GiD – SAFIR_Thermal_2D User Interface

1. Create a GiD project of type *Safir_Thermal_2d*

From the pull down menus select:

Data->Problem type->SAFIR2007-> Safir_Thermal_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.

The GiD SAFIR_Thermal_2d Interface allows also creating a TORSION file**.**

2. Create the geometry of the cross-section.

Units of length must be meters.

The cross section must be designed in the xy-plane.

In terms of SAFIR the GiD y-coordinate equals the SAFIR Y-coordinate (the first global coordinate), the GiD x-coordinate is the second global coordinate, which is in SAFIR denoted as Z-coordinate.

GiD provides several menus for creating geometry (see GiD reference manual and online Help). Also geometry can be imported from DXF and IGES files.

Create the geometry as one or several closed contours and create from those contours NURBS surfaces. The surfaces are finally used for meshing.

For I-type profiles GiD provides a macro in the pull-down menu: *Cross_section*

Note: GiD indicates surfaces by magenta offset-curves to the bounding contours. In the case of cross sections consisting of different materials and surfaces it is sometimes difficult to select surfaces by picking the offset contour. Then it is advisable to use layers (see GiD-Reference Manual), which can be switched on and off.

3. Assign materials

Form the pull-down menu select:

Data->Materials

GiD displays the following dialog box:
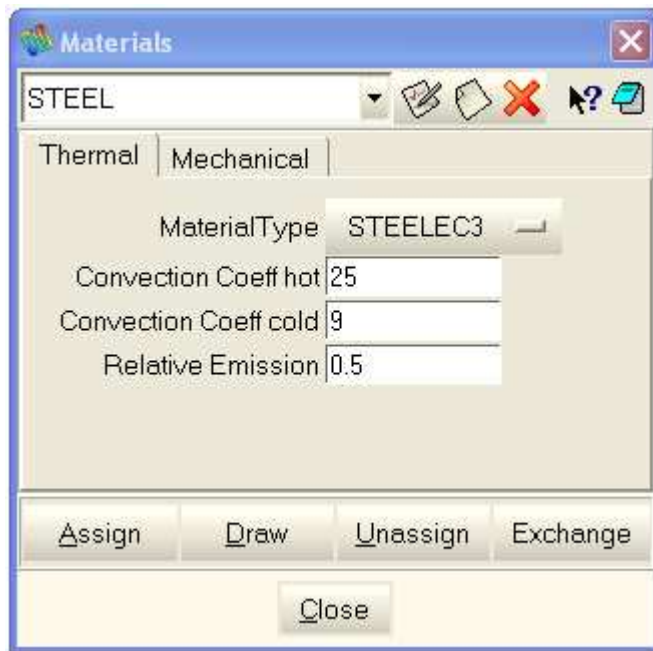
Select a material and *Assign* it to a surface. Use the *Draw* button to display the assigned material in a filled color mode.
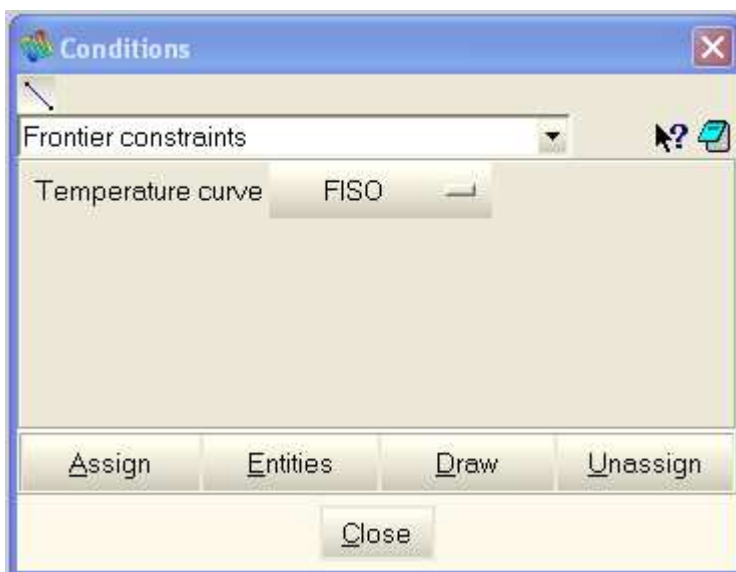
Note: Mechanical properties are only used in case of TORSION calculation.

4. Assign constraints

Frontier constraints

Frontier constraints can be assigned by:
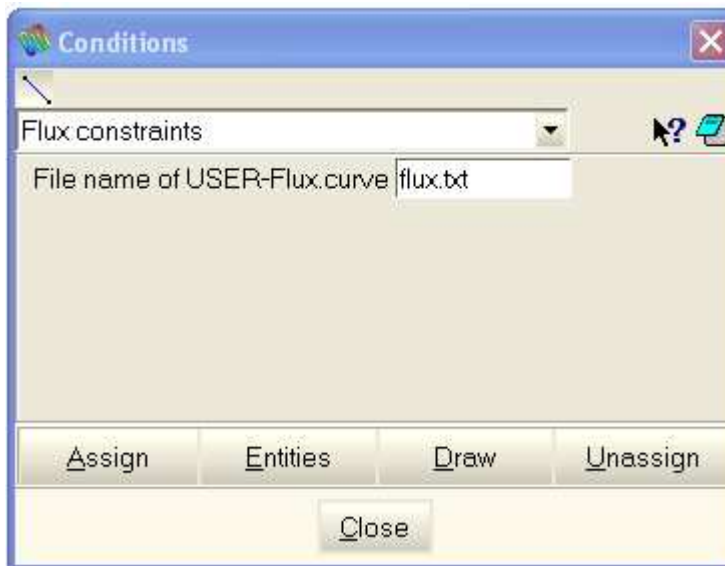
Data->Conditions->Frontier constraints

Select one of the predefined SAFIR - temperature curves or select *USER* and enter the name of a user defined temperature curve. Assign the temperature curve to the boundary curve of the cross section. The button *Draw/color* lets you draw the frontier

in color. If you use a user defined temperature curve you must place this file in the *project-name.gid* directory before calculation starts.

Flux constraints

Flux constraints can be assigned by:
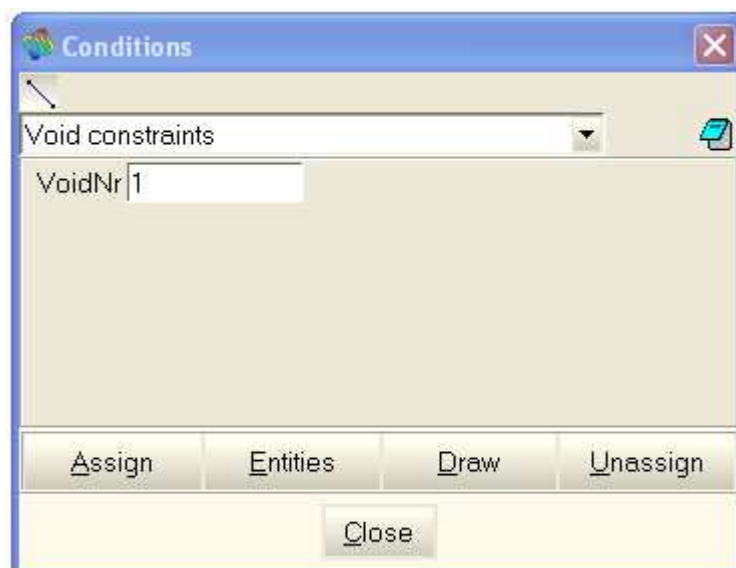
Data->Conditions->Flux constraints

For a MAKE.TEMHA calculation you may enter instead of a filename HASEMI.

Void constraints

Void constraints can be assigned by:

Data->Conditions->Void constraints

You must assign this constraint by picking the entities of the contour surrounding the void. A restriction is that this contour must be designed in a counter-clockwise manner. To verify this you can display the curve normal vectors:

Utilities->Swap normals->Lines

All normal vectors should point to the inside of the void. Vectors, which have the wrong orientation, can be reversed by picking the curve.

If there are several voids in the cross section, they must be consecutively numbered starting with 1, in the text-box *VoidNr*.

In the *Problem Data* – dialog box (see point 7) you must enter the number of voids (NVOID) and the maximum number of elements in any void (NFRONTIERVOID).

5. Creating the mesh

You can create either triangular or quadrilateral meshes. Quadrilateral meshes can also be structured.

The default element type is the Triangle. For Quadrilateral elements select:

Meshing->Element type->Quadrilateral

Select the surfaces to which this element type is to be assigned.

To create the mesh select form the pull down menu:

Meshing->Generate mesh

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.

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:

View->Mode->Geometry

View->Mode->Mesh

6. Input general data for SAFIR

From the pull down menu select:

Data->Problem data

For MAKE.TEM the dialog box looks as follows.

GiD displays a dialog box, where you can enter general data SAFIR uses for the calculation. All variables have the same name as in the SAFIR reference manual und have predefined values. When you click with the right mouse button on the variable name, GiD displays an online help.

TETA: Parameter for the time integration (0 < TETA < 1)

TINITIAL: Temperature of the structure at time t = 0.

Renumber: From the combo-box you can select NORENUM, RENUM,

RENUMPERM, RENUMGEO, READRENUM.

Type of calculation: MAKE.TEM , MAKE.TEMHA, and TORSION .

The file-name of the result-files for the MAKE.TEM and MAKE.TOR calculation is

automatically given by SAFIR by expanding the GiD-Project name with .TEM or

TOR.

 For a MAKE.TEM calculation the MAKE.TOR calculation can be also started

automatically and the result of the .TOR – file will be inserted into the .TEM – file by the GiD-SAFIR interface.

To insert the torsion properties manually into the .TEM file you must use an editor. (See appendix A)

Global Center (Y0,Z0): first and second global coordinate of the center of the cross section for the structural calculation. (Note: in the GiD window Y0 is the vertical y-Coordinate, Z0 is the horizontal x-Coordinate of the cross section center)

Center of Torsion (Yc,Zc): Coordinate of the center of torsion (The y,x Coordinates in the GiD window)

NVOID: Enter the number of voids in the cross section

NFRONTIERVOID: Enter the maximum number of surfaces (i.e.: sides of an element) Enclosing the internal void.
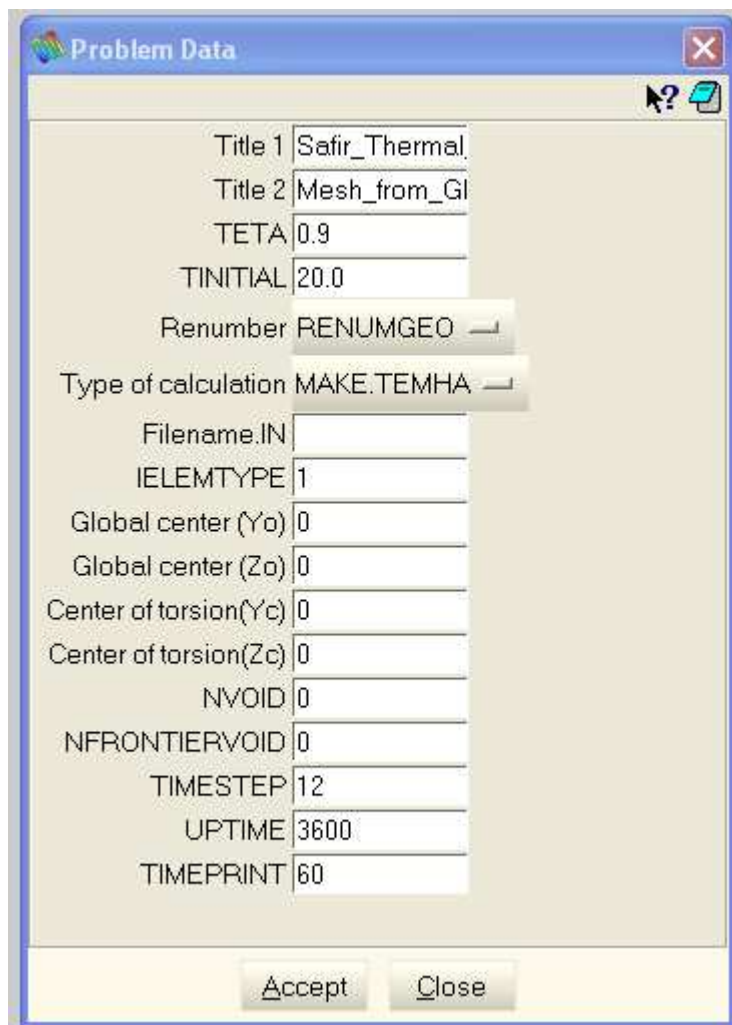
TIMESTEP: Time step in seconds.

UPTIME: End time in seconds.

TIMEPRINT: Time step for printing results.

When you select MAKE.TEMHA the dialog box looks slightly different.

Filename.IN : Enter the name of the complete .IN file of the structural analysis. This file must be present in the GiD-project directory when the analysis is started. Also the file HASEMI.txt must be located in this directory (see the SAFIR2007 manual)

IELEMTYPE: is the number in the structural input file of the beam type to be analysed in this thermal analysis. SAFIR creates for each beam of this element type two .TEM files (bxxxx_1.Tem and bxxxx_2.TEM , xxxx is the beam element number).

7. File the project

To save the project select:

Files->Save or type *Ctrl-s*
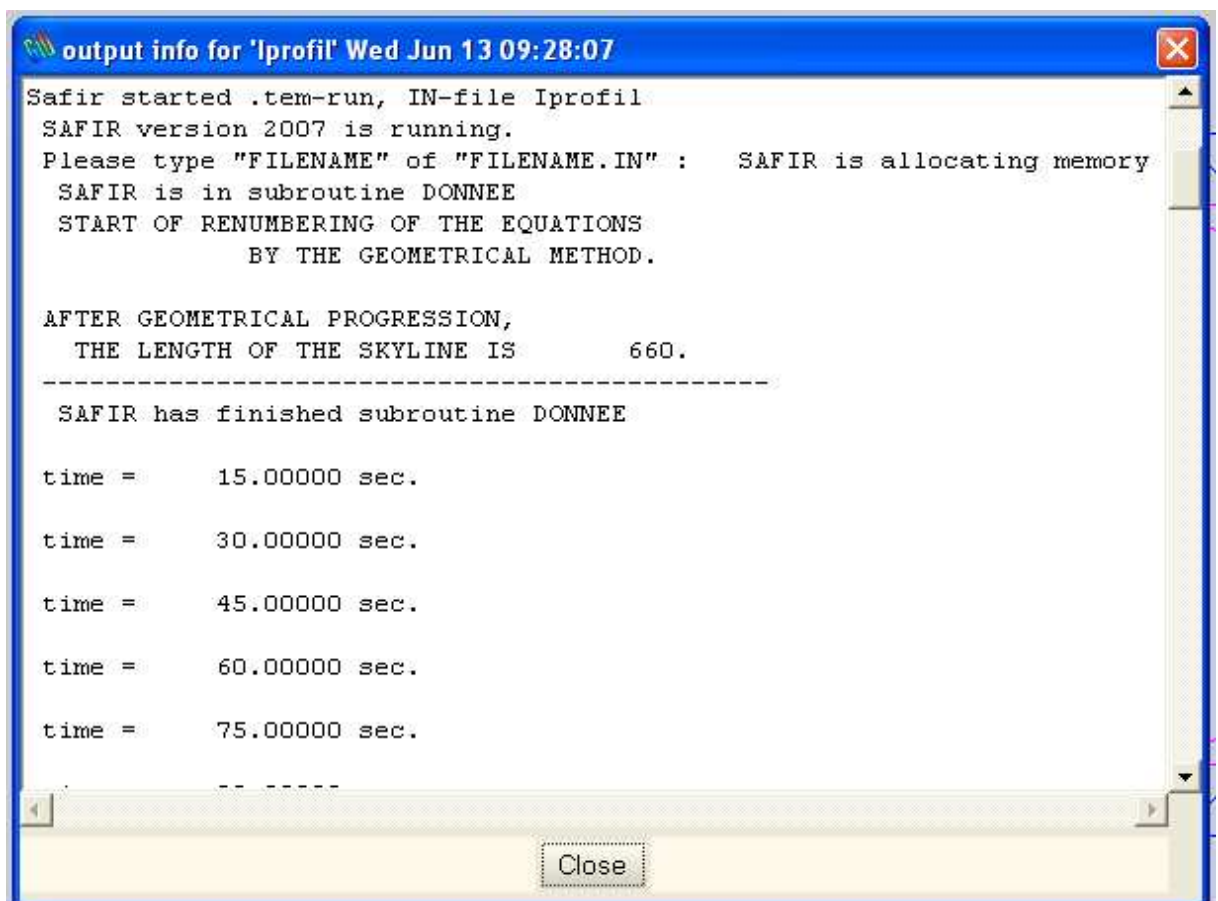
8. Create the SAFIR input-file and run SAFIR

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
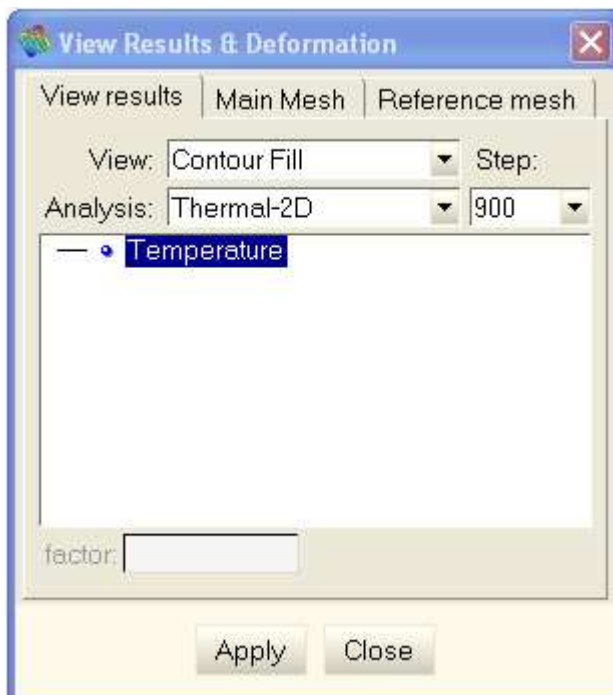
9. Post processing

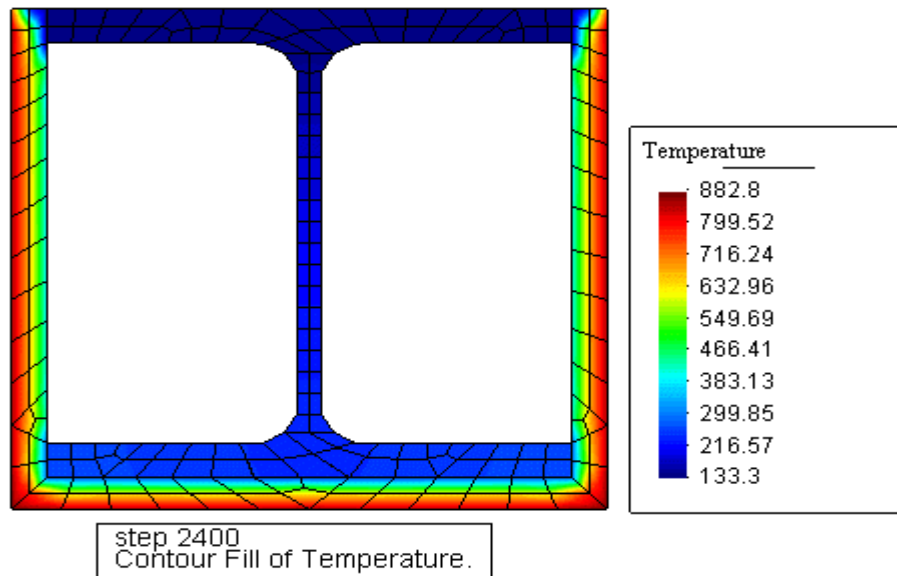For post processing in GiD select from the pull down menu:

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

From the pull down menu select:

Windows->View results

From the *View Button* select *Contour fill,* then select a time step and press *Apply*

step 2400
Contour Fill of Temperature.

To leave the Postprocessor select:

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

Appendix A.

Introduce the torsion properties in the .TEM file.

1) Open the .TEM file which had been created during the temperature analysis and open the .TOR file, which has been created during this torsion analysis. They look very similar (the number of fibres, NFIBERBEAM, must be the same in the 2 files), with the difference that the value of the warping function and of the torsional stiffness is present in the torsion file.

2) From the .TOR file, copy the group of lines starting with

```
w
```

and finishing with

```
According to the principle of virtual works,
     GJ = 0.919990E+08
```

3) Insert this group of lines in the .TEM file just before this line

```
HOT
```

The .TEM file is now ready to be used in a 3D structural analysis.