

## GiD – SAFIR 2D-Structural Analysis User Interface

### 1. Create a GiD project of type *Safir\_Structural\_2d*

From the pull down menus select:

*Data->Problem type->SAFIR2007-> Safir\_Structural\_2d*

Save the project by selecting from the pull down menu:

*File->Save as...*

GiD creates a directory with the entered *project-name* expanded by *.gid* and places a number of help-files in this directory.

### 2. Create the system geometry

Units must be meters. Construct the system geometry in the XY-Plane using GiD geometry commands.

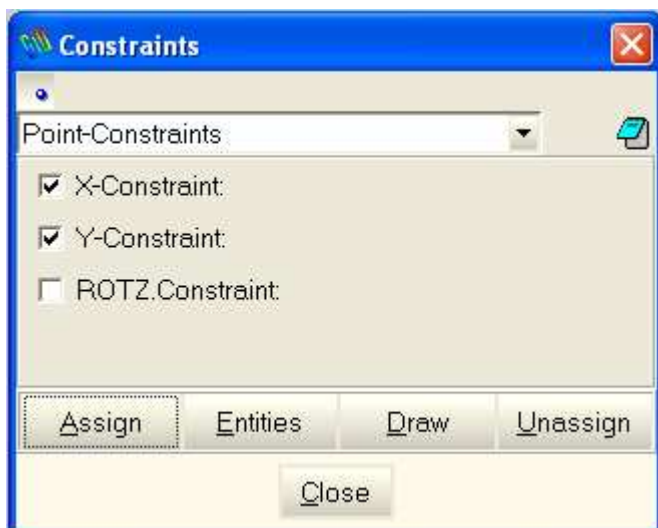
*Geometry->Create->Line*

For details on how to create geometry, look to the GiD reference manual or online-help.

### 3. Define constraints for supports

From the pull down menu select: *Data-> Constraints*

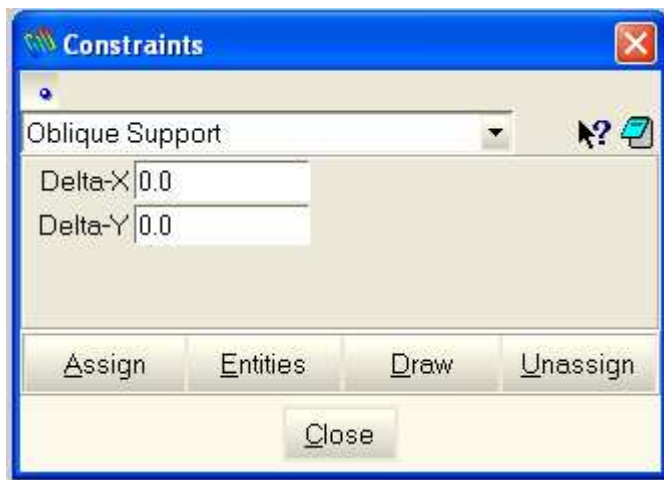
The following dialog box is displayed:



Select the appropriate constraints you need for a support, click the button *Assign* to assign the constraints to a point (or node) of the structure.

**Note:** All constraints are defined in the GLOBAL coordinate system.

To define an *Oblique Support* select:

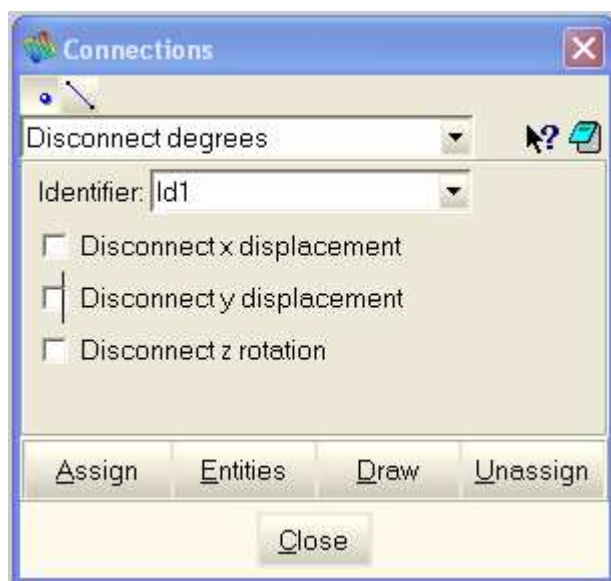


Enter values for Delta-X and Delta-Y and assign this property to a point (or node).

Delta-X, Delta-Y define the direction of the line on which this point can move.

#### 4. Define Connections

To disconnect certain degrees of freedom on the end-node of a beam select from the pull down menu select: *Data-> Connections*



To disconnect some degrees in one point, select the degrees to disconnect and give an Identifier. Then, assign the same identifier using condition *Disconnect\_Id* to all the beam elements that must form a rigid group disconnected from the node.



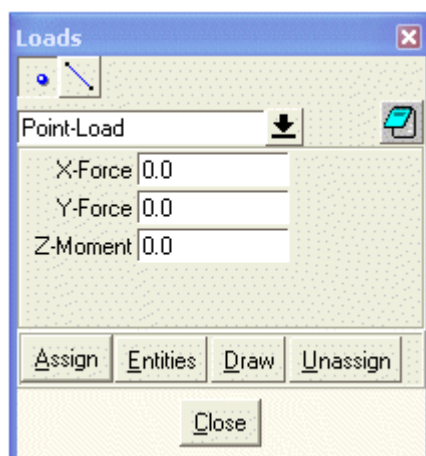
**Example:** if the z-rotation is marked in one node with identifier 'Id1', and the same identifier 'Id1' is assigned to several beam elements that are connected this node then, these beams will be rigid between them and will rotate freely related to the rest of the beams.

## 5. Define Loads

From the pull down menu select:

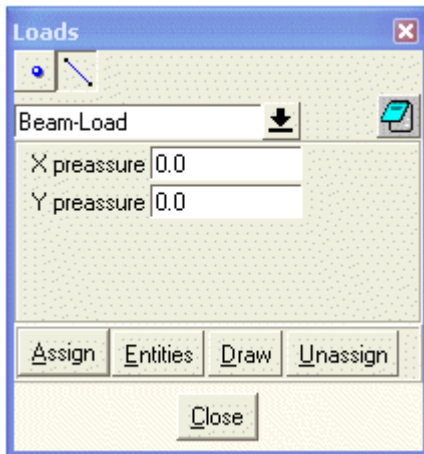
*Data->Conditions->Loads*

The following dialog box is displayed:



Enter Forces in Newton (N) , Moments in Nm and assign it to points (nodes) of the system geometry.

To assign distributed loads click to the line-icon of the dialog box:



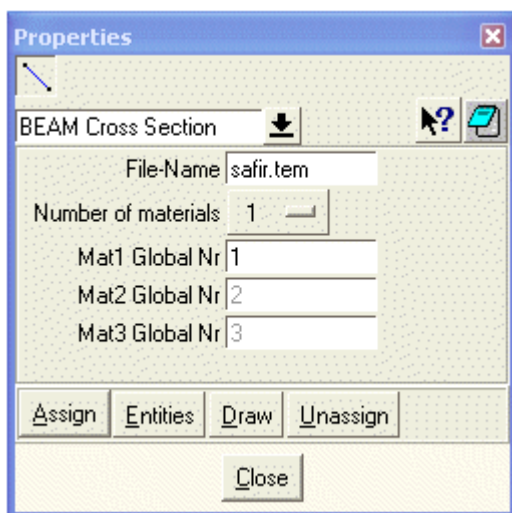
Enter values in N/m to assign distributed loads to a system line. After meshing this load is automatically transferred to all beam-elements of this system line.

To view assigned loads click to the button *Draw* and select *Colors*.

## 6. Assign the temperature files

Form the pull down menu select:

*Data->Conditions->Properties*



Enter in the field *File-Name* the name of the Temperature file (.TEM file ) for the cross-section of the beam. Enter the *number of materials* used in this cross-section. For each local material number ( this is the material number in the .TEM file ) of this section enter the global material number in the structure (this is the material number in the *Problem data -> General ->Material* ).

Assign the data to a system-line (or beam elements if you are in the mesh-display).

**Note:** If a member has several temperature zones, it is advisable to divide already the system line in adequate sections.

The orientation of the cross section's Y-Axis is in direction of the normal vector of the system line. To display the normal vector select from the pull down menu:

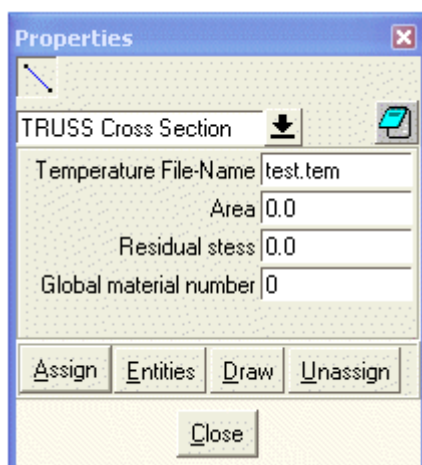
*Utilities->Draw Normals->Lines*

GiD displays the normal vector in magenta and also the direction vector of the line in light blue. If necessary you can change the direction by typing SWAP in the command line. Select the line to swap its sense.

To display assigned properties click the *Draw* button and select *Color*.

**Note:** All .TEM files must be copied into the *project-name.gid* directory before you start the calculation with SAFIR.

To assign Properties to TRUSS elements click to the arrow-icon and select TRUSS Cross Section:



Enter the file name of the temperature curve.

Enter the area of the cross section in square meters.

Enter the Global material number.

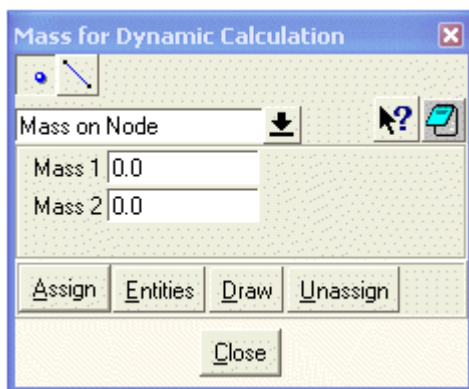
Assign the properties to a system line (or to beam elements).

## 7. Assign mass to nodes and/or beams

In the case of DYNAMIC calculation you can assign masses to nodes.

Select from the pull down menu:

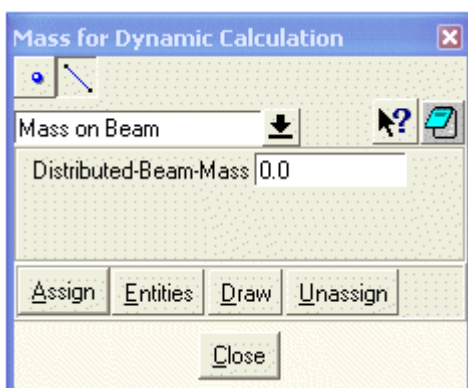
*Data->Conditions->Mass for Dynamic Calculation*



*Mass 1* is the mass [kg] linked to degree of freedom 1 of the node.

*Mass 2* is the mass [kg] linked to degree of freedom 2 of the node.

To assign self weight mass to beam elements click to the line icon .



*Distributed-Beam-Mass* is the self weight in kg/m of the beam.

Use *Assign* to assign this property to a system line.

## 8. Assign general problem data

Form the pull down menu select:

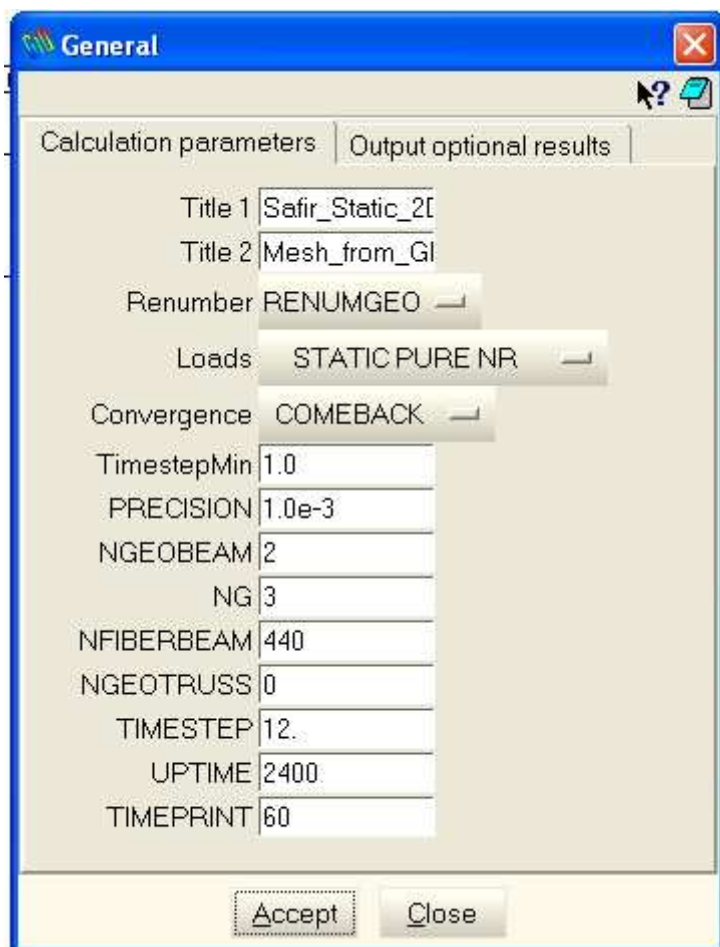
*Data->Problem Data*

This Dialog Box provides 2 tabs:

- *Calculation Parameters*
- *Output optional results*

Parameters you can set here are described in more detail in the SAFIR Ref. Manual.

*Title*: Allows to enter 2 text lines, which are used as title in SAFIR



*Loads:* Allows to select the type of calculation. You can select from the pull down list

- STATIC PURE\_NR
- STATIC APPR\_NR
- DYNAMIC PURE\_NR
- DYNAMIC APPR\_NR
- STATICCOLD PURE\_NR
- STATICCOLD APPR\_NR

*Convergence:* You can select COMEBACK and NOCOMEBACK

*TimestepMin:* Is the minimum time step in case of COMEBACK

*Precision:* Is the small value which must be reached for convergence.

*NGEOBEAM:* Number of beam cross sections having the same geometry, materials and temperature history. ( or in other words, this is the number of .TEM files )

*NG:* Number of integration points for beam elements in longitudinal direction ( can be 2 or 3, for HASEMI calculation NG must be 2 )

*NFIRBERBEAM:* Enter the maximum number of fibers ( elements ) in any cross section (.TEM file)

*NGEOTRUSS:* Enter the number of truss cross sections having the same geometry, materials and temperature history.

*TIMESTEP:* Enter the initial time step in seconds.

*TIMSTEPMAX:* this field is the maximum time step and is used only in dynamic calculation.

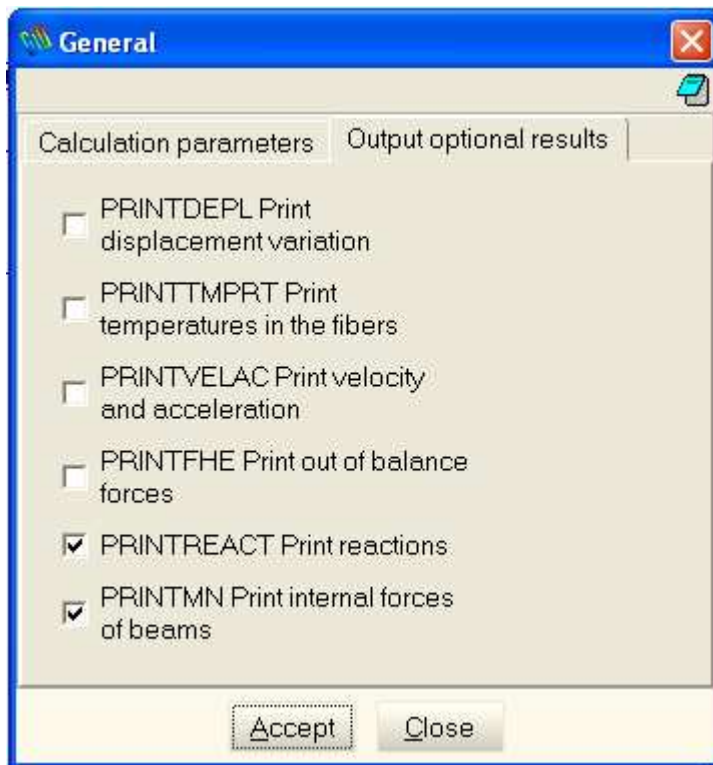
*UPTIME:* Enter the time limit for the calculation in seconds.

*TIMEPRINT:* Enter the time step for the output of results in seconds.

After you change any value of this dialog box you must push the *Accept Data* button ore *Close* to leave the dialog box without changes.



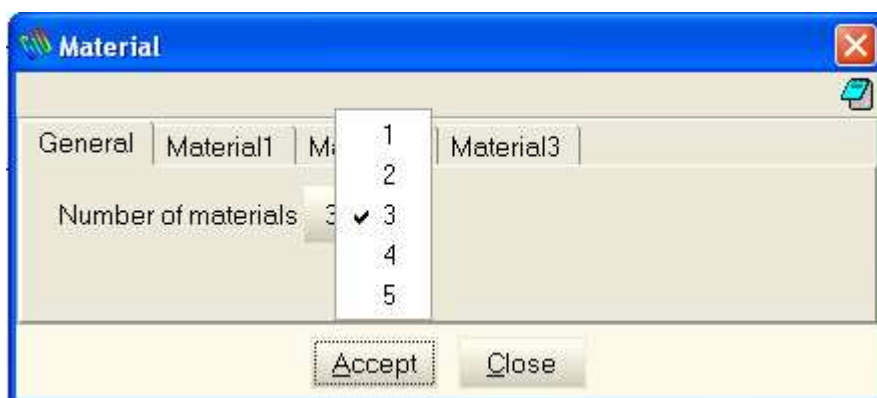
Selecting the tab *Output optional results* shows the following dialog box:



## 9. Defining material

To define materials used in the structure select following menu:

*Data-> Material*



In the General Tab. you have to select :

*Number of materials*: Enter the number of materials used in the structure.



For each of the material used in the structure define materials Material1, Material2, ...

From the pull down list you can select the following SAFIR materials:

- STEELEC3
- STEELEC3DC
- STEELEC2
- PSTEELA16
- SILCONC\_EN
- CALCONC\_EN
- INSULATION

For steels enter values for E-Modulus, Poisson ratio and Yield strength.

For concrete materials the mask changes and you can enter Poisson ratio, Compressive strength and Tension strength.

For material ELASTIC you can enter the E-Modulus and the Poisson ratio.



**Note:** The materials you define here are the Global Material numbers you assign to the local materials in Property Dialog box ( see above under point 5. )

## 10. Create the mesh

To create the mesh select form the pull down menu:

*Meshing-> Generate*

GiD displays a dialog box where you can enter the element size, which is used in the case of non-structured mesh. GiD displays the number of nodes and elements it created and displays the mesh.

If you want to control the number of elements, you can use structured meshing. Select from the pull down menu:

*Meshing->Structured->Lines*

Enter the number of elements to assign to lines.

To switch from mesh-view to geometry-view you can use the last Icon of the GiD-Tool box on the left side of the display, or use the pull down menu:

*Geometry->View Geometry*

*Meshing->Mesh view* or type *Ctrl-m*

## 11. File the project

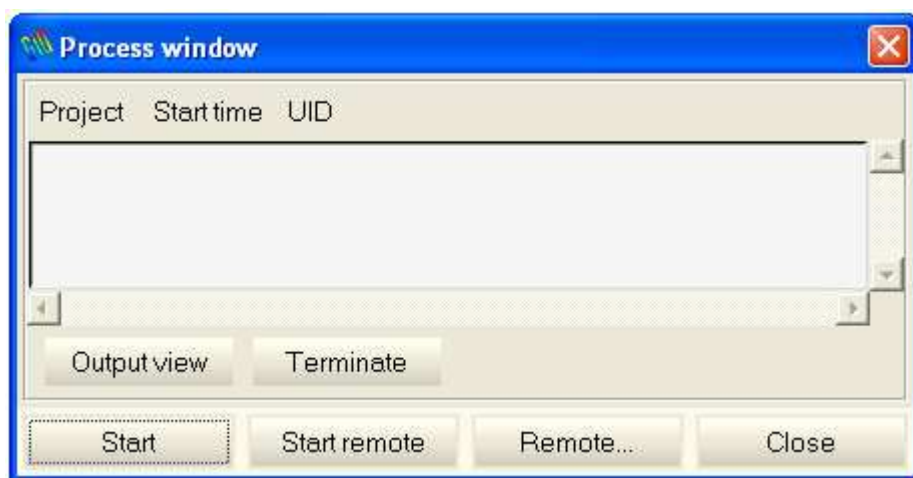
To save the project select:

*Files->Save* or type *Ctrl-s*

## 12. Create the SAFIR input-file and run SAFIR

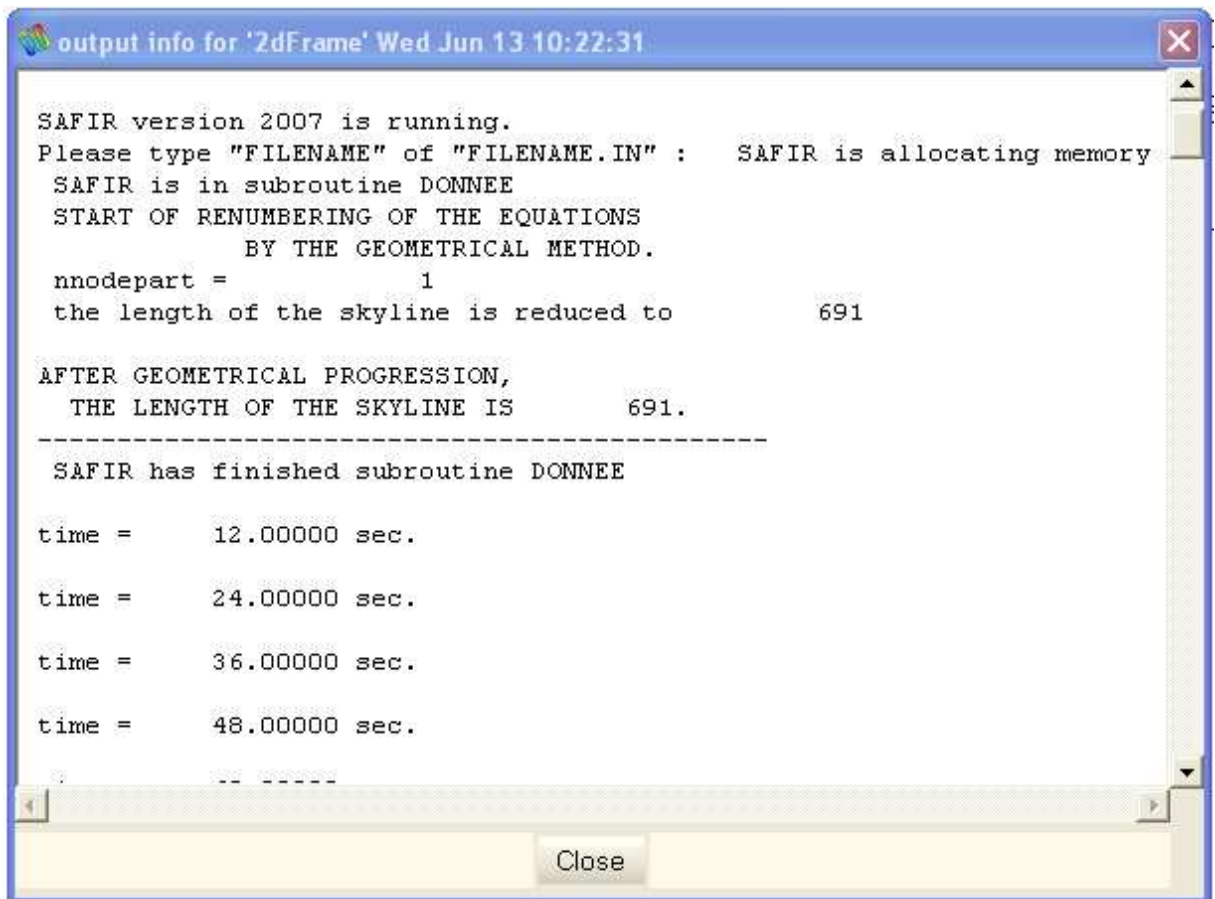
From the pull down menu select:

*Calculate->Calculate window*



GiD displays the process-Window. Click the *Start* button to start the calculation.

Click the *Output view* Button to display a window, where you can watch the progress of the calculation and also error messages of SAFIR.



When SAFIR has finished the calculation GiD displays a dialog box, which lets you directly start the GiD-Postprocessor.

If you prefer post processing with Diamond2004, remember that GiD has placed the SAFIR out-file and the tem-file in the *project-name.gid* directory.

### 13. Post processing

Postprocessing can be done with **Diamond**. The .OUT file is located in the *project-name.gid* directory . The file name is ***project-name.OUT***

For post processing with GiD select from the pull down menu:

*Files->Postprocess* or click the Postprocessor Icon in the tool box.

Select from the *View* pull-down list *Scalar Line Diagram* and in the results tree the component which you want to display. In the *Step* pull-down list select the time-step and press *Apply*:

