

# How to use STAAD more efficiently

---

This document contains the answers to the questions raised by sasa3k at

[http://communities.bentley.com/products/structural/structural\\_analysis\\_design/f/5932/p/89565/254765.aspx#254765](http://communities.bentley.com/products/structural/structural_analysis_design/f/5932/p/89565/254765.aspx#254765)

## Question 1:

1. Lack of physical members accounts for a significant time waste. Any hints on how to overcome this would be much appreciated.

## Answer:

It is indeed possible to generate the model as a physical model and wait until the end of the modeling to convert it to the mathematical model. This approach can perhaps reduce the time spent on modeling. What we mean by that is the following.

The first step as you know is creating the geometry of the model. Whether you are doing this in a CAD software like Microstation or AutoCAD and importing the DXF file, or generating the entire geometry inside the STAAD GUI, do not draw the individual segments, but instead, create each line to represent the full physical member. For example, if you have a 24-m long beam spanning 3 bays with each bay being 8m wide, draw a single line for the full 24 m span without intersecting it with the columns at the 0m, 8m, 16m and 24m locations. In the geometry thus created, the beams and columns won't have an intersection point, but, you can wait until the end of the modeling to remedy that.

If you are accustomed to creating groups, this would be a good time to do that. The groups would thus consist of the physical beams, columns and physical slabs or walls.

Assign the section properties to these physical entities.

If the connection between the beams and columns is pinned, that would require a breaking up of the lines, and you'll have to wait till the end to do that. But if it is a moment connection, no additional effort is required.

Add the loads, steel / concrete design parameters, etc. to the physical lines and complete assigning all the data. Save the file and make a backup (important).

You now have a number of lines that represent the physical members. Select all of them, go to the Geometry menu (on top of the screen), choose *Intersect Selected Members* followed by *Intersect*. A connection will be created between all the lines where they cross each other. This is the segmentation process that involves breaking up the physical beams into individual segments. If you don't want it to be done on the entire model in one go, you can select the lines a few at a time and repeat the process.

When the program intersects them, it will ensure that data such as loads which were specified on the physical members are converted to loads on the individual segments. The member and element groups would be modified in such a manner that the physical member numbers would be replaced with those of the segments.

If meshing is involved, that too can be done at the end. The loads on the parent slab or wall are automatically converted to loads on the individual elements.

It is possible that some things can go wrong using this approach. We do not have a list at this time, so, if you encounter any, if you can let us know, we will get it rectified if possible. You mentioned that you experienced some problems with floor loads. We are examining this and will make improvements if we find any errors.

In the Select menu, the last item is called *Highlight Entities Sequentially*. You can use this tool to visually inspect the segmentation. The program will highlight each member segment and each plate element one at a time automatically.

After the segmentation, you have to assign the member end releases to model the shear connections.

Of course, it is important to inspect the model to verify that the data is accurately converted from physical object data to segmental data, especially the loads, design parameters, etc. Switch on the load display, navigate through the Design Parameters page keeping the “Highlight Assigned Geometry” switched on.

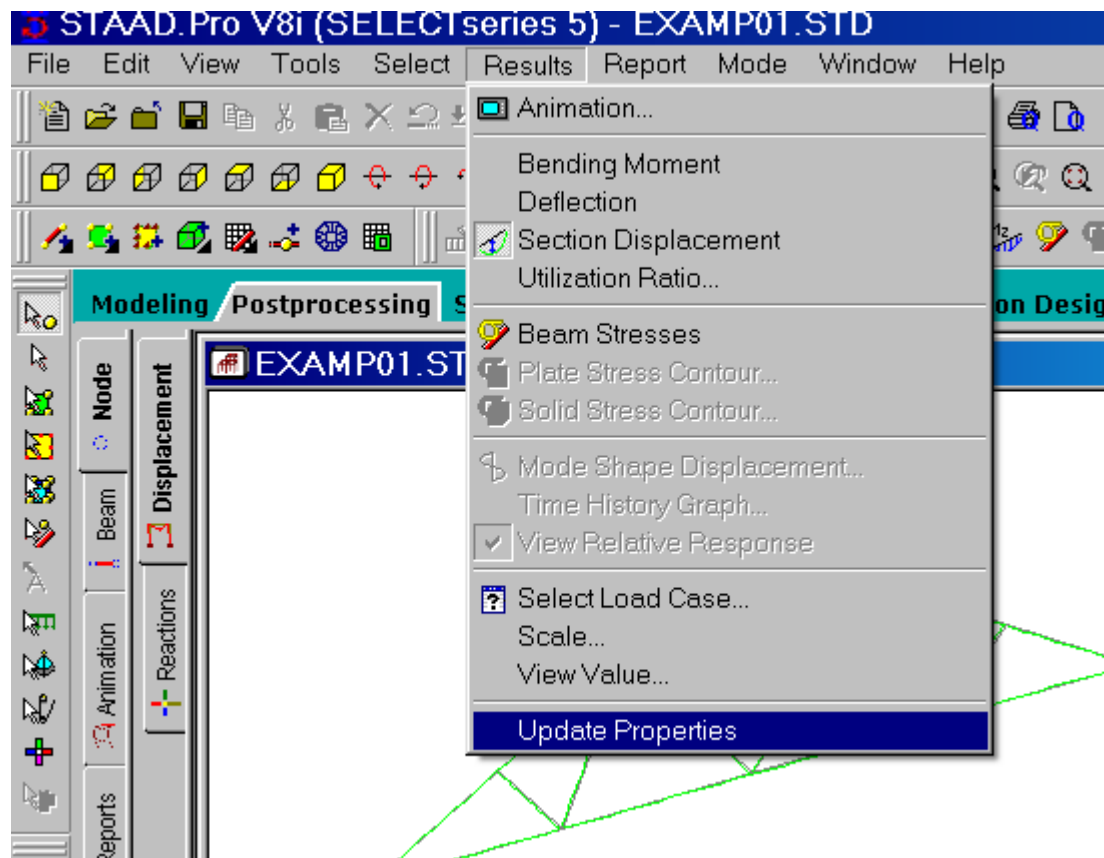
#### Question 2:

2. There is no facility to directly assign design properties to the members. Users have to go to the design tab, select the members, add the design commands, and then assign the properties. Another significant time waste.

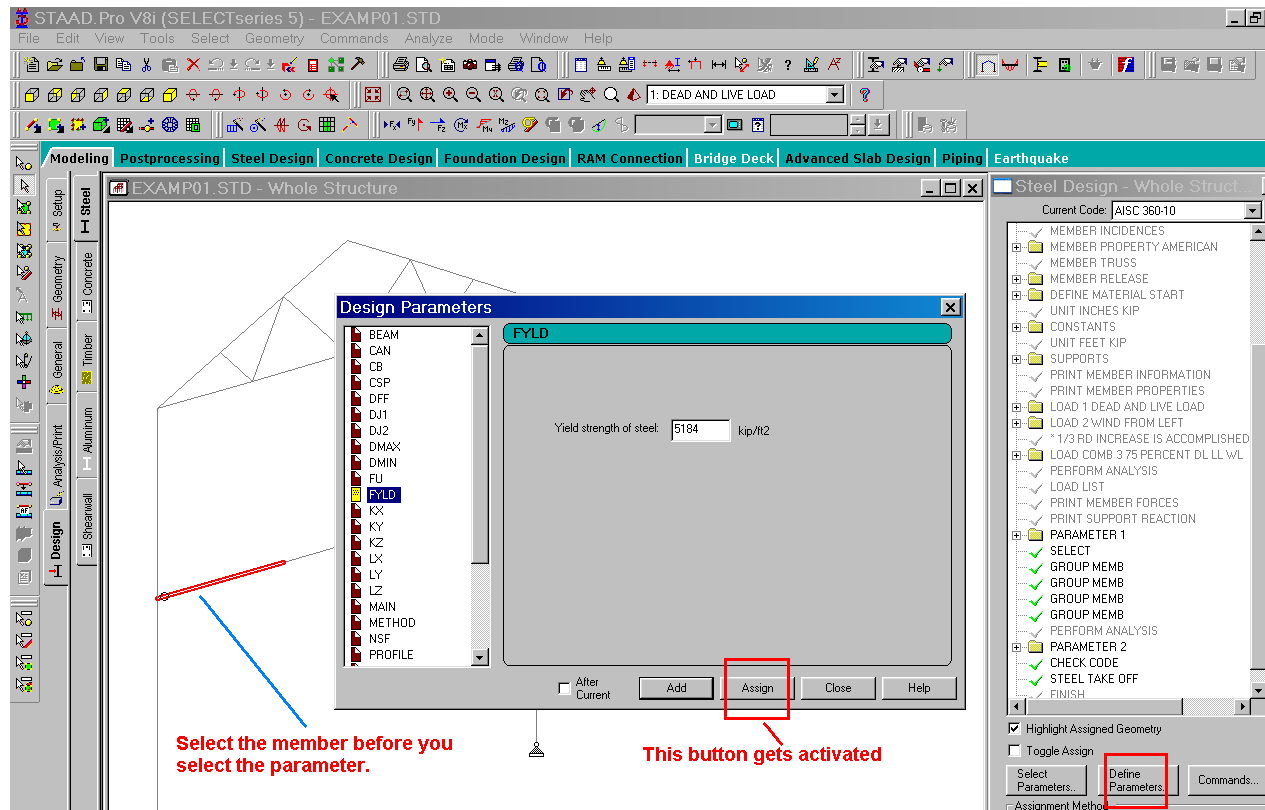
Answer:

When you say design properties, are you referring to the sections chosen through a member selection /grouping? Or are you referring to design parameters like FYLD, LY, LZ, etc.

If it is the former, there is an option called Update Properties in the Results menu in the post-processing mode as shown in the next figure.



If you are referring to assigning parameters, the two step method can be converted to one-step by selecting the members before you select the parameter you wish to assign.



The Assign button will then get activated and the parameter can be directly assigned to the selected member from within that dialog box.

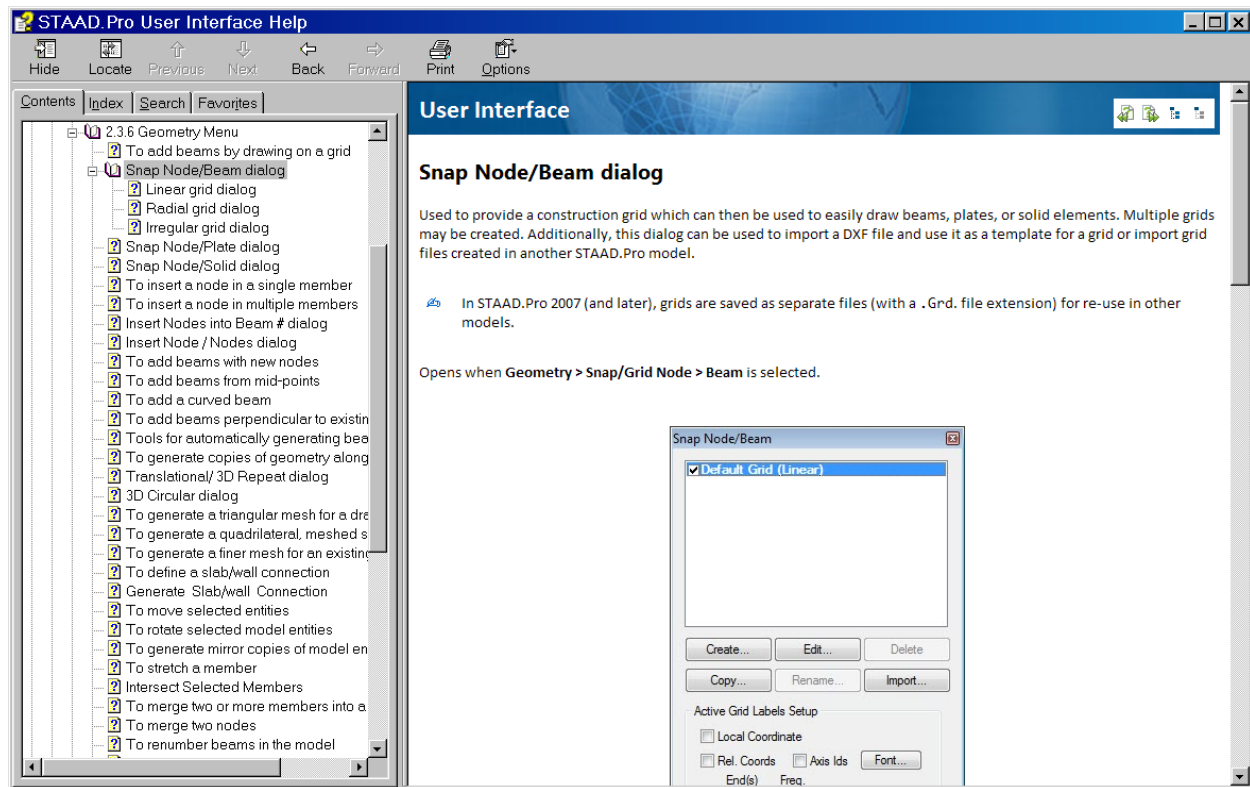
If you are asking for other methods of assigning like double clicking a member and assigning a parameter using the Member Query screen, this is on the development list for a future version.

### Question 3:

3. There are no tools to create grid or construction lines. For complex floors it is very hard to see where the columns are and properly frame the floor. I usually end up printing the floor plans, sketch the members by hand and then draw them back in STAAD. Any other suggestions?

Answer:

There are some rudimentary tools for creating grids which are explained in the STAADGUI manual as shown below.



It may not be as sophisticated as what you can get from CAD software or programs specifically geared towards generating complex geometry. Nevertheless, there are a number of options such as grids in global and non-global planes, grids with lines at unequal spacing, radial grids, etc. which are explained in the aforementioned page.

If you find these to be inadequate, you can use a CAD program – Microstation or AutoCAD for example – to generate the geometry of the model, export it to a DXF file, import the DXF into STAAD.Pro, and then assign the rest of the STAAD input data from within STAAD's GUI. Bentley also has programs like AECOSIM Building Designer than can generate most of the STAAD model and has physical member modeling features.