Multi-loaded Multi-span Beam Module Walk-Through

To begin sizing a Multi-loaded Multi-span Beam in StruCalc you will need to open the Multi-loaded Multi-span Beam design module. This is done by one of three ways. The first is by going to the top tool bar and clicking Design Modules, then moving your mouse down to Beam Design, and finally moving your mouse over and clicking on Multi-loaded Multi-span Beam (see Figure 1).

The second way to open the roof beam module is by clicking on the Multi-loaded Multi-span Beam button located on the bottom tool bar of StruCalc (see Figure 2).

The third and final way to open the Multi-loaded Multi-span Beam module is by holding down the “Shift” key and hitting the “F5” key.

Once the module is open there is some basic information that is needed before a suitable beam size can be determined. Most of this information is entered in the middle portion of the page which is called the module (see Figure 3) and the rest is entered on the left hand side of the screen (see Figure 5 and Figure 6), under what is called the Navigation Tool Bar. In any module of StruCalc the most efficient way to do the design is to work from the top of the screen down. To begin the discussion on sizing a multi-loaded multi-span beam let’s start with the information to be entered in the module:

- Starting at the top, you have the Location box; here you will enter the name of the member or the location. This will be the name under which you save the member in the project. By default, StruCalc will give each member a name; in Figure 3 the default name is “Multi-Loaded Multi-Span Beam 1.”

- Moving down you have the Span Length boxes. You may use up to three spans. This Module can also be used as a simply supported beam if you only use the center span. As you can see from Figure 3, a center span length of 15 ft is automatically entered when first opened.
• Next are the **Unbraced Length - Top** and **Unbraced Length - Bottom** boxes. Here you will enter the unbraced length along the top of the beam and along the bottom of the beam for each span. It is extremely important to make sure you know what these lengths are and enter them in accordingly.

• Moving down there are three large boxes that become buttons when their corresponding span length is entered. When you first open the module, the left one says “Please enter a span length above to activate the left span”. The center one says “Enter center span loads below”, and the right one says “Please enter a span length above to activate the right span”. If the center span is the only span that has a length entered, all of the loading detail boxes below will apply to the center span. If you have more than one span length entered, you need to make sure you have the correct button selected before you start entering in loading details. You will enter the loading information for each span individually and independently of the other spans.

• To the left and right of the span length buttons mentioned above are two check boxes: one called **Left Cantilever** and the other **Right Cantilever**. You will need to check one or both of these if you would like the respective span of the beam to be cantilevered.

• Moving down to the left you will come to the **Point Loads** window. Within this window, you have the ability to enter in six different point loads, each with its corresponding live load, dead load, and location. The location of the load is measured from the left side and is specific to the individual span.

• To the right of the point loads window is the **Beam End Elevation Difference (left end)** box. If your beam is not perfectly horizontal you will need to account for the end elevation difference here.

• Moving down you have two boxes to enter in the **Uniform Live Load** and **Uniform Dead Load**. This is where you enter in a uniform dead or live load that is pertinent to the design. These uniform loads cover the entire length of the span you have selected.

• Below the uniform load boxes are two buttons. The first button is to **Clear All Loads** for all spans entered. The other is the **Trap Wizard** button. This button allows you to auto calculate trapezoidal and uniform loads that cross multi-spans.

• Moving down is the **Trapezoidal Loads** window, where there are several options available in this design window. In this window you can enter up to four separate trapezoidal loads. Using the **Trap Wizard** button will allow you to visualize the details associated with trapezoidal loading.

As a point of clarification, if you are ever concerned about the location of your loads or their magnitudes, just click on the “Loading Diagram” in the bottom left of the screen (Figure 4). This will show exactly what is happening with the beam you are designing.

Now that the information in the module has been entered we can focus on the left hand side of the screen (the Navigation Tool Bar). The first group of information to be specified is under the **Member Codes** section and is the Building Code, NDS, and Steel Design Specification (see Figure 5).

The next group of information to be specified is also located in the Navigation Tool Bar. This information is under the **Material Properties** tab (see Figure 6). A description of this information is as follows, again working from the top of the screen down:
The following section will discuss **Wood design**.

- The first box in this window is the material type. There are five choices: three wood and two steel. We are only concerned with the wood for now.
- Moving down to the next box you have the species. This box will identify the species of wood to be used and is usually set forth by the designer.
- Moving down to the next box you have the grade of lumber. The grade to be used will depend on the specific design and is subject to change based on the parameters of loading.
- To the right of the grade box is the **Round** check box. By clicking this box you allow the design to be done with a round piece of lumber as opposed to your typical dimensional lumber. StruCalc calculates the equivalent square section for design.
- Moving down there is a button that will say **Wet** or **Dry**. Use wet when the moisture content is expected to be above 19%. For anything else use dry.
- To the right of the wet/dry button is a box that will allow you to enter the number of laminations you wish to use. An example of numerous laminations would be three 2x4s nailed together.
- To the right of the lamination box is the width of the member you wish to use. To the right of the width box is the depth of the member you wish to use. In figure 6 it shows one lamination of a 2x4.
- Moving down is the **Flitch Plate** check box. When you click this box you will be given the opportunity to enter the parameters of the steel you wish to use in your flitch plate. The first box is named **Plate Number**, in this box you place the number of steel plates you wish to use. In the next box named **Plate Width** you will enter the width of the individual steel plates.
- Moving to the right is the **Flat Use** check box. This is used if you want to rotate the orientation of the material you are using (i.e. from a 2x4 to a 4x2). This is available for solid sawn lumber only.
- Moving down you have the **Apply the Incising Factor** check box, which is available for solid sawn only. If you are using pressure treated lumber that has been incised then check this box. An incised piece of pressure treated lumber can be identified by the numerous small ½ inch indentations spaced approximately 2 inches apart.
- Next we have **Check Unbalanced Loads**. StruCalc automatically keeps track of alternate span loading and keeps track of the worst shear, moment and deflection. With the unbalanced loads box checked the program will make sure the most conservative design is used.
- Moving down is the **Repetitive Use** check box. When a load is spread over several members an increase in the Fb value is allowed. Reference the building code for details.
- Moving down to the next box you have **Notch Depth**. You can only use the notch depth option when solid sawn, glulam, or structural composite beams are being designed. StruCalc limits the size of the notches based on the NDS being used.
- Moving down to the next box you have **Live Load Deflection** and after that **Total Load Deflection**. These are load deflection criterion set forth by the code. StruCalc automatically selects the minimums, but they can be overridden if desired.
- Moving down to the next box you have the **Duration Factor** box. This figure is also set forth by the code, and depends on the loads that the beam will be designed for. You can also override this number if needed.
The following section will discuss Steel design; again we will begin at the top and work down.

- The first box in this window is the type of steel you will use. Here you are given two choices: steel and tube steel.
- Moving down to the next box you have the shape. This box will identify the shape of steel to be used and this is usually set forth by the designer.
- Moving down to the next box you have the size of steel to be used.
- Moving down to the next box you have the grade of steel. Based on the steel beam shape, StruCalc will select the recommended grade as specified by the AISC steel manual.
- Next you have the $Cb$ box. This is to be used when steel or tube steel rafters are being designed. This is the moment gradient and is conservatively taken to be 1.0.
- Next we have Check Unbalanced Loads. StruCalc automatically keeps track of alternate span loading and keeps track of the worst shear, moment and deflection. With the unbalanced loads box checked the program will make sure the most conservative design is used.
- Moving down to the next box you have Live Load Deflection and after that Total Load Deflection. These are load deflection criterion set forth by the code. StruCalc automatically sets them. They can be overridden if desired.

Once this basic information has been entered, structural members can be determined. There are two ways members can be sized: the first is to utilize the AutoSize function of StruCalc, and the second is to have a size in mind, and check the adequacy.

The AutoSize function of StruCalc is the easiest way to determine a member size for a given loading. To use the AutoSize function:

- First, click the AutoSize button on the left of the StruCalc screen (see Figure 7).
- A material and species (or manufacturer, for structural composite members) will have to be selected before the AutoSize feature can run.
- Then, select a grade (for wood member) or a depth (for steel members). Note, that multiple grades or depths can be selected by holding down the “Ctrl” key and clicking the grades or depths in the list.
- After the grades or depths are selected click on the Run button (see Figure 8) to run the AutoSize feature.
- Once the AutoSize feature has been run, it will come up with a solution list based on the grades or depths that the user has selected, and will show the percent adequate of each solution. From this list you should be able to find a solution you are comfortable with. Choose the solution you desire, then click the Select & Return button (see Figure 9) and the program will return to the design screen.

Down at the bottom of the screen it will give you the percentage of adequacy or inadequacy and the controlling factor. If you put the cursor on the word adequate you will see Figure 10 (the # 140.1% will be unique to your design). This box is the final output with the controlling factor.
The next way to determine a member size would be to simply estimate a size, and **click** the **Calculate** button (see Figure 11) located on the bottom tool bar. This will perform all the calculations necessary to determine a beam’s adequacy and return a block of information that would look like Figure 10, if the member is adequate. If the member is inadequate the green circle would be red, and instead of a check mark, there will be an x. In this case, select a new member size and again **click** the **Calculate** button. Repeat this process until you get an adequate result.  

At this point the solution has been generated, and decided on, and all that is left to do is save the analysis. There are three different ways to save an analysis: the first way to save an analysis is by going to the top tool bar and **clicking** File, then moving your mouse down and **clicking** on **Save Multi-Loaded Multi-Span Beam 1** (see Figure 12).

The second way to save an analysis is to **click** on the **Save** button (see Figure 13) located on the bottom tool bar.  

The third and final way to save the analysis would be by holding down the “Ctrl” key on your keyboard and hitting the “S” key on your keyboard.