

Timber Beam-Column Design

Project Description:

The challenge here is to choose a section for the beam-column shown below. The section should be a standard dressed timber made from No. 1 graded Douglas Fir-Larch.



Building the Model

The member is 16 feet long. It is pinned at its base and fixed in the X-direction at its top. For a preliminary size, use a 5x5 Standard Dressed Timber. For the material, from the Visually Graded Timbers/Beams and Stringers category choose Douglas Fir-Larch for the species and No. 1 for the grade.

If you need help creating the model please consult the VisualAnalysis User's Guide (**Help | Contents**).

Loading

Create the following load cases:

Service Cases:

Dead loads: Apply an axial force of ten kips as shown above.

Wind +X loads: Apply a uniform transverse load of 0.3 kips/ft in the X-direction.

Factored Combinations:

Use the **Load | Load Case Manager** menu to create the ASCE 7-05 ASD load combinations.

Information on applying loads and creating factored combinations is available in the VisualAnalysis User's Guide (**Help | Contents**).

Setup the Design Parameters

Select the beam-column in the design view to make it active for editing using the Modify tab of the Project Manager. Go to the Modify tab and set up the following parameters.

General:

Leave the General parameters unchanged.

Design As:

As mentioned above, we want to design the member as a Standard Dressed Timber, using Douglas Fir-Larch No.1 for the species and grade. Make sure the Shape Category and Material are set up properly.

Bracing:

Select unbraced for all bracing options. Assume the member is part of a braced system in both the y and z directions. With this in mind leave both the Sidesway y and Sidesway z parameters set to No. Leave the k factors unchanged so they are calculated automatically.

Beam Deflections:

Assume a deflection limit of L/240 was specified for the Wind Loading. With this in mind, choose Member Span Ratio as the Strong Deflection type and enter 240 for the 'E or W' (Seismic or Wind) value.

Column Deflections:

Create Modify Filter Grids Cut Views	
Modify:	Design Group
General:	
Design this Group?	Create Table...
Design As	
Member Type:	Other
Shape Category:	NDS Wood\Timber (Std.
Show Advanced?	<input checked="" type="checkbox"/> Yes
Bracing	
Weak (y):	Unbraced
Strong (z):	Unbraced
Sidesway (y):	<input type="checkbox"/> No
Sidesway (z):	<input type="checkbox"/> No
Override K?	<input type="checkbox"/> No
Deflections	
Strong (dy)	Member Span Ratio
'L only':	1
'E or W':	240
'D + L':	1
Other:	1
Weak (dz)	None
Size Constraints	
Configuration	
Options	
Beam	
Beam Type	Simply supported
Loading Pattern	Uniform
Custom Len. Betw. Infl. Pl	<input type="checkbox"/> No
End Notch Type	No Notches
Advanced	

No drift limits were specified so leave the Column Deflections unchecked.

Size Constraints:

No size limits are given, so leave the Depth and Width limit boxes unchecked.

Configuration:

The member is not a repeat member and it does not have bolt holes.

Options:

Leave the Temperature Range at $T \leq 100$ F. Also leave the High Moisture Content, Pressure Treated, Qualifies for Buckling Stiffness Factor, and Qualifies for Incise Factor boxes unchecked.

Beam:

Choose Simply Supported for the Beam Type. For the Loading Pattern, choose Uniform. Leave the Custom Length Between Inflection Points option unchecked. The member does not have notches.

Advanced:

Leave the Advanced parameters unchanged.

Design Load Cases

Go to **Design | Choose Load Cases** to set up the load case for design purposes. Make sure all of the Building Code Combinations are marked to be checked for strength. Make sure the Wind +X loads case is marked for serviceability so that the deflection can be checked for this case.

Once the design group is set up, analyze the model.

Designing the Member

After analyzing, switch to a design view if you aren't currently in one. Select one of the members and choose **Design | Design Selected Group**. The Wood NDS ASD Design Selection dialog will be brought up, with a list of acceptable sizes. Choose a size and **click** OK.

After accepting the design, the unity value that appears in the design view should have a ~ in front of it indicating that it is a preliminary value based on the analysis results with the member in place. To get the real updated unity value you must synchronize the design changes.

Synchronizing Design Changes

To synchronize the design changes, select **Design | Synchronize Design Changes**. You will be prompted to re-analyze, select "Yes", and when it finishes re-analyzing go back to a Design View and review the unity check. It should no longer have the ~ in front of it, indicating that is a final unity value.

If the unity value is greater than one the member has failed and you need to reiterate the design process. The closer the unity value is to one the more efficient, but less conservative the member is.

Reporting Design Checks

To report the extreme design checks, **double-click** on the member in the design view. As you scroll down through the report you will eventually reach the Extreme Checks area. In this area the extreme cases for design checks such as Combined Stresses and Strong Deflections are reported.

Extreme Checks Only

Combined Stresses Check:

Member Name	Load Case #	Offset ft	State	Ft' or Fb1* or Fb2* or Fb1** or Fb2** or						ft or			Unity Check
				Fc' Ksi	FcE1 Ksi	FcE2 Ksi	Fb1' Ksi	Fb2' Ksi	FbE Ksi	fc Ksi	fb1 Ksi	fb2 Ksi	
M1	5	8.00	Comp.	0.36702	1.71040	0.39122	2.0912	1.5984	9.53530	1.159830	0.95027	0	0.69

Strong Flexure Check:

Member Name	Load Case #	Offset ft	Fb Ksi	CD	CM	Ct	CL	CF	Fb'		fb	Unity Check
									Ksi	Ksi		
M1	2	8	1.3500	1.60	1.00	1.00	0.97	1.00	2.09120	0.95027	0.45	