SAFIR[®] training session – level 1

Johns Hopkins University, Baltimore

Example: 3D structural model of a building frame with pinned connections

"3D building frame with concrete columns and steel beams"

T. Gernay & J.M. Franssen



1. General description

This example deals with a 3D frame structure of 8 m by 8 m by 4 m. The columns are reinforced concrete section of 30 cm x 30 cm, with 4 φ 20 steel bars with an axis cover of 30mm, heated on 3 faces. The beams are IPE 300 steel profiles, heated on 3 faces. The columns are fixed, while the beams are pinned. The beams are subjected to uniformly distributed loads of 10 kN/m.

Steel for IPE300: yield strength 355 MPa

Steel for reinforcement bars: yield strength 500 MPa

Concrete: compressive strength 30 MPa; tensile strength 1 MPa

2. Section for IPE300 steel beam

The model created in a previous exercise will be used here.



The .tem file of this model ('*ipe300.tem*') will need to be copy-pasted in the folder with the structural model.

3. Section for concrete column

The model created in a previous exercise will be used here.



The .tem file of this model ('rc30x30.tem') will need to be copy-pasted in the folder with the structural model.

4. Create model for the 3D structure

4.1. Create a new project for structural 3D analysis

From the pull down menu select:

Data->Problem type->SAFIR2016->Safir_Structural_3d



To save the project select (or use icons on the left):

Files->Save $or \bigtriangleup$ or [Ctrl + s]

Note: If Caps lock is active on your keyboard, shortcut do not work

Enter a file name, e.g.: 3DBuild

GiD creates a directory with the name *3DBuild.gid*

GiD creates a number of system files in this directory.

When you start the SAFIR calculation the Safir .*IN* and .*OUT* files will be created in this directory.

4.2. Copy-Paste the section files in the structural analysis directory

GiD has created the directory 3DFrame.gid

The structural input file, which will be created in this directory, will require the information from the section files. Therefore, these sections files need to be located in the same directory.

Copy and paste the files 'ipe300.tem' and 'rc30x30.tem' in the directory 3DFrame.gid

4.3. Create the system geometry (3D structure)

The view is by default in the x-y plane. Here, the plane of the frame will be defined in the x-z plane.

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

View->Rotate->isometric

Or if you want to define a point of view by your own use:

View->Rotate->Trackball

or [F7]

or 🔨

or 🍤

Create the system lines:

Geometry->Create->Straight Line

Enter in the command line (at the bottom of the widows) successively the coordinates of the 5 nodes that define the first frame in the command box. After typing the coordinates of a node, click *[Enter]* to validate.

and the second sec					
Leaving line Enter points	creation. to define	0 new lii line (ESC	nes C to lea	ive)	
Command:	000				
Press [Ente	er]	0.001			
Command:	003				
Press [Ente	er]				
Command:	404				
Press [Ente	er]				
Command:	803	_			
Press [Ente	er]				
1					
Command:	800				
Press [Ente	er]				
		1	.1 11		

Then press [*Esc*] to leave the line creation menu.

You should see this in GiD:



To create the 2 other frames:

Utilities -> Copy or [Ctrl + c]And fill as shown below:

Сору			×	
Enti	ties type:	Lines	•	
Transfo	rmation:	Translation	-	
-First poin	ıt			
Num:	х: 0.0			
	y: 0.0			
٠	z: 0.0			
Second p	oint			
Num:	х: 0.0			
	y: 4.0			
	7. 0.0			
	2. 0.0			
Collap:	se			
Do	extrude:	No	- I	
Create	contacts			
Maintain layers				
Multiple copies: 2				
Select Cancel				

Then click on *select* and select the line and press [Esc]

To see nodes and beams numbers select: *View->Label->All*

Safir_Structural_3D x64	Project: 3DBuild (SAFIR\Safir_Structural_3d			
Files View Geometry Utilities	Data Mesh Calculate Structure About	roblem Type Help		
■ Safe, Structural, 3D x4 Files: View Geometry Utilities ♥ ● ● ●	Project 3DBuild GAFIRSafir Structural 3d Dta Mesh Calculate Structure About Copy Entities type: Lines • Transformation: Translation • First point Num: x [0.0] • x [0.0] Second point Num: x [0.0] • x [0.0] Collapse Do extrude: No • Create contacts Multiple copies: 2 Select Cancel	oblem Type Help		
₩ ** ** **	Select Cancel		12 15	
Selected 4 Lines Geometry has 8 new lines10 new poir	nts. Leaving			
Command:	-			
Zoom: 1x	Nodes: 0, Elements: 0	Render: Normal	Layers: 1	(-3.2288, 8.

Finally, create the last system lines:

Geometry->Create->Straight Line

Press *Ctrl* + *A* to switch the pointer to the mode: '*pick an existing point*'. Click on Node 2, then Node 7, then Node 12. Press *[Esc]*. Repeat with Node 3, then Node 8, then Node 13. Press *[Esc]*. Repeat with Node 4, then Node 9, then Node 14. Press *[Esc]*.

(Note: do not forget to press Ctrl + A, otherwise the line would not be connected to the nodes although it might look like it goes through the nodes).

or 🔨



4.4. Define constraints for the supports

From the pull down menu select

Data->Constraints Select Point Constraints

Tick all the boxes for a fully fixed condition. Assign these constraints to *POINT 1*, *POINT 5*, *POINT 6*, *POINT 10*, *POINT 11*, *POINT 15*, and press [*Esc*].



In the dial box, with *Draw->Colors* you can display the constraints. Press *Finish* or *[Esc]* to leave this view mode.

4.5. Assign the loads

From the pull down menu select *Data->Loads*

Assign a distributed load of -10 000 N/m in Z direction on the beams, using the load function *FLOAD*. Press *Finish* or *[Esc]* to validate.

Loads		×
• 🔨 🕤		
Beam-Load	~	k? 🥏
X Press	ure 0.0	
Y Press	ure 0.0	
Z Press	ure -10000.0	
LOAD FUNCTI	ON FLOAD	-
<u>A</u> ssign	<u>E</u> ntities <u>D</u> raw	<u>U</u> nassign
	<u>C</u> lose	



4.6. Create Local Axes

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



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 Viewoutput window of the calculation run.

The objective is to create 5 local axes (LAXU, LAXD, LAY, LAZU, LAZD) as shown in the figure below.



From the pull down menu select:

Data->Local Axes->Define

Enter the name of the new local axe *LAZD* Select *3 points XZ*

Dialog window		×
Choose	se definition mo	de or delete
<u>3</u> Points XZ	Xand Angle	<u>D</u> elete
	<u>C</u> ancel	

Select **Point 2** as the local axis center. Note: Press "CLTR + A" to allow the selection of an existing point with the mouse.

Select *Point 1* as the point in positive x axis.

Select *Point* 7 as the third node points to the positive direction along the z axis.

Press Enter

You should see this in GiD. This local axis *LAZD* will be used for Columns 1, 5 and 9.



Repeat the operations to create the local axe LAZU

Select *Point 5* as the local axis center.
Select *Point 4* as the point in positive x axis.
Select *Point 10* as the point in positive z axis.
Press *Enter*This local axis *LAZU* will be used for Columns 4, 8, 12.

Repeat the operations to create the local axe LAY

Select *Point 7* as the local axis center. Select *Point 12* as the point in positive x axis. Select *Point 9* as the point in positive z axis. Press *Enter*

This local axis LAY will be used for Beams 13, 14, 15, 16, 17, 18.



From the pull down menu select: *Data->Local Axes->Define* Enter the name of the new local axe *LAXU* Select *X* and Angle

Select *Point 2* as the local axis center. *Note: Press "CLTR* + A" to allow the selection of an existing point with the mouse. Select *Point 3* as the point in positive x axis. Enter *90* in the command box as the angle. Press *Enter* From the pull down menu select: *Data->Local Axes->Define* Enter the name of the new local axe *LAXD* Select *X* and Angle

Select *Point 3* as the local axis center. Select *Point 4* as the point in positive x axis. Enter *90* in the command box as the angle. Press *Enter*

To draw local axes select:

Data->Local Axes->Draw all



4.7. Define the global materials

From the pull down menu select Data->Material

There are 3 materials in the model:

- The steel for the profile IPE300: STEELEC3EN
- The steel for the reinforcement of the concrete columns: STEELEC2EN
- The concrete for the columns: SILCON_ETC

In the section IPE300: MAT 1 = STEELEC3EN In the section RC30x30: MAT 1 = STEELEC2EN, MAT 2 = SILCON_ETC



In the structural model, we decide that the order is:

MAT 1 = STEELEC3EN MAT 2 = STEELEC2EN MAT 3 = SILCON_ETC

Material ×
Temperatures General Material1 Material2 Material3
Number of materials 3 🔻
<u>Accept</u>

Material	×
	2
Temperatures General Material1 Material2 Material3	
Material1 STEELEC3EN 👻	
Mat1 E-Modulus 2.1e11	
Mat1 Poisson ratio 0.3	
Mat1 Yield strength 3.55e8	
Mat1 Max Temperature 1200.	
Mail Rate Declease field strength 0.	
Accept Close	
Material	×
	9
Temperatures General Material1 Material2 Material3	
	- 1
Material2 STEELEC2EN 👻	
Mat2 E-Modulus 2.1e11	
Mat2 Vield strength 5 0e8	
Mat2 Max Temperature 1200.	
Mat2 Rate Decrease Yield Strength 0.	
Mat2 Process Fabrication HOTROLLED 💌	
Mat2 Class Ductility CLASS B 👻	
Accept Close	
	_
Material	x
	2
Temperatures General Material1 Material2 Material3	
Material3 SILCON ETC 👻	
Mat3 Poisson ratio 0.2	
Mat3 Compressive strength 3.0e7	
Mat3 Tension strength 1.0e6	

Note: a description of the material *SILCON_ETC* is given in the following paper:

Gernay, T., & Franssen, J. M. (2012). A formulation of the Eurocode 2 concrete model at elevated temperature that includes an explicit term for transient creep. *Fire Safety Journal*, *51*, 1-9.

4.8. Define the properties (i.e. assign temperature files)

From the pull down menu select Data->Properties

The objective is to assign the .tem file named ipe300 and rc30x30 to the model lines.



In the dial box of *Data->Properties*, change the File-Name: *safir.tem* to the temperature file (.TEM file) of the cross-section, in this case *ipe300.tem*.

Change Local-Axes from -Automatic- to LAXU

Keep the number of materials to 1. The Mat1 Global Nr is 1 (it is the STEELEC3EN). Assign the *ipe300.tem* section to the beams 2, 6 and 10.



Then, keep the file name to *ipe300.tem* but change *Local-Axes* to *LAXD*. Keep 1 material with the Mat1 Global Nr equal to 1. Assign to the beams 3, 7 and 11.



Then, keep the file name to *ipe300.tem* but change *Local-Axes* to *LAY*. Keep 1 material with the Mat1 Global Nr equal to 1.

Assign to the beams 13, 14, 15, 16, 17, 18.



Then, change the file name to *rc30x30.tem*. Change *Local-Axes* to *LAZU*. Specify 2 materials. The Mat1 Global Nr is 2 (it is STEELEC2EN). The Mat2 Global Nr is 3 (SILCON_ETC). Assign to columns 4, 8, 12.



Finally, keep the file name to *rc30x30.tem* but change *Local-Axes* to *LAZD*. Keep 2 materials with Mat1 Global Nr is 2 and Mat2 Global Nr is 3. Assign to columns 1, 5 and 9.



You can draw the local axes of the beams to check the model. Select *Draw -> All Conditions -> Include Local Axes*

4.9. Assign the RELAX conditions

From the pull down menu select *Data->Relaxations*

The objective is to assign pinned conditions at the end of all the beams in the model (i.e. no transfer of strong axis bending moment between the beams and the columns).



After sele	ecting I	Data->	Rela.	xations,	this	window	appears:
	··· 6						

Relaxations				×
$\mathbf{}$				
Relax Beam DOF			~	k? 🕗
Node1 Node2				
N1 DOF1 -1]			
N1 DOF2 -1				
N1 DOF3 -1]	ł		
N1 DOF4 -1	Node 1	Node 2		
N1 DOF5 -1				
N1 DOF6 -1	View > Nor	mals > Lines		
N1 DOF7 -1]			
<u>A</u> ssign	<u>Entities</u>	<u>D</u> raw	<u>U</u> nassign	
		<u>C</u> lose		

For each *Beam*, the user can apply *Relaxations* conditions to either extremity (*Node1* or *Node2*), according to any of the 7 degree of freedom at this end node.

The degrees of freedom are to be considered in *Local Axes* coordinates!

To identify *Node1* from *Node2*, arrows are drawn on the beam elements of the structure, as indicated on the picture in the window here above.

Assign a value of "0" to N1 DOF6 and assign to Beams 2, 6 and 10. Press *Finish* or *[Esc]* to validate.



Assign a value of "0" to N2 DOF6 and assign to Beams 3, 7 and 11. Press *Finish* or *[Esc]* to validate.



Assign a value of "0" to both N1 and N2 DOF6. Assign to Beams 13, 14, 15, 16, 17, 18. Press *Finish* or *[Esc]* to validate.



You can draw the *Relaxations* to check the model. Select *Draw -> Colors*



4.10. Assign the mass

To define the mass for dynamic calculation, select from the pull down menu:

Data-> Mass

Select *Mass on Beam* and put 100 kg/m as Distributed-Beam-Mass and 2 as Rotational-Inertia. Assign to all the beam elements.

Mass for Dynamic	Calculation		×	
• 🔨 🟹				
Mass on Beam			× 💦 🕗	
Distributed-Beam-	Mass 100.0			
Rotational-Inertia 2.0				
Assign	Entities	<u>D</u> raw	<u>U</u> nassign	
Close				

4.11. Define the general problem data

Select from the pull down menu: Data->Problem Data And fill as shown below

General ×	General
	R la lui a constant actional resulta
Calculation parameters Output optional results Title 1 Safir_Static_3[Title 2 Mesh_from_G SOLVER PARDISO NCORES 1 Loads DYNAMIC PURE NR Convergence COMEBACK TIMESTEPMIN 0.002 Consider max displacement PRECISION 1.0e-3 NGEOBEAM 2 NG 2 NFIBERBEAM 491 NGEOTRUSS 0 NGEOSHELL 0 NGSHELLTHICK 3 NREBARS 0 TIMESTEP 2 UPTIME 3600 TIMESTEPMAX 36. TIMEPRINT 2 General modified	Calculation parameters Output optional results PRINTTMPRT Print temperatures in the fibers PRINTVELAC Print velocity and acceleration PRINTREACT Print reactions PRINTMN Print internal forces of beams PRNEIBEAM Print stiffness in beams PRINTSHELL Print stresses in shell elements PRNNXSHELL Print membrane forces in shell elemen PRNMXSHELL Print bending moments in shell elemets PRNEASHELL Print Bending stiffness in shell elements PRNEISHELL Print strains and stresses in shell elements PRNSTRAIN Print strains and stresses in shell elements
<u>A</u> ccept <u>C</u> lose	Accept Close

4.12. Define the mesh

Select Mesh -> Structured -> Lines -> Assign number of cells



Assign 10 elements. Select all the lines. Press [Esc] to validate.

Enter value window			
0	Enter number of cells to assign to lines		
	10		
[Assign	Close	

Select Generate Mesh and then View Mesh



4.13. Start the calculation

From the pull down menu select: Calculate->Calculate window Click the Start button

You can follow the progress of the calculation by selecting Calculate->View process info

4.14. Check the results

Open the .XML file in Diamond to check the model. Plot the support conditions, applied loads, deflected shape, membrane forces, etc.



