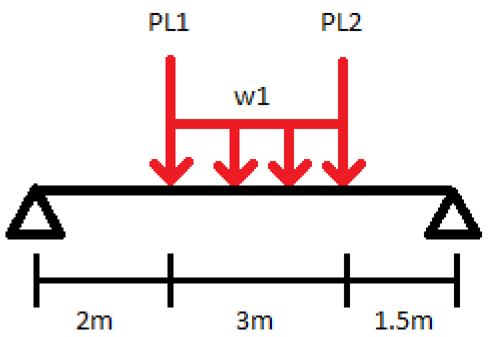
Linking Analysis to Member Designs

In this tutorial:

- Linking and Analysing a timber beam using the Timber Member Design
- Unlinking and Linking a steel beam using the Steel Member Design
- Change the material using the Change Material feature
- Further analysis of steel beam using Structural Toolkit Analysis

Step 1 – Linking and Analysing a timber beam using the Timber Member Design

Using the **Analysis module** in **Structural Toolkit**, we can analyse members that may have **up to three point loads** or **partial UDL's** on certain parts of the beam.



Load	DL	LL
PL1	2.0kN	3.0kN
PL2	1.0kN	1.0kN
w1	0.5kN	1.5kN

Figure 1 – Beam Problem

To analyse the above beam, first open a **Timber Member Design**.

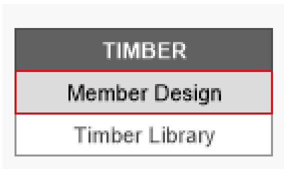


Figure 2 – Timber Member Design

Within the **Timber Member Design**, we will switch over to the **Analysis** first to get the maximum result values.

Click on the **[Analysis…]** button to open an **Analysis module**.

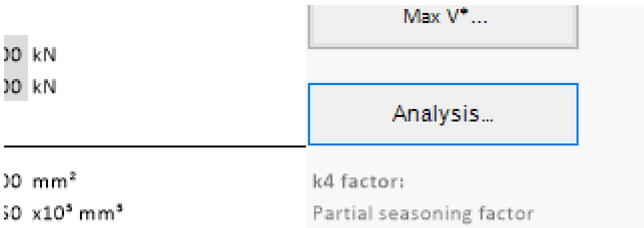


Figure 3 – Timber Member Design Analysis Button

Notice in the **Project Tree,** the **Timber Member** and **Timber Member (Ana)** are both underlined, indicating that they are linked members.

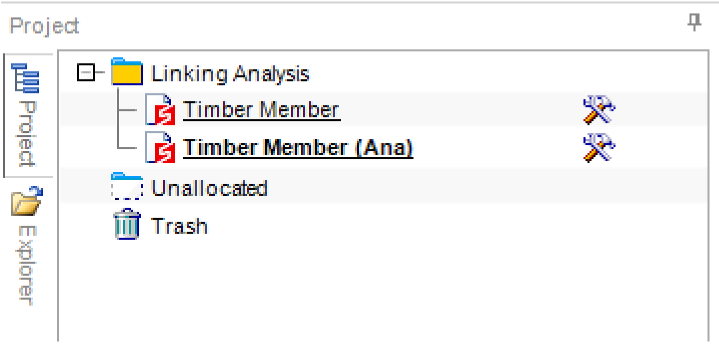


Figure 4 – Project Tree

Input the **geometry** and **loading** of the beam, starting with the **Span** and **Span type** of **(S)** for a simply supported beam, noting that the size and sectional properties automatically transfer (do not edit these manually, they will reset when closed and re-opened as they are linked.)

ANALYSIS V5.00

Geometry for (Timber Member (Ana)): simple beam

Description =	2 / 300mm x 45mm Hyspan	
Span (L) =	6500	mm
Span type =	S	(S)imple, (E)xterior, (I)nterior, (C)antilever, (P)ropped, (F)ixed, (O)ther

Figure 5 – Analysis Geometry

Next input the loads of the beam, using the **Partial UDL** section to indicate a start and end length for the UDL.

Loading

	Uniform loads (kN/m)		
Uniform loads	UDL	Partial 1	Partial 2
Dead load (wdl) =		0.50	
Live load (wll) =		1.50	
Start from LHS (mm) =	0	2000	
End from LHS (mm) =	6500	5000	
S.Wt =	0.18	kN/m	
Ultimate load (w*) =	0.21	2.85	0.00

	Point loads (kN)		
Point loads	PL 1	PL 2	PL 3
Dead load (pdl) =	2.00	1.00	
Live load (pll) =	3.00	1.00	
Pos. from LHS (mm) =	2000	5000	
Ultimate load (p*) =	6.90	2.70	0.00
Include S.Wt =	Y	(Y)es,(N)o	

Figure 6 – Analysis Loads

Self weight is included by default, but may be turned off using the selection input. Also enter the loading type which affects the short and long term factors associated with strength (for timber) and deflections.

Live Load type =	Floor	mber Domestic)	
Short term LL (Ψsu) =	1.00	(Ψsp) =	1.00
Long term LL (Ψlu) =	0.33	(Ψlp) =	0.40
Actual LL (Ψsa) =	1.00	(Ψla) =	0.33

Figure 7 – Analysis Loads

Following our inputs, we have the results at the end of the page, where we can see the **bending moment diagram**, **shear force diagram** and **deflection diagram** (gross immediate with the creep/durational factors not applied).

Graphs * Deflections are Gross Ig immediate - assessment of long term effects to be considered

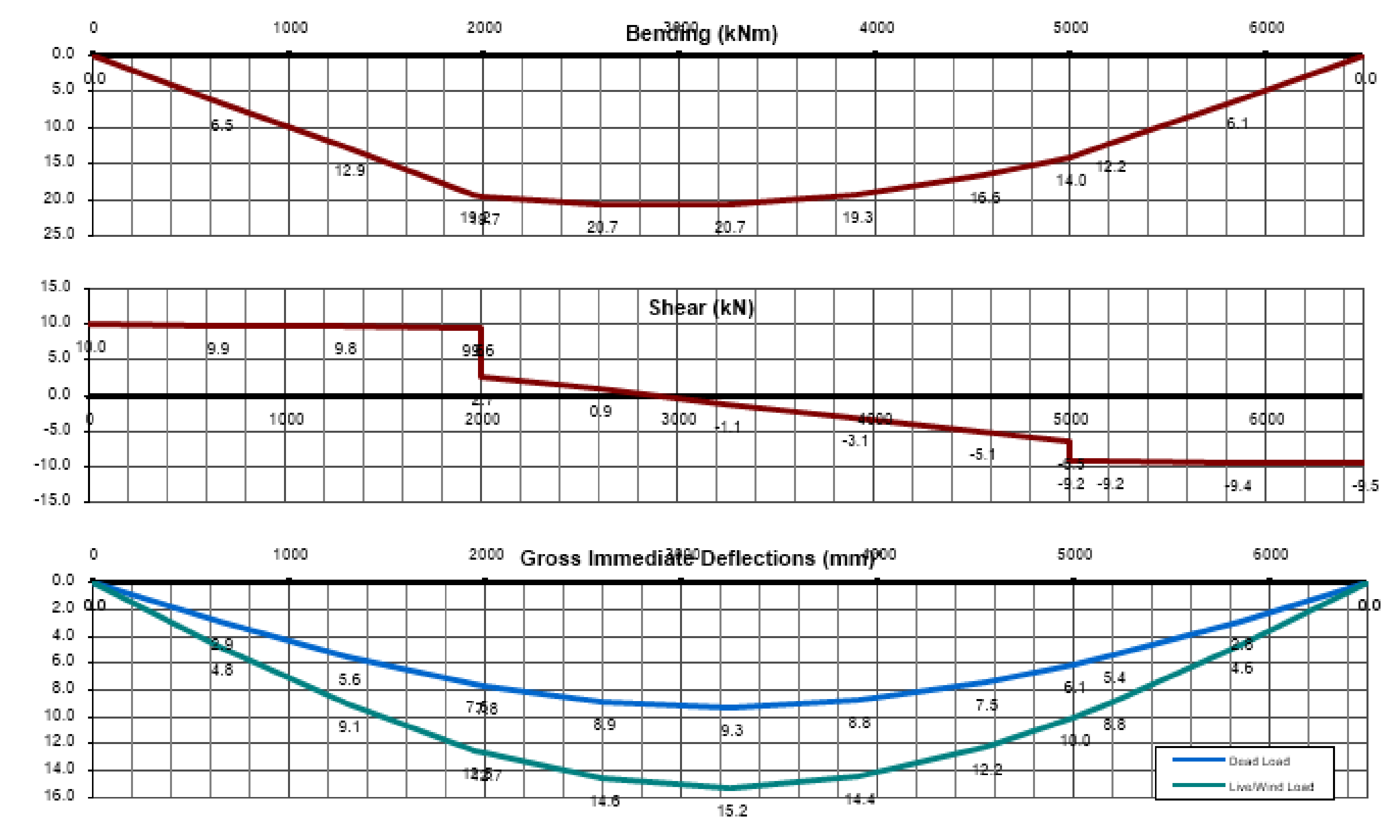


Figure 8 – Analysis Result Diagrams

We can also find the left, right, maximum and results at a specific location from the table above.

Results at midspan Position of result (x) = 3250 mm

1.20*G+1.50*Q analysed - 1.35*G case to be checked

	Left	At x	Right	Max	At	Min	At	Units	Span /
Rdl	2.89		2.77					kN	
Rll	4.38		4.12					kN	
R*	10.04		9.50					kN	
M*	0.00	20.65	0.00	20.86	2885	0.00	0	kNm	
V*	10.04	-1.12	-9.50	10.04	0			kN	
δdl	0.00	9.27	0.00	9.27	3250	0.00	0	mm	702
δll	0.00	15.21	0.00	15.21	3250	0.00	0	mm	427
δdl+Ψs*δll	0.00	24.48	0.00	24.48	3250	0.00	0	mm	266

δPll/δTot.ll = 0.43

Figure 9 – Analysis Result Values

These values, particularly the maximums, we use to design the member.

Click the **[Switch to Design…]** button to switch back to the **Timber Member**.

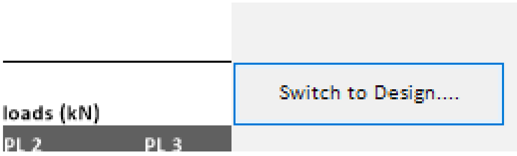


Figure 10 – Analysis Switch to Design

Assume that the **bending lateral restraints (Layb)** remain at **600mm**. (For a floor beam this could be 450mm if laterally supported by joists and for a roof beam perhaps 600mm or 900mm depending if the rafters laterally restrain the member).

Figure 11 – Timber Member Design Restraints

Now change the **Bending & Shear** values to the **Analysis** values using the drop down menu. For this tutorial, change the values to **Critical (C)** which will import the maximum bending and maximum shear values

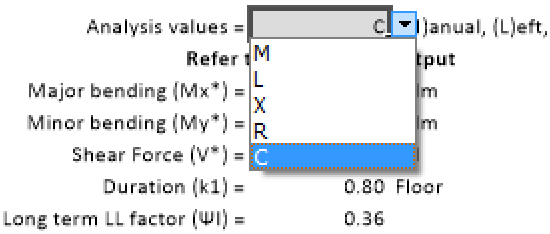


Figure 12 – Timber Member Analysis Values

The default timber beam has failed and therefore we must choose a larger section.

TIMBER MEMBER V5.01		Furr Consulting Pty Ltd
Beam:	(Timber Member) 240mm x 45mm F17 KD HW	
Bending:	M* = 20.18kNm > øM(0.80) = 12.10kNm	No Good (1.67)
Shear:	V* = 9.62kN < øV(0.80) = 17.63kN	OK (0.55)
Axial:	No compression, No tension	
Combined:	Refer below	No Good (2.78)
Deflection:	δdl = L/64 (102mm), δll = L/116 (56mm), δ(dl+Ψs*ll) = L/41 (158mm) at 3250mm from LHS	Warning

Figure 13 – Timber Member Summary

Press the **[Select…]** button to choose a larger section size.

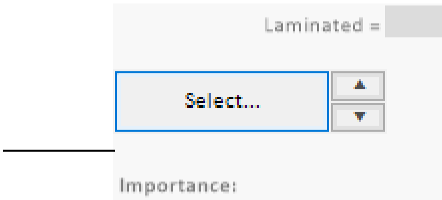


Figure 14 – Timber Member Select Member

Try a double **400 x 45 Hyspan LVL section.** Note that the double section is laminated by default, and the toggle input on the notes area can be changed to consider these as 2 singular members.

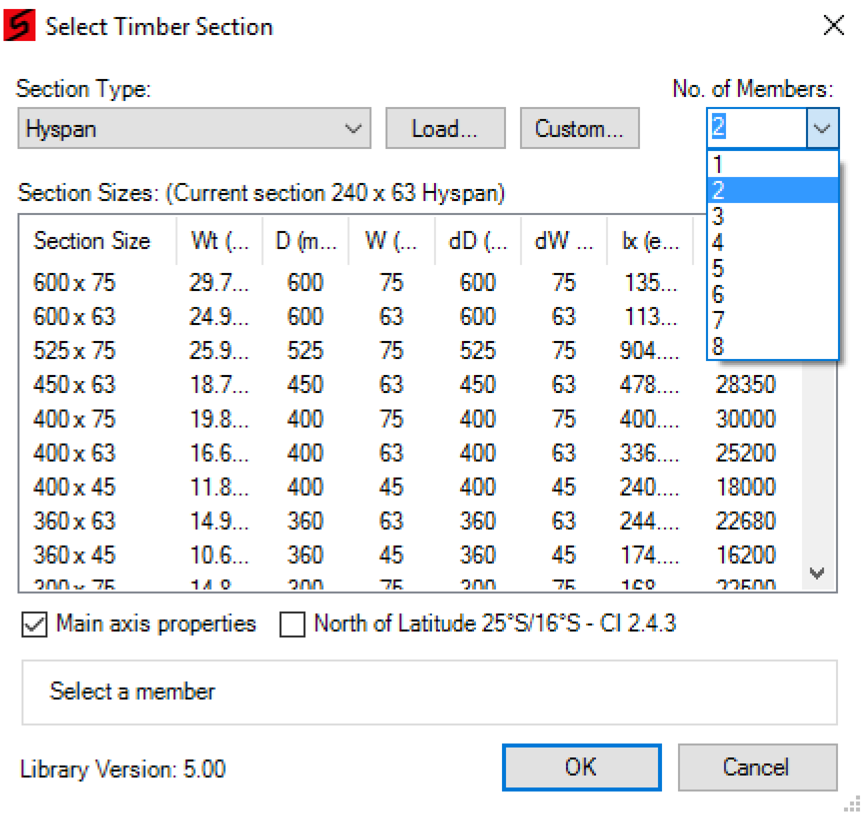


Figure 15 – Timber Selection Dialog

The **2-400 x 45 Hyspan LVL** is **satisfactory**, however the size of the beam suggests that a steel alternative may be preferred.

TIMBER MEMBER V5.01 Furr Consulting Pty Ltd

Beam:	(Timber Member) 2 / 400mm x 45mm Hyspan (Laminated)	
Bending:	M* = 21.23kNm < øM(0.80) = 69.24kNm	OK (0.31)
Shear:	V* = 10.27kN < øV(0.80) = 79.49kN	OK (0.13)
Axial:	No compression, No tension	
Combined:	Refer below	OK (0.31)
Deflection:	δdl = L/505 (13mm), δll = L/1013 (6mm), δ(dl+Ψs*ll) = L/337 (19mm) at 3250mm from LHS	OK

Figure 16 – Timber Member Summary

Step 2 - Unlinking and Linking a steel beam using the Steel Member Design

We can use the same analysis to find an **alternative steel section**. This can be done several ways, by manually unlinking and re-linking, (this step) or by Changing Materials (Step 3 to follow).

First **unlink** the analysis from the **Timber Design Module** by using the **[Remove Link]** button under the **[Members]** tab.

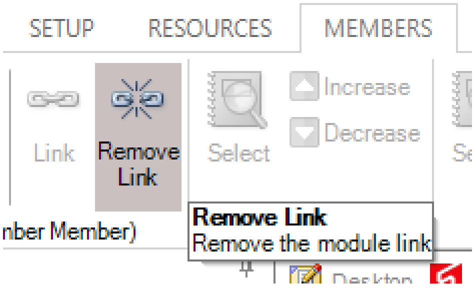


Figure 17 – Remove Link Button

We can now create a new **Steel Member Design** module to link to the analysis

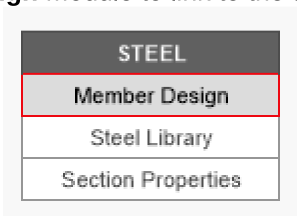


Figure 18 – Steel Member Design

Once opened, again go to the **[Members]** tab and this time select the **[Link]** button, and choose the **Timber Member (Ana)**.

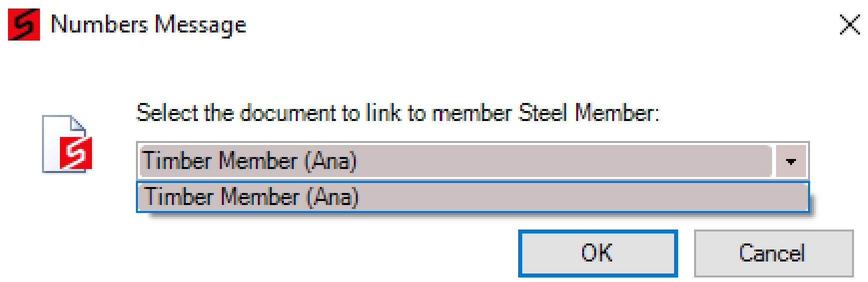


Figure 19 – Link Member Selection

You will also get a message to rename the analysis member to align with the linked member, so press yes to have the name changed.

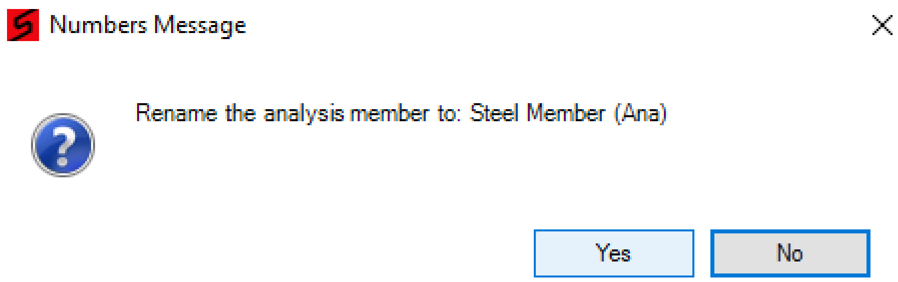


Figure 20 – Rename Analysis Member

Once again change the analysis values in the **Steel Member** to **(C) Critical**.

Anthony Furr Software
ABN 74 992 513 430
97 Mt Pleasant Road
Nunawading, Victoria 3131
P 03 9878 4684
F 03 9878 4685
www.structuraltoolkit.com.au
support@structuraltoolkit.com.au

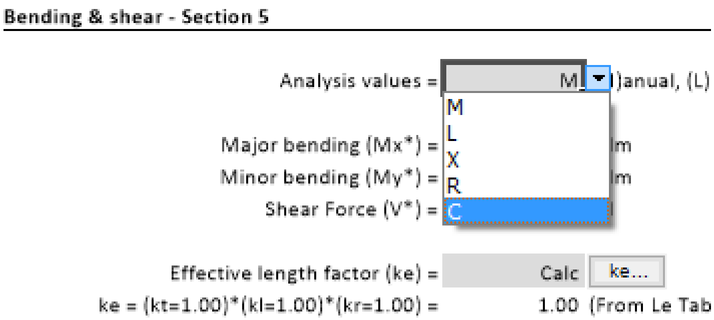


Figure 21 – Steel Member Analysis Values

The exact same analysis is now linked to the **Steel Member** and a section size can be selected. Furthermore, the timber is now redundant and can be removed from the project.

Assume a restraint at the first PL, so a segment of 2000mm and 4500mm. The **αm is calculated as 1.21** for the right segment using AS4100 section 5.6. Here we used the **[αm]** tab to put in the ¼ points to calculate the αm value. The smaller 2000mm segment will have an αm of 1.75 so will be less critical.

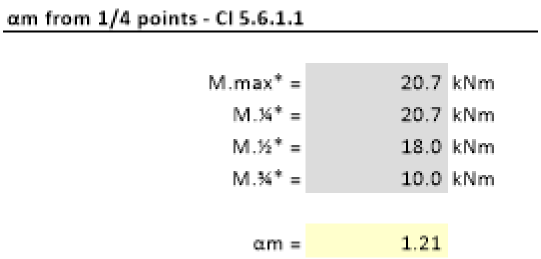


Figure 22 – Steel Member Segment αm

Press the **[Select…]** button, and choose a **200 x 75 PFC** member.

STEEL MEMBER V5.01 Furr Consulting Pty Ltd

Section:	**(Steel Member) 200x75PFC (G300)**	
Bending:	**Mx* = 21.2kNm < øMb(4500,αm=1.21) = 31.1kNm, øMb(αm=1) = 25.7kNm**	**OK (0.68)**
	No minor bending	
Shear:	**V* = 10.2kN < øVvm = 207.4kN**	**OK (0.05)**
Compression:	**No compression**	
Tension:	**No tension**	
Deflection:	**δdl = L/957 (7mm), δll = L/611 (11mm), δ(dl+Ψs*ll) = L/416 (16mm) at 3250mm from LHS**	**OK**

Bending & shear at critical locations - Section 5 Max. restraint (2.5% flange force) = 2.8 kN

Analysis values = C (M)anual, (L)eft, Position (X) from analysis, (R)ight, (C)ritical
Refer to the analysis output (M*to include first order amplification as required - Cl 4.4.2)
Major bending (Mx*) = 21.2 kNm
Minor bending (My*) = 0.0 kNm — Minor bending (My*) = 0.0 kNm
Shear Force (V*) = 10.2 kN
Span / Segment Length (L) = 4500 mm
Effective length factor (ke) = Calc [ke...] — αm = 1.21 [αm...]
ke = (kt=1.00)*(kl=1.00)*(kr=1.00) = 1.00 (From Le Tab)
Effective length (Le = L*ke) = 4500 mm
ø = 0.9 Table 3.4
øMsx = 59.7 kNm — Bending (x) = OK (0.68)
øMbx(αm=1) = 25.7 kNm
øMbx = 31.1 kNm — Shear = OK (0.05)
øMsyL = 12.6 kNm — øMsyR = 13.3 kN
øVv = 207.4 kN
øVvm = 207.4 kN — Ix = 19.1 x10⁶ mm⁴
S.Wt = 0.230 kN/m

Figure 23 – Steel Member

A **200 x 75 PFC** member is satisfactory.

Step 3 – Changing material using the Change Material feature

A simpler way to change the material is to use the Change Material feature. If designing in timber, the button will read [Change To Steel], and if steel [Change To Timber].

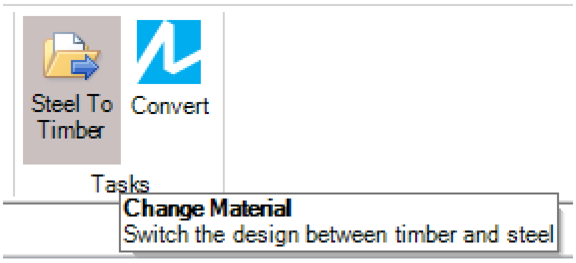


Figure 24 – Change Material

Using this method, the design is copied and renamed to a unique name (including any attached analysis). This works for floor beams, roof beams, propped beams and linked member designs.

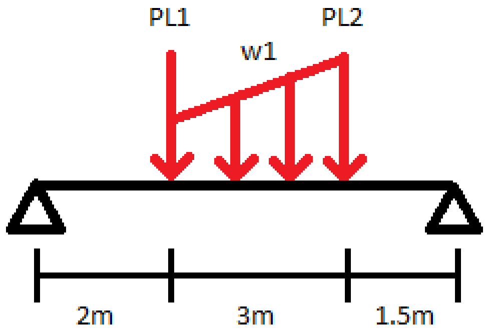
NOTE: The analysis's are now separate, so if loads are updated, both the linked designs' analysis data have to be updated.

Step 4 - Further analysis of steel beam using Structural Toolkit Analysis

If the beam has more than three point loads, is not a single span ie. continuous, has a variable UDL, etc. then we cannot use the Analysis module.

We can however convert the analysis from the Module into **Structural Toolkit Analysis** allowing us to further analyse these conditions and manually transfer maximum results back to the Steel Member Module.

If we use our example, we can change the **UDL to a variable UDL**, starting off with a magnitude of the previous UDL and grading up to the PL2 value.



Load	DL	LL
PL1	2.0kN	3.0kN
PL2	1.0kN	1.0kN
w1 (left)	0.5kN	1.5kN
w1 (right)	1.0kN	1.0kN

Figure 25 – Revised Beam Problem

First select the **[Members**] tab and select the **[Convert]** button to transfer the structure to **Structural Toolkit Analysis**.

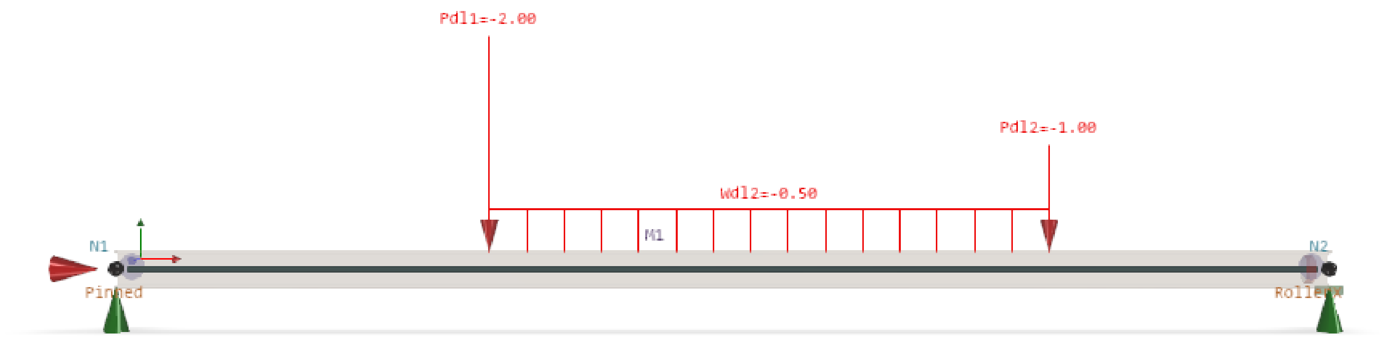


Figure 26 – Structural Toolkit Analysis Beam

If we analyse the bending moment diagram, we can check to verify that all values have transferred across.

Press the **[Linear]** button to perform a **Linear Analysis**.

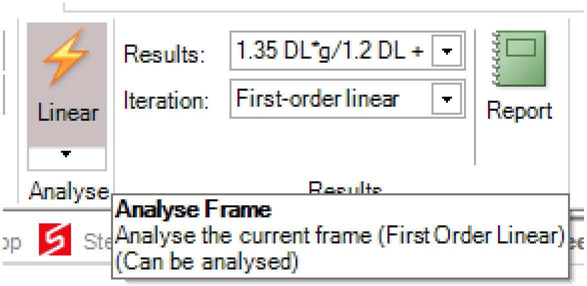
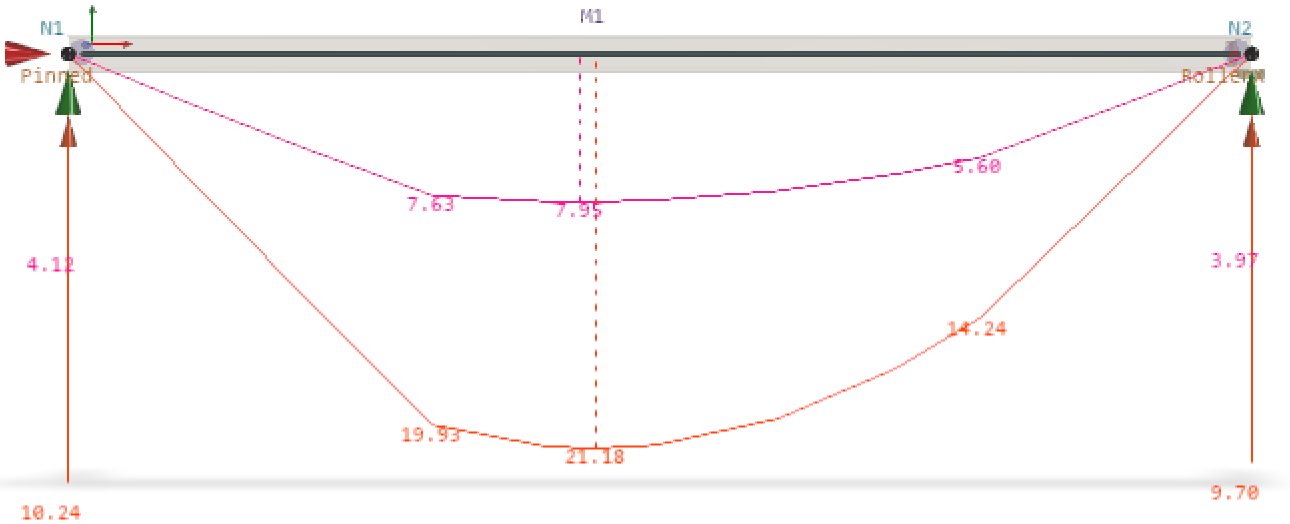


Figure 27 – Linear Analysis Button

The bending moment provides the same results as the Analysis Module (slight discrep,



Results at LHS (Max -ve M) Position of result (x) = 0 mm

1.20*G+1.50*Q analysed - 1.35*G case to be checked

	Left	At x	Right	Max	At	Min	At	Units	
Rdl	3.05		2.94					kN	
Rll	4.38		4.12					kN	
R*	10.24		9.70					kN	
M*	0.00	0.00	0.00	21.18	2890	0.00	0	kNm	
V*	10.24	10.24	-9.70	10.24	0			kN	Span /
δdl	0.00	0.00	0.00	6.80	3250	0.00	0	mm	957
δll	0.00	0.00	0.00	10.65	3250	0.00	0	mm	611
δdl+Ψs*δll	0.00	0.00	0.00	15.62	3250	0.00	0	mm	416

δPll/δTot.ll = 0.43

Graphs * Deflections are Gross Ig immediate - assessment of long term effects to be considered

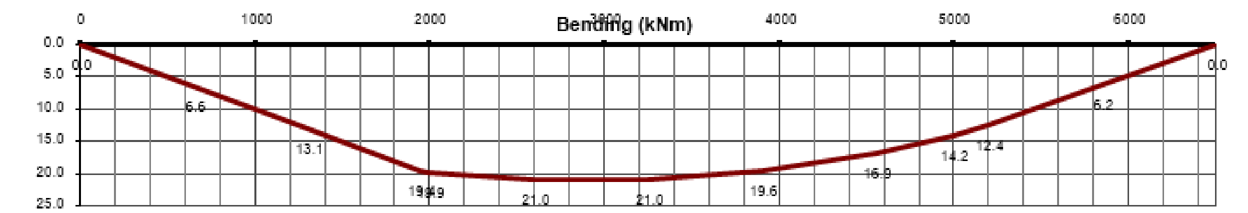


Figure 28 – Bending Moment Comparison

Now we will change the **UDL w1 to a Variable UDL**.

As we already have the values of each end of the UDL, we don't need to create any new Load Definitions, so we can **double click** on the **UDL** to bring up the **Loads dialog**.

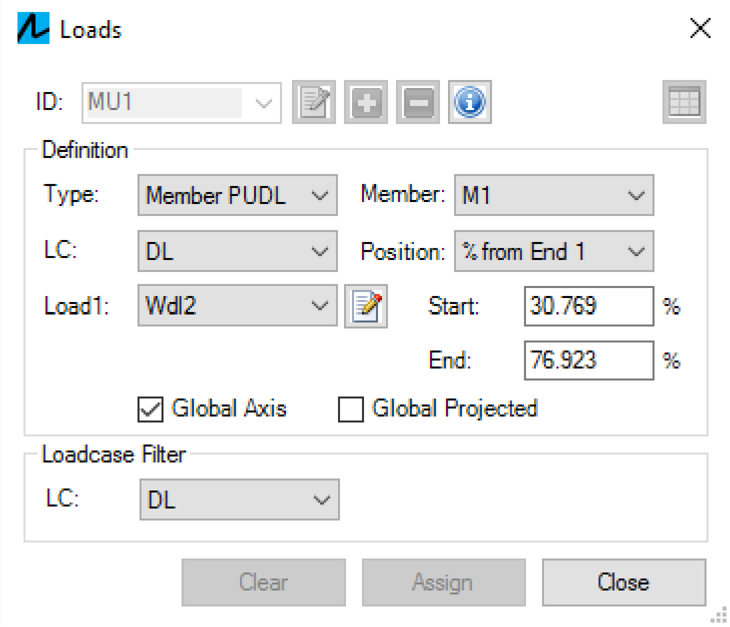


Figure 29 – Loads Dialog

Ensure you have the **DL (Deadload)** case selected and change the **Load Type** to **Member VUDL**, and the **Load 2 definition to Pll1.**

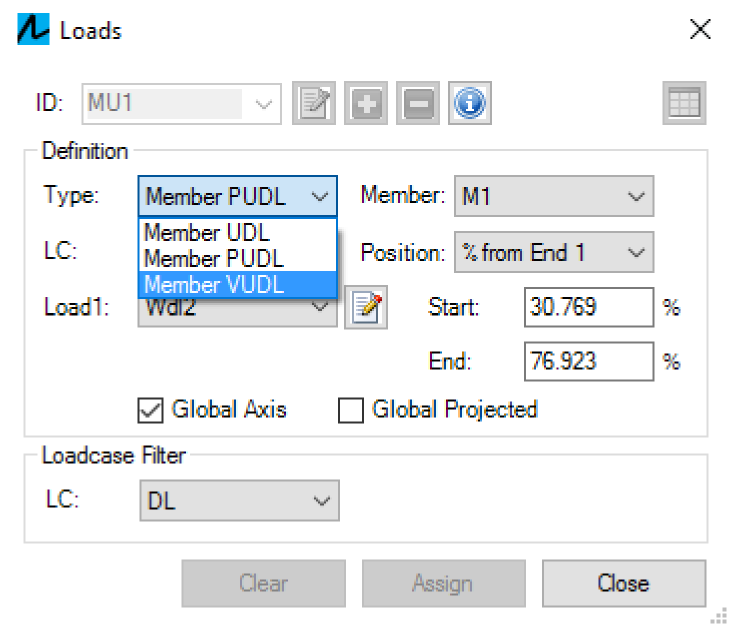


Figure 30 – Member Variable UDL

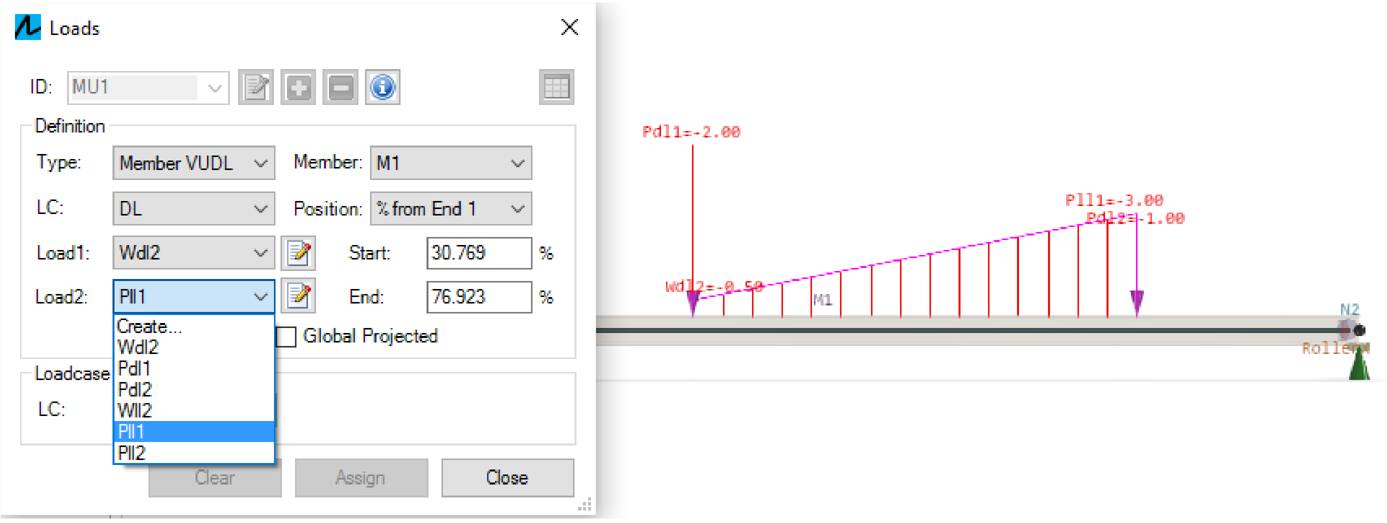


Figure 31 – Load Selection for Member Variable UDL

Change the **LC (Loadcase)** to the **LL (Liveload)** and repeat for the Live load by **changing the Load2 to Pll2**.

Analyse the beam, and turn on all the result cases.

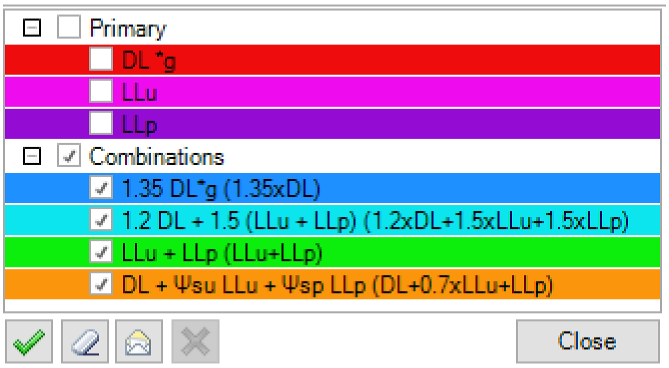


Figure 32 – Analysis Load Combinations

The **maximum bending moment identified is 26.3kNm**.

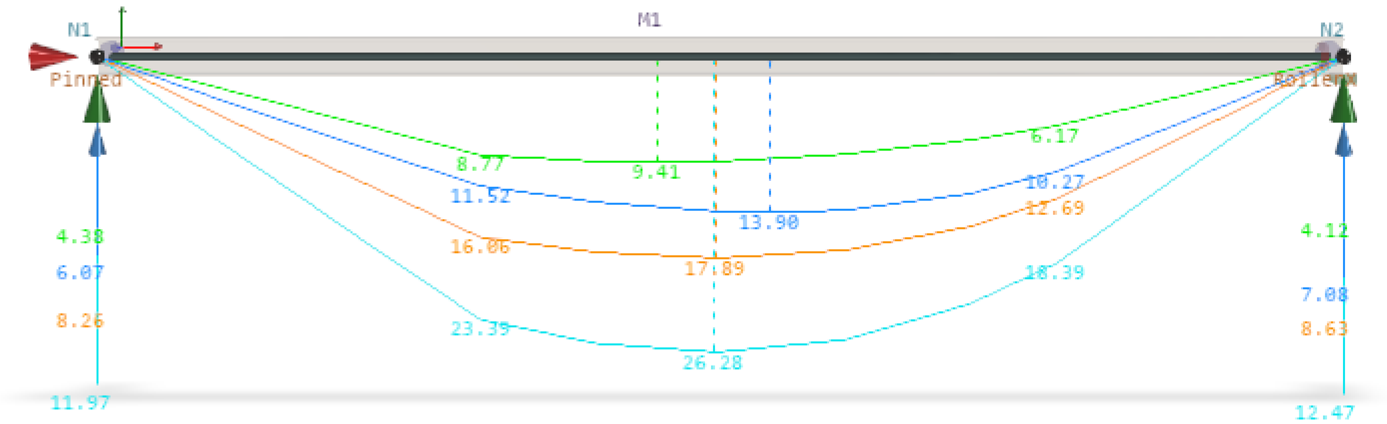


Figure 33 – Revised Bending Moment Diagram

The **maximum shear value identified is 12.50kN**.

Deflections can be checked in the **Structural Toolkit Analysis** itself, while the bending and shear components can be checked in the **Steel Member Design**.

We can modify our existing **Steel Member design** or create a new one.

Change the **Analysis values** within the module to **(M) Manual.**

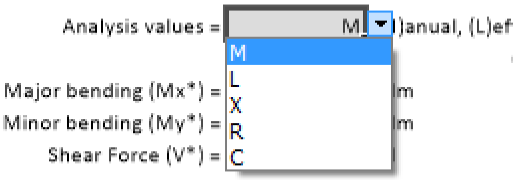


Figure 34 – Steel Member Analysis Values

Input manually the **maximum bending moment of 26.3kNm** and the **maximum shear force of 12.5kN,** noting the αm being recalculated based on ¼ points to 1.17 **.**

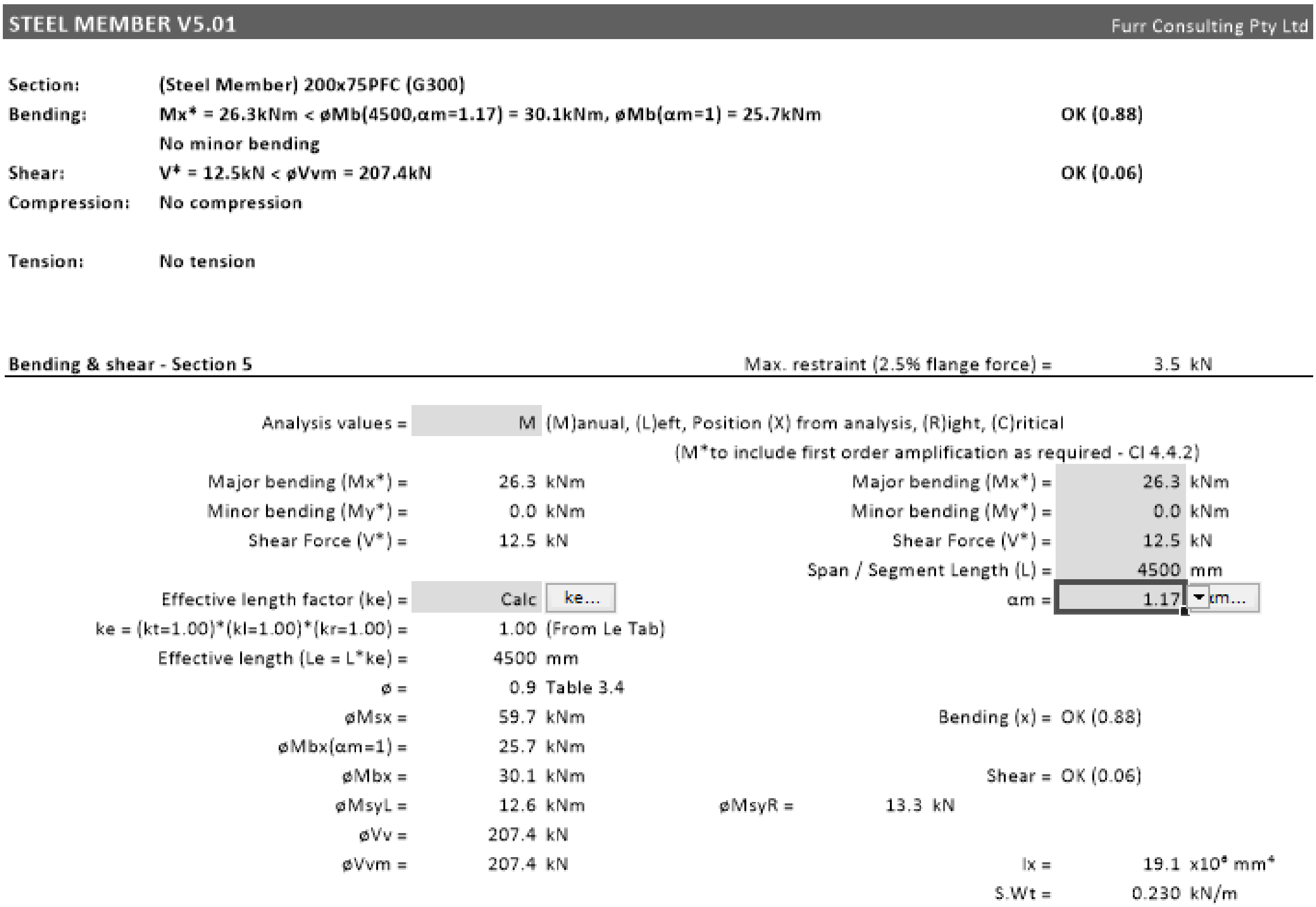


Figure 35 – Revised Steel Member

The beam is satisfactory.

Structural Toolkit

The Essential Design Tool For Australian Structural Engineers

END TUTORIAL

V5.0.1.2

Anthony Furr Software
ABN 74 992 513 430

97 Mt Pleasant Road
Nunawading, Victoria 3131

P 03 9878 4684
F 03 9878 4685

www.structuraltoolkit.com.au
support@structuraltoolkit.com.au