which has the SI option selected by default. The second question defines a text field called Title, and its default value is Default_title.

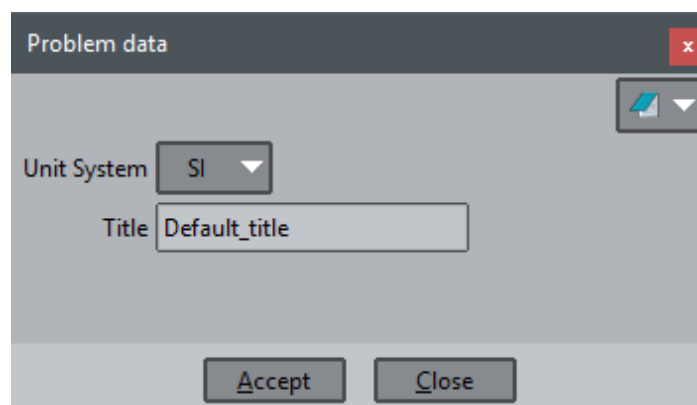
••To end the file, add the following line:

```
END PROBLEM DATA
```

The whole file is as follows:

```
PROBLEM DATA
QUESTION: Unit_System#CB#(SI,CGS,User)
VALUE: SI
QUESTION: Title
VALUE: Default_title
END PROBLEM DATA
```

•The options defined in the .prb file can be managed in the Problem Data window (found in the **Data** menu) in the Preprocessing component of GiD.



Materials file (.mat)

Files with the extension .mat include the definition of different materials through their properties. These are base materials as they can be used as templates during the Preprocessing step for the creation of newer ones.

You can define as many materials as you wish, with a variable number of fields. None of the unused materials will be taken into consideration when writing the data input files for the solver. Alternatively, they can be useful for generating a materials library.

Conversely to the case of conditions, the same material can be assigned to different levels of geometrical entity (lines, surfaces or volumes) and can even be assigned directly to the mesh elements.

In a similar way to how a condition is defined, a material can be considered as a group of fields containing its name, its corresponding properties and their values.

The format of the file is as follows:

```
MATERIAL: material_name
QUESTION: field_name['#CB#'(...,optional_value_i,...)]
VALUE: default_field_value
...
QUESTION: field_name['#CB#'(...,optional_value_i,...)]
VALUE: default_field_value
END MATERIAL
MATERIAL: material_name
...
END MATERIAL
```

All the fields must be filled with words, where a word is considered as a string of characters without any blank spaces. The strings signaled between quotes are literal and the ones within brackets are optional. The interface is case-sensitive, so any uppercase letters must be maintained. The default_field_value entry and various optional_value_i entries can be alphanumeric, integers or real numbers, depending on their type.

Note: There are other options available to expand the capabilities of the Materials window (see [Special fields](#)).

Example: Creating the materials file

Here is an example of how to create a materials file, explained step by step:

- Create and edit the file (problem_type_name.mat in this example) inside the problem_type_name directory (where all your problemtype files are located). As you can see, except for the extension, the names of the file and the directory are the same.
- Create the first material, which starts with the line:

```
MATERIAL: Air
```

The parameter is the arbitrary name of the material. A unique material name is required for this into this materials file (do not use blank spaces in the name of the material).

- The next two lines define a property of the material and its default value:

```
QUESTION: Density
VALUE: 1.0
```

- You can add as many properties as you wish. To end the material definition, add the following line:

```
END MATERIAL
```

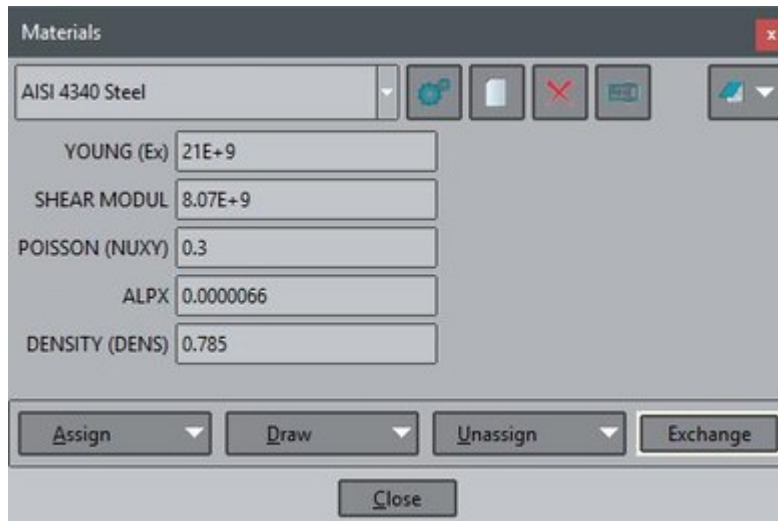
In this example we have introduced some materials; the .mat file would be as follows:

```

MATERIAL: Air
QUESTION: Density
VALUE: 1.01
END MATERIAL
MATERIAL: AISI_4340_Steel
QUESTION: YOUNG_(Ex)
VALUE: 21E+9
QUESTION: SHEAR_MODUL
VALUE: 8.07E+9
QUESTION: POISSON_(NUXY)
VALUE: 0.3
QUESTION: ALPX
VALUE: 0.0000066
QUESTION: DENSITY_(DENS)
VALUE: 0.785
END MATERIAL
MATERIAL: Concrete
QUESTION: Density
VALUE: 2350
END MATERIAL

```

- The materials defined in the .mat file can be managed in the Materials window (found in the **Data** menu) in the Preprocessing component of GiD.



Special fields

•Array fields

Fields of conditions, problem data or materials could store an array of values, and the length of this array is not predefined, could be set at runtime.

For example, if a material has a variable property (an example would be where a property was dependent on temperature and was defined with several values for several temperatures) a table of changing values may be declared for this property. When the solver evaluates the problem, it reads the values and applies a suitable property value.

The declaration of the table requires two lines of text:

The first is a QUESTION line with a list of alphanumeric values between parentheses.

```
QUESTION: field_name(column_title_1,...,column_title_n)
```

These values are the names of each of the columns in the table so that the number of values declared is the number of columns.

This first line is followed by another with the default data values. It starts with the words VALUE: #N#, and is followed by a number that indicates the quantity of elements in the matrix and, finally, the list of values.

```
VALUE: #N# number_of_values value_1 ... value_m
```

The number of values m declared for the matrix obviously has to be the number of columns n multiplied by the number of rows to be filled.

e.g.

```
MATERIAL: Steel
QUESTION: TypeId
VALUE: Metal
STATE: Hidden
QUESTION: Internal_Points(X,Y,Z)
VALUE: #N# 3 0.0 0.0 0.0
HELP: Internal points coordinates
END MATERIAL
```

and example writing the values of this material from the .bas template: [TEMPLATE FILES](#)

```
*loop materials
*if(strcmp(Matprop(TypeId),"Metal")==0)
*set var N=Matprop(Internal_Points,int)
X Y Z
*for(i=1;i<=N;i=i+3)
*Matprop(Internal_Points,*i) *Matprop(Internal_Points,*operation(i+1)) *Matprop(Internal_Points,
*operation(i+2))
*end for
*endif
*end materials
```

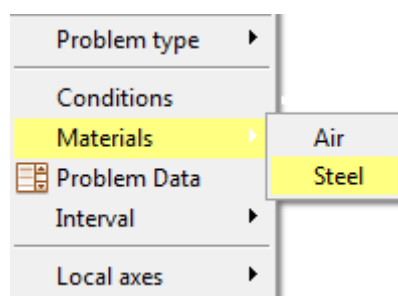
Note that in the example a hidden field named 'TypeId' is used to identify this and its derivated materials.

•Aesthetic fields:

These fields are useful for organizing the information within data files. They make the information shown in the data windows more readable. In this way you can better concentrate on the data properties relevant to the current context.

- Book:** With the **Book** field it is possible to split the data windows into other windows. For example, we can have two windows for the materials, one for the steels and another for the concretes:

```
BOOK: Steels
...
All steels come here
...
BOOK: Concretes
...
All concretes come here
...
```



The same applies to conditions. For general and interval data the **book** field groups a set of properties.

- Title:** The **Title** field groups a set of properties on different tabs of one book. All properties appearing after this field will be included on this tab.


```

TITLE: Basic
...
Basics properties
....
TITLE: Advanced
...
Advanced properties
....

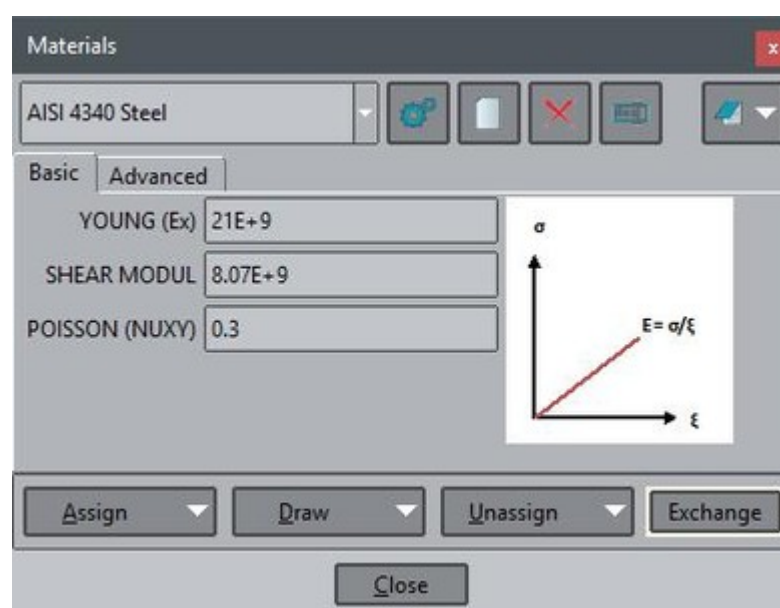
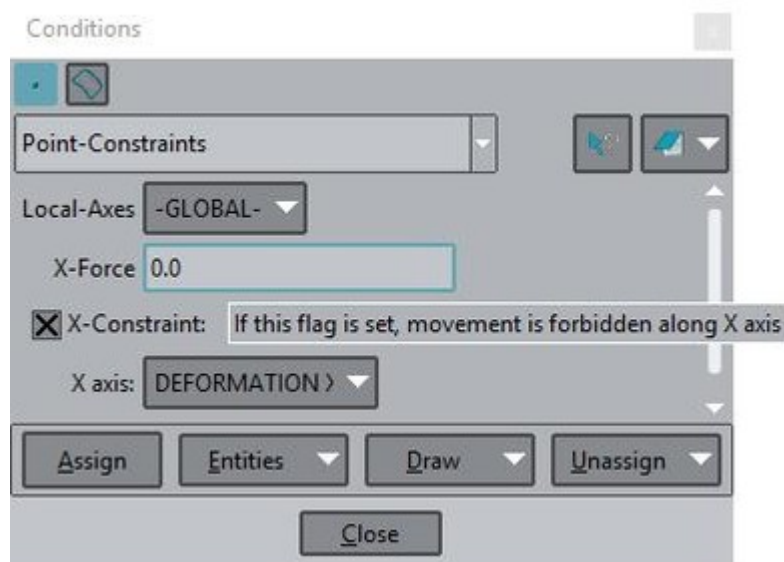
```

•**Help:** With the **Help** field it is possible to assign a description to the data property preceding it. In this way you can inspect the meaning of the property through the help context function by holding the cursor over the property or by right-clicking on it.

```

QUESTION: X-Constraint#CB#(1,0)
VALUE: 1
HELP: If this flag is set, movement is forbidden along X axis

```

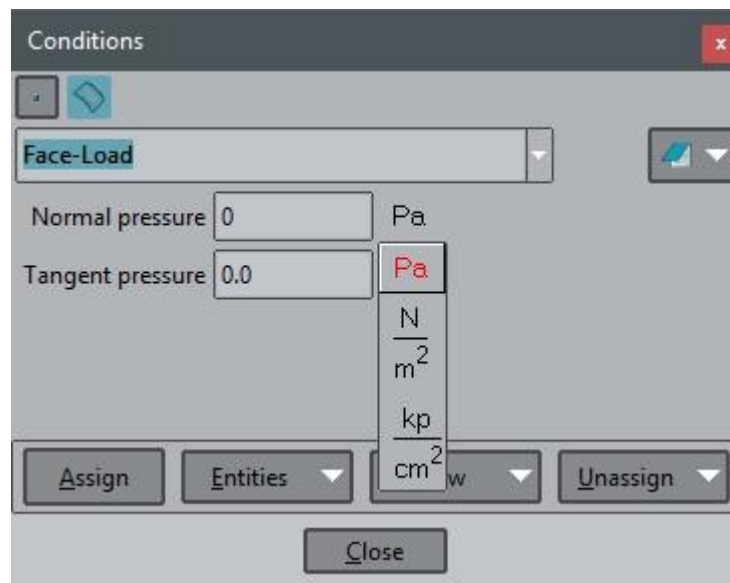


•**Image:** The **Image** field is useful for inserting descriptive pictures in the data window. The value of this field is the file name of the picture relating to the problem type location.

IMAGE: young.png

•**Unit field:** With this feature it is possible to define and work with properties that have units. **GiD** is responsible for the conversion between units of the same magnitude

```
....
QUESTION: Normal_pressure#UNITS#
VALUE: 0.0Pa
....
```



•**Dependencies:** Depending on the value, we can define some behavior associated with the property. For each value we can have a list of actions. The syntax is as follows:

```
DEPENDENCIES <V1>,[ TITLESTATE,<Title>,<State> ],<A1>,<P1>,<NV1>,...,<An>,<Pn>,<NVn> ) ... ( <Vm>,<Am>,<Pm>,<NVm>,... )
```

Where:

<Vi> is the value that triggers the actions. A special value is **#DEFAULT#**, which refers to all the values not listed.

[**TITLESTATE**,<Title>,<State>] this argument is optional. **Titlestate** should be used to show or hide book labels. Many **Titlestate** entries can be given. <Title> is the title defined for a book (TITLE: Title). **State** is the visualization mode: normal or hidden.

<Ai> is the action and can have one of these values: **SET**, **DISABLE**, **HIDE**, **RESTORE**. All these actions change the value of the property with the following differences:

SET assign the value, triggering its dependencies **DISABLE** disables the property

HIDE hides the property

RESTORE brings the property to the enabled and visible state.

Note: **SET** will trigger its dependencies always, also if **#CURRENT#** value is used. **DISABLE**, **HIDE** and **RESTORE** usually maintain **#CURRENT#** value, but if other value is used really is like do also a **SET** and dependencies will be triggered also.

<Pi> is the name of the property to be modified.

<NVi> is the new value of <Pi>. A special value is **#CURRENT#**, which refers to the current value of <Pi>.

Example, when Some_check is 0 the next field Type is hidden.

```

QUESTION: Some_check#CB#(1,0)
VALUE: 0
DEPENDENCIES: (0,HIDE,Type,#CURRENT#)
DEPENDENCIES: (1,RESTORE,Type,#CURRENT#)
QUESTION: Type#CB#(OPTION_1,OPTION_2)
VALUE: OPTION_1

```

Example with titlestate:

```

...
TITLE: General
QUESTION: Type_of_Analysis:#CB#(FILLING,SOLIDIFICATION)
VALUE: SOLIDIFICATION
DEPENDENCIES: (FILLING,TITLESTATE,Filling-Strategy,normal,RESTORE,
Filling_Analysis,GRAVITY,HIDE,Solidification_Analysis,#CURRENT#)
DEPENDENCIES: (SOLIDIFICATION,TITLESTATE,Filling-Strategy,hidden,HIDE,
Filling_Analysis,#CURRENT#,RESTORE,Solidification_Analysis,#CURRENT#)
TITLE: Filling-Strategy
QUESTION: Filling_Analysis:#CB#(GRAVITY,LOW-PRESSURE,FLOW-RATE)
VALUE: GRAVITY
QUESTION: Solidification_Analysis:#CB#(THERMAL,THERMO-MECHANICAL)
VALUE: THERMAL
...

```

•**State:** Defines the state of a field; this state can be: disabled, enabled or hidden. Here is an example:

```

...
QUESTION: Elastic modulus XX axis
VALUE: 2.1E+11
STATE: HIDDEN
...

```

•**#MAT#('BookName'):** Defines the field as a material, to be selected from the list of materials in the book 'BookName'. Here is an example:

```

QUESTION:Composition_Material#MAT#(BaseMat)
VALUE:AISI_4340_STEEL

```

•TKWIDGET: TkWidget

The Tkwidget special field mechanism allow to customize with Tcl scripting language condition or material fields. some Tcl procedures are predefined in dev_kit.tcl to be used for common cases, like show current layers, materials, pick a point or node, or select a filename.

•Layer field:

```

...
QUESTION: your_question
VALUE: your_layername
TKWIDGET: GidUtils::TkwidgetGetLayername

```

Declare in the tkwidget field to use the Tcl procedureGidUtils::TkwidgetGetLayername, e.g:

Material field:

```

...
QUESTION: your_question
VALUE: your_materialname
TKWIDGET: GidUtils::TkwidgetGetMaterialname

```

Declare in the tkwidget field to use the Tcl GidUtils::TkwidgetGetMaterialname e.g:

•Pick point or node field

Declare in the tkwidget field to use the Tcl procedure `GidUtils::TkwidgetPickPointOrNode` , e.g.

```
...
QUESTION: your_question
VALUE: your_node_id
TKWIDGET: GidUtils::TkwidgetPickPointOrNode
```

•Select filename field

```
...
QUESTION: your_question
VALUE: your_filename
TKWIDGET: GidUtils::TkwidgetGetFilenameButton
```

Declare in the tkwidget field to use the Tcl procedure `GidUtils::TkwidgetGetFilenameButton` , e.g.

•Vector field

```
...
QUESTION: your_question
VALUE: vx vy vz
TKWIDGET: GidUtils::TkwidgetGetVector3D
```

To pack in a single line the three components of a vector, internally stored in the same question as a list of three real numbers, e.g.

•Text widget instead of entry widget

```
...
QUESTION: your_question
VALUE: your_value
TKWIDGET: GidUtils::TkwidgetText
```

To replace the standard single-line entry widget with a multi-line text widget.

•Configure the entry field widget

```
...
QUESTION: your_question
VALUE: your_value
TKWIDGET: GidUtils::TkwidgetEntry CONFIGURE {-width 20}
```

procedure to change some configuration of the `ttk::entry` widget. `GidUtils::TkwidgetEntry CONFIGURE {-<option> <value>}`

Unit System file (.uni)

When GiD is installed, the file units.gid is copied within the GiD directory. In this file a table of magnitudes is defined. For each magnitude there is a set of units and a conversion factor between the unit and the reference unit. The units systems are also defined. A unit system is a set of magnitudes and the corresponding unit.

```
BEGIN TABLE
LENGTH : m, 100 cm, 1e+3 mm
...
STRENGTH : kg*m/s^2, N, 1.0e-1 kp
END

BEGIN SYSTEM(INTERNATIONAL)
LENGTH : m
MASS : kg
STRENGTH : N
...
TEMPERATURE : Cel
END
```

The syntax of the unit file (problem_type_name.uni) within the problem type is similar. It can include the line:

```
USER DEFINED: ENABLED
```

(or DISABLED)

meaning that the user is able (or not able) to define his own system unit within the project. If the line does not appear in the file the value is assumed to be ENABLED.

It is possible to ignore all units systems defined by default inside the file units.gid:

```
USE BASE SYSTEMS: DISABLED
```

(or ENABLED)

With the command HIDDEN: 'magnitude', 'magnitude' certain magnitudes will not be displayed in the Problem units window.

```
HIDDEN: strength, pressure
```

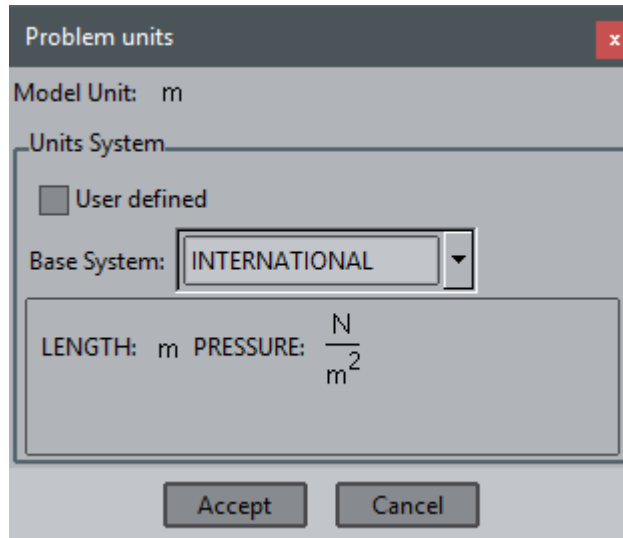
If the problem type uses a property which has a unit, then GiD creates the file project_name.uni in the project directory. This file includes the information related to the unit used in the geometric model and the unit system used. The structure of this file is:

```
MODEL: km
PROBLEM: USER DEFINED
BEGIN SYSTEM
LENGTH: m
PRESSURE: Pa
MASS: kg
STRENGTH: N
END
```

In this file, **MODEL** refers to the unit of the geometric and mesh model of preprocess and **PROBLEM** is the name of the units system used by GiD to convert all the data properties in the output to the solver. If this name is **USER DEFINED**, then the system is the one defined within the file. The block

```
BEGIN SYSTEM
...
END
```

corresponds to the user-defined system.



Unit system: It is possible to define more than one 'unit sytem'. When the user select a unit sytem it mean than when writting to the calculation file the fields with units (material properties, conditions, general data, model length) they will be written converted to the reference unit of this magnitude for the selected unit system.

Hide some units depending on unit system:

The tcl procedure Units::SetUnitsDisallowed allow to specify a list of units to be disallowed (not used in graphical windows)

proc Units::SetUnitsDisallowed { basic_units_to_disallow }
example

```
proc GiD_Event_InitProblemtype { dir } {
    My_On_AfterChangeModelUnitSystem [GiD_Units get system]
}

proc GiD_Event_AfterChangeModelUnitSystem { old_unit_system new_unit_system } {
    My_On_AfterChangeModelUnitSystem $new_unit_system
}

proc My_On_AfterChangeModelUnitSystem { new_unit_system } {
    if { $new_unit_system == "IMPERIAL" } {
        Units::SetUnitsDisallowed {m cm mm Mm km kg ton kton N kp kN MN Nm kNm MNm Pa kPa MPa GPa}
    } elseif { [string range $new_unit_system 0 8] == "INTERNATIONAL" } {
        Units::SetUnitsDisallowed {in Mi miles ft lb lbf kip lbfft lbfin psi ksi Gal}
    } else {
        W "unexpected unit system $new_unit_system"
    }
}
```

Conditions symbols file (.sim)

Files with the extension .sim comprise different symbols to represent some conditions during the preprocessing stage. You can define these symbols by creating ad hoc geometrical drawings and the appropriate symbol will appear over the entity with the applied condition every time you ask for it.

One or more symbols can be defined for every condition and the selection will depend on the specified values in the file, which may be obtained through mathematical conditions.

The spatial orientation can also be defined in this file, depending on the values taken by the required data. For global definitions, you have to input the three components of a vector to express its spatial direction. GiD takes these values from the corresponding conditions window. The orientation of the vector can be understood as the rotation from the vector (1,0,0) towards the new vector defined in the file.

For line and surface conditions, the symbols may be considered as local. In this case, GiD does not consider the defined spatial orientation vector and it takes its values from the line or surface orientation. The orientation assumes the vector (1,0,0) to be the corresponding entity's normal.

These components, making reference to the values obtained from the adequate conditions, may include C-language expressions. They express the different field values of the mentioned condition as `cond(type,i)`, where type (real or int) refers to the type of variable (not case-sensitive) and i is the number of the field for that particular condition.

Example: Creating the Symbols file

Here is an example of how to create a symbols file. Create and edit the file (`problem_type_name.sim` in this example) inside the `problem_type_name` directory (where all your problem type files are located). Except for the extension, the names of the file and the directory must be the same.

The contents of the `problem_type_name.sim` example should be the following:

```
cond Point-Constraints
3
global
cond(int,5)
1
0
0
Support3D.geo
global
cond(int,1) && cond(int,3)
1
0
0
Support.geo
global
cond(int,1) || cond(int,3)
cond(int,3)
cond(int,1)*(-1)
0
Support-2D.geo
cond Face-Load
1
local
fabs(cond(real,2)) + fabs(cond(real,4)) + fabs(cond(real,6))>0.
cond(real,2)
cond(real,4)
cond(real,6)
Normal.geo
```

This is a particular example of the .sim file where four different symbols have been defined. Each one is read from a `***.geo` file. There is no indication of how many symbols are implemented overall. GiD simply reads the whole file from beginning to end.

The `***.geo` files are obtained through GiD. You can design a particular drawing to symbolize a condition and this drawing will be stored as `problem_name.geo` when saving this project as `problem_name.gid`. You do not need to be concerned about the size of the symbol, but should bear in mind that the origin corresponds to the point (0,0,0) and the reference vector is (1,0,0). Subsequently, when these `***.geo` files are invoked from `problem_type_name.sim`, the symbol drawing appears scaled on the display at the entity's location.

Nevertheless, the number of symbols and, consequently, the number of `***.geo` files can vary from one condition to another. In the previous example, for instance, the condition called Point-Constraints, which is defined by using `cond`, comprises three different symbols. GiD knows this from the number 3 written below the condition's name. Next, GiD looks to see if the orientation is relative to the spatial axes (global) or moves together with its entity (local). In the example, the three symbols concerning point constraints are globally oriented.

Imagine that this condition has six fields. The first, third and fifth field values express if any constraint exist along the X-axis, the Y-axis and the Z-axis, respectively. These values are integers and in the case that they are null, the degree of freedom in question is assumed to be unconstrained.

For the first symbol, obtained from the file Support3D.geo, GiD reads `cond(int,5)`, or the Z-constraint. If it is false, which means that the value of the field is zero, the C-condition will not be satisfied and GiD will not draw it. Otherwise, the C-condition will be satisfied and the symbol will be invoked. When this occurs, GiD skips the rest of the symbols related to this condition. Its orientation will be the same as the original drawing because the spatial vector is (1,0,0).

All these considerations are valid for the second symbol, obtained from the file Support.geo, but now GiD has to check that both constraints (&&) - the X-constraint and the Y-constraint - are fixed (their values are not zero).

For the third symbol, obtained from the file Support-2D.geo, only one of them has to be fixed (||) and the orientation of the symbol will depend on which one is free and which one is fixed, showing on the screen the corresponding direction for both degrees of freedom.

Finally, for the fourth symbol, obtained from the file Normal.geo, it can be observed that the drawing of the symbol, related to the local orientation will appear scaled according to the real-type values of the second, fourth and sixth field values. Different types of C-language expressions are available in GiD. Thus, the last expression would be equivalent to entering `'(fabs(cond(real,2))>0. || fabs(cond(real,4))!=0. || fabs(cond(real,6))>1e-10)'`.

Note: As previously mentioned, GiD internally creates a `project_name.geo` file when saving a project, where it keeps all the information about the geometry in binary format. In fact, this is the reason why the extension of these files is `.geo`. However, the file `project_name.geo` is stored in the `project_name.gid` directory, whereas these user-created `***.geo` files are stored in the `problem_type_name.gid` directory.

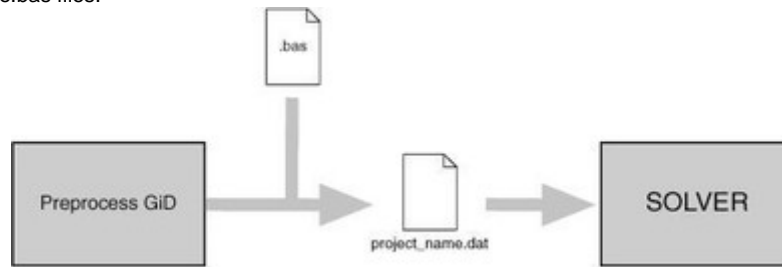
Template files

Once you have generated the mesh, and assigned the conditions and the materials properties, as well as the general problem and intervals data for the solver, it is necessary to produce the data input files to be processed by that program.

To manage this reading, GiD is able to interpret a file called `problem_type_name.bas` (where `problem_type_name` is the name of the working directory of the problem type without the `.bas` extension).

This file (template file) describes the format and structure of the required data input file for the solver that is used for a particular case. This file must remain in the `problem_type_name.gid` directory, as well as the other files already described - `problem_type_name.cnd`, `problem_type_name.mat`, `problem_type_name.prb` and also `problem_type_name.sim` and `***.geo`, if desired.

In the case that more than one data input file is needed, GiD allows the creation of more files by means of additional `***.bas` files (note that while `problem_type_name.bas` creates a data input file named `project_name.dat`, successive `***.bas` files - where `***` can be any name - create files with the names `project_name-1.dat`, `project_name-2.dat`, and so on). The new files follow the same rules as the ones explained next for `problem_type_name.bas` files.



These files work as an interface from GiD's standard results to the specific data input for any individual solver module. This means that the process of running the analysis simply forms another step that can be completed within the system.

In the event of an error in the preparation of the data input files, the programmer has only to fix the corresponding `problem_type_name.bas` or `***.bas` file and rerun the example, without needing to leave GiD, recompile or reassign any data or re-mesh.

This facility is due to the structure of the template files. They are a group of macros (like an ordinary programming language) that can be read, without the need of a compiler, every time the corresponding analysis file is to be written. This ensures a fast way to debug mistakes.

Commands used in the .bas file

List of bas commands: (all these commands must be prefixed by a character *)

Add

Break

Clock Cond CondElemFace CondHasLocalAxes CondName CondNumEntities CondNumFields

*ElmsCenter ElmsConec ElmsLayerName ElmsLayerNum ElmsMat ElmsMatProp ElmsNnode ElmsNnodeCurt ElmsNNodeFace
ElmsNNodeFaceCurt ElmsNormal ElmsNum ElmsRadius ElmsType ElmsTypeName Else Elself End Endif*

FaceElmsNum FaceIndex FactorUnit FileId For Format

GenData GlobalNodes GroupColorRGB GroupFullName GroupName GroupNum GroupNumEntities GroupParentName GroupParentNum

If Include IntFormat IntvData IsQuadratic

LayerColorRGB LayerName LayerNum LayerNumEntities LocalAxesDef LocalAxesDefCenter LocalAxesNum LocalNodes Loop LoopVar

MaterialLocalNum MatNum MatProp MessageBox

Ndim Nelem Nintervals NLocalAxes Nmats Nnode NodesCoord NodesLayerName NodesLayerNum NodesNum Npoin

Operation

RealFormat Remove

Set SetFormatForceWidth SetFormatStandard

Tcl Time

Units

WarningBox

Single value return commands

When writing a command, it is generally not case-sensitive (unless explicitly mentioned), and even a mixture of uppercase and lowercase will not affect the results.

- ***npoin, *ndime, *nnode, *nelem, *nmats, *nintervals.** These return, respectively, the number of points, the dimensions of the project being considered, the number of nodes of the element with the highest number, the number of elements, the number of materials and the number of data intervals. All of them are considered as integers and do not carry arguments (see *format, *intformat), except *nelem, which can bring different types of elements. These elements are: Point, Linear, Triangle, Quadrilateral, Tetrahedra, Hexahedra, Prism, Pyramid, Sphere, depending on the number of edges the element has, and All, which comprises all the possible types. The command *nmats returns the number of materials effectively assigned to an entity, not all the defined ones.
- ***GenData.** This must carry an argument of integer type that specifies the number of the field to be printed. This number is the order of the field inside the general data list. This must be one of the values that are fixed for the whole problem, independently of the interval (see [Problem and intervals data file \(.prb\)](#)). The name of the field, or an abbreviation of it, can also be the argument instead. The arguments REAL or INT, to express the type of number for the field, are also available (see *format, *intformat, *realformat, *if). If they are not specified, the program will print a character string. It is mandatory to write one of them within an expression, except forstrcmp and strcasecmp. The numeration must start with the number 1.

Note: Using this command without any argument will print all fields

- ***IntvData.** The only difference between this and the previous command is that the field must be one of those fields varying with the interval (see [Problem and intervals data file \(.prb\)](#)). This command must be within a loop over intervals (see *loop) and the program will automatically update the suitable value for each iteration.

Note: Using this command without any argument will print all fields

- ***MatProp.** This is the same as the previous command except that it must be within a loop over the materials (see *loop). It returns the property whose field number or name is defined by its argument. It is recommended to use names instead of field numbers.

If the argument is 0, it returns the material's name.

Note: Using this command without any argument will print all fields

Caution: If there are materials with different numbers of fields, you must ensure not to print non-existent fields using conditionals.

- ***MaterialLocalNum** To get the local material number from its global id or its name.

The local material id is the material number for the calculation file, taking into account the materials applied to mesh elements) It has a single argument, an integer of the material global number or its name.

Example:

```
*set var i_material=3
*MaterialLocalNum(*i_material)
*MaterialLocalNum(Steel)
```

- ***ElemsMatProp.** This is the same as Matprop but uses the material of the current element. It must be within a loop over the elements (see *loop). It returns the property whose field number or name is defined by its argument. It is recommended to use names instead of field numbers.

Example:

```
*loop elements
*elemsnum *elemsmat *elemsmatprop(young)
*end elements
```

- ***Cond.** The same remarks apply here, although now you have to notify with the command *set (see *set) which is the condition being processed. It can be within a loop (see *loop) over the different intervals should the conditions vary for each interval.

Note: Using this command without any argument will print all fields

- ***CondName.** This returns the conditions's name. It must be used in a loop over conditions or after a *set cond command.
- ***CondNumFields.** This returns the number of fields of the current condition. It must be used in a loop over conditions or after *setcond
- ***CondHasLocalAxes.** returns 1 if the condition has a local axis field, 0 else
- ***CondNumEntities.** You must have previously selected a condition (see *set cond). This returns the number of entities that have a condition assigned over them.
- ***ElemsNum:** This returns the element's number.

***NodesNum:** This returns the node's number.

***MatNum:** This returns the material's number.

***ElemsMat:** This returns the number of the material assigned to the element.

All of these commands must be within a proper loop (see ***loop**) and change automatically for each iteration. They are considered as integers and cannot carry any argument. The number of materials will be reordered numerically, beginning with number 1 and increasing up to the number of materials assigned to any entity.

- ***FaceElemsNum:** must be inside a ***loop faces**, and print the element's number owner of the face

- ***FaceIndex:** must be inside a ***loop faces**, and print the face index on the element (starting from 1)

- ***LayerNum:** This returns the layer's number.

***LayerName:** This returns the layer's name.

***LayerColorRGB:** This returns the layer's color in RGB (three integer numbers between 0 and 256). If parameter (1), (2) or (3) is specified, the command returns only the value of one color. RED is 1, GREEN is 2 and BLUE is 3.

The commands ***LayerName**, ***LayerNum** and ***LayerColorRGB** must be inside a loop over layers; you cannot use these commands in a loop over nodes or elements.

Example:

```
*loop layers
*LayerName *LayerColorRGB
*Operation(LayerColorRGB(1)/255.0) *Operation(LayerColorRGB(2)/255.0) *Operation(LayerColorRGB(3)/255.0)
*end layers
```

- ***NodesLayerNum:** This returns the layer's number. It must be used in a loop over nodes.

***NodesLayerName:** This returns the layer's name. It must be used in a loop over nodes.

***ElemsLayerNum:** This returns the layer's number. It must be used in a loop over elems.

***ElemsLayerName:** This returns the layer's name. It must be used in a loop over elems.

- ***LayerNumEntities.** You must have previously selected a layer (see ***set layer**). This returns the number of entities that are inside this layer.

- ***GroupNum:** This returns the group's index number.

***GroupFullName:** This returns the full group's name, including parents separated by //. e.g: a//b//c

***GroupName:** This returns only the tail group's name. e.g: c (if group's doesn't has parent then is the same as the full name)

***GroupColorRGB:** This returns the group's color in RGB (three integer numbers between 0 and 256). If parameter (1), (2) or (3) is specified, the command returns only the value of one color. RED is 1, GREEN is 2 and BLUE is 3.

***GroupParentName:** This returns the name of the parent of the current group

***GroupParentNum:** This returns the index of the parent of the current group

These commands must be inside a loop over groups, or after set group.

Example:

```
*loop groups
*groupnum "*"GroupFullName" ("*groupname" parent:*groupparentnum) *groupcolorrgb
*set group *GroupName *nodes
*if(GroupNumEntities)
nodes: *GroupNumEntities
*loop nodes *onlyingroup
*nodesnum
*end nodes
*end if
*set group *GroupName *elems
*if(GroupNumEntities)
elements: *GroupNumEntities
*loop elems *onlyingroup
*elemsnum
*end elems
*end if
*set group *GroupName *faces
*if(GroupNumEntities)
faces: *GroupNumEntities
*loop faces *onlyingroup
*faceelemsnum:*faceindex
*end faces
*end if
*end groups
```

- ***GroupNumEntities.** You must have previously selected a group (see *set group). This returns the number of entities that are inside this group.
- ***LoopVar.** This command must be inside a loop and it returns, as an integer, what is considered to be the internal variable of the loop. This variable takes the value 1 in the first iteration and increases by one unit for each new iteration. The parameter elems, nodes, materials, intervals, used as an argument for the corresponding loop, allows the program to know which one is being processed. Otherwise, if there are nested loops, the program takes the value of the inner loop.
- ***Operation.** This returns the result of an arithmetical expression what should be written inside parentheses immediately after the command. This operation must be defined in C-format and can contain any of the commands that return one single value. You can force an integer or a real number to be returned by means of the parameters INT or REAL. Otherwise, GiD returns the type according to the result.

The valid C-functions that can be used are:

```
+, -, *, /, %, (, ), =, <, >, !, &, |, numbers and variables
sin
cos
tan
asin
acos
atan
atan2
exp
fabs
abs
pow
sqrt
log
log10
max
min
strcmp
strcasncmp
```

The following are valid examples of operations:

```
*operation(4*elemsnum+1)
*operation(8(loopvar-1)+1)
```

Note: There cannot be blank spaces between the commands and the parentheses that include the parameters.

Note: Commands inside *operation do not need * at the beginning.

- ***LocalAxesNum.** This returns the identification name of the local axes system, either when the loop is over the nodes or when it is over the elements, under a referenced condition.
- ***nlocalaxes.** This returns the number of the defined local axes system.
- ***IsQuadratic.** This returns the value 1 when the elements are quadratic or 0 when they are not.
- ***Time.** This returns the number of seconds elapsed since midnight.
- ***Clock.** This returns the number of clock ticks (approx. milliseconds) of elapsed processor time.

Example:

```
*set var t0=clock
*loop nodes
*nodescoord
*end nodes
*set var t1=clock
elapsed time=*operation((t1-t0)/1000.0) seconds
```

- ***Units('magnitude').** This returns the current unit name for the selected magnitude (the current unit is the unit shown inside the unitwindow).

Example:

```
*Units(LENGTH)
```

- ***FactorUnit('unit').** This returns the numeric factor to convert a magnitude from the selected unit to the basic unit.

• Example:

```
*FactorUnit(PRESSURE)
```

- ***FileId** returns a long integer representing the calculation file, written following the current template.

This value must be used to provide the channel of the calculation file to a tcl procedure to directly print data with the GiD_File fprintf special Tcl command.

Multiple values return commands

These commands return more than one value in a prescribed order, writing them one after the other. All of them except LocalAxesDef are able to return one single value when a numerical argument giving the order of the value is added to the command. In this way, these commands can appear within an expression. Neither LocalAxesDef nor the rest of the commands without the numerical argument can be used inside expressions. Below, a list of the commands with the appropriate description is displayed.

- ***NodesCoord.** This command writes the node's coordinates. It must be inside a loop (see *loop) over the nodes or elements. The coordinates are considered as real numbers (see *realformat and *format). It will write two or three coordinates according to the number of dimensions the problem has (see *Ndime).

If *NodesCoord receives an integer argument (from 1 to 3) inside a loop of nodes, this argument indicates which coordinate must be written: x, y or z. Inside a loop of nodes:

*NodesCoord writes three or two coordinates depending on how many dimensions there are.

*NodesCoord(1) writes the x coordinate of the actual node of the loop.

*NodesCoord(2) writes the y coordinate of the actual node of the loop.

*NodesCoord(3) writes the z coordinate of the actual node of the loop.

If the argument real is given, the coordinates will be treated as real numbers.

Example: using *NodesCoord inside a loop of nodes

```
Coordinates:
Node X Y
*loop nodes
*format "%5i%14.5e%14.5e"
*NodesNum *NodesCoord(1,real) *NodesCoord(2,real)
*end nodes
```

This command effects a rundown of all the nodes in the mesh, listing their identifiers and coordinates (**x** and **y**).

The contents of the project_name.dat file could be something like this:

```
Coordinates:
Node      X      Y
1  -1.28571e+001  -1.92931e+000
2  -1.15611e+001  -2.13549e+000
3  -1.26436e+001  -5.44919e-001
4  -1.06161e+001  -1.08545e+000
5  -1.12029e+001   9.22373e-002
...
```

*NodesCoord can also be used inside a loop of elements. In this case, it needs an additional argument that gives the local number of the node inside the element. After this argument it is also possible to give which coordinate has to be written: x, y or z.

Inside a loop of elements:

*NodesCoord(4) writes the coordinates of the 4th node of the actual element of the loop.

*NodesCoord(5,1) writes the x coordinate of the 5th node of the actual element of the loop.

*NodesCoord(5,2) writes the y coordinate of the 5th node of the actual element of the loop.

*NodesCoord(5,3) writes the z coordinate of the 5th node of the actual element of the loop.

- ***ElmsConec.** This command writes the element's connectivities, i.e. the list of the nodes that belong to the element, displaying the direction for each case (anti-clockwise direction in 2D, and depending on the standards in 3D). For shells, the direction must be defined. However, this command accepts the argument swap and this implies that the ordering of the nodes in quadratic elements will be consecutive instead of hierarchical. The connectivities are considered as integers (see *intformat and *format).

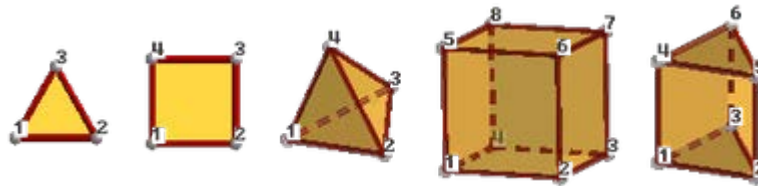
If *ElemsConec receives an integer argument (beginning from 1), this argument indicates which element connectivity must be written:

```
*loop elems
all connectivities: *elemscane
first connectivity *elemscane(1)
*end elems
```

Note: In the first versions of GiD, the optional parameter of the last command explained was invert instead of swap, as it is now. It was changed due to technical reasons. If you have an old .bas file prior to this specification, which contains this command in its previous form, when you try to export the calculation file, you will be warned about this change of use. Be aware that the output file will not be created as you expect.

•***GlobalNodes.** This command returns the nodes that belong to an element's face where a condition has been defined (on the loop over the elements). The direction for this is the same as for that of the element's connectivities. The returned values are considered as integers (see *intformat and *format). If *GlobalNodes receives an integer argument (beginning from 1), this argument indicates which face connectivity must be written.

So, the local numeration of the faces is:



Triangle: (1-2) (2-3) (3-1)

Quadrilateral: (1-2) (2-3) (3-4) (4-1)

Tetrahedra: (1-2-3) (2-4-3) (3-4-1) (4-2-1)

Hexahedra: (1-2-3-4) (1-4-8-5) (1-5-6-2) (2-6-7-3) (3-7-8-4) (5-8-7-6)

Prism: (1-2-3) (1-4-5-2) (2-5-6-3) (3-6-4-1) (4-5-6)

Pyramid: (1-2-3-4) (1-5-2) (2-5-3) (3-5-4) (4-5-1)

•***LocalNodes.** The only difference between this and the previous one is that the returned value is the local node's numbering for the corresponding element (between 1 and nnode).

•***CondElemFace.** This command returns the number of face of the element where a condition has been defined (beginning from 1). The information is equivalent to the obtained with the localnodes command

•***ElemsNnode.** This command returns the number of nodes of the current element (valid only inside a loop over elements).

Example:

```
*loop elems
*ElemsNnode
*end elems
```

•***ElemsNnodeCurt.** This command returns the number of vertex nodes of the current element (valid only inside a loop over elements). For example, for a quadrilateral of 4, 8 or 9 nodes, it returns the value 4.

•***ElemsNnodeFace.** This command returns the number of face nodes of the current element face (valid only inside a loop over elements only in cond, with a previous *set cond of a condition defined over face elements).

Example:

```
*loop elems
*ElemsNnodeFace
*end elems
```

- ***ElemsNNodeFaceCurt.** This command returns the short (corner nodes only) number of face nodes of the current element face (valid only inside a loop over elements onlyincond, with a previous *set cond of a condition defined over face elements).

Example:

```
*loop elems
*ElemsNNodeFaceCurt
*end elems
```

- ***ElemsType:** This returns the current element type as a integer value: 1=Linear, 2=Triangle, 3=Quadrilateral, 4=Tetrahedra, 5=Hexahedra, 6=Prism, 7=Point, 8=Pyramid, 9=Sphere, 10=Circle. (Valid only inside a loop over elements.)

- ***ElemsTypeName:** This returns the current element type as a string value: Linear, Triangle, Quadrilateral, Tetrahedra, Hexahedra, Prism, Point, Pyramid, Sphere, Circle. (Valid only inside a loop over elements.)

- ***ElemsCenter:** This returns the element center. (Valid only inside a loop over elements.)

Note: This command is only available in GiD version 9 or later.

- ***ElemsRadius:** This returns the element radius. (Valid only inside a loop over sphere or Circle elements.)

Note: This command is only available in GiD version 8.1.1b or later.

- ***ElemsNormal.** This command writes the normal's coordinates. It must be inside a loop (see *loop) over elements, and it is onlydefined for triangles, quadrilaterals, and circles (and also for lines in 2D cases).

If *ElemsNormal receives an integer argument (from 1 to 3) this argument indicates which coordinate of the normal must be written: x, y or z.

- ***LocalAxesDef.** This command returns the nine numbers that define the transformation matrix of a vector from the local axes system to the global one.

Example:

```
*loop localaxes
*format "%10.4lg %10.4lg %10.4lg"
x'=*LocalAxesDef(1) *LocalAxesDef(4) *LocalAxesDef(7)
*format "%10.4lg %10.4lg %10.4lg"
y'=*LocalAxesDef(2) *LocalAxesDef(5) *LocalAxesDef(8)
*format "%10.4lg %10.4lg %10.4lg"
z'=*LocalAxesDef(3) *LocalAxesDef(6) *LocalAxesDef(9)
*end localaxes
```

- ***LocalAxesDef(EulerAngles).** This is as the last command, only with the EulerAngles option. It returns three numbers that are the3 Euler angles (radians) that define a local axes system (ϕ, θ, ψ)

$$\begin{pmatrix} \cos\psi \cos\phi - \sin\psi \cos\theta \sin\phi & \cos\psi \sin\phi + \sin\psi \cos\theta \cos\phi & \sin\psi \sin\theta \\ -\sin\psi \cos\phi - \cos\psi \cos\theta \sin\phi & -\sin\psi \sin\phi + \cos\psi \cos\theta \cos\phi & \cos\psi \sin\theta \\ \sin\theta \sin\phi & -\sin\theta \cos\phi & \cos\theta \end{pmatrix} \begin{pmatrix} v_x \\ v_y \\ v_z \end{pmatrix}$$

How to calculate X[3] Y[3] Z[3] orthonormal vector axes from three euler angles angles[3]


```

cosA=cos(angles[0])
sinA=sin(angles[0])
cosB=cos(angles[1])
sinB=sin(angles[1])
cosC=cos(angles[2])
sinC=sin(angles[2])

X[0]= cosC*cosA - sinC*cosB*sinA
X[1]= -sinC*cosA - cosC*cosB*sinA
X[2]= sinB*sinA

Y[0]= cosC*sinA + sinC*cosB*cosA
Y[1]= -sinC*sinA + cosC*cosB*cosA
Y[2]= -sinB*cosA

Z[0]= sinC*sinB
Z[1]= cosC*sinB
Z[2]= cosB

```

How to calculate euler angles angles[3] from X[3] Y[3] Z[3] orthonormal vector axes

```

if (Z[2]<1.0-EPSILON && Z[2]>-1.0+EPSILON){
    double senb=sqrt(1.0-Z[2]*Z[2]);
    angles[1]=acos(Z[2]);
    angles[2]=acos(Z[1]/senb);
    if (Z[0]/senb<0.0) angles[2]=M_2PI-angles[2];
    angles[0]=acos(-Y[2]/senb);
    if (X[2]/senb<0.0) angles[0]=M_2PI-angles[0];
} else {
    angles[1]=acos(Z[2]);
    angles[0]=0.0;
    angles[2]=acos(X[0]);
    if (-X[1]<0.0) angles[2]=M_2PI-angles[2];
}

```

***LocalAxesDefCenter.** This command returns the origin of coordinates of the local axes as defined by the user. The "Automatic" local axes do not have a center, so the point (0,0,0) is returned. The index of the coordinate (from 1 to 3) can optionally be given to LocalAxesDefCenter to get the x, y or z value.

Example:

```

*LocalAxesDefCenter
*LocalAxesDefCenter(1) *LocalAxesDefCenter(2) *LocalAxesDefCenter(3)

```

Specific commands

*\ To avoid line-feeding you need to write *\\, so that the line currently being used continues on the following line of the file filename.bas.

- *# If this is placed at the beginning of the line, it is considered as a comment and therefore is not written.
- ** In order for an asterisk symbol to appear in the text, two asterisks ** must be written.
- *Include. The include command allows you to include the contents of a slave file inside a master .bas file, setting a relative path from the Problem Type directory to this secondary file.

Example:

```
*include includes\execntrlmi.h
```

Note: The *.bas extension cannot be used for the slave file to avoid multiple output files.

- *MessageBox. This command stops the execution of the .bas file and prints a message in a window; this command should only be used when a fatal error occurs.

Example:

```
*MessageBox error: Quadrilateral elements are not permitted.
```

- *WarningBox. This is the same as MessageBox, but the execution is not stopped.

Example:

```
WarningBox Warning: Bad elements. A STL file is a collection of triangles bounding a volume.
```

The following commands must be written at the beginning of a line and the rest of the line will serve as their modifiers. No additional text should be written.

- *loop, *end, *break. These are declared for the use of loops. A loop begins with a line that starts with loop (none of these commands is case-sensitive) and contains another word to express the variable of the loop. There are some lines in the middle that will be repeated depending on the values of the variable, and whose parameters will keep on changing throughout the iterations if necessary. Finally, a loop will end with a line that finishes with *end. After *end, you may write any kind of comments in the same line. The command **break inside a *loop or *for block, will finish the execution of the loop and will continue after the *end line.

The variables that are available for *loop are the following:

- *elems, *nodes, *faces, *materials, *conditions, *layers, *groups, *intervals, *localaxes. These commands mean, respectively, that the loop will iterate over the elements, nodes, faces of a group, materials, conditions, layers, groups, intervals or local axes systems. The loops can be nested among them. The loop over the materials will iterate only over the effectively assigned materials to an entity, in spite of the fact that more materials have been defined. The number of the materials will begin with the number 1. If a command that depends on the loop is located outside it, the number will also take by default the value 1.

After the command *loop:

- If the variable is **nodes**, **elems** or **faces**, you can include one of the modifiers: ***all**, ***OnlyInCond**, ***OnlyInLayer** or ***OnlyInGroup**. The first one signifies that the iteration is going to be performed over all the entities. The ***OnlyInCond** modifier implies that the iteration will only take place over the entities that satisfy the relevant condition. This condition must have been previously defined with ***set cond**. ***OnlyInLayer** implies that the iteration will only take place over the entities that are in the specified layer; layers must be specified with the command ***set Layer**. ***OnlyInGroup** implies that the iteration will only take place over the entities that are in the specified group; group must be specified inside a loop groups with the command ***set Group *GroupName *nodes|elems|faces**, or ***set Group <name>**, with <name> the full name of the group. By default, it is assumed that the iteration will affect all the entities.
- If the variable is **material** you can include the modifier ***NotUsed** to make a loop over those materials that are defined but not used. If the variable is **conditions** you must include one of the modifiers: ***Nodes**, ***BodyElements**, ***FaceElements**, ***Layers** or ***Groups**, to do the loop on the conditions defined over this kind of mesh entity, or only the conditions declared 'over layers' or only the ones declared 'over groups'.
- If the variable is **layers** you can include modifiers: **OnlyInCond** if before was set a condition defined 'over layers'

•If the variable is **groups** you can include modifiers: **OnlyInCond** if before was set a condition defined 'over groups' (e.g. inside a *loop conditions *groups)

Example 1:

```
*loop nodes
*format "%5i%14.5e%14.5e"
*NodesNum *NodesCoord(1,real) *NodesCoord(2,real)
*end nodes
```

This command carries out a rundown of all the nodes of the mesh, listing their identifiers and coordinates (x and y coordinates).

Example 2:

```
*Set Cond Point-Weight *nodes
*loop nodes OnlyInCond
*NodesNum *cond(1)
*end
```

This carries out a rundown of all the nodes assigned the condition "Point-Weight" and provides a list of their identifiers and the first "weight" field of the condition in each case.

Example 3:

```
*Loop Elems
*ElemsNum *ElemsLayerNum
*End Elems
```

This carries out a rundown of all the elements and provides a list of their identifier and the identifier of the layer to which they belong.

Example 4:

```
*Loop Layers
*LayerNum *LayerName *LayerColorRGB
*End Layers
```

This carries out a rundown of all the layers and for each layer it lists its identifier and name.

Example 5:

```
*Loop Conditions OverFaceElements
*CondName
*Loop Elems OnlyInCond
*elemsnum *condelemface *cond
*End Elems
*End Conditions
```

This carries out a rundown of all conditions defined to be applied on the mesh 'over face elements', and for each condition it lists its name and for each element where this condition is applied are printed the element number, the marked face and the condition field values.

Example 6:

```
*loop intervals
interval=*loopvar
*loop conditions *groups
*if(condnumentities)
condition name=*condname
*loop groups *onlyincond
*groupnum *groupname *cond
*end groups
*end if
*end conditions
*end intervals
```

This do a loop for each interval, and for each condition defined 'over groups' list the groups where the condition was applied and its values.

•***if, *else, *elseif, *endif.** These commands create the conditionals. The format is a line which begins with ***if** followed by an expression between parenthesis. This expression will be written in C-language syntax, value return commands, will not begin with *****, and its variables must be defined as integers or real numbers (see ***format, *intformat, *realformat**), with the exception of **strcmp** and **strcasecmp**. It can include relational as well as arithmetic operators inside the expressions.

The following are valid examples of the use of the conditionals:

```
*if((fabs(loopvar)/4)<1.e+2)
*if((p3<p2)|p4)
*if((strcasecmp(cond(1),"XLoad")==0)&&(cond(2)!=0))
```

The first example is a numerical example where the condition is satisfied for the values of the loop under 400, while the other two are logical operators; in the first of these two, the condition is satisfied when $p3 < p2$ or $p4$ is different from 0, and in the second, when the first field of the condition is called XLoad (with this particular writing) and the second is not null.

If the checked condition is true, GiD will write all the lines until it finds the corresponding ***else, *elseif** or ***endif** (***end** is equivalent to ***endif** after ***if**). ***else** or ***elseif** are optional and require the writing of all the lines until the corresponding ***endif**, but only when the condition given by ***if** is false. If either ***else** or ***elseif** is present, it must be written between ***if** and ***endif**. The conditionals can be nested among them.

The behaviour of ***elseif** is identical to the behaviour of ***else** with the addition of a new condition:

```
*if(GenData(31,int)==1)
...(1)
*elseif(GenData(31,int)==2)
...(2)
*else
...(3)
*endif
```

In the previous example, the body of the first condition (written as 1) will be written to the data file if **GenData(31,int)** is 1, the body of the second condition (written as 2) will be written to the data file if **GenData(31,int)** is 2, and if neither of these is true, the body of the third condition (written as 3) will be written to the data file.

Note: A conditional can also be written in the middle of a line. To do this, begin another line and write the conditional by means of the command *****.

•***for, *end, *break.** The syntax of this command is equivalent to ***for** in C-language.

```
*for(varname=expr.1;varname<=expr.2;varname=varname+1)
*end for
```

The meaning of this statement is the execution of a controlled loop, since varname is equal to expr.1 until it is equal to expr.2, with the value increasing by 1 for each step. varname is any name and expr.1 and expr.2 are arithmetical expressions or numbers whose only restrictions are to express the range of the loop.

The command ***break** inside a ***loop** or ***for** block, will finish the execution of the loop and will continue after the ***end** line.

Example:

```
*for(i=1;i<=5;i=i+1)
variable i=*i
*end for
```

•***set**. This command has the following purposes:

- *set cond**: To set a condition.
- *set Layer "layer name" *nodes|elems**: To set a layer.
- *set Group "group name" *nodes|elems|faces**: To set a group. (inside a ***loop** groups can use ***GroupName** as "group name", to get the name of the group of the current loop)
- *set elems**: To indicate the elements.
- *set var**: To indicate the variables to use.

It is not necessary to write these commands in lowercase, so ***Set** will also be valid in all the examples.

***set cond.**: In the case of the conditions, GiD allows the combination of a group of them via the use of ***add cond**. When a specific condition is about to be used, it must first be defined, and then this definition will be used until another is defined. If this feature is performed inside a loop over intervals, the corresponding entities will be chosen. Otherwise, the entities will be those referred to in the first interval.

It is done in this way because when you indicate to the program that a condition is going to be used, GiD creates a table that lets you know the number of entities over which this condition has been applied. It is necessary to specify whether the condition takes place over the ***nodes**, over the ***elems** or over ***layers** to create the table.

So, a first example to check the nodes where displacement constraints exist could be:

```
*Set Cond Volu-Cstrt *nodes
*Add Cond Surf-Cstrt *nodes
*Add Cond Line-Cstrt *nodes
*Add Cond Poin-Cstrt *nodes
```

These let you apply the conditions directly over any geometric entity.

***Set Layer "layer name" *elems|nodes**

***Add Layer "layer name"**

***Remove Layer "layer name"**

This command sets a group of nodes. In the following loops over nodes/elements with the modifier ***OnlyInLayer**, the iterations will only take place over the nodes/elements of that group.

Example 1:

```
*set Layer example_layer_1 *elems
*loop elems *OnlyInLayer
N°:*ElemsNum Name of Layer:*ElemsLayerName N° of Layer : *ElemsLayerNum
*end elems
```

Example 2:

```
*loop layers
*set Layer *LayerName *elems
*loop elems *OnlyInLayer
N°:*ElemsNum Name of Layer:*ElemsLayerName N° of Layer : *ElemsLayerNum
*end elems
*end layers
```

In this example the command `*LayerName` is used to get the layer name.

There are some modifiers available to point out particular specifications of the conditions.

If the command `*CanRepeat` is added after `*nodes` or `*elems` in `*Set cond`, one entity can appear several times in the entities list. If the command `*NoCanRepeat` is used, entities will appear only once in the list. By default, `*CanRepeat` is off except where one condition has the `*CanRepeat` flag already set.

A typical case where you would not use `*CanRepeat` might be:

```
*Set Cond Line-Constraints *nodes
```

In this case, when two lines share one endpoint, instead of two nodes in the list, only one is written.

A typical situation where you would use `*CanRepeat` might be:

```
*Set Cond Line-Pressure *elems *CanRepeat
```

In this case, if one triangle of a quadrilateral has more than one face in the marked boundary then we want this element to appear several times in the elements list, once for each face.

Other modifiers are used to inform the program that there are nodes or elements that can satisfy a condition more than once (for instance, a node that belongs to a certain number of lines with different prescribed movements) and that have to appear unrepeated in the data input file, or, in the opposite case, that have to appear only if they satisfy more than one condition. These requirements are achieved with the commands `*or(i,type)` and `*and(i,type)`, respectively, after the input of the condition, where `i` is the number of the condition to be considered and `type` is the type of the variable (integer or real).

For the previous example there can be nodes or elements in the intersection of two lines or maybe belonging to different entities where the same condition had been applied. To avoid the repetition of these nodes or elements, GiD has the modifier `*or`, and in the case where two or more different values were applied over a node or element, GiD only would consider one, this value being different from zero. The reason for this can be easily understood by looking at the following example. Considering the previous commands transformed as:

```
*Set Cond Volu-Cstrt *nodes *or(1,int) *or(2,int)
*Add Cond Surf-Cstrt *nodes *or(1,int) *or(2,int)
*Add Cond Line-Cstrt *nodes *or(1,int) *or(2,int)
*Add Cond Poin-Cstrt *nodes *or(1,int) *or(2,int)
```

where `*or(1,int)` means the assignment of that node to the considered ones satisfying the condition if the integer value of the first condition's field is different from zero, and `*or(2,int)` means the same assignment if the integer value of the second condition's field is different from zero). Let us imagine that a zero in the first field implies a restricted movement in the direction of the X-axis and a zero in the second field implies a restricted movement in the direction of the Y-axis. If a point belongs to an entity whose movement in the direction of the X-axis is constrained, but whose movement in the direction of the Y-axis is released and at the same time to an entity whose movement in the direction of the Y-axis is constrained, but whose movement in the direction of the X-axis is released, GiD will join both conditions at that point, appearing as a fixed point in both directions and as a node satisfying the four expressed conditions that would be counted only once.

The same considerations explained for adding conditions through the use of `*add cond` apply to elements, the only difference being that the command is `*add elems`. Moreover, it can sometimes be useful to remove sets of elements from the ones assigned to the specific conditions. This can be done with the command `*remove elems`. So, for instance, GiD allows combinations of the type:

```
*Set Cond Dummy *elems
*Set elems(All)
*Remove elems(Linear)
```

To indicate that all dummy elements apart from the linear ones will be considered, as well as:

```
*Set Cond Dummy *elems
*Set elems(Hexahedra)
*Add elems(Tetrahedra)
*Add elems(Quadrilateral)
*Add elems(Triangle)
```

The format for *set var differs from the syntax for the other two *set commands. Its syntax is as follows:

*Set var varname = expression

where varname is any name and expression is any arithmetical expression, number or command, where the latter must be written without *** and must be defined as Int or Real.

A Tcl procedure can also be called, but it must return a numerical result. The following are valid examples for these assignments:

```
*Set var kol=cond(1,real)
*Set var ko2=2
*Set var S1=CondNumEntities

*Set var p1=elemsnum()
*Set var b=operation(p1*2)
*tcl(proc MultiplyByTwo { x } { return [expr {$x*2}] })*\
*Set var a=tcl(MultiplyByTwo *p1)
```

•***intformat**, ***realformat**, ***format**. These commands explain how the output of different mathematical expressions will be written to the analysis file. The use of this command consists of a line which begins with the corresponding version, *intformat, *realformat or *format (again, these are not case-sensitive), and continues with the desired writing format, expressed in C-language syntax argument, between double quotes (").

The integer definition of *intformat and the real number definition of *realformat remain unchanged until another definition is provided via *intformat and *realformat, respectively. The argument of these two commands is composed of a unique field. This is the reason why the *intformat and *realformat commands are usually invoked in the initial stages of the .bas file, to set the format configuration of the integer or real numbers to be output during the rest of the process.

The *format command can include several field definitions in its argument, mixing integer and real definitions, but it will only affect the line that follows the command's instance one. Hence, the *format command is typically used when outputting a listing, to set a temporary configuration.

In the paragraphs that follow, there is an explanation of the C format specification, which refers to the field specifications to be included in the arguments of these commands. Keep in mind that the type of argument that the *format command expects may be composed of several fields, and the *intformat and *realformat commands' arguments are composed of an unique field, declared as integer and real, respectively, all inside double quotes:

A format specification, which consists of optional and required fields, has the following form: **%[flags][width][.precision]type**. The start of a field is signaled by the percentage symbol (%). Each field specification is composed of: some flags, the minimum width, a separator point, the level of precision of the field, and a letter which specifies the type of the data to be represented. The field type is the only one required.

The most common flags are:

- To left align the result
- + To prefix the numerical output with a sign (+ or -)
- # To force the real output value to contain a decimal point.

The most usual representations are integers and floats. For integers the letters d and i are available, which force the data to be read as signed decimal integers, and u for unsigned decimal integers.

For floating point representation, there are the letters e, f and g, these being followed by a decimal point to separate the minimum width of the number from the figure giving the level of precision. The number of digits after the decimal point depends on the requested level of precision.

Note: The standard width specification never causes a value to be truncated. A special command exists in GiD: *SetFormatForceWidth, which enables this truncation to a prescribed number of digits.

For string representations, the letter s must be used. Characters are printed until the precision value is reached.

The following are valid examples of the use of **format**:

```
*Intformat "%5i"
```

With this sentence, usually located at the start of the file, the output of an integer quantity is forced to be right

aligned on the fifth column of the text format on the right side. If the number of digits exceeds five, the

representation of the number is not truncated.

```
*Realformat "%10.3e"
```

This sentence, which is also frequently located in the first lines of the template file, sets the output format for the real numbers as exponential with a minimum of ten digits, and three digits after the decimal point.

```
*format "%10i%10.3e%10i%15.6e"
```

This complex command will specify a multiple assignment of formats to some output columns. These columns are generated with the line command that will follow the format line. The subsequent lines will not use this format, and will follow the general settings of the template file or the general formats: `*IntFormat`, `*RealFormat`.

• ***SetFormatForceWidth**, ***SetFormatStandard** The default width specification of a "C/C+" format, never causes a value to be truncated.

`*SetFormatForceWidth` is a special command that allows a figure to be truncated if the number of characters to print exceeds the specified width.

`*SetFormatStandard` changes to the default state, with truncation disabled.

For example:

```
*SetFormatForceWidth
*set var num=-31415.16789
*format "%8.3f"
*num
*SetFormatStandard
*format "%8.3f"
*num
```

Output:

```
-31415.1
```

```
-31415.168
```

The first number is truncated to 8 digits, but the second number, printed with "C" standard, has 3 numbers after the decimal point, but more than 8 digits.

Tcl This command allows information to be printed using the Tcl extension language. The argument of this command must be a valid Tcl command or expression which must return the string that will be printed. Typically, the Tcl command is defined in the Tcl file (.tcl, see [TCL AND TK EXTENSION](#) for details).

Example: In this example the `*Tcl` command is used to call a **Tcl** function defined in the problem type .tcl file. That function can receive a variable value as its argument with `*variable`. It is also possible to assign the returned value to a variable, but the Tcl procedure must return a numerical value.

In the .bas file:


```
*set var num=1
*tcl(WriteSurfaceInfo *num)
*set var num2=tcl(MultiplyByTwo *num)
```

In the .tcl file:

```
proc WriteSurfaceInfo { num } {
    return [GiD_Info list_entities surfaces $num]
}

proc MultiplyByTwo { x } {
    return [expr {$x*2}]
}
```

General description

All the rules that apply to filename.bas files are also valid for other files with the .bas extension. Thus, everything in this section will refer explicitly to the file filename.bas. Any information written to this file, apart from the commands given, is reproduced exactly in the output file (the data input file for the numerical solver). The commands are words that begin with the character *. (If you want to write an asterisk in the file you should write **.) The commands are inserted among the text to be literally translated. Every one of these commands returns one (see [Single value return commands](#)) or multiple (see [Multiple values return commands](#)) values obtained from the preprocessing component. Other commands mimic the traditional structures to do loops or conditionals (see [Specific commands](#)). It is also possible to create variables to manage some data. Comparing it to a classic programming language, the main differences will be the following:

- The text is reproduced literally, without printing instructions, as it is write-oriented.
 - There are no indices in the loops. When the program begins a loop, it already knows the number of iterations to perform.
- Furthermore, the inner variables of the loop change their values automatically. All the commands can be divided into three types:
- Commands that return one single value. This value can be an integer, a real number or a string. The value depends on certain values that are available to the command and on the position of the command within the loop or after setting some other parameters. These commands can be inserted within the text and write their value where it corresponds. They can also appear inside an expression, which would be the example of the conditionals. For this example, you can specify the type of the variable, integer or real, except when using strcmp or strcmp. If these commands are within an expression, no * should precede the command.
 - Commands that return more than one value. Their use is similar to that of the previously indicated commands, except for the fact that they cannot be used in other expressions. They can return different values, one after the other, depending on some values of the project.
 - Commands that perform loops or conditionals, create new variables, or define some specifications. The latter includes conditions or types of element chosen and also serves to prevent line-feeding. These commands must start at the beginning of the line and nothing will be written into the calculations file. After the command, in the same line, there can be other commands or words to complement the definitions, so, at the end of a loop or conditional, after the command you can write what loop or conditional was finished.

The arguments that appear in a command are written immediately after it and inside parenthesis. If there is more than one, they will be separated by commas. The parentheses might be inserted without any argument inside, which is useful for writing something just after the command without inserting any additional spaces. The arguments can be real numbers or integers, meaning the word REAL or the word INT (both in upper- or lowercase) that the value to which it points has to be considered as real or integer, respectively. Other types of arguments are sometimes allowed, like the type of element, described by its name, in the command *set elem, or a chain of characters inserted between double quotes " for the C-instructions strcmp and strcmp. It is also sometimes possible to write the name of the field instead of its ordering number.

Example:

Below is an example of what a .bas file can be. There are two commands (***nelem** and ***npoin**) which return the total number of elements and nodes of a project.

```
%%% Problem Size %%%
Number of Elements & Nodes:
*nelem *npoin
```

This .bas file will be converted into a project_name.dat file by GiD. The contents of the project_name.dat file could be something like this:

```

%%% Problem Size %%%
Number of Elements & Nodes:
5379 4678

```

(5379 being the number of elements of the project, and 4678 the number of nodes).

Detailed example - Template file creation

Below is an example of how to create a Template file, step by step.

Note that this is a real file and as such has been written to be compatible with a particular solver program. This means that some or all of the commands used will be non-standard or incompatible with the solver that another user may be using.

The solver for which this example is written treats a line inside the calculation input file as a comment if it is prefixed by a \$ sign. In the case of other solvers, another convention may apply.

Of course, the real aim of this example is familiarize you with the commands GiD uses. What follows is the universal method of accessing GiD's internal database, and then outputting the desired data to the solver.

It is assumed that files with the .bas extension will be created inside the working directory where the problem type file is located. The filename must be problem_type_name.bas for the first file and any other name for the additional .bas files. Each .bas file will be read by GiD and translated to a .dat file.

It is very important to remark that any word in the .bas file having no meaning as a GiD compilation command or not belonging to any command instructions (parameters), will be written verbatim to the output file.

First, we create the header that the solver needs in this particular case.

It consists of the name of the solver application and a brief description of its behaviour.

```

$-----
CALSEF: PROGRAM FOR STRUCTURAL ANALYSIS

```

What follows is a commented line with the ECHO ON command. This, when uncommented, is useful if you want to monitor the progress of the calculation. While this particular command may not be compatible with your solver, a similar one may exist.

```

$-----
$ ECHO ON

```

The next line specifies the type of calculation and the materials involved in the calculation; this is not a GiD related command either.

```

$-----
LINEAR-STATIC, SOLIDS

```

As you can see, a commented line with dashes is used to separate the different parts of the file, thus improving the readability of the text.

The next stage involves the initialization of some variables. The solver needs this to start the calculation process.

The following assignments take the first (parameter (1)) and second (parameter (2)) fields in the general problem, as the number of problems and the title of the problem.

The actual position of a field is determined by checking its order in the problem file, so this process requires you to be precise.

Assignment of the first (1) field of the Problem data file, with the command *GenData(1):

```

$-----
$ NUMBER OF PROBLEMS: NPROB = *GenData(1)
$-----

```

Assignment of the second (2) field assignment, *GenData(2):

```
$ TITLE OF THE PROBLEM: TITULO= *GenData(2)
$-----
```

The next instruction states the field where the starting time is saved. In this case, it is at the 10th position of the general problem data file, but we will use another feature of the *GenData command, the parameter of the command will be the name of the field.

This method is preferable because if the list is shifted due to a field being added or subtracted, you will not lose the actual position. This command accepts an abbreviation, as long as there is no conflict with any other field name.

```
$-----
$ TIME OF START: TIME= *GenData(Starting_time)
$-----
```

Here comes the initialization of some general variables relevant to the project in question - the number of points, the number of elements or the number of materials.

The first line is a description of the section.

```
$ DIMENSIONS OF THE PROBLEM:
```

The next line introduces the assignments.

```
DIMENSIONS :
```

This is followed by another line which features the three variables to be assigned. NPNOD gets, from the *npoin function, the number of nodes for the model; NELEM gets, from *nelem, either the total number of elements in the model or the number of elements for every kind of element; and NMATS is initialized with the number of materials:

```
NPNOD= *npoin, NELEM= *nelem, NMATS= *nmats, \
```

In the next line, NNODE gets the maximum number of nodes per element and NDIME gets the variable *ndime. This variable must be a number that specifies whether all the nodes are on the plane whose Z values are equal to 0 (NDIME=2), or if they are not (NDIME=3):

```
NNODE= *nnode, NDIME= *ndime, \
```

The following lines take data from the general data fields in the problem file. NCARG gets the number of charge cases, NGDLN the number of degrees of freedom, NPROP the properties number, and NGAUSS the gauss number; NTIPO is assigned dynamically:

```
NLOAD= *GenData(Load_Cases), *\
```

You could use NGDLN= *GenData(Degrees_Freedom), *, but because the length of the argument will exceed one line, we have abbreviated its parameter (there is no conflict with other question names in this problem file) to simplify the command.

```
NGDLN= *GenData(Degrees_Fre), *\
NPROP= *GenData(Properties_Nbr), \
NGAUS= *GenData(Gauss_Nbr) , NTIPO= *\
```

Note that the last assignment is ended with the specific command * to avoid line feeding. This lets you include a conditional assignment of this variable, depending on the data in the General data problem.

Within the conditional a C format-like strcmp instruction is used. This instruction compares the two strings passed as a parameter, and returns an integer number which expresses the relationship between the two strings. If the result of this operation is equal to 0, the two strings are identical; if it is a positive integer, the first argument is greater than the second, and if it is a negative integer, the first argument is smaller than the second.

The script checks what problem type is declared in the general data file, and then it assigns the coded number for this type to the NTIPO variable:

```
*if(strcmp(GenData(Problem_Type), "Plane-stress")==0)
1 *\
*elseif(strcmp(GenData(Problem_Type), "Plane-strain")==0)
2 *\
*elseif(strcmp(GenData(Problem_Type), "Revol-Solid")==0)
3 *\
*elseif(strcmp(GenData(Problem_Type), "Solid")==0)
4 *\
*elseif(strcmp(GenData(Problem_Type), "Plates")==0)
5 *\
*elseif(strcmp(GenData(Problem_Type), "Revol-Shell")==0)
```

```
6 *\
*endif
```

You have to cover all the cases within the if sentences or end the commands with an elseif you do not want unpredictable results, like the next line raised to the place where the parameter will have to be:

```
$ Default Value:
*else
0 *\
*endif
```

In our case this last rule has not been followed, though this can sometimes be useful, for example when the problem file has been modified or created by another user and the new specification may differ from the one we expect.

The next assignment is formed by a string compare conditional, to inform the solver about a configuration setting.

First is the output of the variable to be assigned.

```
, IWRIT= *\
```

Then there is a conditional where the string contained in the value of the Result_File field is compared with the string "Yes". If the result is 0, then the two strings are the same, while the output result 1 is used to declare a boolean TRUE.

```
*if(strcmp(GenData(Result_File),"Yes")==0)
1 ,*\
```

Then we compare the same value string with the string "No", to check the complementary option. If we find that the strings match, then we output a 0.

```
*elseif(strcmp(GenData(Result_File),"No")==0)
0 ,*\
*endif
```

The second to last assignment is a simple output of the solver field contents to the INDSO variable:

```
INDSO= *GenData(Solver) , *\
```

The last assignment is a little more complicated. It requires the creation of some internal values, with the aid of the *set cond command.

The first step is to set the conditions so we can access its parameters. This setting may serve for several loops or instructions, as long as the parameters needed for the other blocks of instructions are the same.

This line sets the condition Point-Constraints as an active condition. The *nodes modifier means that the condition will be listed over nodes. The *or(...) modifiers are necessary when an entity shares some conditions because it belongs to two or more elements.

As an example, take a node which is part of two lines, and each of these lines has a different condition assigned to it. This node, a common point of the two lines, will have these two conditions in its list of properties. So declaring the *or modifiers, GiD will decide which condition to use, from the list of conditions of the entity.

A first instruction will be as follows, where the parameters of the *or commands are an integer - (1, and (3, in this example - and the specification int, which forces GiD to read the condition whose number position is the integer.

In our case, we find that the first (1) field of the condition file is the X-constraint, and the third (3) is the Y-constraint:

GiD still has no support for substituting the condition's position in the file by its corresponding label, in contrast to case for the fields in the problem data file, for which it is possible.

```
*Set Cond Surface-Constraints *nodes *or(1,int) *or(3,int)
```

Now we want to complete the setting of the loop, with the addition of new conditions.

```
*Add Cond Line-Constraints *nodes *or(1,int) *or(3,int)
*Add Cond Point-Constraints *nodes *or(1,int) *or(3,int)
```

Observe the order in which the conditions have been included: firstly, the surface constraints with the *Set Cond command, since it is the

initial sentence; then the pair of *Add Cond sentences, the line constraints; and finally, the point constraints sentence. This logical hierarchy forces the points to be the most important items.

Last of all, we set a variable with the number of entities assigned to this particular condition.

Note that the execution of this instruction is only possible if a condition has been set previously.

```
NPRES= *CondNumEntities
```

To end this section, we put a separator in the output file:

```
$-----
```

Thus, after the initialization of these variables, this part of the file ends up as:

```
$ DIMENSIONS OF THE PROBLEM:
DIMENSIONES :
NPNOD= *npoin, NELEM= *nelem, NMATS= *nmats, \ NNODE=
*nnode, NDIME= *ndime, \
NCARG= *GenData(Charge_Cases), *\
NGDLN= *GenData(Degrees_Fre), *\
NPROP= *GenData(Properties_Nbr), \
NGAUS= *GenData(Gauss_Nbr) , NTIPO= *\
*if(strcmp(GenData(Problem_Type),"Tens-Plana")==0)
1 *\
*elseif(strcmp(GenData(Problem_Type),"Def-Plana")==0)
2 *\
*elseif(strcmp(GenData(Problem_Type),"Sol-Revolver")==0)
3 *\
*elseif(strcmp(GenData(Problem_Type),"Sol-Tridim")==0)
4 *\
*elseif(strcmp(GenData(Problem_Type),"Placas")==0)
5 *\
*elseif(strcmp(GenData(Problem_Type),"Laminas-Rev")==0)
6 *\
*endif
, IWRIT= *\
*if(strcmp(GenData(Result_File),"Yes")==0)
1 ,\
*elseif(strcmp(GenData(Result_File),"No")==0)
0 ,\
*endif
INDSO= *GenData(Solver) , *\
*Set Cond Surface-Constraints *nodes *or(1,int) *or(3,int)
*Add Cond Line-Constraints *nodes *or(1,int) *or(3,int)
*Add Cond Point-Constraints *nodes *or(1,int) *or(3,int)
NPRES=*CondNumEntities
$-----
```

After creating or reading our model, and once the mesh has been generated and the conditions applied, we can export the file (project_name.dat) and send it to the solver.

The command to create the .dat file can be found on the File -> Export -> Calculation File GiD menu. It is also possible to use the keyboard shortcut Ctrl-x Ctrl-c.

These would be the contents of the project_name.dat file:

```

$-----
CALSEF: PROGRAM FOR STRUCTURAL ANALYSIS
$-----
$ECHO ON
$-----
LINEAR-STATIC, SOLIDS
$-----
$NUMBER OF PROBLEMS:
NPROB = 1
$-----
$ PROBLEM TITLE
TITLE= Title_name
$-----
$DIMENSIONS OF THE PROBLEM
DIMENSIONS :
  NPNOD=    116 ,  NELEM=  176 ,  NMATS=   0 ,  \
  NNODE=     3 ,  NDIME=   2 ,  \
  NCARG=     1 ,  NGDLN=   1 ,  NPROP=   5 ,  \
  NGAUS=     1 ,  NTIPO=   1 ,  IWRT=    1 ,  \
  INDSO=    10 ,  NPRES=   0
$-----

```

This is where the calculation input begins.

Formatted nodes and coordinates listing

As with the previous section, this block of code begins with a title for the subsection:

```
$ NODAL COORDINATES
```

followed by the header of the output list:

```
$ NODE COORD.-X COORD.-Y COORD.-Z
```

Now GiD will trace all the nodes of the model:

```
*loop nodes
```

For each node in the model, GiD will generate and output its number, using *NodesNum, and its coordinates, using *NodesCoord.

The command executed before the output *format will force the resulting output to follow the guidelines of the specified formatting.

In this example below, the *format command gets a string parameter with a set of codes: %6i specifies that the first word in the list is coded as an integer and is printed six points from the left; the other three codes, all %15.5f, order the printing of a real number, represented in a floating point format, with a distance of 15 spaces between columns (the number will be shifted to have the last digit in the 15th position of the column) and the fractional part of the number will be represented with five digits.

Note that this is a C language format command.

```

*format "%6i%15.5f%15.5f%15.5f"
*NodesNum *NodesCoord
*end nodes

```

At the end of the section the end marker is added, which in this solver example is as follows:

```
END_GEOMETRY
```

The full set of commands to make this part of the output is shown in the following lines.

```

GEOMETRY
$ ELEMENT CONNECTIVITIES
$ ELEM. MATER. CONNECTIVITIES
*loop elems
  *elemsnum *elemsmat *elemsConec
*end elems
$ NODAL COORDINATES
$ NODE COORD.-X COORD.-Y COORD.-Z
*loop nodes
  *format "%6i%15.5f%15.5f%15.5f"
  *NodesNum *NodesCoord
*end
END_GEOMETRY

```

The result of the evaluation is output to a file (project_name.dat) to be processed by the solver program.

The first part of the section:

```
$-----
GEOMETRY
$ ELEMENT CONNECTIVITIES
$ ELEM. MATER. CONNECTIVITIES
  1      1      73      89      83
  2      1      39      57      52
  3      1      17      27      26
  4      5       1       3       5
  5      5       3      10       8
  6      2      57      73      67
  .      .       .       .       .
  .      .       .       .       .
  .      .       .       .       .
176     5      41      38      24
```

And the second part of the section:

```
$ NODAL COORDINATES
$ NODE COORD.-X COORD.-Y COORD.-Z
  1      5.55102      5.51020
  2      5.55102      5.51020
  3      4.60204      5.82993
  4      4.60204      5.82993
  5      4.88435      4.73016
  6      4.88435      4.73016
  .      .           .
  .      .           .
  .      .           .
116     -5.11565      3.79592
END GEOMETRY
```

If the solver module you are using needs a list of the nodes that have been assigned a condition, for example, a neighborhood condition, you have to provide it as is explained in the next example.

Elements, materials and connectivities listing

Now we want to output the desired results to the output file. The first line should be a title or a label as this lets the solver know where a loop section begins and ends. The end of this block of instructions will be signalled by the line `END_GEOMETRY`.

```
GEOMETRY
```

The next two of lines give the user information about what types of commands follow.

Firstly, a title for the first subsection, `ELEMENTAL CONNECTIVITIES`:

```
$ ELEMENTAL CONNECTIVITIES
```

followed by a header that preceeds the output list:

```
$ ELEM. MATER. CONNECTIVITIES
```

The next part of the code concerns the elements of the model with the inclusion of the `*loop` instruction, followed in this case by the `elems` argument.

```
*loop elems
```

For each element in the model, GiD will output: its element number, by the action of the `*elemsnum` command, the material assigned to this element, using the `*elemsmat` command, and the connectivities associated to the element, with the `*elemsConec` command:

```
*elemsnum *elemmat *elemsConec
*end elems
```

You can use the swap parameter if you are working with quadratic elements and if the listing mode of the nodes is non-hierarchical (by default, corner nodes are listed first and mid nodes afterwards):

```
*elemsnum *elemmat *elemsConec(swap)
*end elems
```

Nodes listing declaration

First, we set the necessary conditions, as was done in the previous section.

```
*Set Cond Surface-Constraints *nodes *or(1,int) *or(3,int)
*Add Cond Line-Constraints *nodes *or(1,int) *or(3,int)
*Add Cond Point-Constraints *nodes *or(1,int) *or(3,int)
NPRES=*CondNumEntities
```

After the data initialization and declarations, the solver requires a list of nodes with boundary conditions and the fields that have been assigned.

In this example, all the selected nodes will be output and the 3 conditions will also be printed. The columns will be output with no apparent format.

Once again, the code begins with a subsection header for the solver program and a commentary line for the user:

```
BOUNDARY CONDITIONS
$ RESTRICTED NODES
```

Then comes the first line of the output list, the header:

```
$ NODE CODE PRESCRIPTED VALUES
```

The next part the loop instruction, in this case over nodes, and with the specification argument *OnlyInCond, to iterate only over the entities that have the condition assigned. This is the condition that has been set on the previous lines.

```
*loop nodes *OnlyInCond
```

The next line is the format command, followed by the lines with the commands to fill the fields of the list.

```
*format "%5i%1i%1i%f%f"
*NodesNum *cond(1,int) *cond(3,int) *\
```

The *format command influence also includes the following two if sentences. If the degrees of freedom field contains an integer equal or greater than 3, the number of properties will be output.

```
*if(GenData(Degrees_Freedom_Nodes,int)>=3)
*cond(5,int) *\
*endif
```

And if the value of the same field is equal to 5 the output will be a pair of zeros.

```
*if(GenData(Degrees_Free,int)==5)
0 0 *\
*endif
```

The next line outputs the values contained in the second and fourth fields, both real numbers.

```
*cond(2,real) *cond(4,real) *\
```

In a similar manner to the previous if sentences, here are some lines of code which will output the sixth condition field value if the number of degrees of freedom is equal or greater than three, and will output a pair of zeros if it is equal to five.

```
*if(GenData(Degrees_Free,int)>=3)
*cond(6,real) *\
*endif
*if(GenData(Degrees_Free,int)==5)
0.0 0.0 *\
*endif
```

Finally, to end the section, the *end command closes the previous *loop. The last line is the label of the end of the section.


```
*end
END_BOUNDARY CONDITIONS
$-----
```

The full set of commands included in this section are as follows:

```
BOUNDARY CONDITIONS
$ RESTRICTED NODES
$ NODE CODE          PRESCRIPTED VALUES
*loop nodes *OnlyInCond
*format "%5i%1i%1i%f%f"
    *NodesNum *cond(1,int) *cond(3,int) *\
*if(GenData(Degrees_Free,int)>=3)
*cond(5,int) *\
*endif
*if(GenData(Degrees_Free,int)==5)
0 0 *\
*endif
*cond(2,real) *cond(4,real) *\
*if(GenData(Degrees_Free,int)>=3)
*cond(6,real) *\
*endif
*if(GenData(Degrees_Free,int)==5)
0.0 0.0 *\
*endif
*end
END_BOUNDARY CONDITIONS
$-----
```

Elements listing declaration

First, we set the loop to the interval of the data.

```
*loop intervals
```

The next couple of lines indicate the starting of one section and the title of the example, taken from the first field in the interval data with an abbreviation on the label. They are followed by a comment explaining the type of data we are using.

```
LOADS
TITLE: *IntvData(Charge_case)
$ LOAD TYPE
```

We begin by setting the condition as before. If one condition is assigned twice or more to the same element without including the *CanRepeat parameter in the *Set Cond, the condition will appear once; if the *CanRepeat parameter is present then the number of conditions that will appear is the number of times it was assigned to the condition.

```
*Set Cond Face-Load *elems *CanRepeat
```

Then, a condition checks if any element exists in the condition.

```
*if(CondNumEntities(int)>0)
```

Next is a title for the next section, followed by a comment for the user.

```
DISTRIBUTED ON FACES
$ LOADS DISTRIBUTED ON ELEMENT FACES
```

We assign the number of nodes to a variable.

```
$ NUMBER OF NODES BY FACE NODGE = 2
$ LOADED FACES AND FORCE VALUES
*loop elems *OnlyInCond
ELEMENT=*elemsnum(), CONNECTIV *globalnodes
*cond(1) *cond(1) *cond(2) *cond(2)
*end elems
END_DISTRIBUTED ON FACES
*endif
```

The final section deals with outputting a list of the nodes and their conditions.

Materials listing declaration

This section deals with outputting a materials listing.

As before, the first lines must be the title of the section and a commentary:

```
MATERIAL PROPERTIES
$ MATERIAL PROPERTIES FOR MULTILAMINATE
```

Next there is the loop sentence, this time concerning materials:

```
*loop materials
```

Then comes the line where the number of the material and its different properties are output:

```
*matnum() *MatProp(1) *MatProp(2) *MatProp(3) *MatProp(4)
```

Finally, the end of the section is signalled:

```
*end materials
END_MATERIAL PROPERTIES
$-----
```

The full set of commands is as follows:

```
MATERIAL PROPERTIES
$ MATERIAL PROPERTIES FOR MULTILAMINATE
*loop materials
*matnum() *MatProp(1) *MatProp(2) *MatProp(3) *MatProp(4)
*end materials
END_MATERIAL PROPERTIES
$-----
```

The next section deals with generating an elements listing.

Nodes and its conditions listing declaration

As for previous sections, the first thing to do set the conditions.

```
*Set Cond Point-Load *nodes
```

As in the previous section, the next loop will only be executed if there is a condition in the selection.

```
*if(CondNumEntities(int)>0)
```

Here begins the loop over the nodes.

```
PUNCTUAL ON NODES
*loop nodes *OnlyInCond
*NodesNum *cond(1) *cond(2) *\
```

The next *if sentences determine the output writing of the end of the line.

```
*if(GenData(Degrees_Free,int)>=3)
*cond(3) *\
*endif
*if(GenData(Degrees_Free,int)==5)
0 0 *\
*endif
*end nodes
```

To end the section, once again you have to include the end label and the closing *endif.

```
END_PUNCTUAL ON NODES
*endif
```

Finally, a message is written if the value of the second field in the interval data section inside the problem file is equal to "si" (yes).

```
*if(strcasecmp(IntvData(2),"Si")==0)
SELF_WEIGHT
*endif
```

To signal the end of this part of the forces section, the following line is entered.

```
END_LOADS
```

Before the end of the section it remains to tell the solver what the postprocess file will be. This information is gathered from the *IntvData command. The argument that this command receives (3) specifies that the name of the file is in the third field of the loop iteration of the interval.

```
$-----
$POSTPROCESS FILE FEMV = *IntvData(3)
```

To end the forces interval loop the *end command is entered.

```
$-----
*end nodes
```

Finally, the complete file is ended with the sentence required by the solver.

```
END_CALSEF $-----
```

The preceding section is compiled completely into the following lines:

```
*Set Cond Point-Load *nodes
*if(CondNumEntities(int)>0)
PUNCTUAL ON NODES
*loop nodes *OnlyInCond
    *NodesNum *cond(1) *cond(2) *\
    *if(GenData(Degrees_Free,int)>=3)
    *cond(3) *\
    *endif
    *if(GenData(Degrees_Free,int)==5)
    0 0 *\
    *endif
    *endif
END_PUNCTUAL ON NODES
*endif
*if(strcasecmp(IntvData(2),"Si")==0)
SELF_WEIGHT
*endif
END_LOADS
$-----
$POSTPROCESS FILE
FEMV = *IntvData(3)
$-----
*end nodes
END_CALSEF
$-----
```

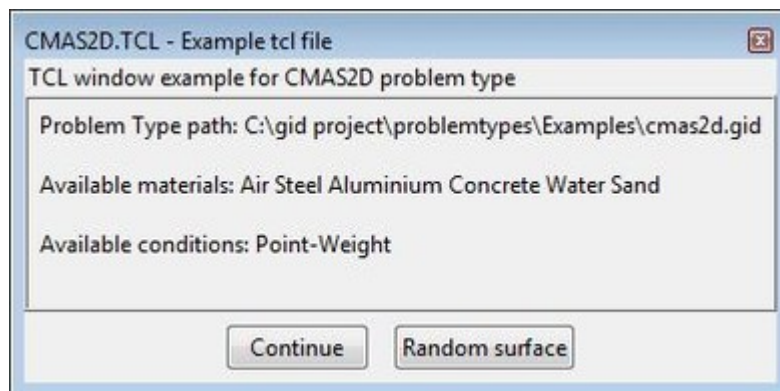
This is the end of the template file example.

Tcl/Tk example

Here is a step by step example of how to create a Tcl/Tk extension. In this example we will create the file **cmas2d.tcl**, so we will be extending the capabilities of the **cmas2d** problem type. The file **cmas2d.tcl** has to be placed inside the **cdmas2d** Problem Type directory.

Note: The **cmas2d** problem type calculates the center of mass of a 2D surface. This problem type is located inside the Problem Types directory, in the GiD directory.

In this example, the **cmas2d.tcl** creates a window which appears when the problem type is selected.



This window gives information about the location, materials and conditions of the problem type. The window has two buttons: **CONTINUE** lets you continue working with the **cmas2d** problem type; **RANDOM SURFACE** creates a random 2D surface in the plane XY.

What follows is the Tcl code for the example. There are three main procedures in the **cmas2d.tcl** file:

```
•proc InitGIDProject {dir}
```

```

proc InitGIDProject {dir } {
    set materials [GiD_Info materials]
    set conditions [GiD_Info conditions over_point]
    CreateWindow $dir $materials $conditions
}

```

This is the main procedure. It is executed when the problem type is selected. It calls the **CreateWindow** procedure.

•**proc CreateWindow {dir mat cond}**

```

proc CreateWindow {dir mat cond} {
    if { [GidUtils::AreWindowsDisabled] } {
        return
    }
    set w .gid.win_example
    InitWindow $w [= "CMAS2D.TCL - Example tcl file"] ExampleCMAS "" "" 1
    if { ![winfo exists $w] } return ;# windows disabled || usemorewindows == 0
    ttk::frame $w.top
    ttk::label $w.top.title_text -text [= "TCL window example for CMAS2D problem type"]
    ttk::frame $w.information -relief ridge
    ttk::label $w.information.path -text [= "Problem Type path: %s" $dir]
    ttk::label $w.information.materials -text [= "Available materials: %s" $mat]
    ttk::label $w.information.conditions -text [= "Available conditions: %s" $cond]
    ttk::frame $w.bottom
    ttk::button $w.bottom.start -text [= "Continue"] -command "destroy $w"
    ttk::button $w.bottom.random -text [= "Random surface"] -command "CreateRandomSurface $w"
    grid $w.top.title_text -sticky ew
    grid $w.top -sticky new
    grid $w.information.path -sticky w -padx 6 -pady 6
    grid $w.information.materials -sticky w -padx 6 -pady 6
    grid $w.information.conditions -sticky w -padx 6 -pady 6
    grid $w.information -sticky nsew
    grid $w.bottom.start $w.bottom.random -padx 6
    grid $w.bottom -sticky sew -padx 6 -pady 6
    if { $::tcl_version >= 8.5 } { grid anchor $w.bottom center }
    grid rowconfigure $w 1 -weight 1
    grid columnconfigure $w 0 -weight 1
}

```

This procedure creates the window with information about the path, the materials and the conditions of the project. The window has two buttons: if **CONTINUE** is pressed the window is dismissed; if **RANDOM SURFACE** is pressed, it calls the **CreateRandomSurface** procedure.

•**proc CreateRandomSurface {w}**

```

proc CreateRandomSurface {w} {
    set ret [tk_dialogRAM $w.dialog [= "Warning"] \
        [= "Warning: this will create a nurbs surface in your current project"] "" 1 [= "Ok"] [= "Cancel"]]
    if {$ret == 0} {
        Create_surface
        destroy $w
    }
}

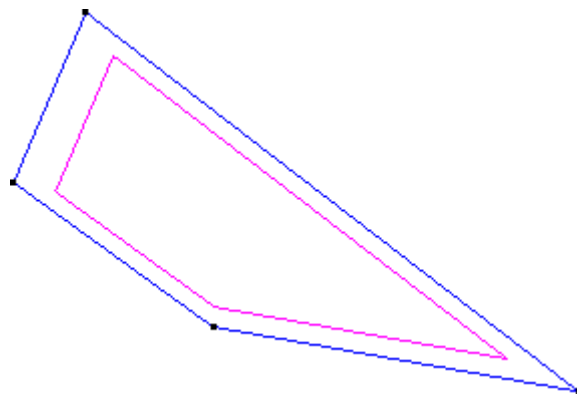
```

This procedure is called when the **RANDOM SURFACE** button is pressed. Before creating the surface, a dialog box asks you to continue with or cancel the creation of the surface. If the surface is to be created, the **Create_surface** procedure is called. Then, the window is destroyed.

```

proc Create_surface {} {
  set a_x [expr rand()*10]
  set a_y [expr rand()*10]
  set b_x [expr $a_x + rand()*10]
  set b_y [expr $a_y + rand()*10]
  set c_x [expr $b_x + rand()*10]
  set c_y [expr $b_y - rand()*10]
  if {$a_y < $c_y} {
    set d_y [expr $a_y - rand()*10]
    set d_x [expr $a_x + rand()*10]
  } else {
    set d_y [expr $c_y - rand()*10]
    set d_x [expr $c_x - rand()*10]
  }
  GiD_Process escape escape escape geometry create line \
    $a_x,$a_y,0.000000 $b_x,$b_y,0.000000 $c_x,$c_y,0.000000 $d_x,$d_y,0.000000 close
  GiD_Process escape escape escape escape geometry create NurbsSurface Automatic \
    4 escape
  GiD_Process 'Zoom Frame escape escape escape escape
}

```



A 2D surface (a four-sided 2D polygon) is created. The points of this surface are chosen at random.