Statically determinate beam analysis

Title

Statically determinate structural analysis for a simply supported beam with an overhang for reaction calculations.

Description

Beam ABCD has a pinned support at A and a roller support at C. It carries two concentrated loads of 15 kN each and a uniformly distributed load of 2 kN/m over the right hand half as shown in the figure below. Determine the Reactions.



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. You can also change units any time in the model from the status bar below as shown in the figure.

💷 Gen 2	2013 - [C:\Users	\Owner\D	esktop\	Work Back	up-new\S	hort Tuto	rials for	students\(Gen\1	Futorial 1- Stati	ically determi	nate beam analys	is\Tu	torial 01] - [M	odel Vie]
: 🔯 <u>F</u> ile	<u>E</u> dit <u>V</u> iew	<u>M</u> odel	<u>L</u> oad	<u>A</u> nalysis	<u>R</u> esults	<u>D</u> esign	M <u>o</u> de	<u>Q</u> uery	Too	ols <u>W</u> indow	<u>H</u> elp		_		
Freque	Grid/Snap	UCS/GCS	View	C Acti	vation			Wizard 1		Unit System.			ad	Building Me	esh Settle
2 📽	1 🗶 📜 📚 🕽	e 😢 🖻	💥 🖗	, 📮 🚬	n, 🏣			1/ 🖉		Preferences.			2		
i 🗅 🚅		+ <u>C</u> +	8	<u>)</u> i 🔒 1	: 🚈 🛛	2. 😼 🗹	24	11 🕟		MGT Comma	nd <u>S</u> hell	Ctrl+F12	•	\$	
Tree Mer	าน			ąΧ	۵ 🚺	Model	view			<u>B</u> ill of Materia	il				
Menu	Tables I. Crown	Morke	Bonor	+ 1						Data Generat	tor	I I	·		
Struc	Unit System								₩	Sectional Pro	perty Calcula	tor			
	Length	Force	(Mass)		Heat				C.	General Sect	ion Designer.				
	⊙ m	O N	(kg)		C cal					Convert Meta	a Files to DXF	Files			
F	Com	€k≬	N (ton)		C kca				忿	Dynamic Rep	ort Generato	r			
÷ 🛼 N	Omm	C k	gf (kg)		01				-	Dynamic Rep	ort Image				
	~	C to	nf (ton)		0.12				1	Dynamic Rep	ort Auto Reg	eneration			
	U II	O Ib	f (lb)		C N					Text Editor		Ctrl+F5			
	C in	O ki	ps (kips/g	a)	🖲 Btu					Graphic Edito	or	Ctrl+F6			
E - E	Temperatur	e								<u>C</u> ustomize		1			
	C Celsius	•	Fahrenhe	eit						Full Screen		Ctrl+U			
	Note : Sele	cted units	are displa	ayed in rele	vant				-						
	dialog boxe units.	s. Values a	are NOT	changed wit	h										
	Set/Chang	e Default	Unit Syst	em											
	OK		Apply	/	Cano	el									

Dimension: Length= 3@2.0m= 6.0m
 Step 4: Go to Model>Structure Wizard>Beam and type in 3@2.0 in the Distances box.
 Press Add. Type 1 in the Material and Section ID entry.

Gen 2013 - [C:\Users\Own	er\Desktop\Work Backup-new\Sho	ort Tutorials for students\Gen\Tutorial 1- Statically determinate beam analysis\Tutorial 01] - [Model Vie]
Eile Edit View Mo	del <u>L</u> oad <u>A</u> nalysis <u>R</u> esults <u>D</u>	<u>D</u> esign M <u>o</u> de Query <u>T</u> ools <u>W</u> indow <u>H</u> elp
Freque Grid/Snap	Structure Type	Wizard Node Element Property BC/Mass Stage Load Building Mesh Settler
* / * 🗏 🛎 😕	Structure <u>W</u> izard	<u>ーー Beam </u> 日本 % 区 × 品 決 ゆ か か
D 📽 🖬 🗙 🗅	User Coordinate System	1 Column 🛛 🖾 Beam Wizard 💽 💽
Tree Menu	<u>G</u> rids ▶	Arch Input/Edit Insert
Menu Tables Group Structure Analysis Geometry La Static Loads Configuration Geometry La Static Loads Configuration Arrow Time History Analysis Moving Load Analy TSettlement Analysis Heat of Hydration A Non-Linear Analysis Results Design Configuration Configuration A Results Configuration Configuratio	Nodes Elements Properties Boundaries Masses Mesh Domajn Dimension Toggle Worktree Grgup Check Structure Data	If Frame Ctrl+Shift+X Iruss Ctrl+Shift+Y Imput Type Type 1 C Type 1 C Type 2 Imput Type Pstances: 3@2.0 m Imput Type C Type 1 C Type 2 Imput Type Pstances: 3@2.0 m Imput Type C Type 1 C Type 2 Imput Type Pstances: 3@2.0 m Imput Type C Type 1 C Type 2 Imput Type Pstances: 3@2.0 m Imput Type C Type 1 C Type 2 Imput Type Imput Type Pstances: 3@2.0 m Imput Type Imput Type Pstances Delete Delete

- *Element type:* Beam element (Beam Wizard from Step 4 generates beam elements automatically)
- *Material: M*odulus of elasticity, $E = 3.0 \times 10^7 \text{ kN/m}^2$

Step 5: Go to **Model>Properties>Material>Add**. Select User defined in the Type of Design and Enter $E=3.0 \times 10^7 \text{ kN/m}^2$. Enter a name for the material and click OK and Close

Ele Edit View M	odel Load Analysis Besu	ts Design Mode Query Iools Window Help	
Freque Grid/Snap	Structure Type	Witzard Node Element Property BC/Mass Stage Load Building Mesh Settlement Result Query	
*273\$	Structure Wizard	▶ ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ●	Material Data
D 📽 🖬 🖂 🕰	User Coordinate System	• <mark>₩ E % # 6 @ @ 9 % % % % @ 2 * 2 * * # </mark> ≜ 1 @ % %	General
Tree Menu	Grids	> lodel View	Material ID 1 Name Material 1
Menu Tables Group	Nodes	Properties	E Fasticity Data
Structure Analysis Configuration	Elements	Material Secton Thickness	Type of Denim Liker Defined
Geometry	Properties	Material ID Name Type Standard D6 Add	Standard None
Static Loads Response Spectrum	Boundaries	Time Dependent Material(Creep/Shrinkape) Function 1 Material 1 User Def. Modify	
Time Hatory Analya	Masses	Time Dependent Materal(<u>Creep/Shrinkage</u>)	User Concrete
III Settlement Analysis	Buigng	Ime Dependent Material (ogn). Strength)	-Time of Material
Composite Section Heat of Hydration A	Mesh	Change Element Dependent Material Property	C Isotropic C Orthotropic DB
Non-Linear Analysis	Domajn	Plastic Material	User Defined
Construction Stage Besuits	Named Plane	T Carting	Modulus of Elesticity : 2.0000e+005 kN/m^2
🕘 🔣 Design	Dimension	Section Stiffness Scale Factor	Poisson's Ratio : 0
Clery 1	Toggle Worktree	Tapered Section Group	Thermal Coefficient : 0.0000e+000 L(P)
	digup	A Thirkness	Weight Density : 0 kN/m^3
	Check Structure Data	Wall Stiffness Scale Factor	Use Mass Density: 0 kN/m^3/g
		H Inebstir Hinne Bronarties	Concrete
		H Inelastic Hinges	Modulus of Elasticity : 0.0000e+000 kN/m^2
		😤 Group Damping : Element Mass & Stiffness Proportional	Poisson's Ratio : 0
		😤 Group Damping : Strain Energy Proportional	Thermal Coefficient : 0.0000e+000 1/01
		C Inelastic Material Properties	Webschersty : 0 Min^3
		Eber Division of Section	(dermas benny) v klum*3ka
		Material Table Ctri+Alt+L	Plastor Data
		Table Ctri+Alt+S	Plastic aterial Name NONE
		Thickness Table Ctri+At+T	Thermal Trans
		Change Element Dependent Material Property Table	Specific Heat 0 Btu/AN*(F)
		Inaliatic lange Table	Heat Conduction : 0 Btu/m*hr*1F1
		1 Frances and a second	Demping Ratio :
			OK Canda Hoppy

• Section Property: $B \times H = 0.5 \text{ m} \times 0.5 \text{ m}$

Step 6: Go to **Model>Properties>Section>Add**. Select Solid Rectangle in the Type of Section Drop down menu and check on User. Enter H=0.5 and B=0.5. Enter a name for the Section, click OK and Close

Freque Grid/Snap	Structure Type	Wizard Node Element Property BC/Mass Star	e Load Building Mesh Settlement Result Query		
*27	Structure Wizard	 1/2/車本全員通本業配×4.2 	****		
	User Coordinate System	• <u>ਬ 🛙 & #</u> 🗊 @ 🖻 와 🗣 🕸 & @ 🖇	- 🕰 🛱 🛍 🗅		
Tree Menu	Grids	Iodel View		Section Data	•
Menu Tables Group Structure Analysis Configuration	Nodes Elements	• •	Properties Material Section Thickness	DB/User	
Geometry	Properties	• 🔟 Material	The lines	Section ID 1	Sold Rectangle
Static Loads Response Spectrum Por Time History Analys Moving Load Analys	Boundaries Masses Builging	Time Dependent Material(Creep/Shrinkage) Eurction Time Dependent Material(Creep/Shrinkage) Time Dependent Material(Comp. Strength)	I Section 1 User 58	Modify Name Section 1 0	User CB ALSC10(US)
Settlement Analysis Gomposite Section Heat of Hydration A Non-Linear Analysis	Mes <u>h</u> Domajn	Time Dependent Material Link Change Element Dependent Material Property Plastic Material		Copy Import	Sect. Name v
Construction Stege Results Delign Query County	Ngmed Plane Dimengion Toggle Worktree Grgup	Section Section Stiffness Scale Factor Stapered Section Group		Renumber	Get Data from Single Anale DB Name AtSC10(US) v Sect. Name v
	Check Structure Data	→ Thickness Wall Stiffness Scale Factor		12	H 0.5 m B 0.5 m
		H Instant: Hinge Properties H Instant: Hinges Coroo, Darrong: Element Mass & Stiffness Proportional Coroo, Darrong: Element Mass & Stiffness Proportional Coroo, Darrong: Element Mass & Stiffness Proportional Distribution Distribution Distribution Element Mass of Section	H103 8105		
		Starteil Table Cri+Ab: Section Table Cri+Ab: Thickness Table Cri+Ab: Manage Element Dependent Natural Property Table Cri+Ab: Wild Stress Scale Factor Table Health: Hings Table	н. 15 Т	Offset : Center-Center Change Offset	Consider Shear Deformation.

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

Gen 2013 - (C/USers/Owner/Deidnep/Work Backup-new/Short Tul Ere Edt Yew Hodel Load Analyse Besuits Desg FrequeOut/Swap UCS/HCS / Vew CArtivisouri X / Silver Control Control Control Inc.	onials for students/Gen/Tutorial1-Statically determinate beam i Mgde Query Isola Window Heb Woard Finder Element Property BC/Hass Stat	nalysis/Tutonal 01 *) - (Mod e Load Buikting Hesh	el V] Settlement Result Query	
			· · · · · · · · · · · · · · · · · · ·	
Tree Menu a × 4 / Mode	View	·		
The Table Card Week Report		-		

• *Boundary Condition:* Simply Supported (Pinned at A and Roller at C)

Step 7: Go to View>Remove Hidden Lines or click \clubsuit to toggle back to the wireframe (line) view. Check on \bullet^n in the toolbar to display node numbers. Use select single \clubsuit to select or highlight node 1.



Step 7: Go to **Model>Boundary>Supports** and check on Dx and Dz and Apply.

Gen 2013 - [C:\Users\Owner\Desktop\Wor	rk Backup-new\Short Tutorials	for students\Gen\Tutorial 1- Statically determin	ate beam analysis\Tutorial 01 *] - [M	odel V]	
📴 Eile Edit. View Model Load An	nalysis <u>R</u> esults <u>D</u> esign M	ode Query Tools <u>W</u> indow <u>H</u> elp			
Freque Grid/Snap UCS/GCS View C	. Activation	Wizard Node Element Property BC/M	lass Stage Load Building Mes	h Settlement Result Query	
"¥ 🖊 🎢 🇮 🌲 🎾 🎦 🎾 🖉 📮] .n 😃 🍺	「「夏南な今日間平米四米	16 X 🔿 > >		
D ≊∎ × Ω•⊇• @ <u>0</u>	● 筆種 之回图 ※	#®®®®♥₽₽₹\$®\$	- 🕹	• ! 🕰 🔒 🔒 🖓 📆 🕞	
Tree Menu 🛛	x 4 🚺 Model View	v			
Node Element Boundary Mass Load					
Supports					
Boundary Group Name					
Default					
Options					
(• Add (Replace (Delete					
Support Type (Local Direction)					
Z					
Ry Dy 7					
Rx					
Rz X					
Dz * A					
E D-AU					
Dx 🔽 Dy 🗆 Dz 🔽					
	0				
R-ALL					
Rx T Ry T Rz T					
Apply Close					

This becomes the Pinned support.

Step 8: Again use Select Single 🗳, this time to highlight or select node 3 and check on Dz only, Apply and Close.



This becomes the Roller support.

• *Load Case 1:* 2 vertically downward concentrated loads P=15 kN are applied at the nodes 2 and 4 in the (-) Z direction.

Step 9: Go to **Loads>Static Load Cases** and define static load cases 'P' (concentrated load) and 'w' (Uniformly Distributed Load). Select Load type as User defined for both of them. Click Close after adding the two load cases.



Step 10: Go to **Load>Nodal Loads** and select load case P. Select the nodes 2 and 4. Enter FZ=-15 kN and press Apply. Click on the Front View icon **a** on the Right to see the front view of the beam.



• *Load Case 2:* A uniformly distributed load, w=2kN/m is applied on the beam over a distance of 3m from end D.

Step 11: Switch off display of node numbers by clicking on \square . Click on \square to display element numbers. Go to **Model>Elements>Divide Elements.** Select the element number 2 using \square . Enter number of divisions as 2 and click Apply and Close.



Step 12: Go to Load>Element Beam Loads. Select the load case 'w' and select Load type as Uniform Load. Enter w=-2kN/m in the Direction Global Z. Select elements 4 and 3 using \therefore . Click Apply and Close. Switch off element number display \therefore .

Gen 2013 -	[C/\Users\Owner\Desktop\Work Backup-new\S	ihort Tutorials for students\Geni Tutorial 1- Statical	ly determinate beam analysis\Tutoria	t 01 *] - [Model V]				14
Tac Fee Eq.	t Yew Model Load Analysis Besuits	Deedu Möge ÖnelA Toos Mulgow F	ieb					
Freque	Pd/Snap10CS/0CS1Vew C1 Activation	Witard Node Element Proc	renty BC/Mass Stape Load Bui	king Nesh Setti	ement Result Query			
\$ [7] 7].	A B P P P P P P P P P P P P P P P P P P	I to of the state of the state	A G V W V & S S					
	× 2.5.90	LE ARE DE ESPACE	19 1		· 🛱 🔒 🐴 🕏	1 1		
Tree Monu	* × 1 / D	Hodel View						
Node Eleme	ent Boundary Main Load							
Element Beam	Loads 🔄							
- Load Case N	470							
-								
Load Group?	ane al unit							
Onlines								
G Add C	Replace (* Delete							
Load Type								
Uniform Loa	els 💌							
11.17	(TTT)							and the second se
No	+++++							
2	22				100	C. A.		
			4		- <u>-</u>	f. 4	 2	
T tcombio	stv							
Lange and the second second								
Direction :								
Take								
(Relativ	e C Absolute							
x1 0								
x3 0	0							
x 4 0	0							
	UNC XEM							
	Antiv Que							

• Analysis: Step 13:Go to Analysis>Perform Analysis or Press F5

Results

Reaction Forces:

Step 14: Go to **Results>Combinations** and enter a combination name. Select type as Add and add a combination COMB 1 = 1.0*P + 1.0*w as shown in the figure below.

🕼 Gen 2013 - [C:\Users\Owner\Desktop\Work Bac	ckup-new\Short Tutorials for students\Gen\Tutorial 1- Statically determinate beam analysis\Tutorial 01] - [Model Vie]	
Eile Edit View Model Load Analysis	s <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 Ac	ti Combinations Ctri+F9 BC/Mass Stage Load Building Mesh Settlement Result Query	1
18 🖊 🎢 🏛 🎓 18 19 12 🖗 📮 🖓	🛛 Reactions	
D 🖻 🖬 🗙 🏊 - 🗅 - I 🖨 🖪 i 🔮 '	ግ Deformations 🔸 🗘 🔹 🖧 📥 🖗 🖗) 📆 🛅
Tree Menu 🏾 🗛 🗙	Load Combinations	
Menu Tables Group Works Report	General Steel Design Concrete Design SRC Design Footing Design - Load Combination List	
· Soldes:5 · Song Benerits:4 · Song Beam:4 · H≣ Properties Material:1	No Name Active Type Description Active LoadCase Factor A COMB 1 Activ Add P(ST) 1.0000 w(ST) 1.0000	
□ I: Material 1 □ I: Material 1 □ I: Section : 1 □ I: Section 1		
		Ĩ.
 □ 10 Static Load Case 1 [P :] □ 10 Nodal Loads : 2 □ 10 Static Load Case 2 [w :] □ 40 Element Beam Loads : 2 	E	E
	· ·	Ļ
	Copy Import Auto Generation Spread Sheet Form Copy into Steel Design	•
	File Name: C:Users\Owner\Desktop\Work Backup-new\Short Tutr Browse Make Load Combination Sheet Close	

Step 15: Click on **Results>Reactions>Reaction Forces/Moments** and select the load combination COMB 1. Select FXYZ. Check o data and click on the box next to Values to change number of decimal points to 2 and click OK to see reactions graphically.



Step 16: Click on **Results>Result Tables>Reactions** and select the load combination COMB 1. Click OK to display the reactions in the table format. Note the Summation of all forces from the Reaction Table.



Hand Calculations:



- (1) To determine H_A ($\Sigma H = 0$) There are no horizontal loads. $\therefore H_A = 0$
- (2) To determine $V_{\rm C}$

Take moments about A:

Note that the moment of the UDL (Uniformly Distributed Load) is the resultant total of UDL ($2 \times 3 = 6 \text{ kN}$) multiplied by the distance from A to the line of action of that resultant (i.e. 4.5 m)

 $\begin{aligned} (\Sigma M_A &= 0) \\ + (15 \times 2) - (V_C \times 4) + (2 \times 3 \times 4.5) + (15 \times 6) &= 0 \\ &\therefore V_C &= +36.75 \ kN \end{aligned}$

(3) To determine V_A $\Sigma V = 0$

$$+V_A - 15 + V_C - (2 \times 3) - 15 = 0$$

+V_A - 15 + (+36.75) - 6 - 15 = 0
∴ V_A = -0.75 kN (*i.e.* 0.75 kN downwards)

(4) Check by taking moments about C:

$$\Sigma M_C = +(V_A \times 4) - (15 \times 2) + (2 \times 3 \times 0.5) + (15 \times 2)$$

= +(-0.75 \times 4) - 30 + 3 + 30 = 0
\times Correct

Comparison of Results

Reaction	Node Number	Theoretical	Midas Gen
H _A	1	0.00	0.00
V _A	1	-0.75	-0.75
V _C	3	36.75	36.75

Unit : kN

Reference

Ray Hulse and Jack Cain, "*Structural Mechanics, Macmillan College Workout Series*", 1st Edition, The Macmillan Press Limited, Houndmills, Basingstoke, Hampshire, RG21 2XS, 1991, Example 1.2, Page 6.