

---

2011

GoBeam User's Guide



Yakov Polyakov  
Spreadsheet Solutions  
12/15/2011

---

## Revision Sheet

Release No.	Date	Revision Description
1.0	12/15/2011	Moving Loads added

---

# GoBeam USER'S GUIDE

## TABLE OF CONTENTS

1. OVERVIEW	
1.1 Introduction	1
1.2 Program Capabilities	1
1.3 Program Limitations	2
1.4 Installation and System Requirements	3
1.5 GoBeam Toolbar	5
2. INPUT DATA	
2.1 Workbook Organization	7
2.2 General Data	7
2.3 Span Data Table	8
2.4 Support Data Table	9
2.5 Loads Data Table	9
2.6 Managing Loads	10
3. AVAILABLE ANALYSIS TYPES	12
4. MOVING LOAD ANALYSIS	
4.1 General	14
4.2 Moving Load Analysis Step-by-Step	14
4.3 Pre-Defined Moving Loads	16
5. SECTION PROPERTIES CALCULATOR	17
6. OUTPUT OF ANALYSIS RESULTS	18
7. STEP-BY-STEP EXAMPLE	19

## 1. OVERVIEW

### 1.1 INTRODUCTION

GoBeam is a MS Excel based software for the structural analysis of continuous beams with or without columns. While creating GoBeam, the author, a practicing structural engineer, had three goals in mind:

- Quick and easy input data allowing rapid, quick, interactive changes
- Features capable of defining and solving real life engineering problems
- Full visual and tabulated output for Moment, Shear, Deflection and Rotation

#### Features

GoBeam is a powerful and easy to use beam analysis program. It requires a minimum amount of effort to enter input data and view the results of the analysis.

GoBeam has been created in Microsoft Excel spreadsheet; however, all calculations are performed using VBA, which works behind the scenes without cluttering the worksheets. The VBA algorithm employs a finite element analysis using the Displacement Method to find forces and displacements in statically indeterminate structures. The analysis results have been rigorously verified and successfully compared to the results of leading commercial finite element analysis software.

### 1.2 PROGRAM CAPABILITIES

#### MODELING

- Supports SI and US customary units
- Capable of analyzing continuous beams with up to 100 spans
- Allows concentrated and linear loads
- Allows free or fixed ends at the extreme left or right outside supports
- Handles knife edge or column type supports
- Capable of elastic supports (vertical springs)
- Allows In-span hinges (moment release)
- Allows variable moment of inertia within a span
- Provides sway frame option (for beams supported on columns)

#### ANALYSIS

1. Linear static analysis
2. Non-linear analysis of beams on elastic foundation (compression only spring supports)
3. Support settlements analysis
4. Computes influence lines of forces, deflections and support reactions
5. Moving load analysis

---

## PRE AND POST-PROCESSING

- Saves and restores load cases
- Combines loads into factored combinations
- Computes maximum/minimum force envelopes
- Allows entering formulas into input data fields
- Convenient tool that calculates and assigns the moment of inertias of typical structural shapes and rolled steel sections
- Computes forces and deflections at any location
- Genuine excel tables allow seamless interface with the user built material design spreadsheets

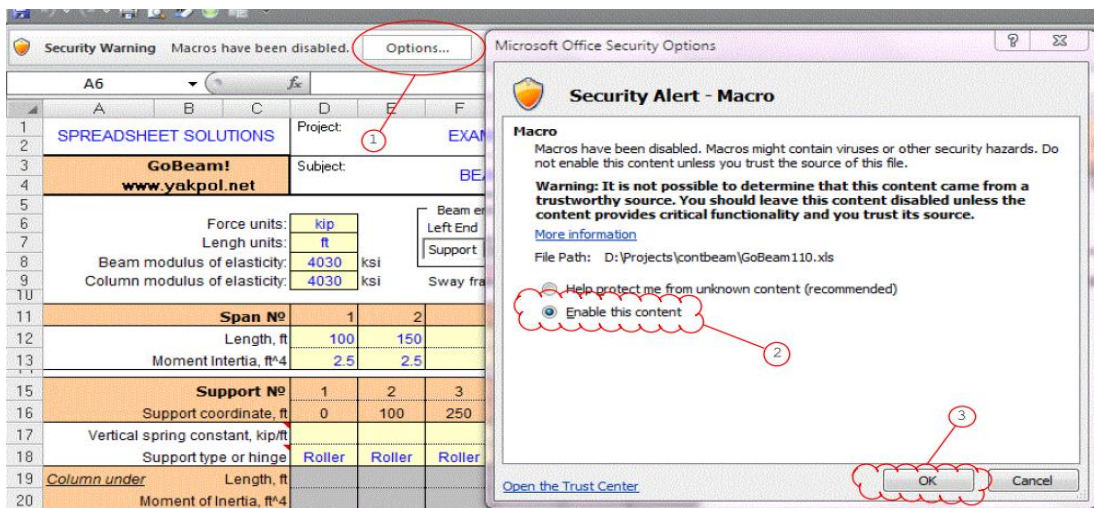
### 1.3 PROGRAM LIMITATIONS

- Maximum number of spans – 100
- Maximum number of line loads in the Load Data Table is unlimited, but only first 100 will be displayed graphically
- Maximum number of load cases stored in database – 20
- Maximum number of load combinations – 10
- Gobeam may not function properly when more than one copy of program is open

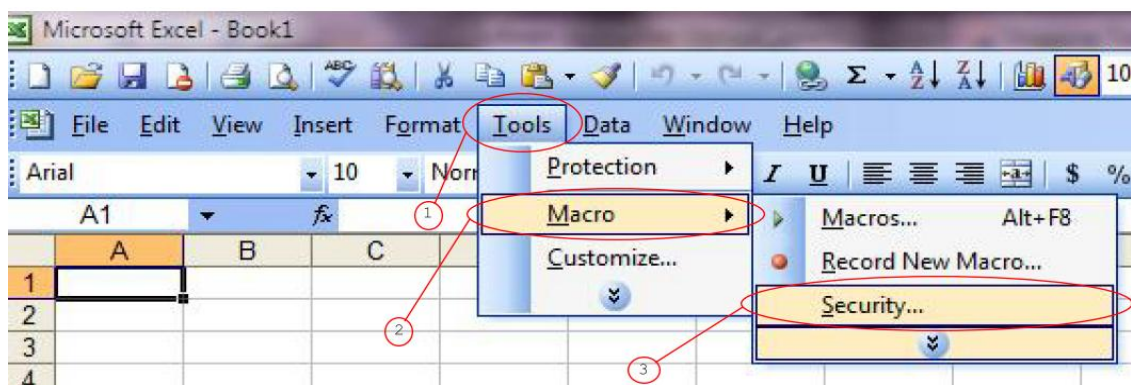
## 1.4 INSTALLATION AND SYSTEM REQUIREMENTS

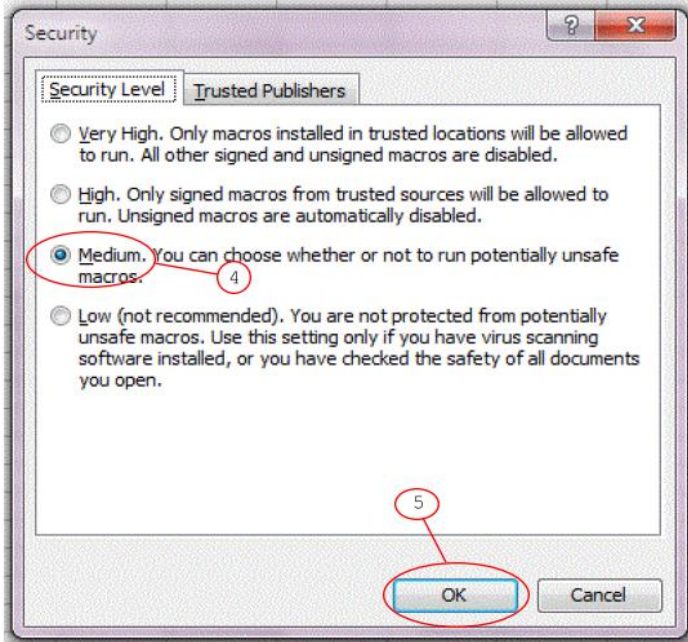
The spreadsheet has been tested to work properly using Microsoft Excel 2003 and 2007 for Windows XP and Windows 7. For compatibility it has been saved in 2003 format with a file extension \*.xls. To install, copy current GoBeam workbook into the working folder on your PC or in a network. If desired, the file can be saved in Excel 2007 format as a macro-enabled workbook (file name extension \*.xlsm).

The Excel settings must be set to enable VBA macros. In Excel 2007, upon opening of the worksheet the Security Warning will appear in the upper left corner of the screen. Mouse click on Options, then select Enable This Content option on Security Alert Macro dialog box, press OK:

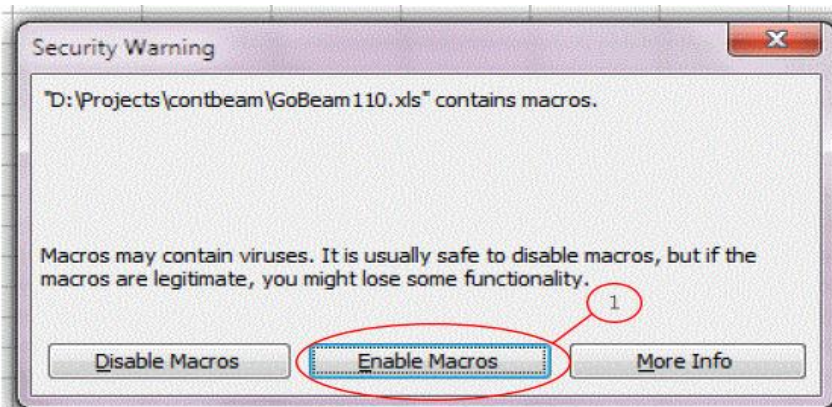


In Excel 2003 set macro security level to medium. Tools/Macros/Security.../Security Level/Medium, press OK:



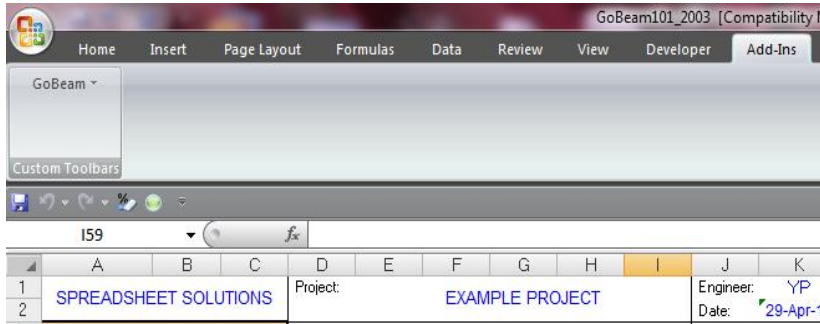


Restart Excel and select Enable Macros option at the prompt:

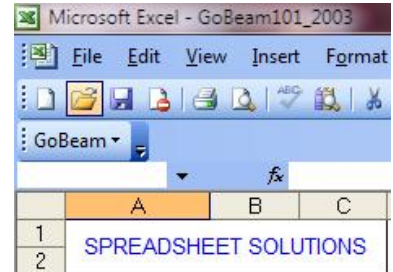


## 1.5 GOBEAM TOOLBAR

Most, but not all commands can be executed using command buttons embedded into the worksheets. The toolbar allows access to all commands and helps navigate through the pages of workbook.

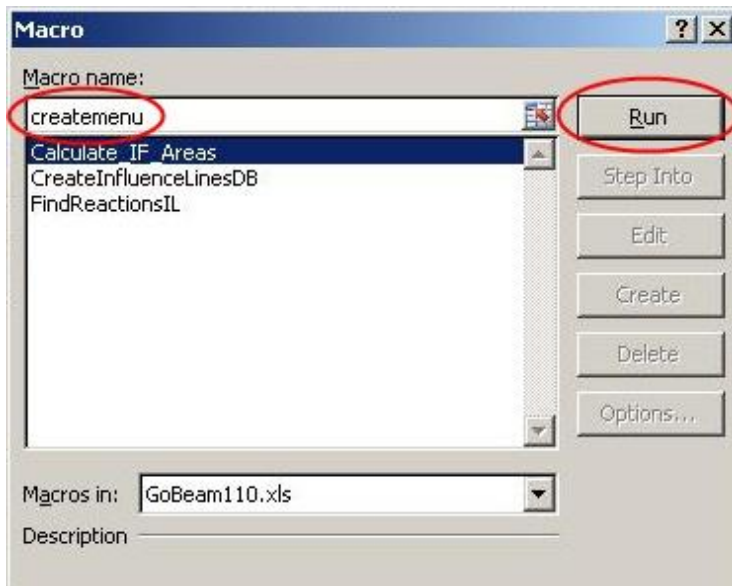


In Excel 2007 the toolbar will be found under the Add-Ins tab of the main menu.



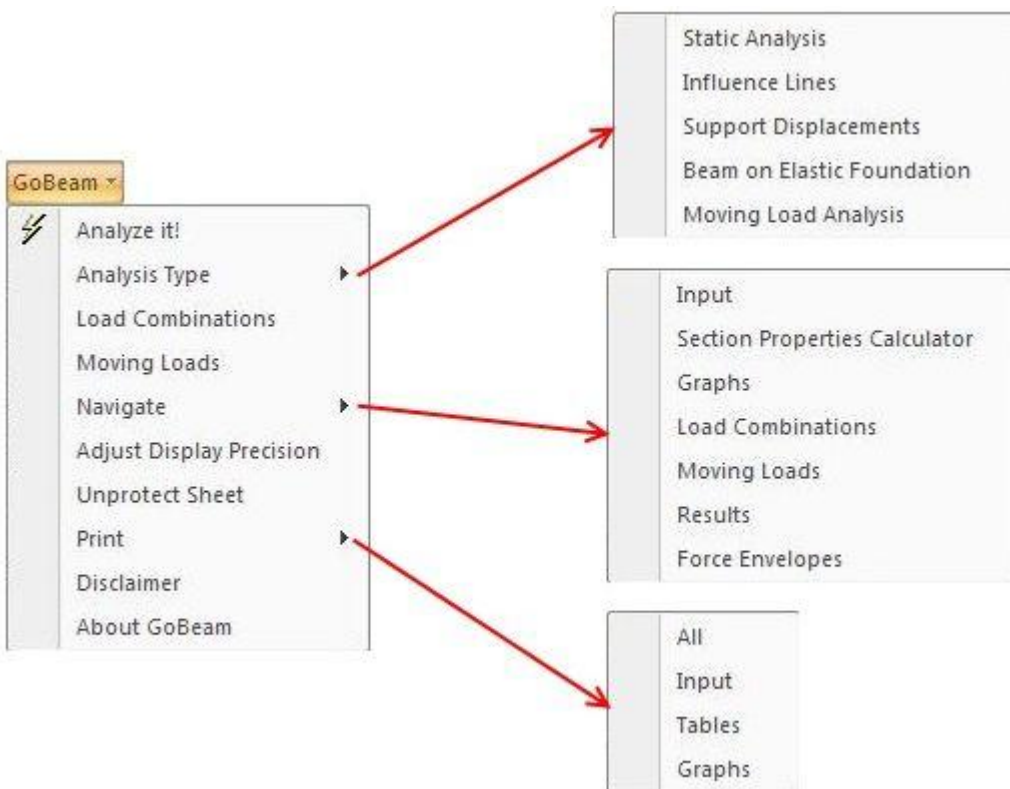
In Excel 2003 the GoBeam tool bar will appear just below the main menu.

To restore the toolbar, open Macro dialog box (2007: Developer/Macros, 2003: Tools/Macro/Macros...), type 'CreateMenu' in the Macro name field and mouse click on 'Run' command button:



## TOOL BAR COMMANDS:

- Analyze it! – executes selected type of analysis
- Analysis Type – selects analysis type
- Load Combinations – opens Load Case Manager dialog box
- Moving Loads – opens Moving Loads dialog box
- Navigate – allows access to workbook pages
- Unprotect Sheet – unprotects active worksheet
- Display Precision – opens dialog box controlling floating point accuracy of the numerical results
- Print – opens the Print dialog box
- Disclaimer – displays a disclaimer
- About GoBeam – displays the license information of the owner and contact details of the developer



GoBeam ToolBar with expanded 'Analysis Type', 'Navigate', and 'Print' buttons:

## 2. INPUT DATA

### 2.1 WORKBOOK ORGANIZATION

Navigating around the spreadsheet:

- Worksheet "Input", enter beam parameters and loads containing the following data tables:
  - Span data table
  - Support data table
  - Load table
- Worksheet "Sections" calculates moment of inertia of various cross-sections and assigns it to spans or to columns.
- Worksheet "Results" contains tabulated results for a single load case static analysis or influence lines.
- Worksheet "Envelopes" contains tables of maximum/minimum force envelopes.
- Worksheet "Graphs" displays force and displacement diagrams.
- Worksheet "LoadComb" is used to define load combinations.
- Worksheet "Trucks" is used to define moving loads.

### 2.2 GENERAL DATA

Force units:   
Length units:   
Beam modulus of elasticity:  ksi  
Column modulus of elasticity:  ksi

Beam end restraints  
Left End:  Right End:   
Sway frame:

Moment Diagram  
 Positive above  
 Positive below

Number of increments per span:   
Number of spans to display:

- Measurement units (Force: kip, lb, kN, N; Length: ft, in, m, mm)  
Select the available US and metric force and length units.
- Modulus of elasticity (ksi, MPa)  
Enter separately the modulus of elasticity for the beam(s) and for the column(s).
- Number of increments per span (1 to 100)  
Minimum recommended values are 8 for static analysis and 20 for moving loads analysis.  
The program will create the specified number of evenly spaced points along the span to report the results. Extra points will be added at the locations of concentrated loads. For spans shorter than half the longest span the number of increments will be reduced proportionally to span length. For better precision of moving load analysis the minimum number of increments is set to 20.
- Number of spans to display  
This option formats spreadsheet to allow entering more or less spans. It does not change the number of spans in the model which is defined by the number of properly entered span lengths.
- Sway frame  
This option is applicable to the beams supported by columns. When selected, the frame analysis will allow for lateral displacements, otherwise the model will be restrained from lateral movements.
- Beam end restraints  
Each end of the beam can be either:

Free – free to move or rotate (typical cantilever end)

Fixed – no rotations allowed. The vertical movement can be allowed by entering spring constant at the vertical spring support line.

Support – either knife edge or column support.

## 2.3 SPAN DATA TABLE

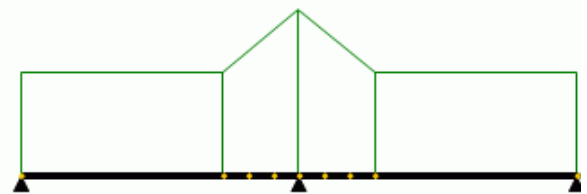
- Length  
The length of each span must be entered in a row of consecutive cells without blanks between. The number of spans will be calculated by the number of cells containing valid span length. To reduce number of spans delete span length values starting from the rightmost column.
- Moment of inertia  
Enter moment of inertia for each span.
- Entering variable moment of inertia within one span.  
Define physical span length as a series of short spans and then set 'Vertical support spring constant' for intermediate supports to zero. Example below presents two-span beam having variable depth near intermediate support. Each physical span is modeled as four spans with different stiffness parameters. Note when support spring constant is set to zero the support symbol on the beam diagram is replaced by the symbol of intermediate node.

Span No	1	2	3	4	5	6	7	8
Length, ft	80	10	10	10	10	10	10	80
Moment Inertia, ft <sup>4</sup>	10	15	28	50	50	28	15	10

Support No	1	2	3	4	5	6	7	8	9
Support coordinate, ft	0	80	90	100	110	120	130	140	220
Vertical spring constant, kip/ft		0	0	0		0	0	0	
Support type or hinge	Roller	Roller	Roller	Roller	Roller	Roller	Roller	Roller	Roller
<u>Column under</u> Length, ft									
Moment of Inertia, ft <sup>4</sup>									
<u>Column above</u> Length, ft									
Moment of Inertia, ft <sup>4</sup>									
Induced support displacements, ft									

Positive direction of loads: ↑ ↺		Loading Title: DL			
Comment	Load Type	WA kip and ft	WB kip/ft	LA ft	LB ft
	Linear	-5	-5	0	80
	Linear	-5	-8	80	110
	Linear	-8	-5	110	140
	Linear	-5	-5	140	220

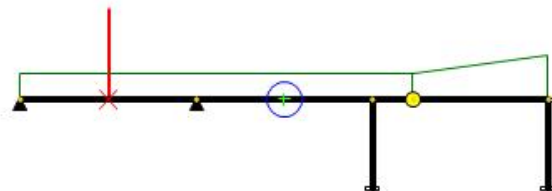


## 2.4 SUPPORT DATA TABLE

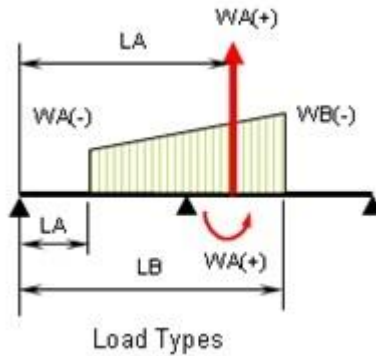
- Support type
  - Roller – knife edge support with vertical restrain only
  - Column – provides vertical and rotational restrains. The spring constant  $KM_z = 4EI/L$  [force\*length/radians], where E – modulus of elasticity, I – moment of inertia, and L – length of the column. Column can be specified below or/and above the beam.
  - Hinge – creates moment release in the beam. Does not provide any support.
- Vertical spring constant  
Vertical elastic spring can be assigned to roller and column support type or to fixed beam end. Data options:
  - No data in this field corresponds to infinitely stiff vertical support.
  - Zero value completely releases vertical restraint at the joint. This option is handy to specify variable moment of inertia within a span.
  - Positive value [Force/Length] creates elastic spring at the joint.
- Induced support displacements  
This line of data is available only for 'Support Displacements' type of analysis. Blank cell will be read as zero value.

## 2.5 LOADS DATA TABLE

Positive direction of loads: ↑ ↺		Loading Title:		DL	
Comment	Load Type	WA kip and ft	WB kip/ft	LA ft	LB ft
	Linear	-3	-3	0	67
	Linear	-3	-5	67	90
	P	-35		15	
	M	56		45	



- Loading Title is an identifier under which load case will be saved in the database.
- Entering load case data  
Each line of data describes a single load. The number of loads is unlimited but only first 100 will be graphically displayed on girder diagram. Empty rows are allowed, the rows with incomplete data are ignored.
- Load Types  
GoBeam permits three load types:
  1. Linear – Linearly distributed vertical load. Parameters:  
WA, WB – load intensity [force/length] on the left and right end respectively. Positive is when acting upward.  
LA, LB – distance from the left end of the beam to the left and right end of load application.
  2. P – Concentrated vertical force. Parameters:  
WA – load intensity [force]. Positive is when acting upward.  
LA – distance from the left end of the beam to the load application.
  3. M – Concentrated moment. Parameters:  
WA – load intensity [force\*length]. Positive is when acting counter clockwise.  
LA – distance from the left end of the beam to the load application.



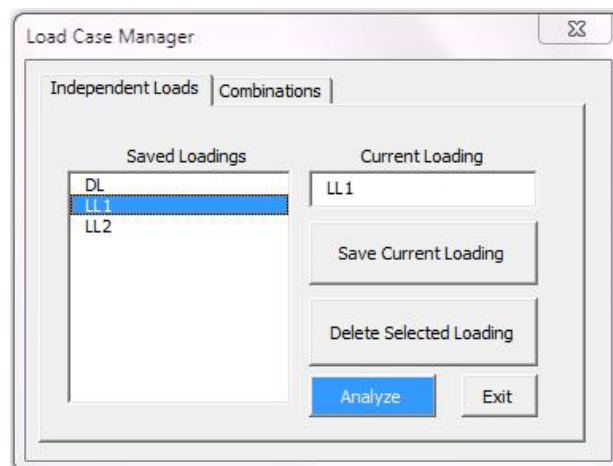
## 2.6 MANAGING LOADS

The load data table can be saved into the database as an independent load case. Later saved load cases can be assembled into the load combinations, and force envelopes. Up to ten load combinations can be specified.

### MANAGING INDEPENDENT LOAD CASES

Open 'Load Case Manager' dialog box by using toolbar (GoBeam/Load Combinations) or by 'Load Combinations' command button on worksheet 'Input'. Select tab Independent Loads.

- To save current (displayed on worksheet 'Input') load case into the database click on command button 'Save current loading'. The load case will be saved under the name of its Title. Enter a new name in the 'Loading Name' text box if desired. If load case with this name already exists in the database it will be overwritten.
- To restore a saved load case, select its name from the list of Saved Loadings and click on the command button 'Make Selected Loading Current'. The load case will be restored in the load data table.
- To delete load case from database select its name from the list and execute command button 'Delete Selected Loading'.

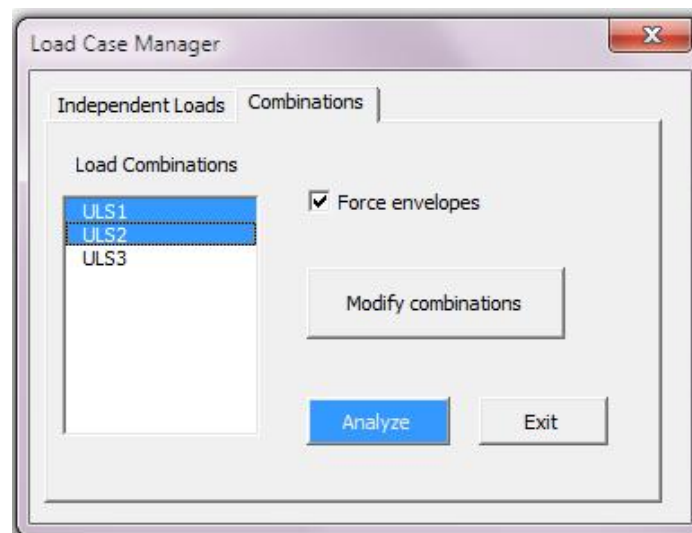


### LOAD COMBINATIONS

- Defining load combinations  
Prior to defining load combinations contributing load cases shall be saved into the database. The load combinations will be defined in the table located in the worksheet 'Load Combinations'. Left column displays the names of saved independent load cases. The names of load combinations shall be entered in the top row of the table. The factors for load cases will reside in the body of the table.

Independent Loading		LOAD COMBINATIONS AND LOAD FACTORS					
Nº	Name	SLS1	SLS2	SLS3	ULS3		
1	DL	1	1	1	1.25		
2	LL1	1	0	1	1.4		
3	LL2	0	1	1	1.4		

- Analyzing load combinations  
Open 'Load Case Manager', tab 'Combinations'. Select pre-defined combination from the list and click on 'Convert into loading' command button. The command will create a load case consisting of all load cases in the combination multiplied by appropriate load factors. The load case will be displayed in the main load data table. It can be treated as independent load case, analyzed, modified and saved.

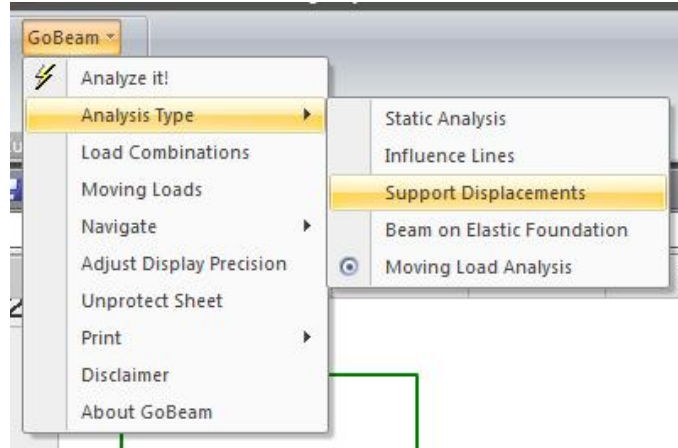
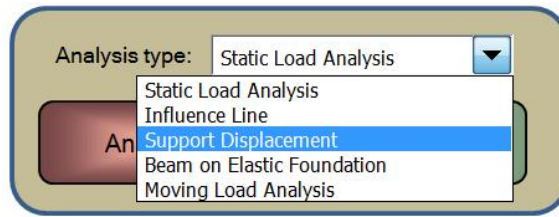


## FORCE ENVELOPES OF SEVERAL LOAD COMBINATIONS

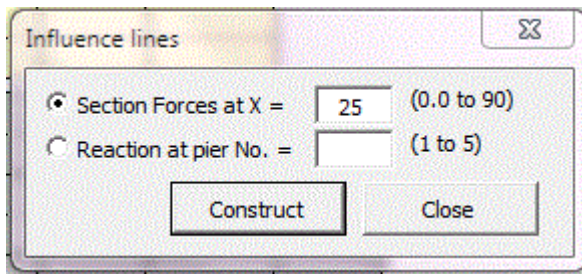
Open 'Load Case Manager' dialog box, tab 'Envelopes'. Select pre-defined load combinations from the list and execute 'Analyze' command button. One by one GoBeam will analyze all selected load combinations and will compute minimum and maximum force envelopes. The results will be displayed on worksheets 'Graphs' and 'Envelopes'.

### 3. AVAILABLE ANALYSIS TYPES

The type of analysis can be selected from GoBeam toolbar or from the list box on worksheet 'Input'.



- Static analysis  
Linear static analysis will be performed for a single load case or for a number of load combinations if force envelopes option is selected (see previous chapter).
- Influence lines  
When this option is selected, the dialog box will prompt to specify section along the beam or a support number for which to construct influence lines of forces and displacements.



- Support displacements  
This option will compute forces in the beam induced by vertical displacements at the support nodes. The displacements shall be entered in the support data table in the row 'Induced support displacements'.  
When support displacements option is selected, the span loads will be ignored. The support displacement cannot be a part of static load combinations.

<b>Support No</b>	1	2	3	4	5
Support coordinate, ft	0	30	60	67	90
Vertical spring constant, kip/ft					
Support type or hinge	Column	Roller	Roller	Hinge	Roller
<i>Column under</i> Length, ft	10				
Moment of Inertia, ft <sup>4</sup>	2				
<i>Column above</i> Length, ft					
Moment of Inertia, ft <sup>4</sup>					
Induced support displacements, ft	0.08		0.08		

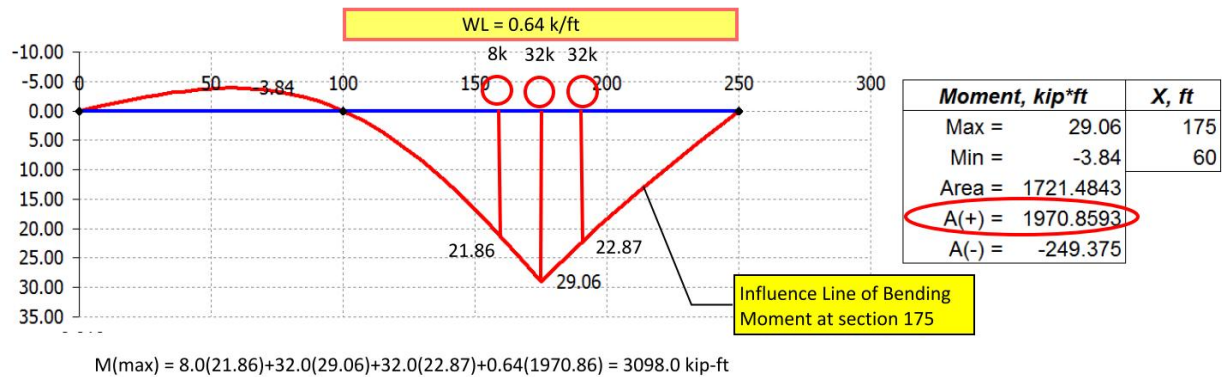
- Beam on elastic foundation  
This option will perform non-linear analysis assuming only positive or zero vertical reactions at ALL supports. This type of analysis is common for grade beams and line footings.
- Moving load analysis (refer to the following chapter)

## 4. MOVING LOAD ANALYSIS

### 4.1 GENERAL

The moving loads simulate highway and railroad vehicles. Typically the design codes describe moving load as a combination of truck and/or lane loads. The truck load comprises a sequence of liberally spaced axle (point) loads which can be placed at any location along the beam in order to produce extreme forces. The lane load is a uniform distributed load patterned in a way to contribute to the extreme forces.

Gobeam implements influence line based moving load analysis. First it creates force influence lines at each section, analyzes them, and then places moving load in the most unfavorable location. The force due to truck is calculated as  $F_{Truck} = \sum P_i y_i$ , where  $P_i$  is a weight of truck  $i^{th}$  axle and  $y_i$  is a value of influence line below that axle. The force due to lane load is calculated as  $F_{Lane} = A_o w_L$ , where  $A_o$  is negative or positive area of influence line and  $w_L$  is the lane load intensity. The figure below demonstrates sample calculations of maximum bending moment in the midspan section of two-span beam.



Note that precision of computations depends on the number of increments within a beam span (minimum 20 recommended for moving load analysis).

### 4.2 MOVING LOAD ANALYSIS STEP-BY-STEP.

- Step 1. Define beam by filling out span and support data tables.
- Step 2. Define moving load.

The moving load (combination of truck and lane) must be saved in the database. Access worksheet 'Trucks' (Gobeam/Navigate/Moving Loads) and enter moving load parameters as demonstrated below.

**Definition of Moving Load (combination of truck and lane loadings)**

Title: **hypotetical moving load** Name stored in database

Description: **HS20-44 Truck**

Input Units:  kip, ft  kN, m Moving load design units are independent from global beam units

**Truck parameters**

Neglect axles not contributing to extreme force

Variable distance between axles:  and

Maximum distance:  ft

Truck load coefficient:

Number of axles:

Nominal weight:  kip

**Lane parameters**

No restrictions Independent from truck placement

Exclude within truck limits Partial lane load (no lane within truck limits)

Trailing behind truck

Last axle to lane dist.:  ft

Load intensity:  kip/ft

Lane load coefficient:  Buffer between lane and truck axle (for partial lane only)

Axle No	Weight kip	Spacing ft
1	8	14
2	32	14
3	32	
4		
5		
6		
7		
8		
9		
10		
11		

Variable axle spacing (e.g. 14 to 30 feet). Note different font.

e.g. impact factor

Maximum 30 axles

Step 3. Save load into the database by executing command button 'Save displayed load'. The new load title will appear on the first line of 'Predefined loads' list box.

**Predefined loads**

- hypotetical moving load
- HS20-44 (AASHTO STD)
- T44 (AS 5100.7)
- HLP 400 (M<0) (AS5100)
- HLP 400 (AS5100.7)
- M1600 (AS 5100.2)
- S1600 (AS 5100.2)
- CL-W Lane (CAN/CSA)
- CL-625 Truck (CAN/CSA)
- Cooper E 80 (AREMA)

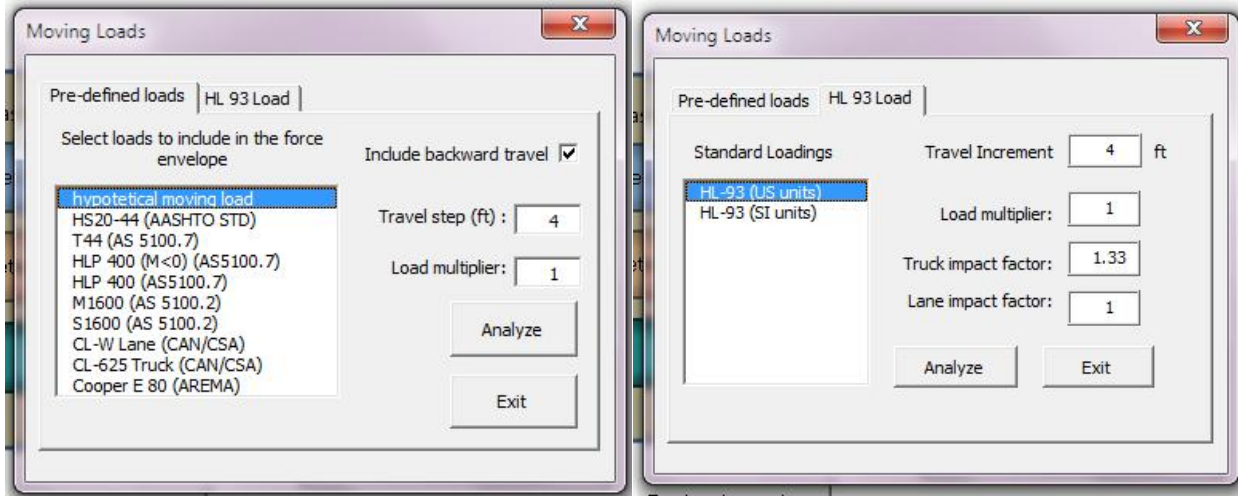
**Database commands**

Save displayed load

Delete selected load

Run !

Step 4. Open 'Moving Loads' dialog box (Gobeam/Moving Loads or command button 'Run'). Select 'Predefined loads' tab, pick desired moving load from the list, enter values for 'travel step' and 'load multiplier' and click on 'Analyze' button to run analysis. For HL93 select 'HL 93 Load' tab, enter relevant parameters and click on 'Analyze' button. The progress bar will monitor processes. When analysis is finished the program will jump back on 'Input' page.



Step 5. Observe force envelopes on pages Input, Graphs and Envelope.

### 4.3 PRE-DEFINED MOVING LOADS

Gobeam comes with the database of moving loads from several international design codes. To examine load data double click on the appropriate line of list box on page 'Trucks' (see previous page). The loads can be modified and saved under the same or a different name.

## 5. SECTION PROPERTIES CALCULATOR

The section properties calculator is located on the worksheet "Sections" which can be accessed directly from Excel tabs, GoBeam/Navigate/Section Properties Calculator menu or by a co-named command button on "Input" worksheet. The dialog allows to define typical beam cross-sections or to select structural rolled steel sections from a few international databases. The figure below shows suggested order of making selections and entering data into this worksheet.

**SECTION PROPERTIES CALCULATOR**

1 Assign properties to:  
 Spans  
 Columns

2 Select Spans: 1, 2, 3  
 All

3 Section type:  
Rectangular  
T-section  
I-section  
Box  
Circular  
Pipe  
Structural steel sections

4 Section dimensions:  
Units: in  
Depth, H = 24  
Web width, Bw = 12  
Slab width, Bs = 68  
Slab thickness, Ts = 6

5

6 Section Properties:  
H = 24 in  
yb = 16.846 in  
A = 624 in<sup>2</sup>  
Ix = 27393 in<sup>4</sup>  
Iy = 159808 in<sup>4</sup>

7 Bending axis: X  
Multiplier: 0.5

8

Diagram: T-section with dimensions H, Bs, Bw, Y, X, ts.

Assign Properties  
Back to Input sheet

**Moment of Inertia in design units: 0.66052 ft<sup>4</sup>**

## 6. OUTPUT OF ANALYSIS RESULTS

### GRAPHICAL OUTPUT

The diagrams of beam forces and displacements due to single load case or maximum/ minimum force envelopes can be viewed on worksheet 'Graphs'. The worksheet can be accessed through GoBeam toolbar: Add-ins/GoBeam/Navigate/Graphs.

### TABULATED OUTPUT

- The tabulated results of beam forces and displacements due to static analysis and influence line analysis are located on worksheet 'Results'. The force envelopes of load combinations or moving load analysis are located on page 'Envelope'.
- ANALYSIS RESULTS table lists the values for shear force, bending moment, deflections and rotations at the number of increments within each span specified on worksheet 'Input'.
- ANALYSIS RESULTS AT CUSTOM LOCATIONS table lists forces at the user specified locations entered in the left column. The values at custom locations are calculated by linear interpolation between the values in the ANALYSIS RESULTS table. The section locations can be changed at any time followed by executing "click here to recalculate" command button positioned above the table.
- REACTIONS table lists support reactions and support displacements.
- EXTREME VALUES table lists maximum and minimum forces and displacements in the beam. Maximum/minimum areas of force diagrams are relevant when Influence Lines type of analysis is selected.

## 7. STEP-BY-STEP EXAMPLE

This example demonstrates analysis of a 3-span reinforced concrete building frame for combinations of dead and live loads. It shows in detail how to build the model, to analyze it for a single load, to create load combinations and to compute envelopes of maximum/minimum forces.

Problem:

Find maximum/minimum factored forces in reinforced concrete continuous beam due to permanent uniformly distributed load (1.4 kip/ft) and patterned distributed live load (0.6 kip/ft).

Parameters:

- Spans 28'+24'+28'
- Beam is built integrally with columns (top and bottom)
- Beam is restrained from horizontal displacements
- Load combination 1.2D+1.6L
- Beam effective cross-section: T-section,  $h = 24"$ ,  $b_w = 12"$ ,  $b_e = 68"$ ,  $t_s = 6"$
- Columns are 11 feet long, 18"x18"
- Use gross moment of inertia for columns and cracked moment of inertia for beams ( $I_{cr}=0.5I_g$ )
- Modulus of elasticity for beams and columns  $E = 3600$  ksi

Step-by-step instructions

- Step 1. Enter beam geometry (refer to Figure 1)
1. Select design units (kip and ft).
  2. Enter modulus elasticity for beam and columns (3600 ksi).
  3. Set end restraints for left and right ends of the beam to type '*Support*'.
  4. Enter length and moment of inertia of each span ( $I=1.0$  for now).
  5. Select 'Column' type for all 4 supports.
  6. Enter column length = 11' for 'columns under' and 'columns above'. For now, set moment of inertia for all spans and columns to 1.0. Correct stiffness will be calculated in Step 2.

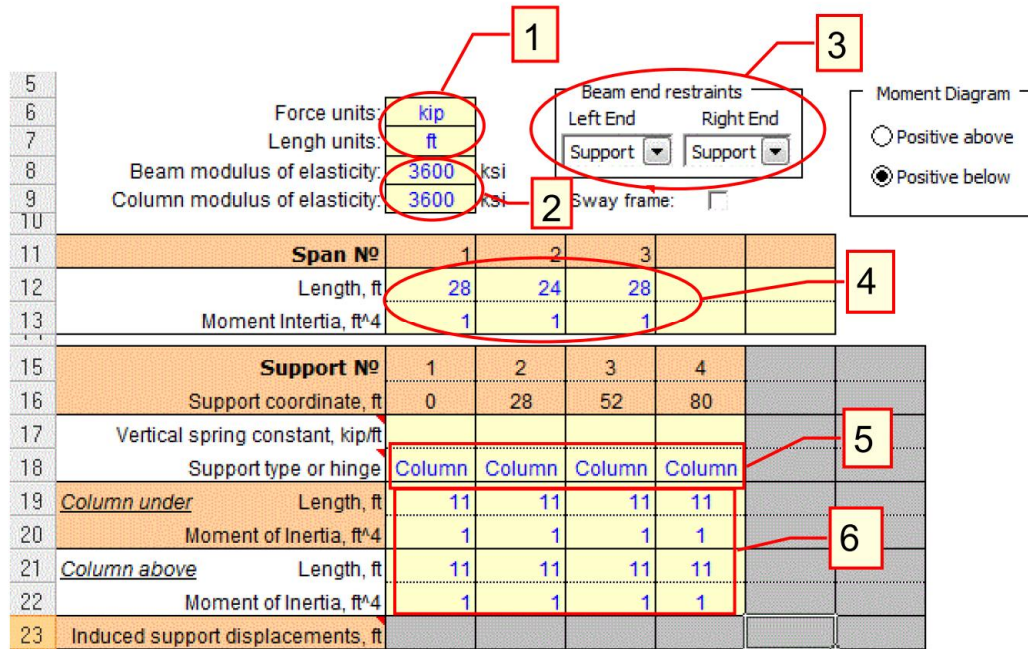


FIGURE 1. WORKSHEET INPUT: SPAN AND SUPPORT DATA TABLES

Step 2. Calculate and assign section properties for beams and columns

Access worksheet 'SECTIONS' (by selecting sheet tab 'SECTIONS', or by clicking on command button 'SECTION PROPERTIES', or using menu: ADD-INS/GOBEAM/NAVIGATE/SECTION PROPERTIES CALCULATOR).

- Set option button 'Assign properties to spans'. Select all spans using check box.
- Make proper selections and fill out data fields in the order shown on Figure 2. Note that multiplier 0.5 accounts for cracked moment of inertia ( $I = 0.5I_g$ ). Click on the command button 'Assign Properties' on the bottom of the page to transfer calculated values of moment of inertia into Span Data table.
- Set option button 'ASSIGN PROPERTIES TO COLUMNS'. Similarly fill out necessary data (see Figure 3) and execute command button 'Assign Properties' to transfer values of column moment of inertia into Support Data table.
- Click on command button 'Back to Input Sheet'. The data tables and model on worksheet 'Input' shown on Figure 4.

## SECTION PROPERTIES CALCULATOR

Assign properties to:

Spans

Columns

Select Spans

All

1

2

3

Section type

Rectangular

T-section

I-section

Box

Circular

Pipe

Structural steel sections

Section dimensions

Units:

Depth, H =

Web width, Bw =

Slab width, Bs =

Slab thickness, Ts =

Section Properties

H = 24 in

yb = 16.846 in

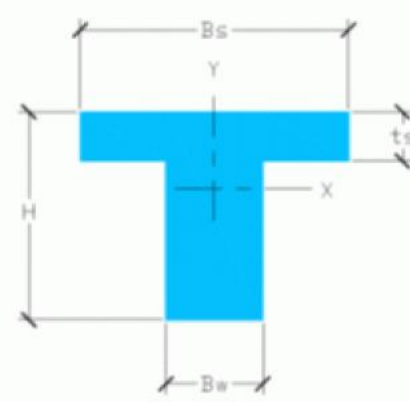
A = 624 in<sup>2</sup>

Ix = 27393 in<sup>4</sup>

Iy = 159808 in<sup>4</sup>

Bending axis:

Multiplier:



**Moment of Inertia in design units: 0.66052 ft<sup>4</sup>**

FIGURE 2. WORKSHEET SECTIONS: CALCULATING BEAM MOMENT OF INERTIA.

## SECTION PROPERTIES CALCULATOR

Assign properties to:

Spans

Columns

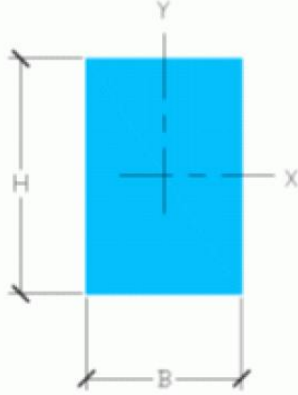
Select Columns  All

1
2
3
4

Section type

Rectangular  
 T-section  
 I-section  
 Box  
 Circular  
 Pipe  
 Structural steel sections

Section dimensions	
Units:	<input type="text" value="in"/>
Depth, H =	<input type="text" value="18"/>
Width, B =	<input type="text" value="18"/>
	<input type="text" value="68"/>
	<input type="text" value="6"/>
	<input type="text" value="0.3"/>
	<input type="text" value="1"/>



Section Properties	
H =	18 in
y <sub>b</sub> =	9 in
A =	324 in <sup>2</sup>
I <sub>x</sub> =	8748 in <sup>4</sup>
I <sub>y</sub> =	8748 in <sup>4</sup>

Bending axis:

Multiplier:

Assign Properties

Back to Input sheet

Moment of Inertia in design units: 0.42188 ft<sup>4</sup>

FIGURE 3. WORKSHEET SECTIONS: CALCULATING COLUMN MOMENT OF INERTIA.

Span №		1	2	3		
Length, ft		28	24	28		
Moment Inertia, ft <sup>4</sup>		0.66052	0.66052	0.66052		
Support №		1	2	3	4	
Support coordinate, ft		0	28	52	80	
Vertical spring constant, kip/ft						
Support type or hinge		Column	Column	Column	Column	
<i>Column under</i>	Length, ft	11	11	11	11	
	Moment of Inertia, ft <sup>4</sup>	0.42188	0.42188	0.42188	0.42188	
<i>Column above</i>	Length, ft	11	11	11	11	
	Moment of Inertia, ft <sup>4</sup>	0.42188	0.42188	0.42188	0.42188	
Induced support displacements, ft						

Positive loads: ↑ ↻		Load Case:			
Comment	Load Type	WA kip and ft	WB kip/ft	LA ft	LB ft

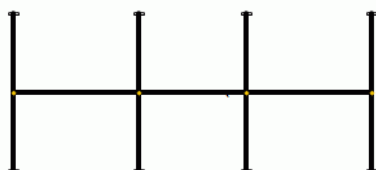
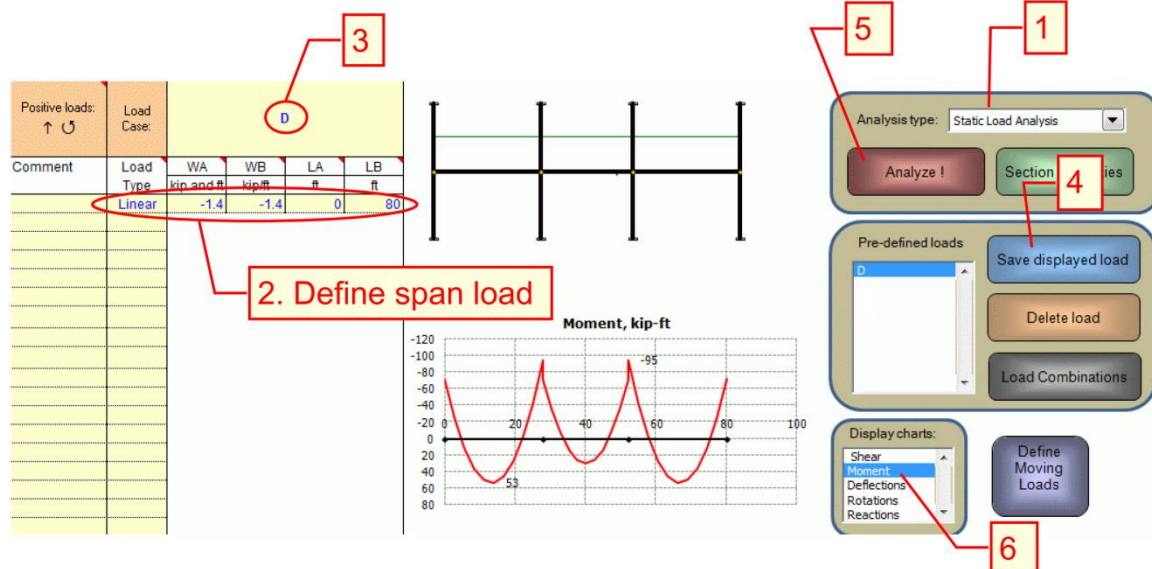


FIGURE 4. WORKSHEET INPUT: COMPLETED SPAN AND SUPPORT DATA TABLES.

Step 3. Define independent load cases

① Set analysis type to 'Static Load Analysis'.

Now we will define four load cases named D, L1, L2 and L3. The first (D) will be a uniformly distributed dead load (Load Type = Linear) of 1.4 k/ft magnitude (WA = WB = - 1.4) extending over the entire length of the beam (from LA = 0 to LB = 80). ② Enter span load data (it takes a single row in the Load Table). ③ Enter 'D' for Load Case Name and ④ click on 'SAVE DISPLAYED LOAD' command button to save it into the database. ⑤ Analyze the structure by executing 'ANALYZE!' button. ⑥ Select a type of force to display a chart of.



3

D

2. Define span load

5

1

4

6

FIGURE 5. WORKSHEET INPUT: INDEPENDENT LOAD CASE 'D'.

Similarly define and save load cases L1, L2 and L3 as shown on Figures 6, 7 and 9 respectively.

Positive loads: ↑ ↻	Load Case:	L1			
Comment	Load Type	WA kip and ft	WB kip/ft	LA ft	LB ft
	Linear	-0.6	-0.6	0	28
	Linear	-0.6	-0.6	52	80

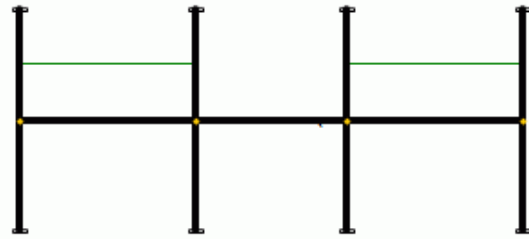


FIGURE 6. LOAD CASE 'L1'.

Positive loads: ↑ ↻	Load Case:	L2			
Comment	Load Type	WA kip and ft	WB kip/ft	LA ft	LB ft
	Linear	-0.6	-0.6	0	52

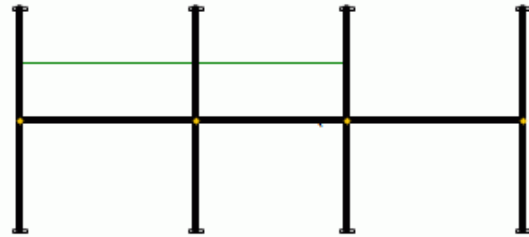


FIGURE 7. LOAD CASE 'L2'.

Positive loads: ↑ ↻	Load Case:	L3			
Comment	Load Type	WA kip and ft	WB kip/ft	LA ft	LB ft
	Linear	-0.6	-0.6	28	52

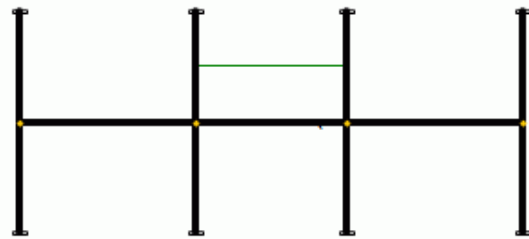


FIGURE 8. LOAD CASE 'L3'.

Step 4. Define load combinations

With all independent loads stored we will define load combinations. The load combination table is located on worksheet 'LoadComb' which can be accessed via main menu (Add-Ins/GoBeam/Navigate/Load Combinations), using LOAD COMBINATIONS command button (Load Combinations/Modify Combinations), or using Excel page tabs on the bottom of the screen. We'll define three load combinations:

$$ULS1 = 1.2D + 1.6L1$$

$$ULS2 = 1.2D + 1.6L2$$

$$ULS3 = 1.2D + 1.6L3$$

Type in names of load combinations in the first row of the table and enter corresponding load factors of the independent load cases listed on the left. Return to Input worksheet.

Independent Loading		LOAD COMBINATIONS AND LOAD FACTORS			
No	Name	ULS1	ULS2	ULS3	
1	D	1.2	1.2	1.2	
2	L1	1.6			
3	L2		1.6		
4	L3			1.6	

FIGURE 9. WORKSHEET LOADCOMB: LOAD COMBINATIONS TABLE.

Step 5. Run analysis and review results

Open Load Combinations dialog box using command button Load Combinations or via main menu: Add-Ins/Load Combinations. Select tab Combinations. Click on one of the load combinations in the list box. The combination will be converted into independent load case and displayed in Load Table and on the graph. Click on Analyze command button to perform static analysis for single load combination and observe charted results.

Step 6. Calculate force envelopes

Open 'Load Case Manager' dialog box by clicking on command button 'Load Combinations' and select tab 'Combinations'. Select 'Force Envelopes' check box. It will let you select multiple load combinations to be analyzed simultaneously. The resulting forces will be organized into maximum and minimum force envelopes. Select all three load combinations and click 'Analyze' button. Wait until analysis completed, it may take a few seconds.

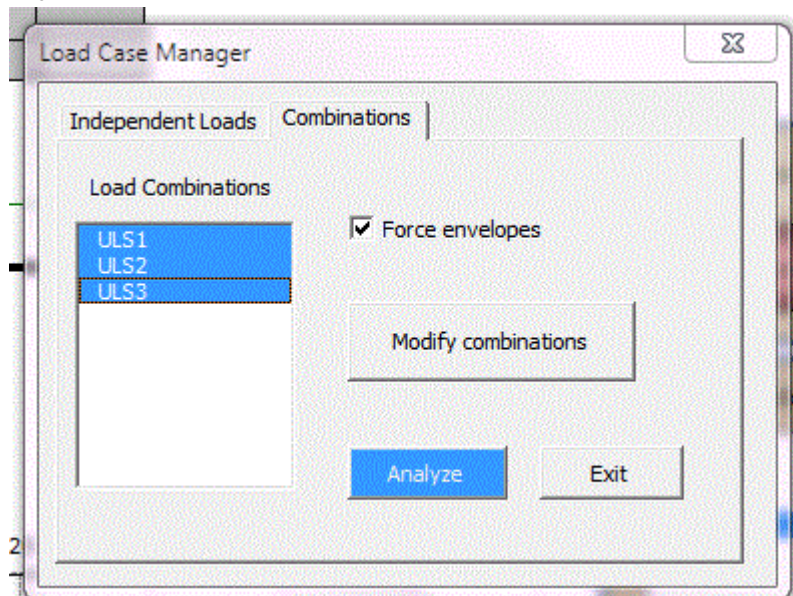


FIGURE 10. LOAD CASE MANAGER DIALOG BOX.

Observe force envelope diagrams on the chart. For complete graphical results access worksheet 'Graphs' (Add-Ins/GoBeam/Navigate/Graphs), for tabulated results access worksheet 'Envelope' (Add-Ins/GoBeam/Navigate/Force Envelopes).

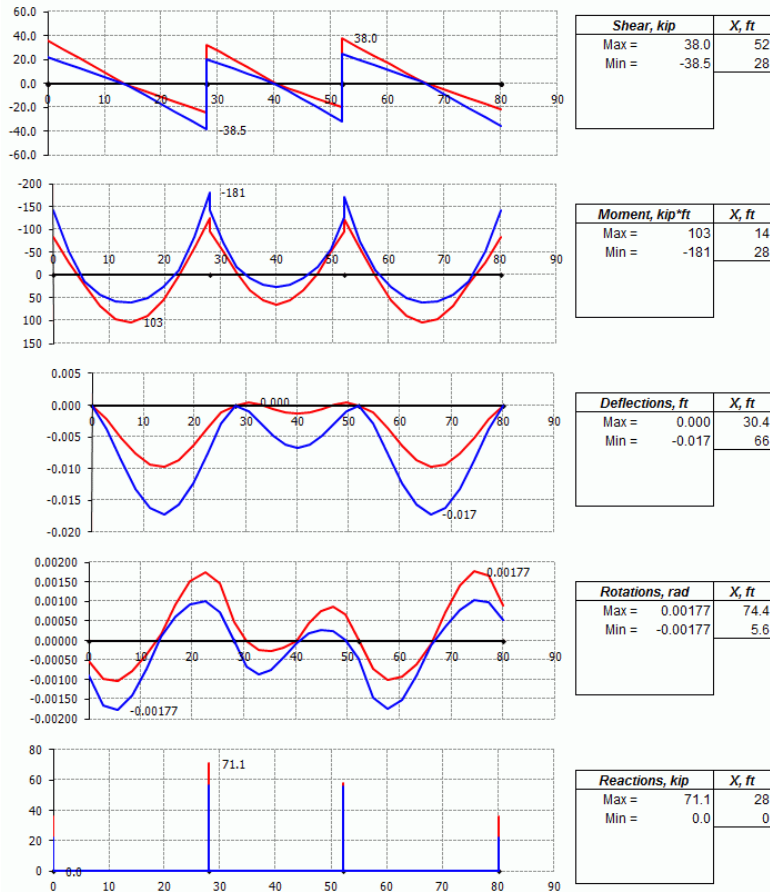


FIGURE 11. WORKSHEET GRAPHS: MAXIMUM/MINIMUM FORCE ENVELOPES.

MAXIMUM ENVELOPE (kip, ft, radians)					MINIMUM ENVELOPE (kip, ft, radians)				
X, ft	Shear	Moment	Deflection s	Rotations	X, ft	Shear	Moment	Deflection s	Rotations
0	35.9	-84	0.000	-0.00053	0	22.1	-141	0.000	-0.00089
1E-06	35.9	-84	0.000	-0.00053	1E-06	22.1	-141	0.000	-0.00089
2.8	28.5	-28	-0.002	-0.00097	2.8	17.4	-51	-0.004	-0.00166
5.6	21.1	20	-0.005	-0.00103	5.6	12.7	14	-0.009	-0.00177
8.4	13.7	68	-0.008	-0.00079	8.4	8.0	43	-0.013	-0.00141
11.2	6.3	96	-0.009	-0.00036	11.2	3.3	58	-0.016	-0.00072
14	-1.1	103	-0.010	0.00015	14	-1.6	61	-0.017	0.00010
16.8	-6.1	90	-0.009	0.00092	16.8	-9.0	51	-0.016	0.00060
19.6	-10.8	56	-0.006	0.00151	19.6	-16.4	27	-0.012	0.00093
22.4	-15.5	1	-0.004	0.00176	22.4	-23.8	-10	-0.008	0.00100
25.2	-20.2	-60	-0.001	0.00147	25.2	-31.2	-84	-0.003	0.00072
28	-24.9	-123	0.000	0.00048	28	-38.5	-181	0.000	-0.00002
28	-24.9	-123	0.000	0.00048	28	-38.5	-181	0.000	-0.00002
28	32.5	-94	0.000	0.00048	28	20.2	-141	0.000	-0.00002
30.4	26.2	-51	0.000	-0.00002	30.4	16.1	-70	-0.001	-0.00065
32.8	19.9	-5	0.000	-0.00025	32.8	12.1	-17	-0.003	-0.00086

FIGURE 12. WORKSHEET ENVELOPE: FORCE ENVELOPES AT DEFAULT LOCATIONS.

