# **Rigid Jointed Frame Analysis**

# Title

Statically determinate structural analysis for a Rigid Jointed Frame

## Description

A pitched-roof portal frame is pinned at supports A and H and members CD and DEF are pinned at the ridge as shown in Figure below. For the loading indicated:

i) Determine the support reactions and

ii) Sketch the axial load, shear force and bending moment diagrams.



Structural geometry and analysis model

## **Finite Element Modelling:**

Analysis Type: 2-D static analysis (X-Z plane)
 Step 1: Go to File>New Project and then go to File>Save to save the project with any name



Step 2: Go to Model>Structure Type to set the analysis mode to 2D (X-Z plane)



• Unit System: kN,m Step 3: Go to Tools>Unit System and change the units to kN and m.



#### • Geometry generation:

Step 4: Go to Model>Nodes>Create Nodes and type in (0,0,0) in the Coordinates box. Press Apply. Switch to the Front view by clicking on  $\cong$  as shown below. Switch on node number from the option  $\mathbb{A}^n$ . Node number 1 will be point A.



*Step 5:* Go to **Model>Element>Extrude Elements**. Select type as Node->Line. Use select single button  $\stackrel{\circ}{!}$  to highlight the node number 1 by clicking on it. Type in (0,0,2.5) in the Equal Distances dx,dy,dz box and enter number of times=2 and click Apply to generate the column.

Gen 2013 - [C:\Users\Owner\Desktop\Work	Backup-new\Short Tutorials for students\Gen\Tutorial 3- Rigid Frame Analysis\Tutorial	03 *] - [Model View]	
Eile Edit View Model Load Anal	ysis <u>R</u> esults <u>D</u> esign M <u>o</u> de <u>Q</u> uery <u>T</u> ools <u>W</u> indow <u>H</u> elp		
Freque   Grid/Snap  UCS/GCS   View C	Activation   Wizard   Node Element   Property   BC/Mass   Stage	e   Load   Building   Me	sh   Settlement   Result   Query
* / * = * * * * * *	●11 気道ならき増业乳園×翌送す	\$ \$ \$	
D≊∎XI2+2+8 <b>D</b>	₺ ፝ቘ ፟⊈ዄ፼ጲ፼፼፼፼፼፼ዾጜጜፙ፼፞ኇ1	- 2	- 🛛 🚔 🔒 🗎 🖓 📸 🟠
Tree Menu 📮	× 4 Con Model View		2 241-00
Node Element Boundary   Mass   Load			
Extrude Elements			
ElStart Number			
Node Number : 4			
Element Number : 13			
Extrude Type			
Source Remove Move			
T Reverse I-J			
Element Attribute Element Type: Beam			
Material :			
1 <u> </u>			
1			
Beta Angle : 0 JDeg]			
Generation Type			
Translate C Rotate C Project			
Equal Distance			
C Unequal Distance			
dx,dy,dz: 0, 0, 2.5 m	$\Delta c$		
Number of Times : 2 🕂	N A		
Merging Tolerance	Message Window		
<u>Apply</u> <u>Close</u>			

*Step 6:* Similar to Step 5, select this time node number 3 and enter dx,dy,dz as (4,0,2) m, number of times=1 and click Apply to generate the pitched roof.



4

*Step 7:* Select node 4 and extrude using dx, dy, dz = (3, 0, -1) and number of times = 2 and Apply to generate the other half of the pitched roof.



*Step 8:* Finally Select node 6 and extrude using dx, dy, dz = (0,0,-3) and number of times=2 and click Apply and Close to generate the other column.



• *Material: M*odulus of elasticity,  $E = 2.0 \times 10^8 \text{ kN/m}^2$ 

*Step 9:* Go to **Properties>Material Properties>Add**. Select User defined in the Type of Design and Enter  $E=2.0 \times 10^8 \text{ kN/m}^2$ . Enter a name for the material and click OK and Close.



• Section Property: BS Column section UC 305x305x158

*Step 10:* Go to **Properties>Section Properties>Add**. Select DB/User tab and select I section type. Select BS4-93 from section DB and select the section UC 305x305x158 from the dropdown. Click OK and Close.

Snapl	Structure Type	Woard Node Element Property   SC/Mas	Stage   Load   Building   Mesh   Settlement   Result	Query	Section Data	6
8 2	Structure Weard	1000年年今日日本省田区1	1 A 4 > 7		DB/Liser	
3	User Coordinate System	· 2 C & F & O O O O O O A & A O S	· 🎎 🛛 · 🛱 🔓 (	1 2 2 2 2	la se la constante de la constante	
	Grids	Iodel View			Section ID   1	I I-Section •
iroup	Hodes	•	Properties	12	Name UC 305x305x158	C User (# 08 854-93 +
	Elements	• /	Material Section Thiduness		-	
8	Properties	• E Hateral				Sect. Name UC 305x305x158
m:7	Boundaries	Time Dependent Material(Creep/Shrinkage) <u>Function</u>	1 UC 305x305x158 D8 I	Pt A00	241	T TOT OF Sector
	Masses	<ul> <li>Time Dependent Material(Creep/Shrinkage)</li> </ul>		Hodry	1 1 12	
: 1 Naterial	Building	<ul> <li>Time Dependent Material(Comp. Strength)</li> </ul>		Delete		- Get. Data from Sindle Ande
0	Mesh	Oracos Element Dependent Material Dependent		Copy	292	C0 None (ASC 10(US) +
rown	Domajn	Plasty Marwal		Import		sectores 1
12	Named Plane	E Denke Herkingsto		Renumber		a 0.3271 a
1	Dimension	L Section		1 2	1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	81 0.3112 m
10	Toggle Worktree	Tapared Section Group		2		ter 0.0158 m
	Grgup	, a month of the second graph.	-	Q the general sector of the se	0 -	#1 0.025 m
	Gheck Structure Data	<ul> <li>Inconstant</li> </ul>				82 0 m
		Itm was strmess scale Factor		4		82 0 m
		H Inelastic Hinge Properties				P1 0.0152 m
		H Inelastic Hinges	H:0.3271 B1:0.3112 bv:0.0158 f1:0.025 B	2:0 172		12 J- m
		Group Lamping : Element Mass & Schness Proportiona	her			
		Instantic Material Researcher				Ti conto thought mater
		Eher Dyson of Section				<ul> <li>Consider pricar perioritation.</li> </ul>
					Offset : Center Outer	
		T Section Table	NLAPLC		Change Offset	
		Thriness Table	HARLES HARLES			
		Change Element Dependent Material Property Table.			Show Calculation Results	OK Cancel Apply
		Wall Stiffness Scale Factor Table			-	
		H Inelastic Hinge Table				
	(D)					

You can see the shape of the section in the model generated. Go to the Works Tree to check the information for your model.



• *Boundary Condition:* Pinned Supports at A and H and a pinned connection between members CD and DEF at point D.

*Step 11:* Use select single  $\overset{\circ}{\square}$  to select or highlight nodes 1 and 8 as shown in figure below. Go to **Model>Boundaries>Supports** and check on Dx and Dz and Apply.



7

This becomes the Pinned support at A and H. DX and DZ provide horizontal and vertical restraint simultaneously. No rotational restraint has been provided.

*Step 12:* Select and display element numbers by clicking on  $\stackrel{n}{\rightarrow}$ . Again use Select Single  $\stackrel{r}{\searrow}$ , this time to highlight or select element 3. Go to **Model>Boundaries>Beam End Release.** Select Fixed-Pinned and Apply and Close.



This applies a moment release or a pinned connection at point D.

### • Load Case:

#### Concentrated Loads:

-Vertically downward loads of 15 kN, 25 kN, 35 kN and 20 kN at points C, D, E and F respectively applied in (-Z) direction.

-Horizontal Loads of 12kN, 8kN, 5kN and 8kN at points B, C, F and G respectively applied in (+X) direction.

*Step 13*: Go to **Load>Static Load Cases** and define static load case 'P'. Select Load type as User defined for both of them. Click Close after adding the load case.

🕼 Gen 2013 - [C:\Use	ers\Owr	ner\Desl	ktop\Work Back	up-new\Short Tutoria	ls for students\G	en\Tutorial 3- Rig	id Frame An	alysis\Tu	torial 03] -	[Model View]
Eile Edit Vie	w <u>M</u> o	odel L	oad Analysis	Results Design	Mode Query	<u>Tools W</u> indow	<u>H</u> elp			
Freque Grid/Sna	p UCS	/GCS 🗖	J Static Load	Cases		F9 Element	Property   B	C/Mass	Stage   Lo	oad   Building   M
ነ 🖓 🎢 🗮 🎓	Static L	oad Ca	ses	Cases Using Load Co	mbinations		ala (14) 🔽		兴 🏟 🝃	• 🔧
🗄 🗅 🚅 🖬 🖂 🗌	_									- 🏒
Tree Menu	Nan	ne :	P			ſ	Add			
Menu   Tables   Gro	Тур	e :	User Define	d Load (USER)	•	-	Modify		A	
Works	Des	cription :	:				Delete	i		
• Nodes : 8	_									4
Elements :		No	Name	Туре		Description	on	<u></u>		
- HE Properties	₽	1	P	User Defined Load	(USER)					
T Material :     T 1 Ma	<u> </u>									
E Section :										
□ ⊥ Boundaries										
Supports :								E		
- lo-d Beam End										
<b>⊳⊲</b> Type										
								-		
	•			m				P.		

Step 13: Go to Load>Nodal Loads and select load case P. Select the node 2 using 2 and enter FX=12 kN and press Apply. Select Node 3 and enter FX=8 kN and FZ = -15 kN and Apply. Select Node 4 and enter FZ=-25 kN and Apply. Select Node 5 and enter FZ=-35 kN and Apply. Select Node 6 and Apply FX=5 kN and FZ= -20 kN. Select node 7 and Apply FX=8 kN.



You can display the values from the Works Tree by Right clicking on Nodal Loads and click Display.

*Uniform Loads:* A uniformly distributed load, UDL of 12 kN/m is applied vertically downward on the member CD.

*Step 14:* Go to **Loads>Element Beam Loads** and select load case P. Select the element number 3 and enter w=-12 kN/m in the Global Z direction and select Projection as Yes and press Apply and Close.



From the display, the UDL looks as if it has been applied over the length of the element 3 but internally it has been applied only on the projected length i.e. X direction length of that element since the Projection 'Yes' was selected while applying the load.

The analysis results for forces and moments in the frame are generated in the finite element software with respect to each element's local axis or centroidal axis. So before running the analysis, we must make sure that the local axes of all elements are aligned properly. Turn on the isometric view using  $\square$ . To display the local axis, go to **View>Display** and select Element tab and check on Local axis and click OK. You will find that the members A to C or the element numbers 1 and 2 have their local y axis inclined in the opposite direction to the rest of the model's y local axis. This may result in opposite signs for bending moments, My.

Check off node numbers and element numbers . Also display only wireframe by clicking on for clarity.



Now to align the local axis properly, go to **Model>Elements>Change Element Parameters** and go to Element Local Axis and select the 2 elements 1 and 2 that need to be changed and enter the change in beta angle (local axis angle)=180 degrees in the Change Mode.

Gen 2013 - [C:\Users\Owner\Desktop\Wo	rk Backup-new∖Short	Tutorials for stude	nts\Gen\Tutorial 3- Ri	gid Frame Analysis\Tuto	rial 03 *] - [Model View]	
Elle Edit View Model Load A	nalysis <u>R</u> esults <u>D</u> e	esign M <u>o</u> de <u>Q</u> ue	ery <u>T</u> ools <u>W</u> indov	v <u>H</u> elp		
Freque Grid/Snap  UCS/GCS   View C.	Activation    Wi	zard   Node Elem	ent Property   BC/I	lass   Stage   Load   Bu	uilding   Mesh   Settlement	Result   Query
1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	📮 🔊 🖧 🎁 🚺	建造谷会	ヨーロンシン	記法会かな		
D 📽 🖬 🗙   오 · 오 - ) 🖨 🐧	品准先 立		🕲 🖻 💅 🖳 G	×代 🛞 🖇 1to3	• 👲 12	• i 🕰 🔒 i 🙈 i
Tree Menu 📮 🗙	4 Model Vi	ew				
Node Elem   Boun   Mass   Load			2	z		
Change Element Parameters			/	- La		
						Ŧ
Node Number : 9						- has
Element Number : 8		Z v				
		J.				
Parameter Type						
C Material ID						
C Section ID						
C Thickness ID						4
C Wall ID						
Element Local Axis						
C Deveen Stream thread						
C Align Element Local						
· Augricialitic cocai	2-					
Mode						
C Assign						
C Ref. Vector						
Beta Angle : 🚺 💌 [Deg]						
C Change						×Z
Ande Increment : 180 [Dec]						
Hige Indeniere Toodi						
Apply Gose						
	-					
						ļ
	No and a second					

You can now undisplay the local axis by going back to View>Display and checking off

Local Axis in the Element tab and click Apply. Switch to front view by clicking on from right.

• Analysis: Step 15: Go to Analysis>Perform Analysis

# Results

• Reaction Forces:

*Step 16:* Click on **Results>Reactions>Reaction Forces/Moments** and select the load case P. Select FXYZ. Check on Values and click on the box in next to Values to change number of decimal points to 2 and click OK to see reactions graphically.



• Beam forces/moment diagrams:

*Step 17:* Click on **Results>Forces>Beam Diagrams** and select the load Case P. Select Fx and Click OK to display the axial force diagram in the members. Note that the '+' sign indicates Tension and '-' indicates Compression.



Step 18: Change to Fz to view the shear force diagram and click Apply.







# Hand Calculations:



(1) To determine support reactions

Taking vertical force sum:  $\Sigma V = \mathbf{0}$ 

$$V_A - 15.0 - (12.0 \times 4.0) - 25.0 - 35.0 - 20.0 + V_H = 0$$
  
 $V_A + V_H = 143$  Equation (1)

Taking horizontal force sum:  $\Sigma H = 0$ 

$$+H_A + 12 + 8.0 + 5.0 + H_H = 0$$
  
 $H_A + H_H = -25.0$  Equation (2)

Taking sum of moments about A:

 $\Sigma M_A = 0$ 

 $\begin{array}{l} (12\times2.5)+(8.0\times5.0)+(12.0\times4.0)(2.0)+(25.0\times4.0)+(35.0\times7.0)\\ +(20.0\times10.0)+(5.0\times5.0)+(8.0\times2.0)-(H_{H}\times1.0)-(V_{H}\times10.0)=0\\ H_{H}+10.0V_{H}=752 \quad Equation \ (3)\\ \end{array}$  Taking sum of moments about the Pin D:  $\Sigma M_{pin}=0 \end{array}$ 

 $+(35.0 \times 3.0) + (20.0 \times 6.0) - (5.0 \times 2.0) - (8.0 \times 5.0) - (H_H \times 8.0) - (V_H \times 6.0) = 0$ 8.0H<sub>H</sub> + 6.0V<sub>H</sub> = 175 Equation (4) From Equations (1),(2),(3) and (4) we obtain:

$$H_A = +4.30 \ kN$$
  
 $V_A = +64.07 \ kN$   
 $H_H = -37.30 \ kN$   
 $V_H = +78.93 \ kN$ 



(2) To determine member forces:

Assuming anticlockwise bending moments to be positive:

$$\begin{split} M_A &= 0 \qquad (Pinned Support - No \ moments) \\ M_B &= -(4.30 \times 2.5) = -10.75 \ kNm \\ M_C &= -(4.30 \times 5.0) - (12.0 \times 2.5) = -51.50 \ kNm \\ M_D &= 0 \qquad (Pin - No \ Moments) \\ M_E &= -(20.0 \times 3.0) + (5.0 \times 1.0) + (8.0 \times 4.0) - (37.3 \times 7.0) + (78.93 \times 3.0) \\ &= -47.31 \ kNm \\ M_F &= +(8.0 \times 3.0) - (37.30 \times 6.0) = -199.80 \ kNm \\ M_G &= -(37.30 \times 3.0) = -111.90 \ kNm \end{split}$$



The values of the end-forces  $F_1$  to  $F_{12} \, \text{can}$  be determined by considering the equilibrium of each member and joint in turn.

Consider Member ABC:  $(\Sigma V = 0)$ 

$-F_1 + 64.07 = 0$
$\therefore F_1 = 64.07 \text{ kN}$
$(\Sigma H = 0$
$+4.30 + 12.0 - F_2 = 0$
$\therefore F_2 = 16.30 \text{ kN}$
Consider Joint C:
$(\Sigma V = 0)$ There is an applied vertical load at joint C=15kN
$-F_1 + F_3 = -15.0$
$\therefore F_3 = 49.07 \text{ kN}$
$(\Sigma H = 0)$ There is an applied horizontal load at joint C=8 kN
$-F_2 + F_4 = +8.0$
$\therefore F_4 = 24.30 \ kN$
Consider Member CD:
$(\Sigma V = 0)$
$+49.07 - (12.0 \times 4.0) + F_5 = 0$
$\therefore F_5 = -1.07 \text{ kN}$
$(\Sigma H = 0)$
$-F_2 + F_4 = +8.0$
$\therefore F_6 = 24.30 \ kN$
Consider Member FGH:
$(\Sigma V = 0)$
$+78.93 - F_{11} = 0$
:. $F_{11} = 78.93 \text{ kN}$
18

 $(\Sigma H = 0)$  $-37.30 + 8.0 + F_{12} = 0$  $\therefore F_{12} = 29.30 \ kN$ 

Consider joint F:

 $(\Sigma V = 0)$  There is an applied vertical load at joint F=20 kN  $F_{11} + F_9 = -20.0$  $\therefore F_9 = 58.93 \text{ kN}$  $(\Sigma H = 0)$  There is an applied horizontal load at joint F=5 kN  $+F_{12} - F_{10} = +5.0$  $\therefore F_{10} = 24.30 \ kN$ Consider member DF:

 $(\Sigma V = 0)$ 

$$+58.93 - 35.0 + F_7 = 0$$
  
 $\therefore F_7 = 23.93 \text{ kN}$ 

 $(\Sigma H = 0)$ 

$$-24.30 + F_8 = 0.0$$
  
 $\therefore F_8 = 24.30 \ kN$ 

The calculated values can be checked by considering the equilibrium at joint D.



$$(\Sigma V = 0)$$
  
-1.07 - 23.93 = -25.0 kN (equal to the applied vertical load at D)

 $(\Sigma H = 0)$ 

$$-24.30 + 24.30 = 0.0$$

The axial force and shear force in member CD can be found from:

Axial load =  $+/-(Horizontal force \times Cos\alpha) +/-(Vertical force \times Sin\alpha)$  $Shear force = + \frac{1}{-(Horizontal force \times Sin\alpha)} + \frac{1}{-(Vertical force \times Cos\alpha)}$ 

The signs are dependent on the directions of the respective forces.

 $\alpha = \tan^{-1}(2.0/4.0) = 26.565^{\circ}$  $\cos \alpha = 0.894; \sin \alpha = 0.447$ 

Member CD:



Shear force = 
$$-(24.30 \times \sin \alpha) + (1.07 \times \cos \alpha) = -9.91 kN$$

Member DEF:

1.07 kN

$$\theta = \tan^{-1}(2.0/6.0) = 18.435^{\circ}$$
  
 $\cos \theta = 0.947; \sin \theta = 0.316$ 



Assume axial compression to be positive. At joint D Axial force =  $+(24.30 \times \cos \theta) + (23.93 \times \sin \theta) = +30.57 \ kN$ Shear force =  $+(24.30 \times \sin \theta) - (23.93 \times \cos \theta) = +14.98 \ kN$ 



#### At joint F

 $\begin{aligned} Axial \ force &= +(24.30\times\cos\theta) + (58.93\times\sin\theta) = +41.63 \ kN\\ Shear \ force &= -(24.30\times\sin\theta) + (58.93\times\cos\theta) = +48.13 \ kN \end{aligned}$ 



# **Comparison of Results**

Reactions	Node Number	Theoretical	Midas Gen
H <sub>A</sub>	1	4.32	4.32
V <sub>A</sub>	1	64.07	64.07
H <sub>H</sub>	8	-37.32	-37.32
V <sub>H</sub>	8	78.93	78.93



## Reference

William M.C. McKenzie, "*Examples in Structural Analysis*", 1st Edition, Taylor & Francis 2 Park Square, Milton Park, Abingdon, Oxon OX14 4RN, 2006, 5.1.2 Example 5.2, Page 361.

Unit : kN,m