Ir Lansival Jean-Baptiste
Université de Liège
Le 28/11/07

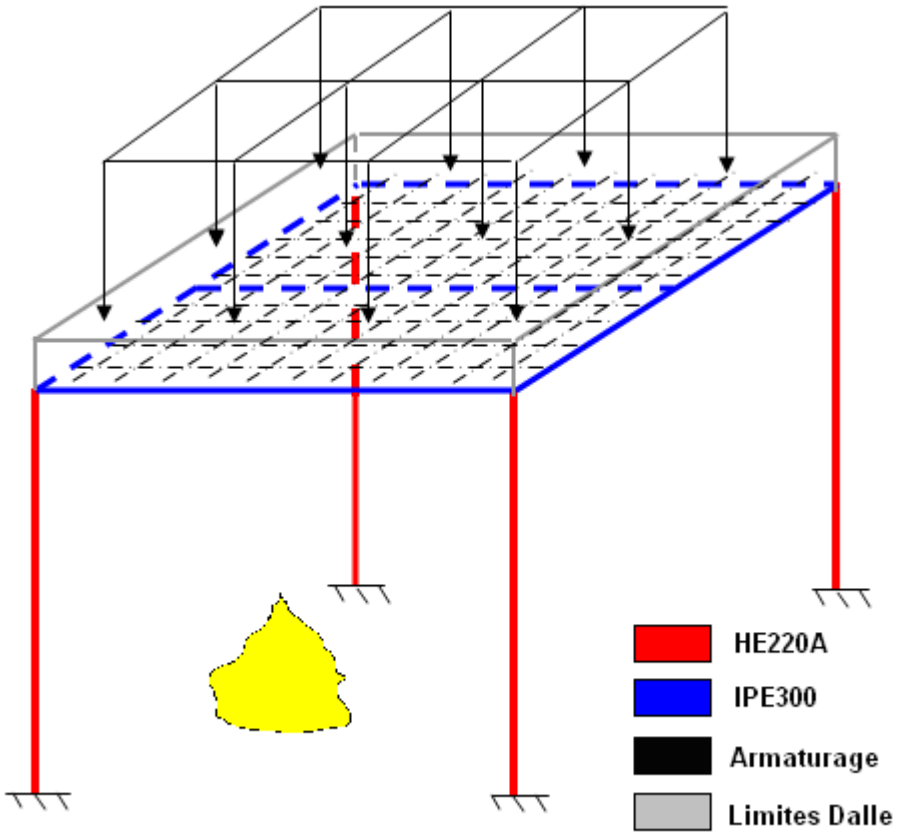
Réalisation d'une simulation 3D et feu HASEMI

BUT

Ce document illustre, sur base d'un cas particulier, la méthodologie et la procédure à suivre pour réaliser une simulation d'une structure en trois dimensions soumise à un feu localisé (*HASEMI*).

PRESENTATION DU CAS PARTICULIER

Il s'agit d'une structure formée de portiques en acier (*HE220A* pour les poteaux et *IPE300* pour les poutres) sur lesquels repose une dalle en béton armé (10 cm d'épaisseur armée de manière orthotropique). Cette structure sera soumise à un chargement vertical uniformément réparti sur la dalle. Les appuis aux pieds des quatre poteaux seront modélisés comme des encastrements.



Dans ce qui suit, nous tâcherons donc d'expliciter la procédure à suivre de manière mener à terme la simulation d'une structure 3D soumise au feu *HASEMI*.
Nous commencerons alors par l'explication de la modélisation de l'ouvrage sous le logiciel **GID** (3D) ainsi que la modélisation des sections sous **WIZARD**, nous donnerons par la suite les modifications à apporter aux fichiers ***.in*** créés par ces logiciels en tenant compte des particularités qu'amène la simulation d'un feu localisé et le passage en 3D (nous verrons alors comment retravailler le fichier ***.in*** modélisant l'ensemble de la structure et ceux modélisant les sections).

PROCEDURE

1. Modélisation de la structure sous GID

But :
- Définir la géométrie de la structure
- Définir les caractéristiques des matériaux
- Définir le chargement
- Donner les appuis
- Réaliser le maillage
- Assigner à chaque élément le type de section qui lui correspond

Procédure: Cfr. "*Tutorial for GID-SAFIR 3D Structural Analysis*" joint à ce document. Notons que ce document présentant la méthodologie à employer pour créer une structure 3D sur le logiciel **GID** a été rédigé sur base d'un cas particulier différent mais similaire celui qui nous occupe ici.

Il en ressort un fichier ***.in*** décrivant la structure et comprenant l'ensemble des informations citées ci-dessus.

Il est à noter que ce fichier *Structural* doit être réalisé **avant** tout autre chose. En effet, le calcul thermique (*HASEMI*) fait référence à ce fichier de manière à déterminer la position de chacun des éléments de la discrétisation et ainsi déterminer le flux qu'il reçoit (fonction de la distance séparant ce dit élément de la projection verticale du foyer sur le plafond de l'ouvrage).

Remarque :
Il conviendra également de vérifier que les nœuds de la dalle et des poutres qui ont la même position possèdent des déplacements égaux (suivant le mode d'assemblage de ces deux éléments Cfr. *SAME*).
Notons également qui peut y avoir une incompatibilité du nombre de degré de liberté de ces nœuds dont on veut imposer les mêmes déplacements. Il convient alors de définir un nombre de degré de liberté pour ces nœuds égal à la valeur maximale entre le nombre de degré de liberté de chacun de ces mêmes nœuds.
Dans l'exemple qui nous occupe, nous devions imposer que les nœuds de la dalle possédant les mêmes coordonnées que les nœuds des poutres

aient les mêmes déplacements. Or **GID** définit 6 DDL pour les SHELL et 7 DDL pour les BEAM. Dés lors, nous avons donner au nœuds des SHELL coïncidant avec les nœuds des poutres un nombre de DDL égal à 7.

2. Modélisation des sections des poutres et poteaux à l'aide de Wizard

But : - Définir la section des poutres et colonnes
- Mailler cette section
- Déterminer les faces exposées et celles qui ne le sont pas
- Donner les caractéristiques de la protection éventuelle

Il en ressort des fichiers ***.in*** comportant ces renseignements.

3. Création du fichier décrivant le feu local

Un fichier intitulé « *hasemi.txt* » doit être créé. Ce dernier contient les informations sur la position du feu, sa puissance, son diamètre etc.

Description du feu local *"HASEMI"*

```
NFIRE  1                                  Number of localised fire

FIRE_POS  2.   2.   0.                    Position
HEIGHT   4.                               Distance between the fire and
DIAMETER                                  the ceiling
        0.     0.                         Time (s) - Fire source diameter (m)
        300.    2.
        400.    3.
        600.    4.
       6000.    4.
END_DIAM
   RHR
      0.    0.                            Time (s) - Rate of heat
      600.  900000.                       release (w)
      6000. 900000.
 END_RHR
```

4. Modifications pour HASEMI et passage de 2D en 3D

Les fichiers obtenus en 2 doivent alors être modifiés de manière à imposer un feu *HASEMI* et pour passer du 2D en 3D.

La prise en compte du feu localisé se fera par l'intermédiaire du remplacement de *F* (dans la rubrique intitulée *FRONTIER*) par *FLUX* et de *FISO* par *HASEMI* (la condition *F0* n'est alors pas nécessaire).
Il convient également de remplacer *MAKE.TEM* par *MAKE.TEMHA* suivit du nom du fichier ***.in*** réalisé en 1 (Structure entière). Il convient également

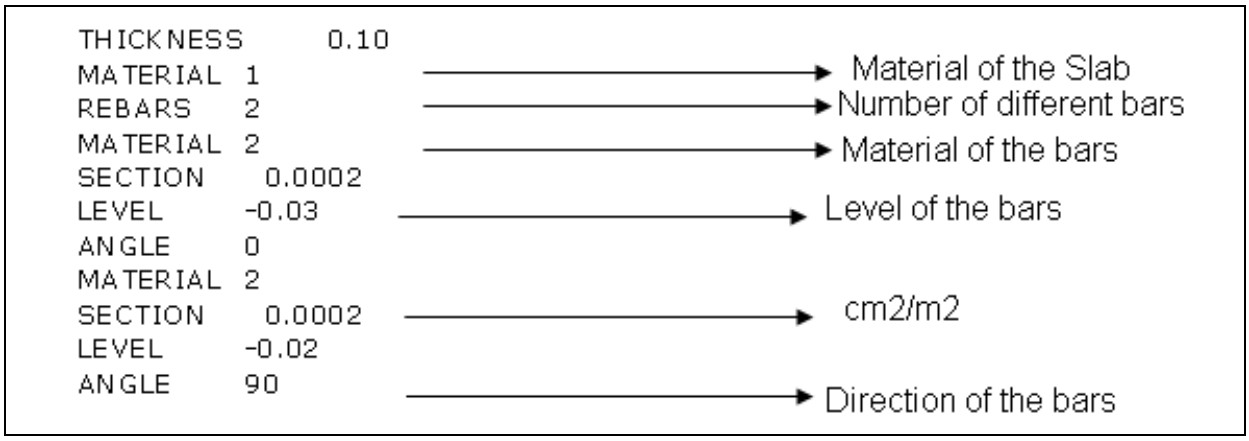
de préciser quel type de section l'on traite (*BEAM_TYPE* ou *SHELL_TYPE* suivit du numéro du type d'élément considéré).
Des indications plus détaillées concernant ce qui précède pourront être consultées dans le document intitulé « *Modification for the mechanical analysis* » (joint à ce document).

Une fois ces modifications réalisées, il suffit de lancer le fichier sur **SAFIR**, les résultats sont les fichiers *…001_1.tem*, *…001_2.tem* etc. (1 et 2 font référence au point d'intégration de Gauss, 001 correspond, pour un type de section donné, au numéro de l'élément considéré).

5. Modélisation de la section de la dalle

Il faut créer un ***.in*** (à l'aide de **GID** par exemple) contenant les informations nécessaires aux calculs telles que la géométrie, le maillage, les matériaux etc.
Notons qu'il ne faut pas ici modéliser l'armature, ceci sera fait dans une prochaine étape.
Les modifications en vue de tenir compte du feu localisé et reprises au point 4 sont également ici d'application.

De manière analogue à ce qui a été dit au point 4, il faut, une fois ces modifications faites, lancer le calcul thermique sur SAFIR. Les résultats obtenus sont un nombre de fichier *S0001_1.tsh*, *S0001_2.tsh* … égal au nombre d'élément SHELL de la dalle.
Il faut alors, dans le premier fichier *…tsh* obtenu (*S0001_1.tsh*), donner les caractéristiques géométriques de la dalle et de son armaturage.



Une fois ces résultats obtenus, il faut alors réaliser le passage en 3D.
Ceci peut être fait en modifiant à nouveau les fichiers ***.in*** obtenus par **WIZARD** (Cfr. Point 2 : section des poutres et poteaux) de manière à calculer la rigidité torsionnelle des poutres et poteaux. Il faut alors supprimer les commandes *TEMPERAT, TETA, TINITIAL, MAKE.TEM* et les remplacer par *TORSION*. Il convient par la suite de remplacer le nom de

fichier ….*OUT* par *….tor*. Il faut également supprimer tout ce qui se trouve sous la rubrique *FRONTIER* ainsi que les commandes *TIME* et *ENDTIME*.
Les propriétés thermiques des matériaux doivent être changée en les propriétés mécaniques pour l'évaluation de la rigidité torsionnelle.
Une fois l'ensemble de ces changements réalisés, il suffit de faire tourner ces fichiers sous **SAFIR** pour en retirer les résultats dans le *….tor*.

Cette opération doit être effectuée pour chaque type de section.
Les résultats du *.tor* seront alors recopier dans le premier fichier bxxxx-1.tem de chaque type de section.
Des renseignements supplémentaires sur ce qui précède pourront être consultés dans le document intitulé « *This Files explains the different steps that have to be … for a 2D analysis* » (ci-joint).

Finalement, il faut changer les noms des fichiers décrivant chacun des éléments dans le fichier ***.in*** obtenu en 1 dans la rubrique *TRANSLATE*.
Il faut remplacer le nom de ces fichiers *.tem* par le premier fichier *bxxxx_1.tem* (et de même pour les SHELL).
Par exemple, HE220A.tem deviendra B0001_1.tem. Si cette poutre a été discrétisée en 40 éléments, le nom IPE300.tem sera remplacé par B0041_1.tem et ainsi de suite. De même, le nom de fichier dalle.in sera remplacé par S0001_01.tsh.

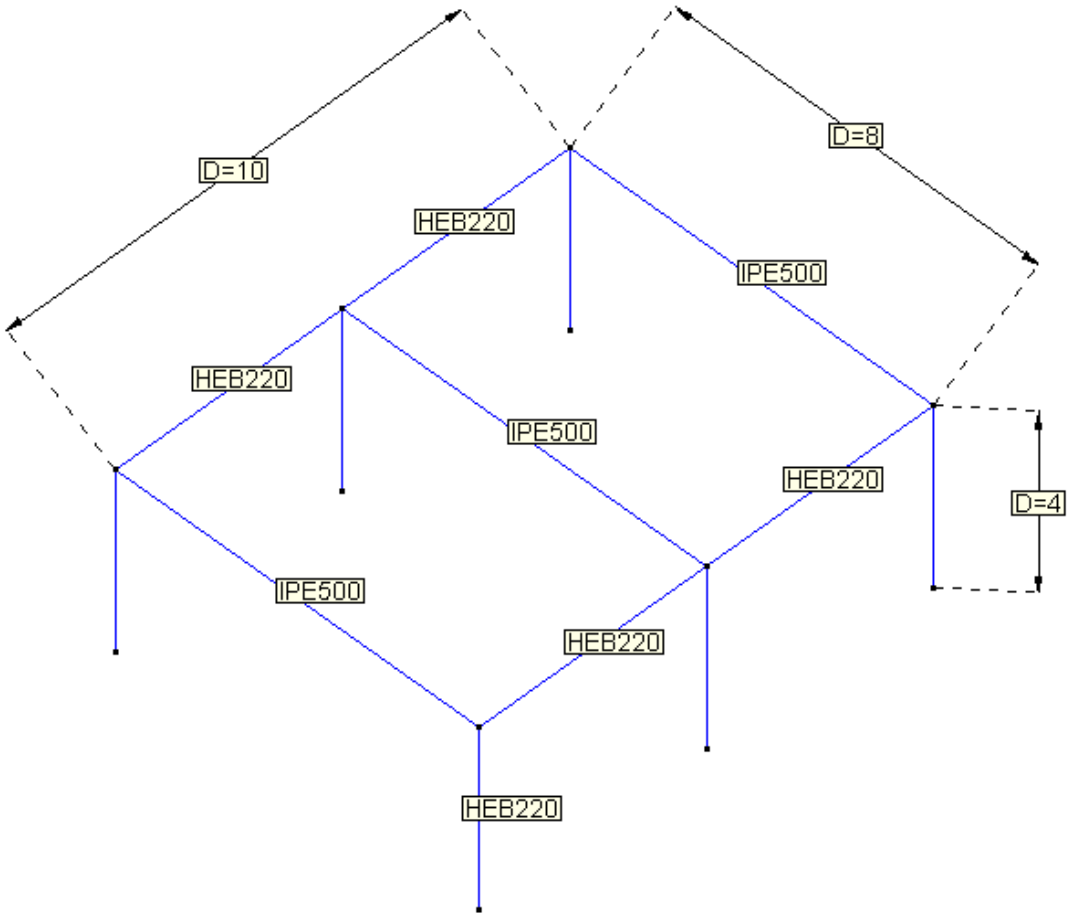
Le fichier ***.in*** contenant l'ensemble des informations de la structure 3D est à présent complet, il peut donc être lancé avec **SAFIR**.

Annexe

- GID2SAFIR3DStructuralTutorial.pdf
- How to go to 3D.pdf
- Modification for the HASEMI fires.pdf

Tutorial for GID-SAFIR 3D Structural Analysis

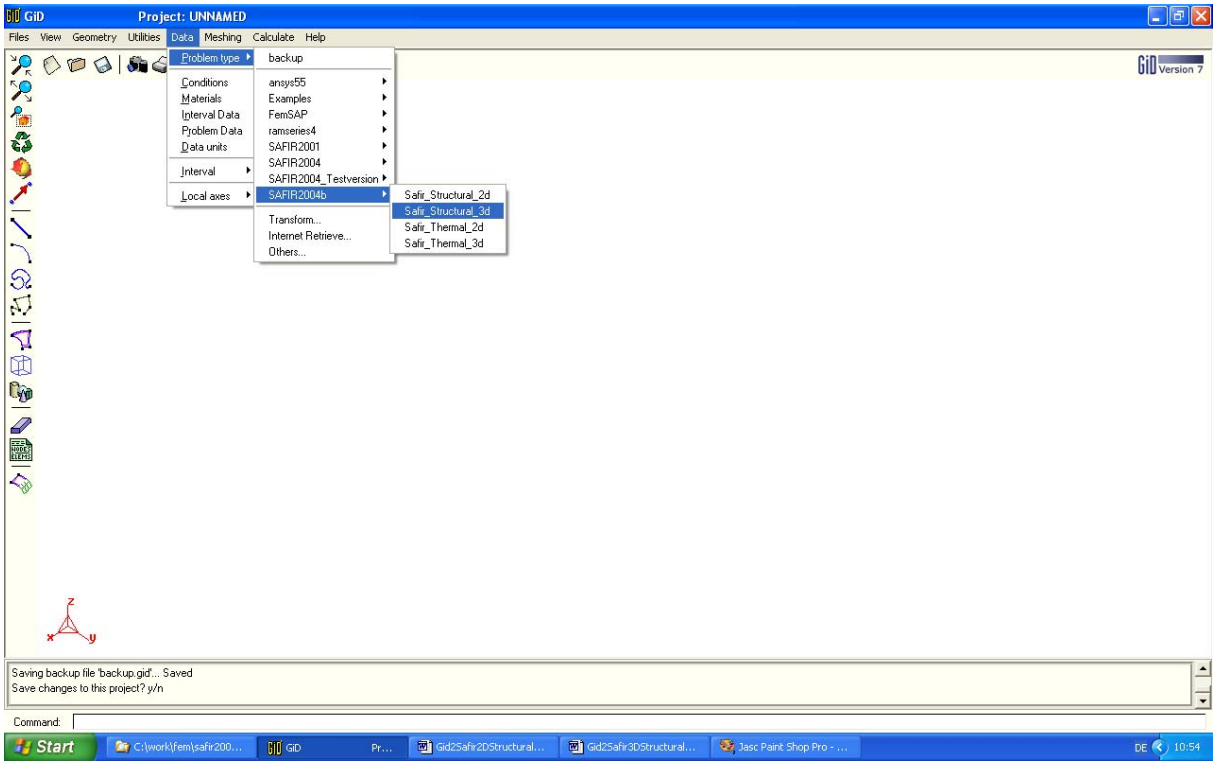
1. **The example is a 3d frame** with the following system geometry and beam profiles. There are 3 temperature zones, the .tem files used are HEB220B0.tem (cold), HEB220B1.tem (750° max), HEB22B2.tem (450°max); IPE500B0 (cold); IPE500B1.tem (750° max), IPE500B2.tem (450° max) and can be found in the gid2safir/examples directory.



2. **Create a new project of type Safir_Structural_3d**

From the pull down menu select:

Data->Problem type->SAFIR2004b->Safir_Structura_3d



Enter a file name, eg.: ***frame3d***

GiD creates a directory with the name ***frame3d.gid***

GiD creates a number of system files in this directory.

When you start the SAFIR calculation the SAFIR **. IN** and**.OUT file** are placed here.

Also before starting a calculation all **.TEM** files must be placed to this directory.

3. **Create the system geometry**

To change to the 3d isometric view select from the pull down menu:

View->Rotate->isometric

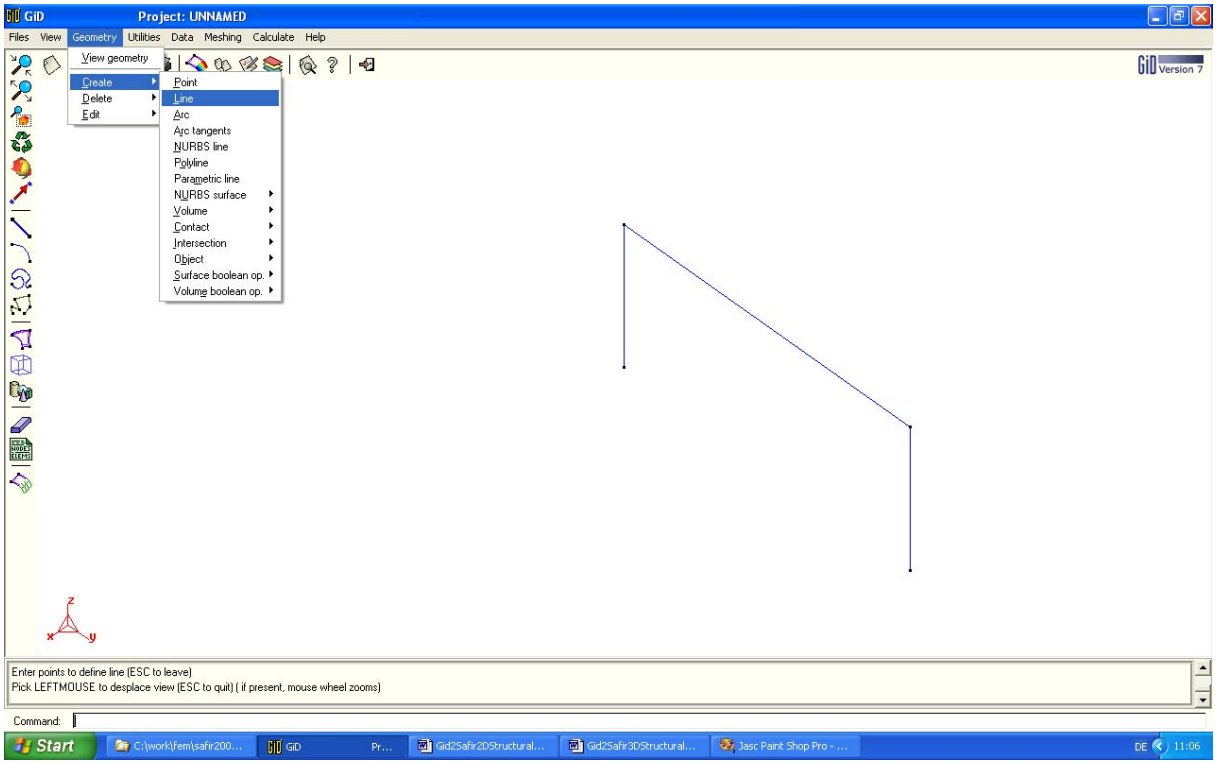
Create the system lines:

Geometry->Create->Line

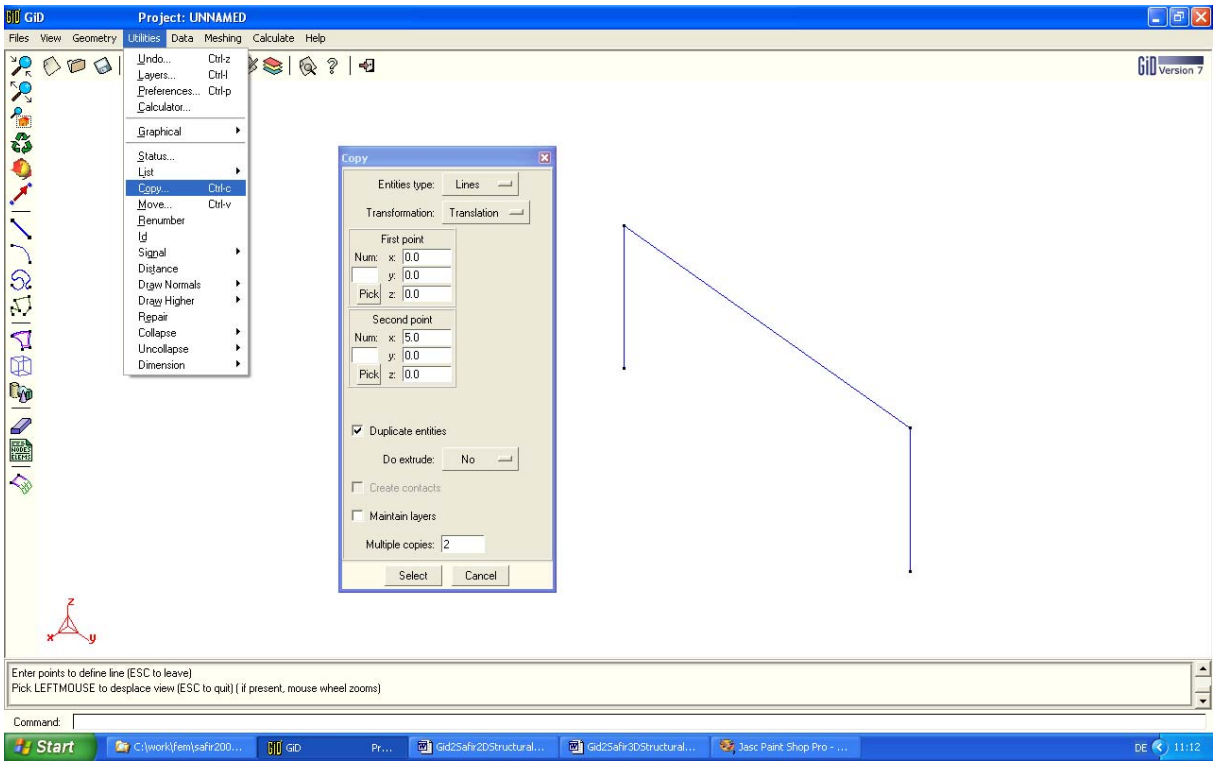
Enter in the command line

0,0,0 0,0,4 @0,8,0 @0,0,-4 Esc

The first frame is created.



Use ***Utilities->Copy*** to create the next 2 frames.

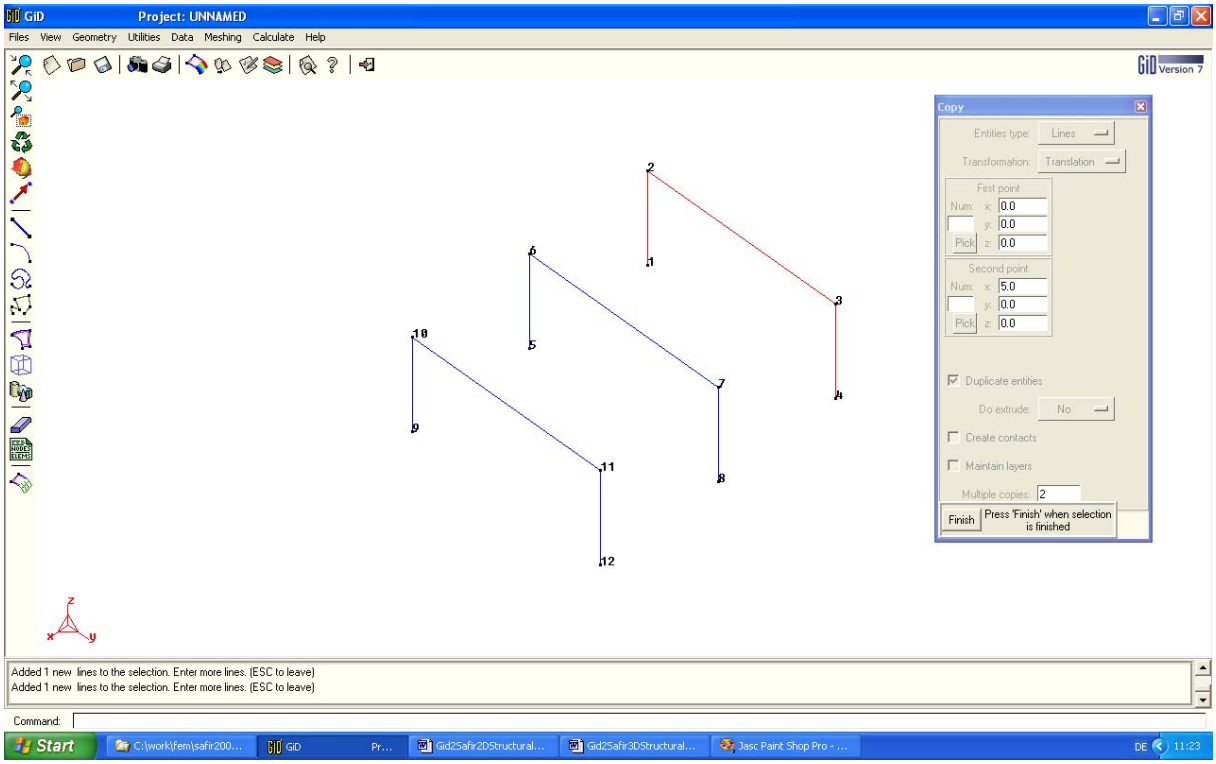


Select for Entity type: ***Lines***

Enter for Second point: ***x = 5***

Enter for multiple copies: ***2***

Select the lines and press ***Finish***



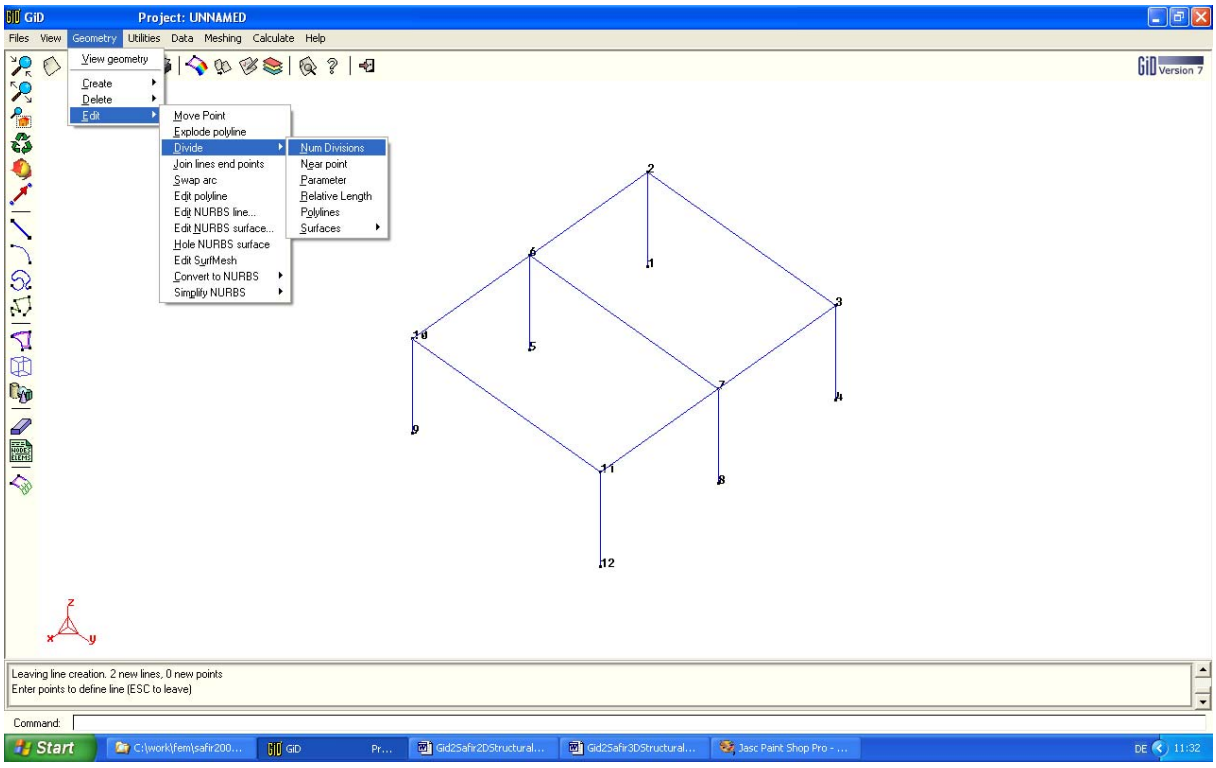
Next we connect the endpoints of the columns in x direction:

Geometry->Create->Line

Enter in the command line **Ctrl-a** and pick point 2 , 6 and 10 and complete with Esc

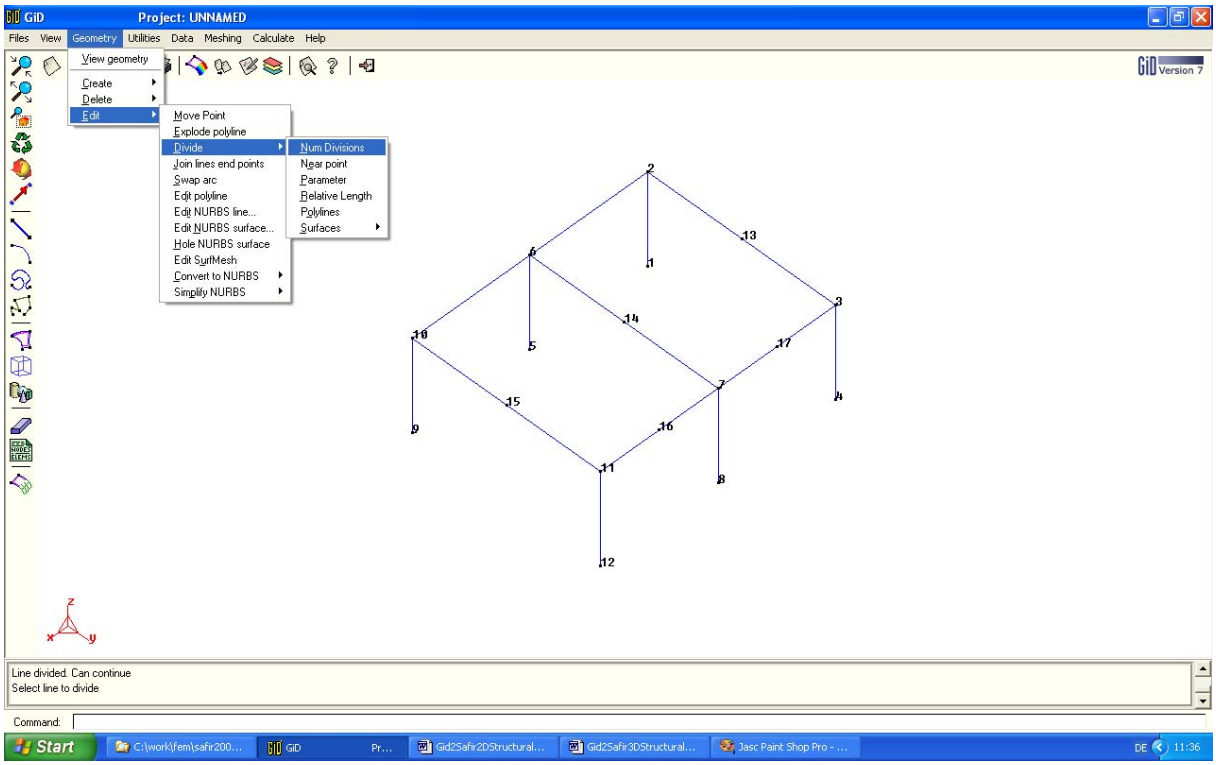
Geometry->Create->Line

Enter in the command line **Ctrl-a** and pick point 3 , 7 and 11 and complete with Esc



Divide the beams 2-3, 6-7, 10-11, 3-7, 7-11 in 2 pieces.

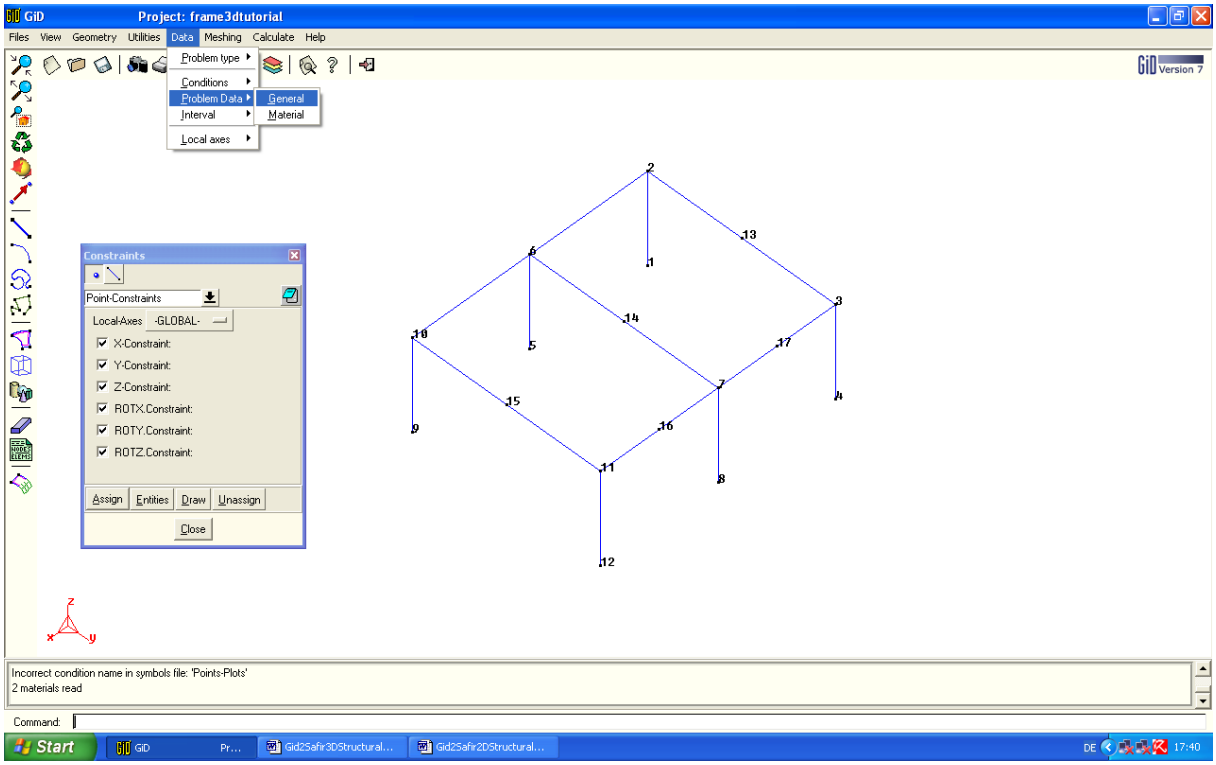
Geometry->Edit->Divide->Num.Divisions



4. **Define constraints for the supports:**

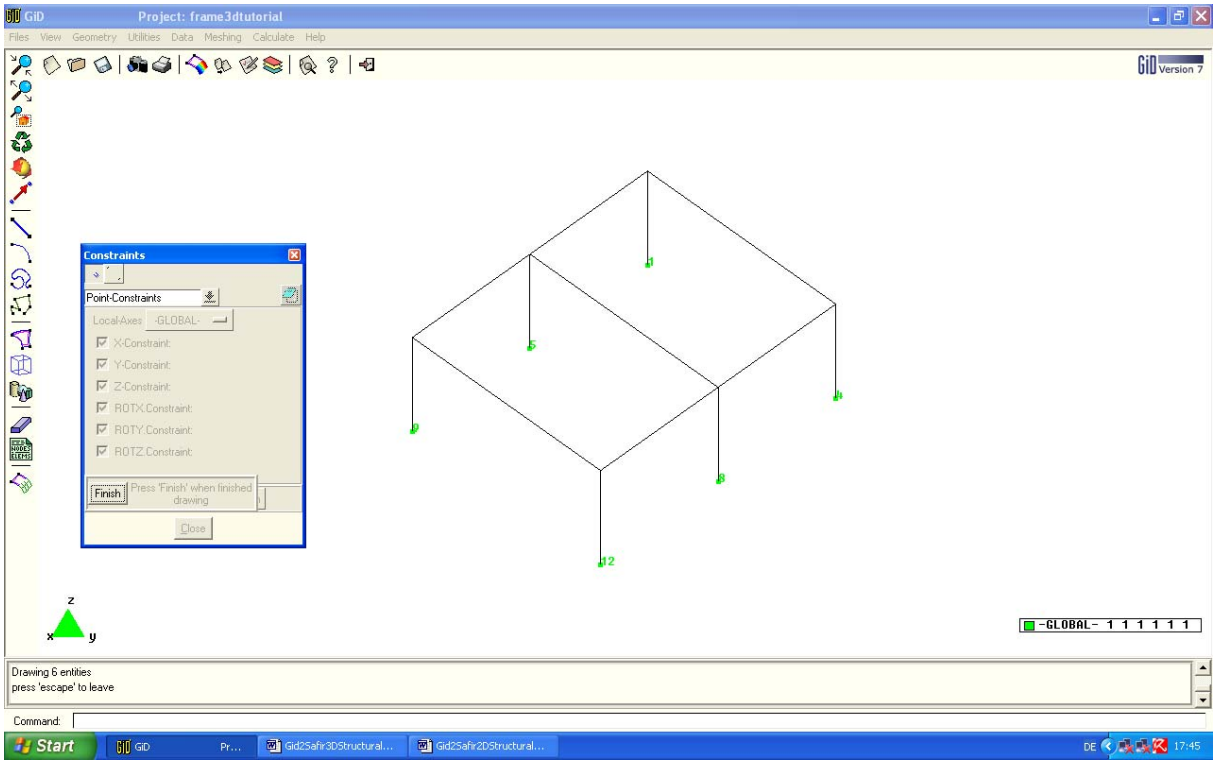
From the pull down menu select

Data->Conditions->Constraints



Click the ***Assign*** button to assign the constraints to the base points of the columns.

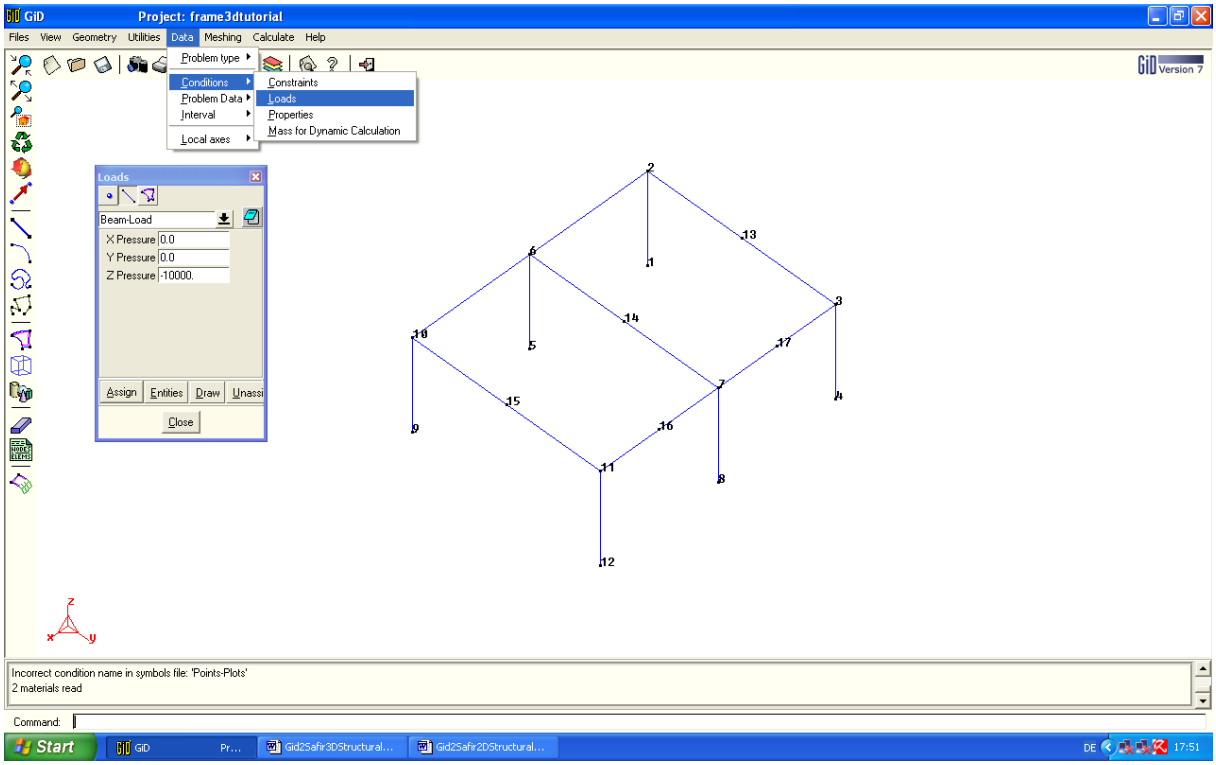
To display the constraints click ***Draw->Colors***



5. **Define loads**

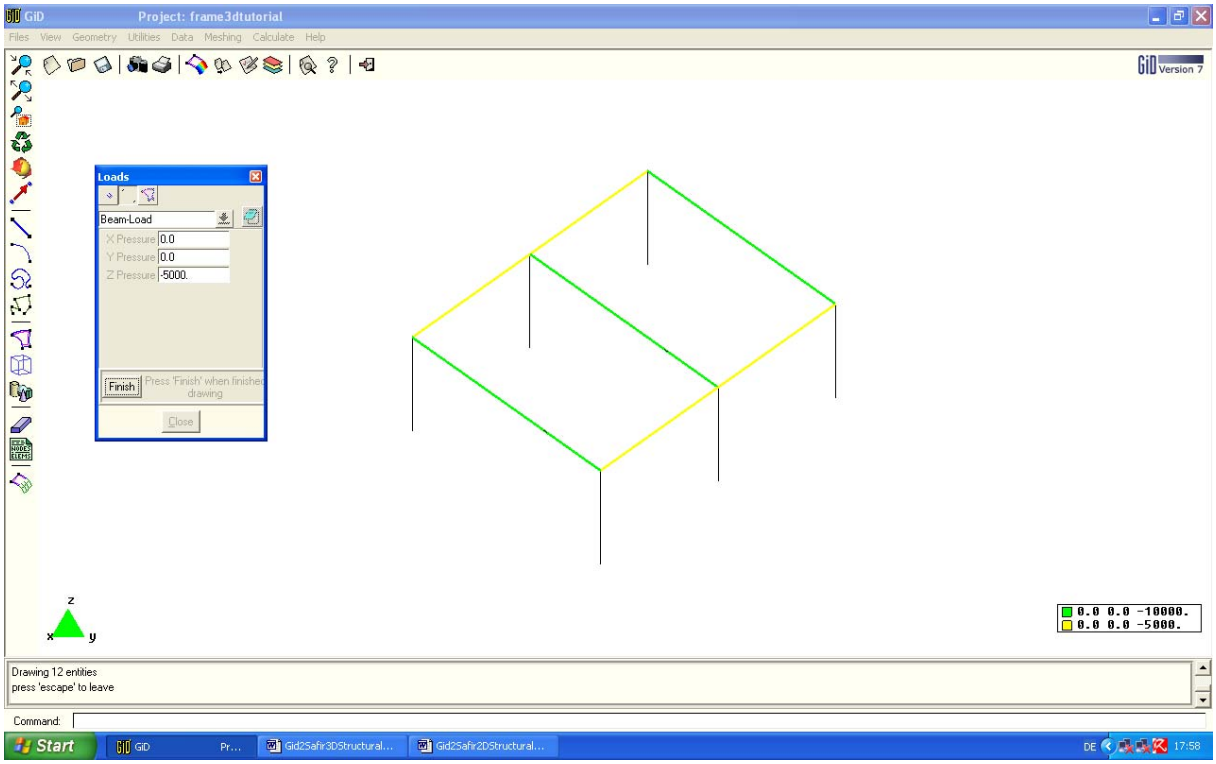
From the pull down menu select:

Data->Conditions->Loads



Select ***Beam-Load*** and enter for Z-Pressure –10000. and ***Assign*** it to all beams in y-direction. Enter a Z-Pressure of –5000. and assign it to all beams in x-direction.

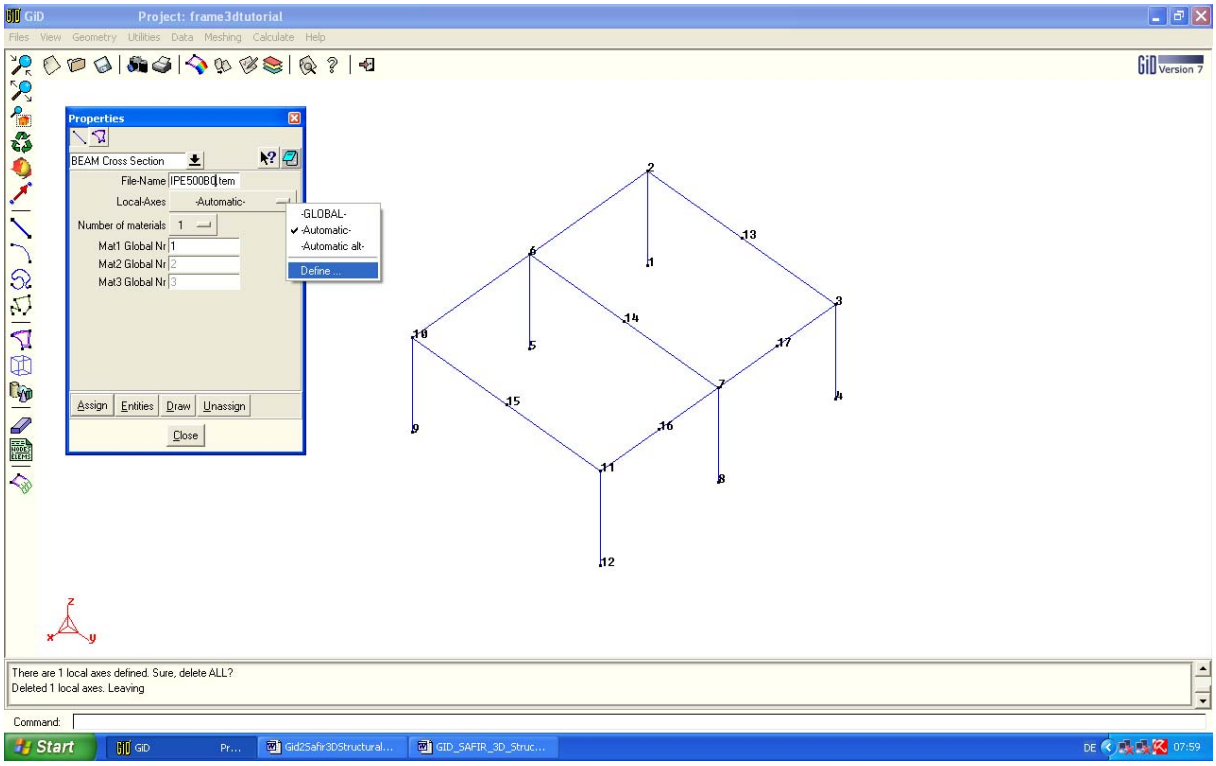
To display the loads select ***Draw->Colors***



6. **Assign temperature files (.TEM files)**

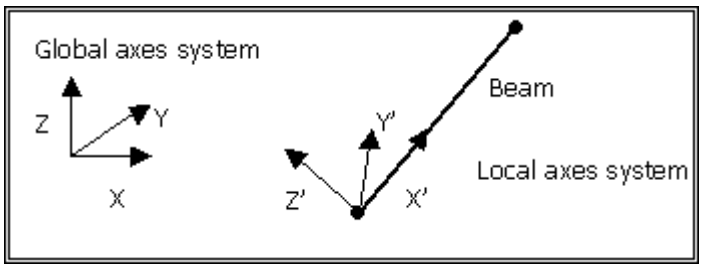
From the pull down menu select:

Data->Conditions->Properties



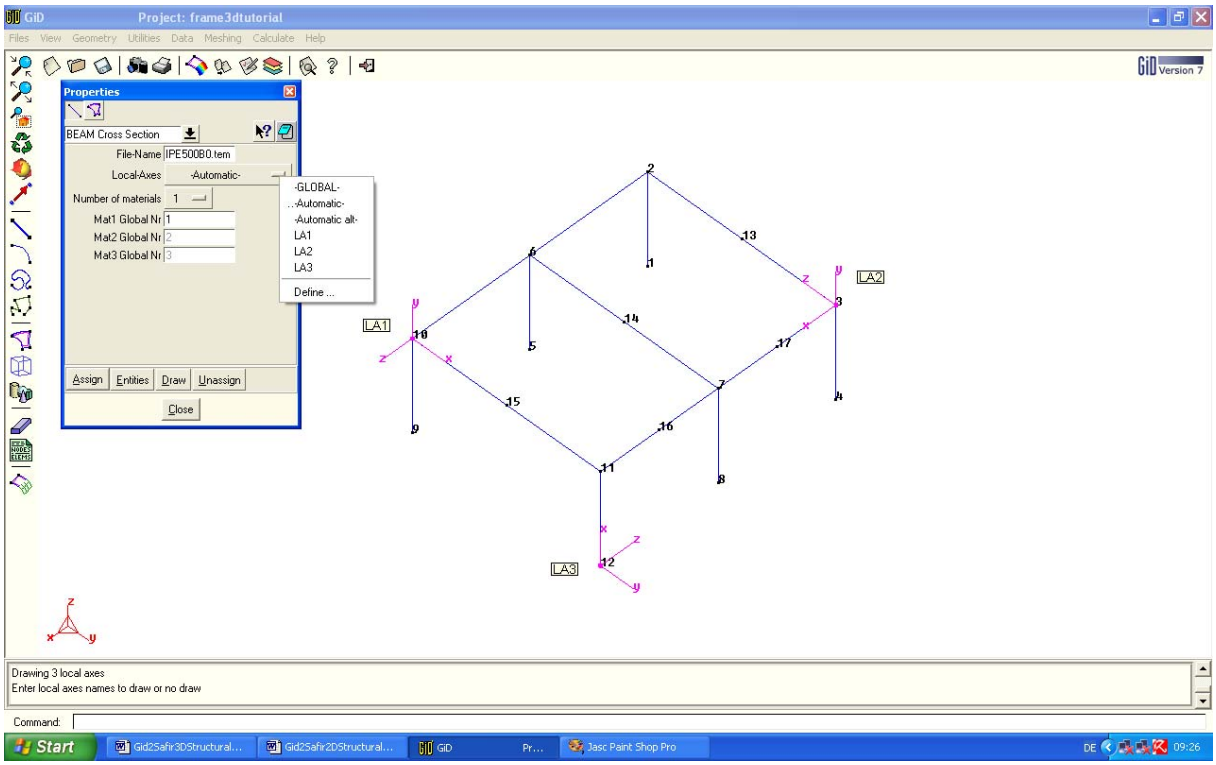
File-Name: *Enter* the name of the temperature file (.TEM file) of the cross-section.

Local Axes: The orientation of the cross-section is controlled by defining a local axes X'Y'Z –system.

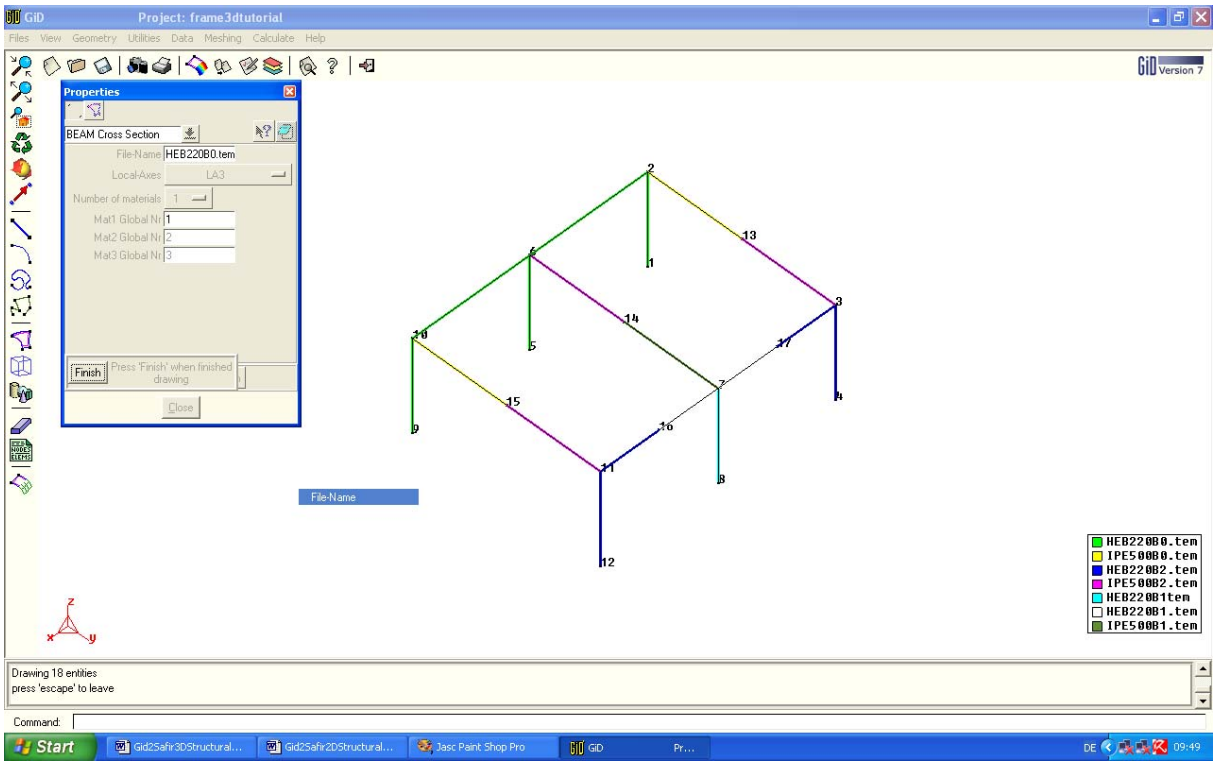


Note: Unlike SAFIR which needs a 4th node to describe the orientation of a cross section on a beam, the GiD-SAFIR interface uses a local X'Y'Z' axes system.
When you start the SAFIR calculation the GiD-SAFIR Interface creates the 4th node in the X'Y' plane. If the center of the local axes is not located on the system line of the beam, the direction vector of the Y'-axis is used together with the starting point of the beam to define the 4th node. However the GiD-SAFIR interface will issue a warning message in the View-output window of the calculation run !!

Define. Create 3 named local axes (LA1,LA2,LA3) either with 3 points or the X' direction and an angle.

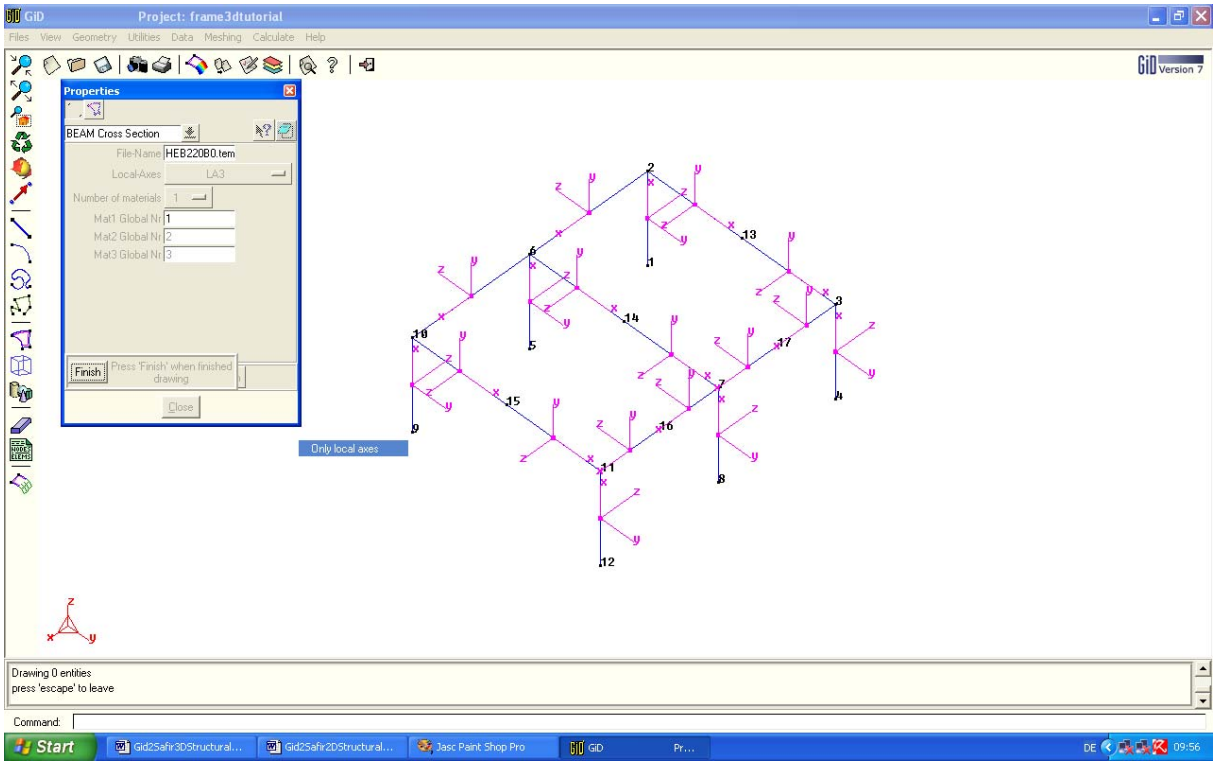


Assign the .tem file names to the system lines:



To display the local axes select in the property dialog box:

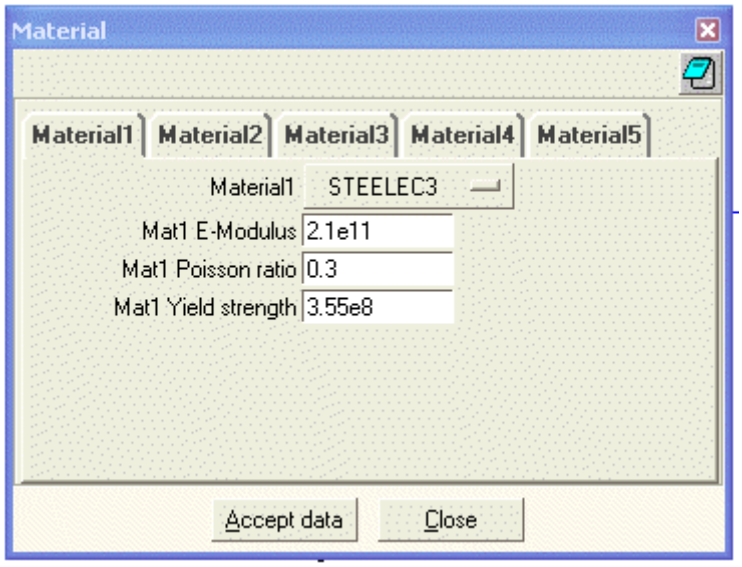
Draw->All conditions->Only local axes



7. Define global materials:

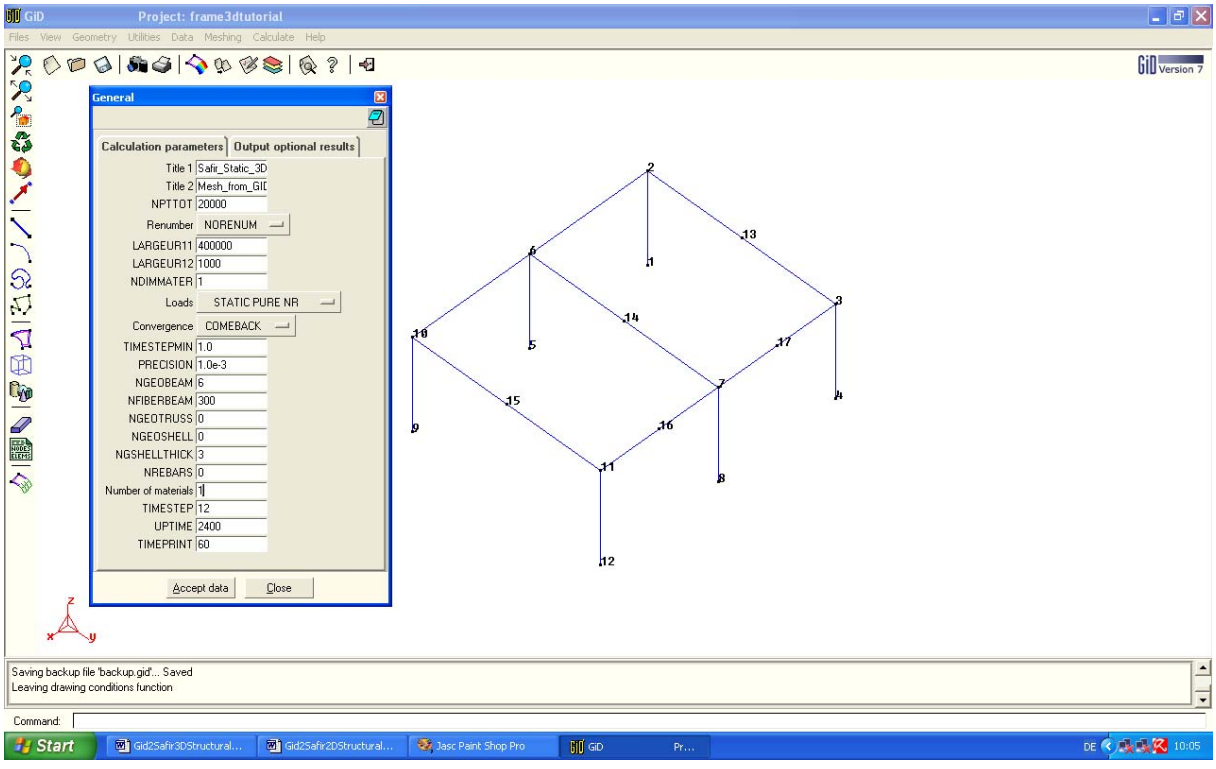
To define material 1 select from the pull down menu:

Data->Problem Data->Material



8. Define general data

Data->Problem Data->General



Enter the following values

NPTTOT = **35000**

NGEOBEAM = **6** (the number of .tem files)

NFIBERBEAM = 300 (max. number of fibers)

Number of materials = 1

TIMSTEP, UPTIME, TIMEPRINT as needed

9. **Generate the mesh:**

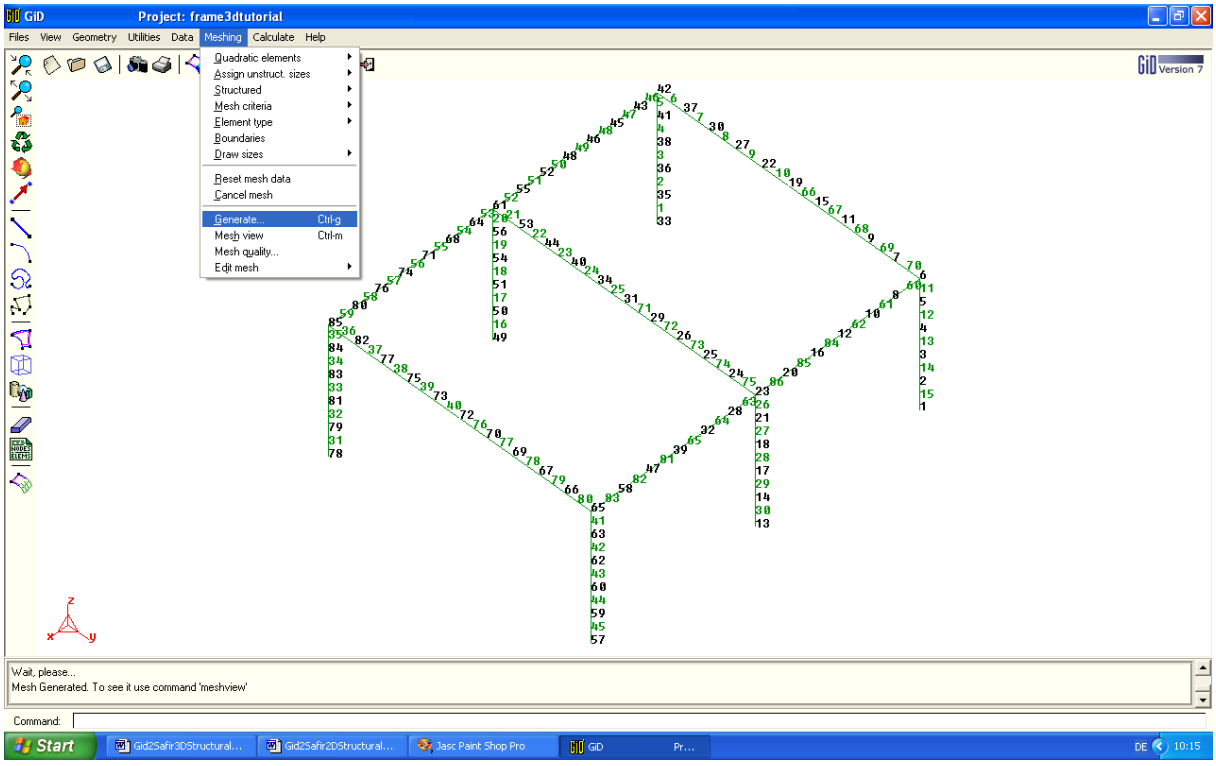
Select from the pull down menu:

Meshing->Generate

Enter the element size to 0.75

To display element and node numbers select from the pull down menu:

View->Label->All



If you are not satisfied with the mesh repeat meshing and change the element size.

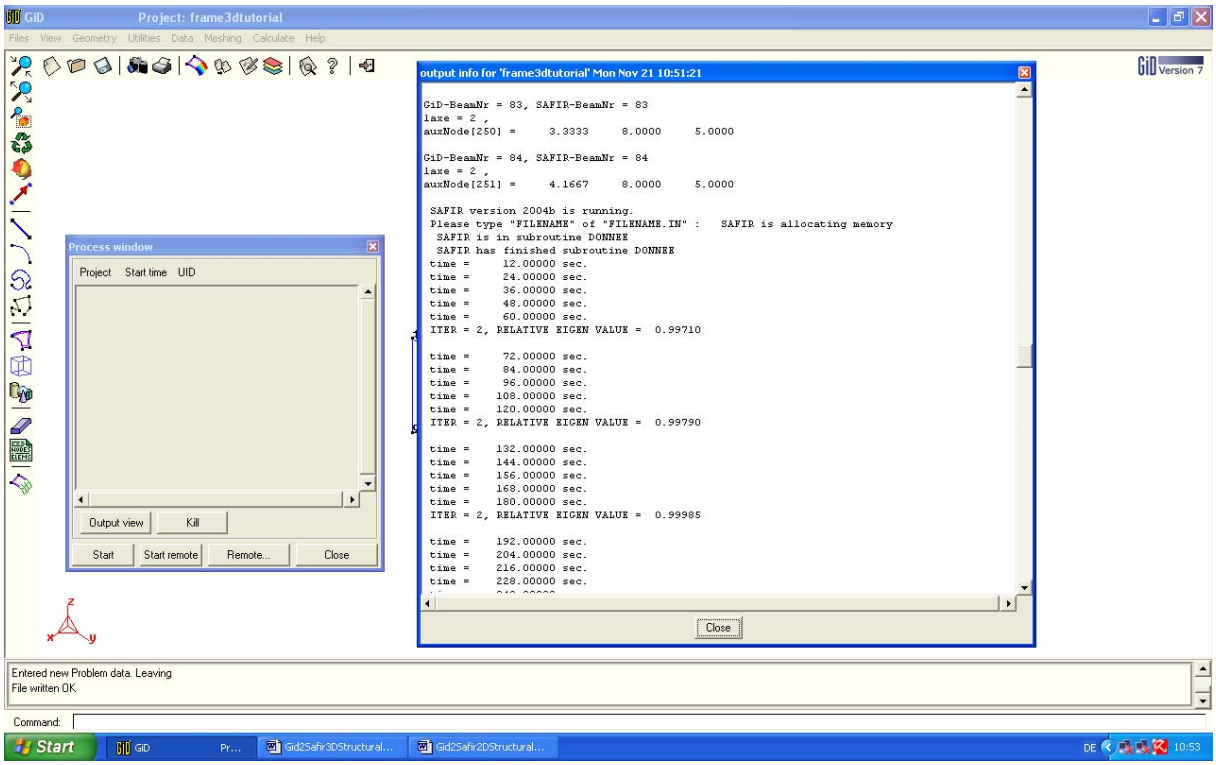
Alternately you can assign each line the number of elements:

Meshing->Structured->Lines

10. Start the calculation.

Calculate->Calculate window

Click the ***Start*** button and then the ***Output view*** button



GiD creates a .IN file in the project directory and starts the calculation.
In the output window you can watch the calculation progress from SAFIR and the GiD interface program which generates GiD postprocessor files from the .OUT file.

Note: If SAFIR found some errors in the .IN file you will also see the error message in this window. This can for example be in the case when you forgot to copy all .TEM files to the project directory, or if you entered a wrong number for NGEOBEM, NFIBERBEAM or Number of materials. Also if one of the numbers NPTTOT, LARGEUR11 and LARGEUR12 is to small, SAFIR prints an error message and suggest the right number.

11. Postprocessing

Postprocessing can be done with **Diamond 2004**. The .OUT file is located in the *project-name.gid* directory . The file name is ***project-name.out***

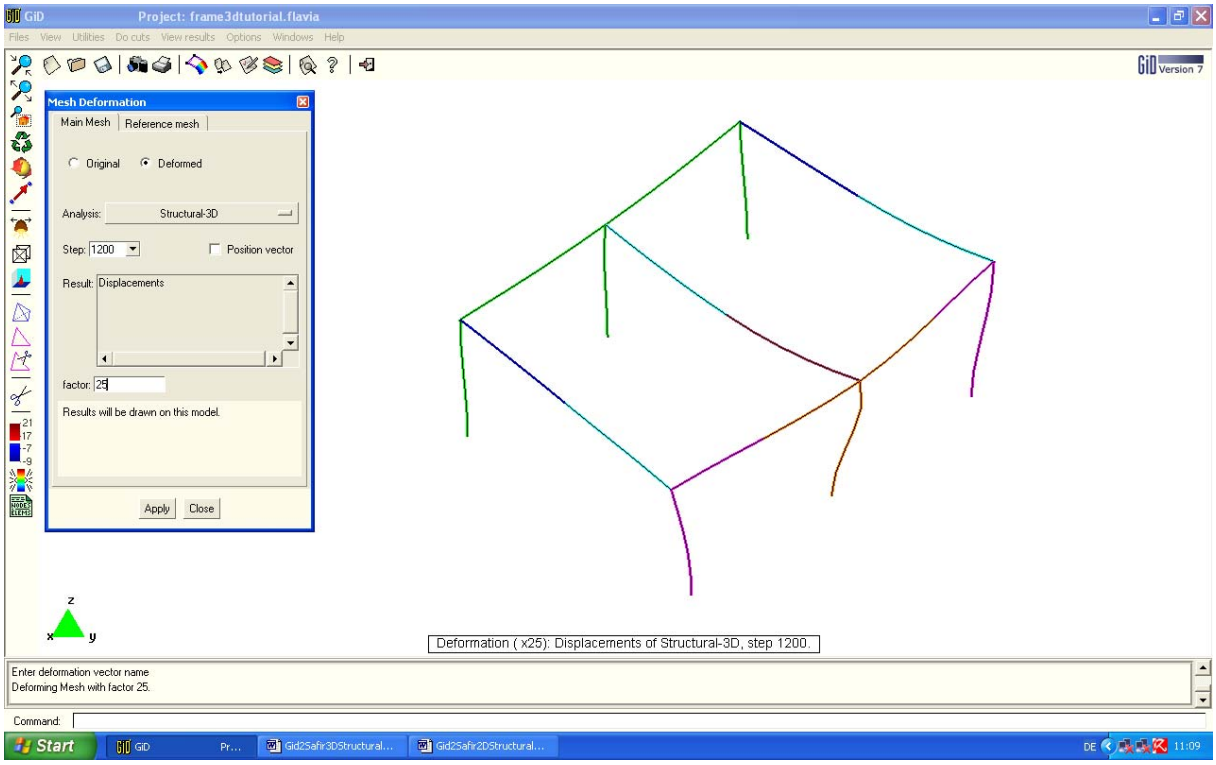
For post processing with GiD select from the pull down menu:

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

To see the deformed mesh select from the pull down menu:

Window-> Deform mesh

Select ***Deformed*** and a time step and press ***Apply***.



For viewing the results select:

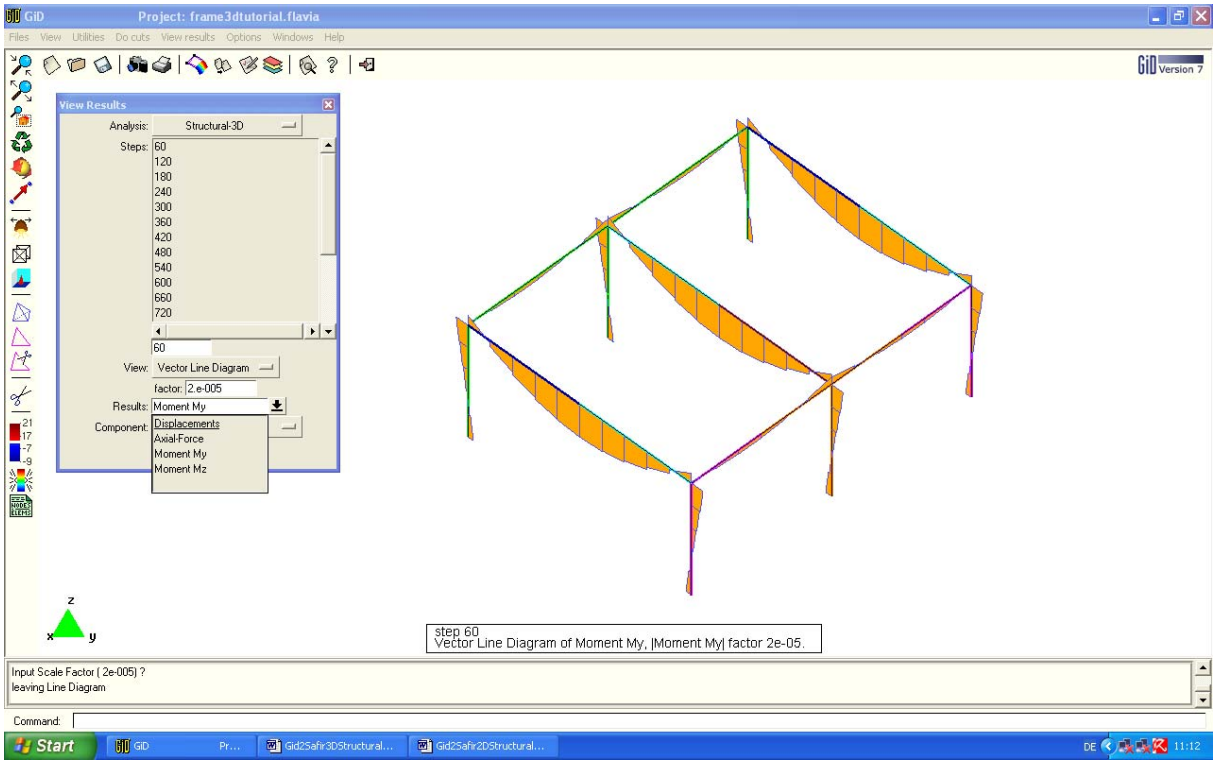
Windows->View results

Select from the ***View*** pull down list ***Vector Line Diagram*** and from Results:

- Displacements
- Axial Force

- Moment My
- Moment Mz

From the Step list select the time step and press ***Apply***.



This files explains the different steps that have to be performed to generate the input files that are necessary for a 3D structural analysis, starting from existing input files that have been previously defined for a 2D analysis

Step 1 : calculate the torsional stiffness in the section

We assume that the starting point is an input file created to calculate the temperatures, say C2T.IN.

1. Copy the existing C2T.IN file and rename it in C2TOR.IN
2. In this new file, change the comment lines (optional)
3. In this new file, replace the commands

```
  TEMPERAT
     TETA       0.9
  TINITIAL     20.0
  MAKE.TEM
by the command
   TORSION
```

4. In this new file, change the name of the file which will be created, for example, replace "c2t.tem" by "c2tor.tor"
5. Delete all the `FRONTIER` and related commands (there is no fire curve applied here). The `SYMMETRY` command now follows immediately the definition of the elements.
6. Delete the `TIME` …. `ENDTIME` commands (there is no need of time steps because the calculation is done only once).
7. Use of symmetries
    a. If the whole section is modelled (no symmetry is used), IT IS ABSOLUTELY REQUIRED to fix the value of the solution to 0 in at least one node of the axis of symmetry. If not, the torsional stiffness will be correctly evaluated, but the warping function at all nodes will be offset by an arbitrary value and this will create amazing results in torsion during the mechanical analysis.
    b. `YSYM` cannot be used in a 3D problem. It means that, if the temperatures where previously calculated using the `YSYM` command because the .TEM file was to be used in a 2D mechanical analysis, it is now necessary to recalculate the temperatures and to replace this `YSYM` command by a `REALSYM` command.
    c. If one part of the section only is discretised and 1 or 2 `REALSYM` commands are used, it is necessary to fix the value of the solution to 0 at all the nodes which are on the axes of symmetry. If there is another axis of symmetry in the solution, but this axis is not used in a `REALSYM` command, the values of the solution must also be fixed to 0 on this axis. This is the example of a square section if ¼ of the section is discretised, then the solution must also be fixed to 0 on the diagonal of the square section.

8. In the MATERIAL command, you must enter the material mechanical properties for each material (the same properties as the one you will use in the mechanical analysis. Don't forget the Poison's ratio).

9. Run SAFIR with this new file. The CPU time is extremely short. The end of the .OUT file looks like this:

```
  G * Ip =   27259615.3846154
According to the principle of virtual works,
     GJ = 0.919990E+08

EIw = 0.856407E+07

 THE STIFFNESS MATRIX HAS BEEN TRIANGULARISED     1 TIMES.
```

GJ is the elastic torsional stiffness of the section at room temperature. In reality, this value will decrease because of cracking even at room temperature (may-be not so much for a column loaded in compression) and will decrease even further during the fire because of the temperature increase and the subsequent decrease in material stiffness. it is not possible to calculate this evolution as a function of time in SAFIR. Only a constant value can be entered. The stiffness can, for instance, be divided by an arbitrary factor of 2. This has to be made in the .TEM file. This means that the simulation is valid provided that torsion is not the may load path. This is not so restrictive in most usual building elements. This would be restrictive in, for example, a cantilever beam submitted to a pure torsion moment.

Step 2. Introduce the torsion properties in the .TEM file.

1) Open the .TEM file which had been created during the temperature analysis and open the new file which has been created during this torsion analysis, "c2tor.tor" in this case. 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 mechanical analysis. If an YSYM command was used when the file was to be used in a 2D calculation, the number of points has now been multiplied by 2, and the information on torsion is now present in the file.

Step 3. Modify the IN file for the mechanical analysis.

1) Copy and rename the .IN file, for example C2M.IN into C3M.IN
2) If a first step, don't modify the `NPTTOT` value. SAFIR will later let you know to which value it has to be increased. The same holds for `LARGEUR11` and `LARGEUR12`.
3) Even if one single member, i.e. a beam or a column, is analysed, the number of nodes has to be increased by 1, here from 21 to 22 (see section 4.2 of the user's manual to see why this node is needed and how you can define the coordinates). In more complex structures, it is possible that more additional nodes are needed.
4) `NDIM` goes from 2 to 3.
5) `NDDLMAX` goes from 3 to 7.
6) The nodes which had 3 DoF have now 7 DoF, see the `FROM` command.
7) Add a `FROM` command which says that the node(s) that you have added, see point 3, has 0 DoF.
8) The number of fibres has to be adapted in the `NFIBERS` command (probably multiply it by 2 if you previously used an `YSYM` command in the thermal calculation).
9) The nodes which were previously present have to be given a third coordinate. In order to have a traditional view with DIAMOND, it is preferable that the new co-ordinate be the Y co-ordinate, see table below. Giving 0 to all these Y coordinates means no initial imperfection in this direction.

2D	=>	3D
X	=>	X
-	=>	Y
Y	=>	Z

10) Give the coordinates to the node(s) that you have added, see point 3
11) Modify the `BLOCK` commands taking into account the new DoF's. they go in the order: X, Y, Z, tetaX, tetaY, tetaZ, warping. Note that 1 line is needed for each blocked node, and 1 blank line at the end of the series.
12) Add the fourth node (the one that you have added) to every beam finite element.
13) Modify the loads in order to reflect the new DoF's. Note that DISTRBEAM have 3 components, pX, pY and pZ. NODELOAD have 6 components, FX, FY, FZ, MX, MY, MZ.

Step 4 : run SAFIR

You may have to adapt some values for the dimensions of the arrays, but SAFIR will let you know. When using Diamond, don't forget to use the TOOL-ViewPoint Tool.

Modification for the mechanical analysis

The name of the .TSH or .TEM files that describe the sections heated by the HASEMI fire is, for each section type, the name of ONE of the relevant .TSH or .TEM file. For example, "`b0017_2.tem`" or "`s0156_3.tsh`".

The information about the torsion properties (for .TEM files) or about the re-bar layers (for the .TSH files) has to be present only in this file, not in the other .TSH or .TEM files of the same section type that describe the temperature at the other points of integration.

As a consequence, all the beam elements of one section type have the same torsion stiffness.

Modification for the thermal analyses

- The command "`MAKE.TEM`" is replaced by the command "`MAKE.TEMHA`" if the temperatures are determined in beam sections, and the command "`MAKE.TSH`" is replaced by the command "`MAKE.TSHHA`" if the temperatures are determined in shell elements shell elements.
- After this command "`MAKE.TXXHA`, the complete name "*filename.in*" of the input file of the mechanical analysis is given (maximum 20 characters) on the next line.

  Note: the input file describing the mechanical analysis must be present when the thermal analysis is run. If the mechanical analysis is 2D, gravity must be pointing downward the Y axis (i.e. Y is vertical upward). If the mechanical analysis is 3D, gravity must be pointing downward the Z axis (i.e. Z is vertical upward).
- The command "`BEAM_TYPE`" ielemtype
  or the command "`SHELL_TYPE`" ielemtype
  is given on the next line, where "ielemtype" is the number in the mechanical input file of the section type (beam or shell) analysed in this thermal analysis.
- The command giving the name of the TEM file is omitted (SAFIR will create the files with the appropriate names such as "b001_1.tem", "b001_2.tem", "b002_1.tem", etc.
- Each surface of the section heated by the HASEMI fire must have a FLUX command and the text "`HASEMI`" where appropriate. For example:

  ```
  FLUX    NO    NO    HASEMI    NO
  ```
  ! Do not use "`HASEMI`" in a "`FRONTIER`" command. There is no need to add a frontier "`F20`" on the same surface (for reemitted energy); it will be automatically added by SAFIR. "`F20`" frontiers can be added on other surfaces, e.g. on the upper unheated side of a slab.

A file called "*hasemi.txt*" must be present in the folder as the other input files. This file describes the local Hasemi fire(s). It's format is:

SERIES 1: comments

Any number of lines, followed by a blank line.

SERIES 2: number of fires

"NFIRE", nfire

where "nfire" is the number of localised Hasemi fires.

The following series have to be written "nfire" times

SERIES 3: position of the fire(s)

"FIRE_POS", posx, posy, [posz]

where posx, posy and posz give the position of the Hasemi fire in the system of coordinates of the mechanical analysis.

SERIES 4: ceiling height

"HEIGHT", hc

where "hc" is the vertical distance from the fire source to the ceiling.

> Note: only this value will influence the severity of the fire on the structure. The vertical position of the fire given in Series 3 is in fact not used in the model

SERIES 5: diameter

"DIAMETER"
time(1), diameter(1)
time(2), diameter(2)
.
.
time(j), diameter(j)
"END_DIAM"

where "j" cannot be bigger than "IDIMTIMSTEP" (at the moment = 20)

SERIES 6: Rate of Heat Release

"RHR"
time(1), RHR(1)
time(2), RHR(2)
.
.
time(i), RHR(i)
"END_RHR"

where "i" cannot be bigger than "IDIMTIMSTEP" (at the moment = 20)

Example of such a file:

```
Description of the local fire by the Hasemi method.

     NFIRE    2           ! Number of localised fires

  FIRE_POS   0.   0.  0.5
    HEIGHT  2.5           ! Distance between the fire and the ceiling
  DIAMETER
             0.        0.  ! Time (s) - Fire source diameter (m)
           300.        3.
           600.        4.
  END_DIAM
       RHR
             0.        0.  ! Time (s) - Rate of heat release (W)
           600.   900000.
   END_RHR

  FIRE_POS  1.5   0.  0.5
    HEIGHT  2.5
  DIAMETER
             0.        0.
           100.        0.
           400.        3.
           600.        4.
  END_DIAM
       RHR
             0.        0.
           100.        0.
           500.   900000.
           600.   900000.
   END_RHR
```

New variables used

`diamhasemi(idimtimstep,2,nfire)`

diamhasemi(i,1,nf): time "`i`" where the diameter is defined for the fire "`nf`"

diamhasemi(i,2,nf): diameter at this time "`i`" for the fire "`nf`"

`hceiling(nfire)`

Distances between the fire source and the ceiling in the Hasemi fire(s)

`ieh`

Index for the loop on the "`ielemhas`" F.E.

`ielemhas`

Number of elements that have the type "`ielemtype"`

`ielemtype`

The number in the mechanical input file of the section type (beam or shell) where the temperatures will be calculated.

`ingb`

Index for the loop on the number of Gauss points

`inumelemhas(nbeammec)`

Vector giving the numbers of the elements (from 1 to ielemhas) that have the type `"ielemtype"`

`lfirsthasemi`

Is TRUE if the temperatures have not yet been calculated in the first Hasemi section

`lhasemi`

Is TRUE if a mechanical calculation is performed under Hasemi fires(s), i.e. if "STATIC_H" or "DYNAMIC_H" have been declared.

`lmatemhasemi`

Is TRUE if a thermal analysis is made in beam sections subjected to Hasemi fire(s).

`lmaketshhasemi`

Is TRUE if a thermal analysis is made in shell thickness subjected to Hasemi fire(s).

`nbeammec`

The number of beam finite elements in the mechanical analysis

`ndimmec`

The number of dimensions in the mechanical analysis

`nfire`

`The` number `of Hasemi fire(s)`

`ngaussbeammec`

The number of gaussian points in the beams of the mechanical analysis

`ngshellthickmec`

The number of gaussian points on the thickness of the shell F.E. in the mechanical analysis

`nnodemec`

The number of nodes in the mechanical analysis

`nodesofFEmec(4,nFEmec)`

Nodes of the relevant F.E. (beam or shell) in the mechanical analysis

`nshellmec`

The number of shell finite elements in the mechanical analysis

`rcoordgausshas(ielemhas,ngaussbeammec,ndimmec)`

rccordgausshas(ih,ng,ndi) is the "nd^{ith}" coordinate, in the mechanical analysis, of the "ng^{th}" point of Gauss in the F.E. that has the Hasemi number "`ih`".

`rcoordgmec(ndimmec,nnodemec)`

Coordinates of the nodes in the mechanical analysis

`rfirepos(nfire,ndimmec)`

Positions of the Hasemi fire(s) in the mechanical analysis

`rhasemiactu(nfire)`

Horizontal distance in the system of coordinates of the mechanical analysis between the Gauss point where the temperatures are calculated and the fire "`nf`".

`rhrhasemi(idimtimestep,2,nfire)`

rhrhasemi(i,1,nf): time "`i`" where the RHR is defined for the fire "`nf`"

rhrhasemi(i,2,nf): RHR at this time "`i`" for the fire "`nf`"