Tower Quick Start +

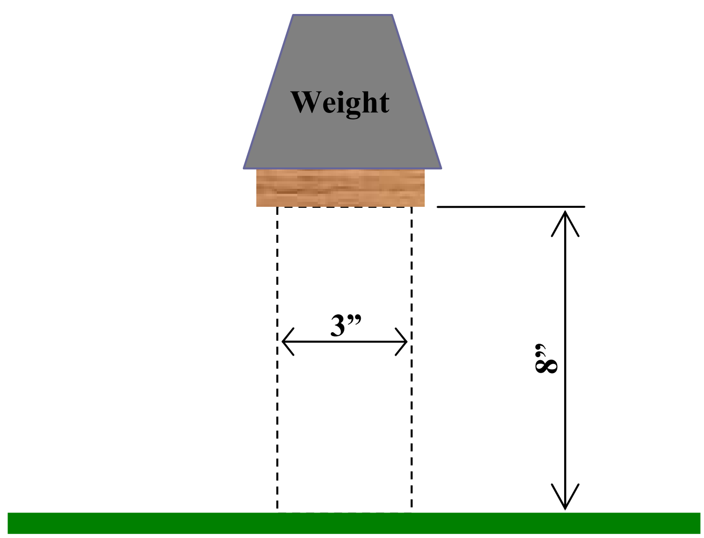
ModelSmart3D can be used to design many types of structures. This chapter is for those who would like to learn how to use the program while working with a model tower.

We titled it "Tower Quick Start +" because it also includes construction & navigation tricks.

Let's start with a simple example.

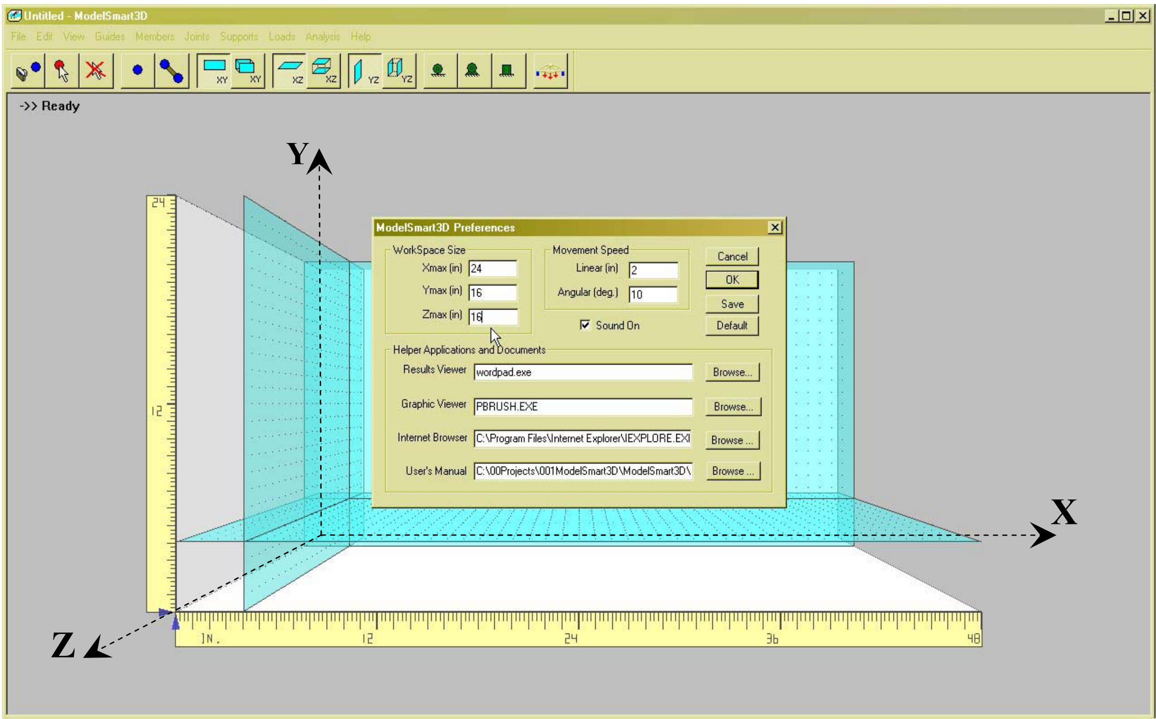
Problem Statement:

Design a tower to support a working gravity load of 10 lbs at a height of 8". The tower must have a safety factor of at least 2.0. The load will be applied to a load block placed atop the tower and measuring 3 ½" x 3 ½" x ¾". The load block is made from #2 KD southern pine and weights 3 oz. (included in the working load). No side of the tower may be wider than 3".



For simplicity, let's construct a tower that is 3" wide on a side and 8" high. (You can adjust the dimensions later to account for the thickness of the members.)

Start ModelSmart3D by double clicking its shortcut.



Next, select the "Edit|Preferences…" menu option and change the WorkSpace (the three sided box, floating planes and rulers) dimensions to :

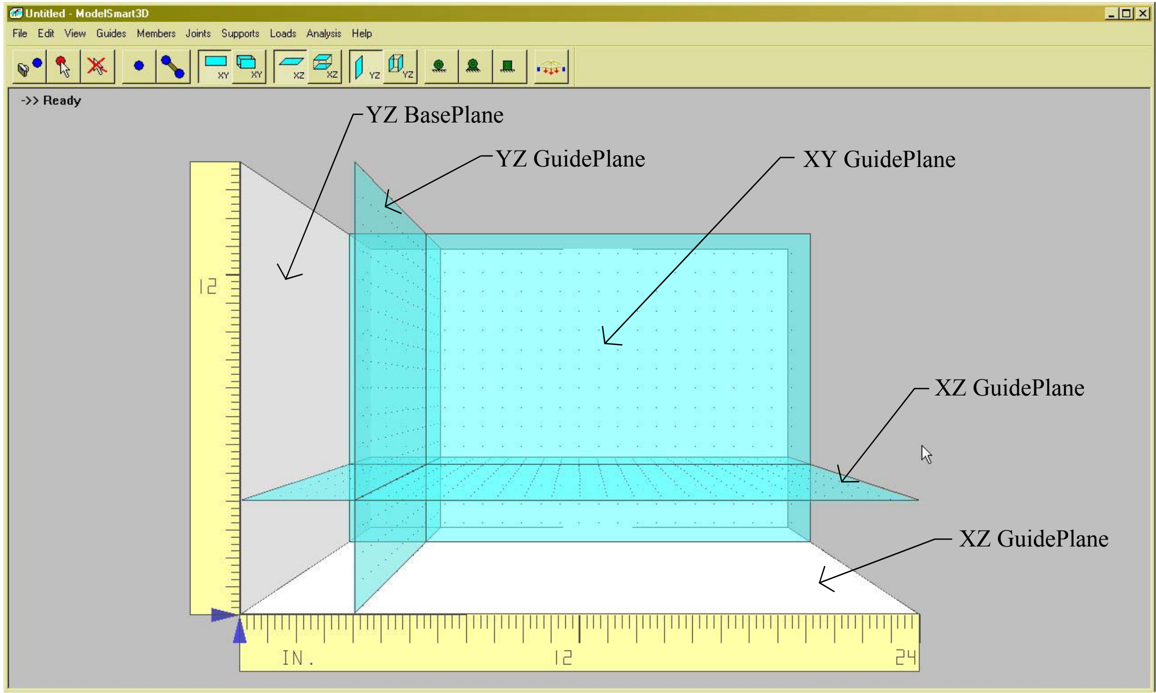
Xmax = 24"
Ymax = 16"
Zmax = 16"

(The 3D Cartesian coordinate system denotes the global sign convention used by the WorkSpace.)

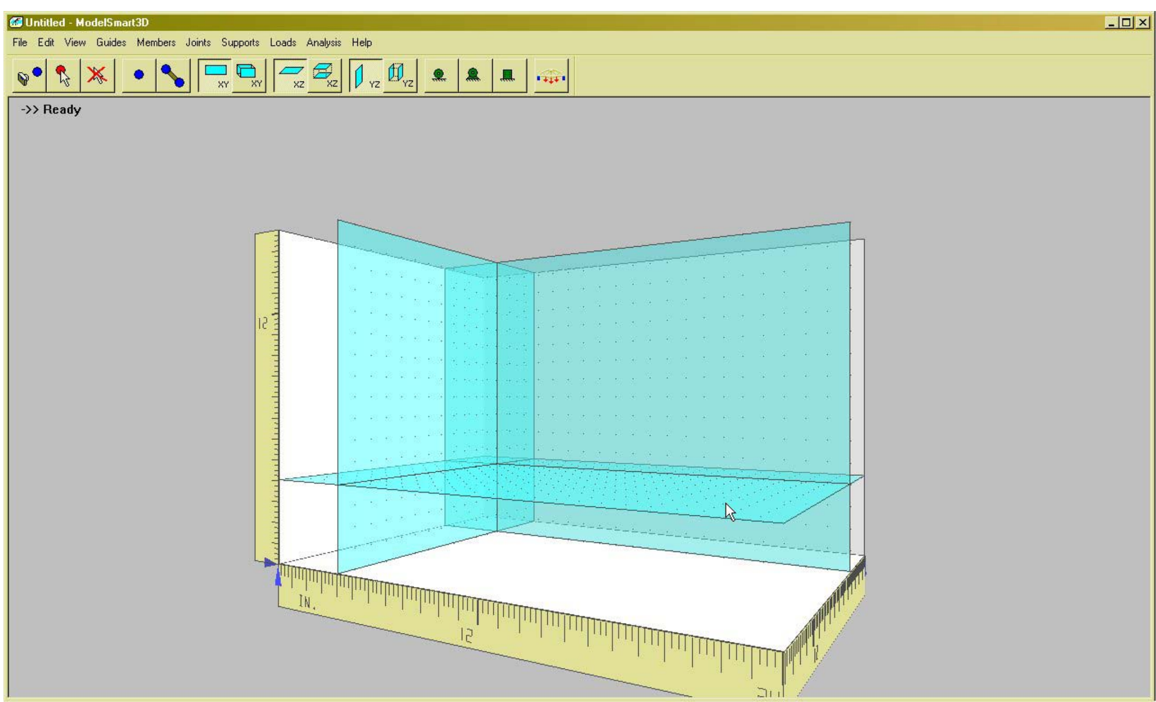
Click the "Save" button then click "OK".

Press "F1" to re-center the WorkSpace. Press the down arrow key a couple of times to move the observer out a little.

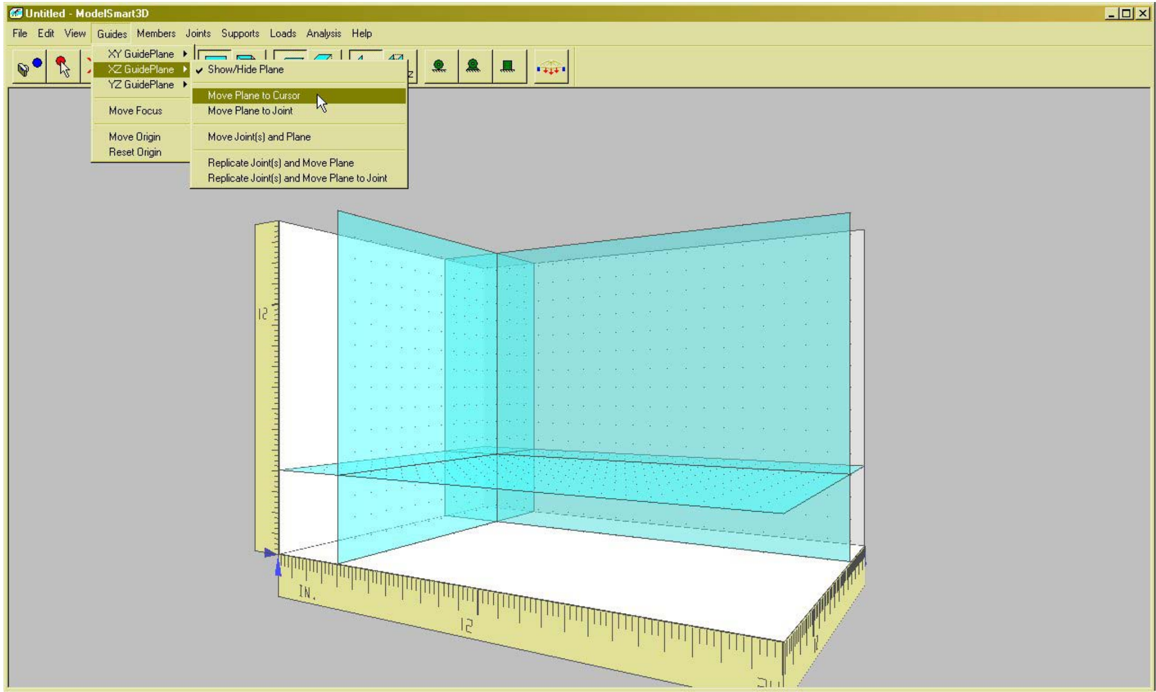
Your screen should now look similar to the follow image:



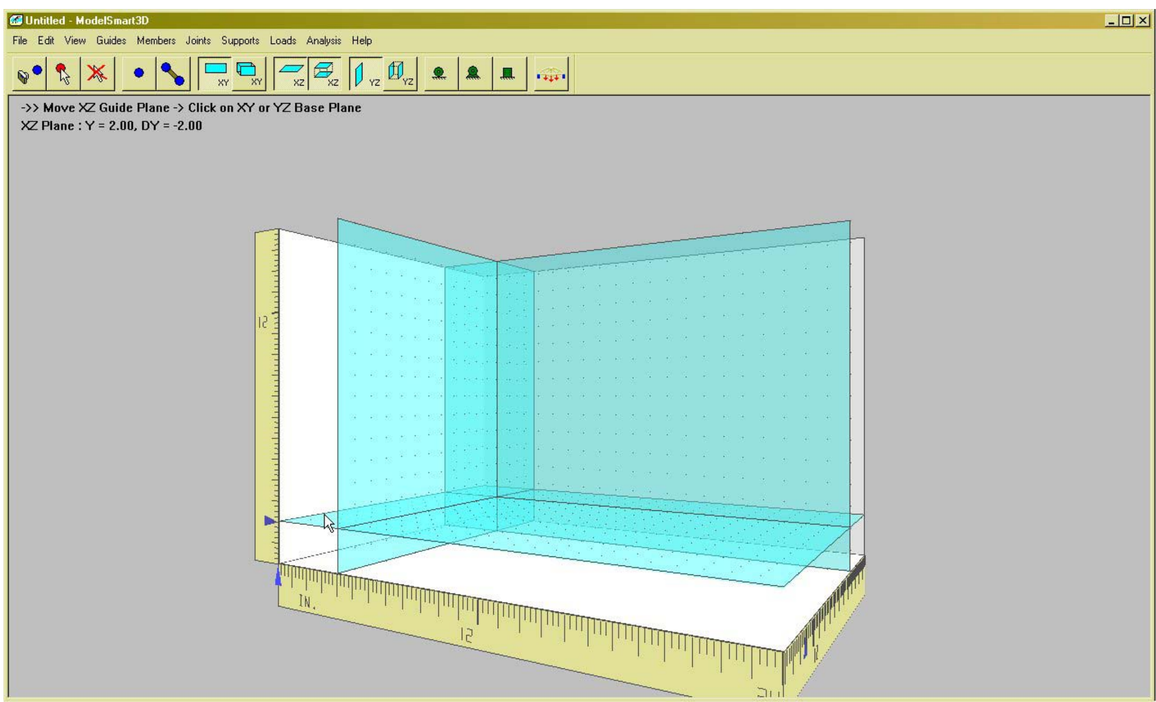
Let's move the XZ GuidePlane (the transparent floating plane used locate the models' joints – see image above) down to Y=2"



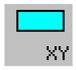
Press the right arrow key a couple of times to rotate the WorkSpace.

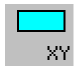


Select the “Guides|XZ GuidePlane|Move GuidePlane to Cursor” menu option

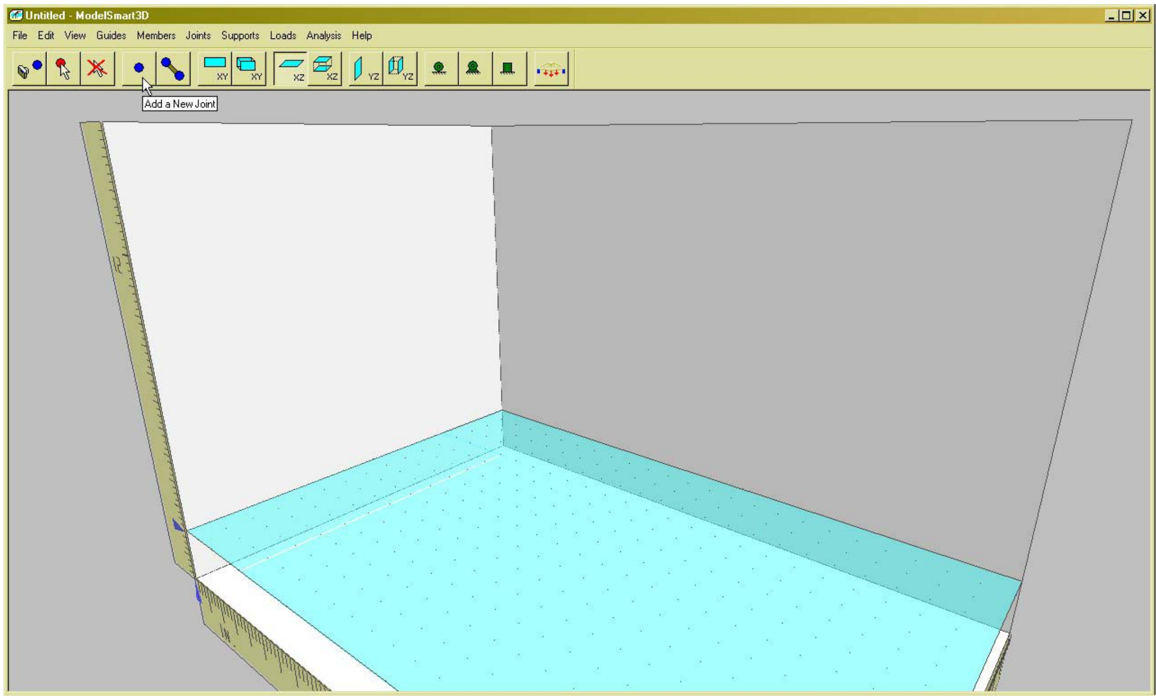


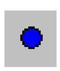
Left click and drag the cursor on the YZ BasePlane (vertical white plane at left). When the second line of text at the upper left of the screen reads Y=2.0”, release the left mouse button. (Do not try to point through the YZ GuidePlane when moving the XZ GuidePlane.)

Click the 

& 

toolbar buttons to toggle off these GuidePlanes. Use the "+" key and the other numeric keypad navigation keys (see below) until your screen looks similar to the screen below.



Click the  "Add Joint" toolbar button to put the program in the "Add Joint" mode.

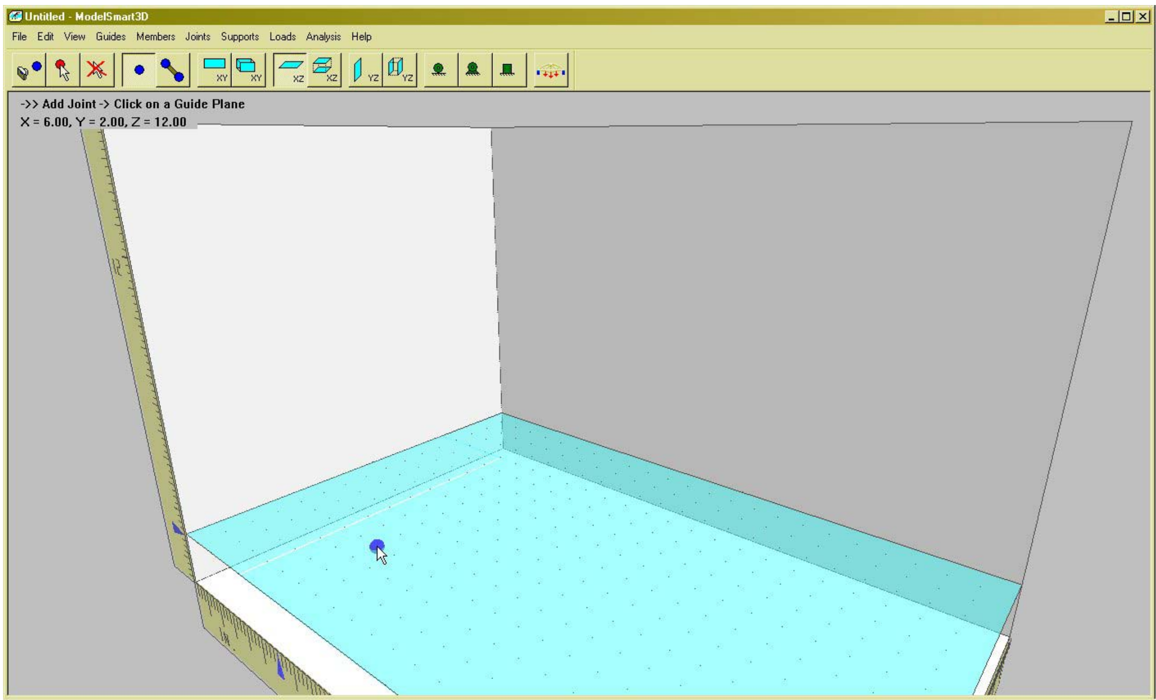
Numeric Keypad Navigation Keys

"/" (division key)	Move the observer's viewpoint directly toward the focal point.
"*" (multiplication key)	Move the observer's viewpoint directly away from the focal point.
"+"(addition key)	Move the observer's viewpoint up while remaining focused on the model.
"-"(minus key)	Move the observer's viewpoint down while remaining focused on the model.
Up Arrow (On Keypad)	Move the observer's viewpoint and focus up.
Down Arrow (On Keypad)	Move the observer's viewpoint and focus down.
Left Arrow (On Keypad)	Move the observer's viewpoint and focus left.
Right Arrow (On Keypad)	Move the observer's viewpoint and focus right.

Move the mouse pointer until the coordinates (second line of text at the upper left of the screen) reads:

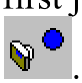
X=6.0, Y=2.0, Z=12.0

Click the left mouse button to place the joint (location where a member starts or ends).



Navigation tool:

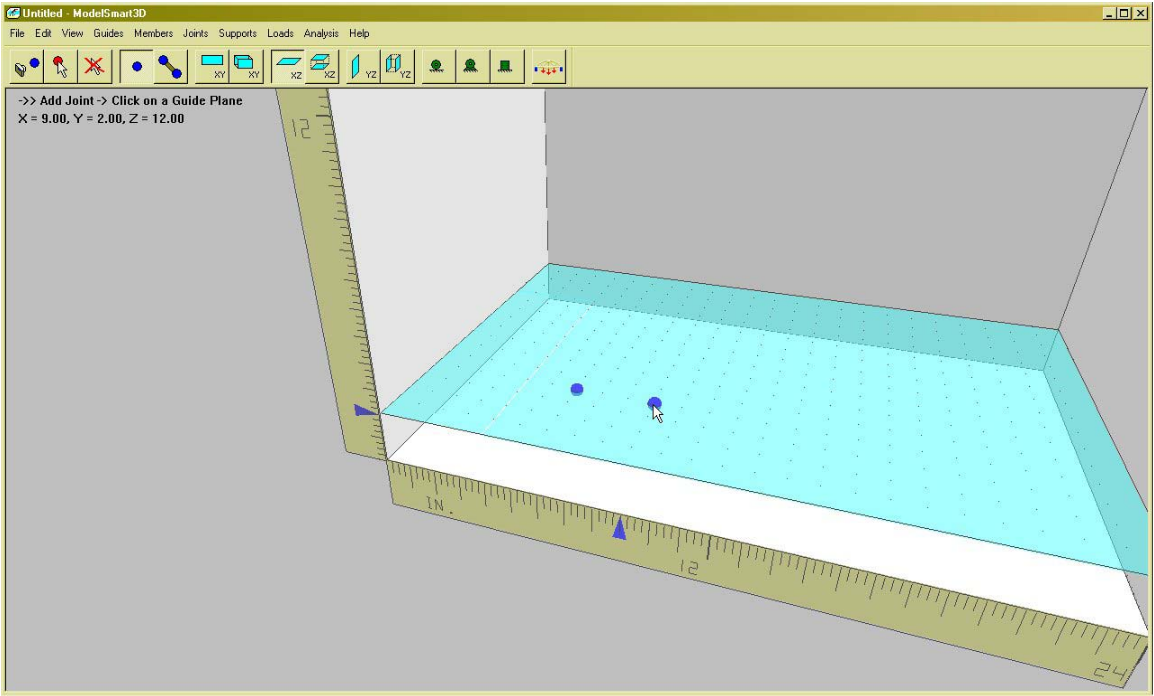
Once the first joint is placed, you can use the "**Focus**" navigation feature.

Click the ."Set Focus" toolbar button at the upper left of the screen

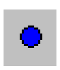
Now, left click the joint that you just placed. The navigation keys will now move the observer relative to the focus joint.

Use the "*" key on the numeric keypad to move directly away from the focus joint and the "/" key (on the numeric keypad) to move directly toward it.

Let's add another joint.



Use the arrow keys to rotate the WorksSpace to the orientation show above.

Click the  "Add Joint" toolbar button again to put the program back in the "Add Joint" mode. Move the mouse pointer until the coordinates reads:

X=9.0, Y=2.0, Z=12.0

Click the left mouse button to place the second joint.

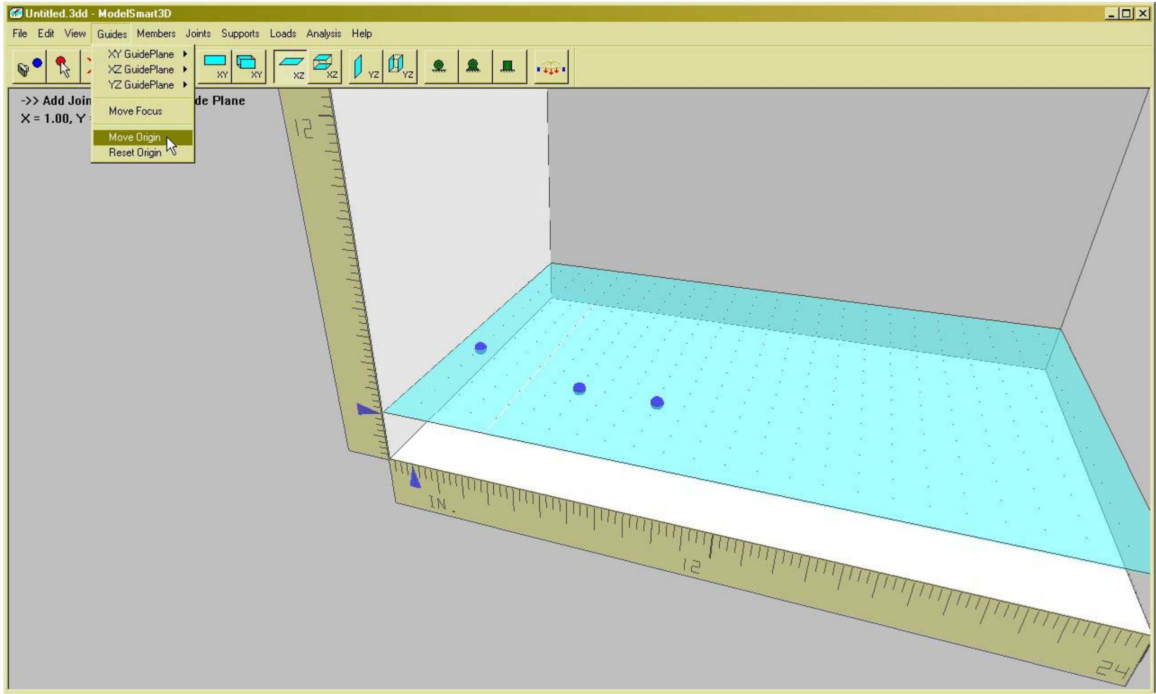
Note: The right mouse button is a navigation only button. Clicking and holding the right mouse button moves the observers eye depending on where the mouse cursor is located on the screen:

Right Mouse Click Navigation Guide

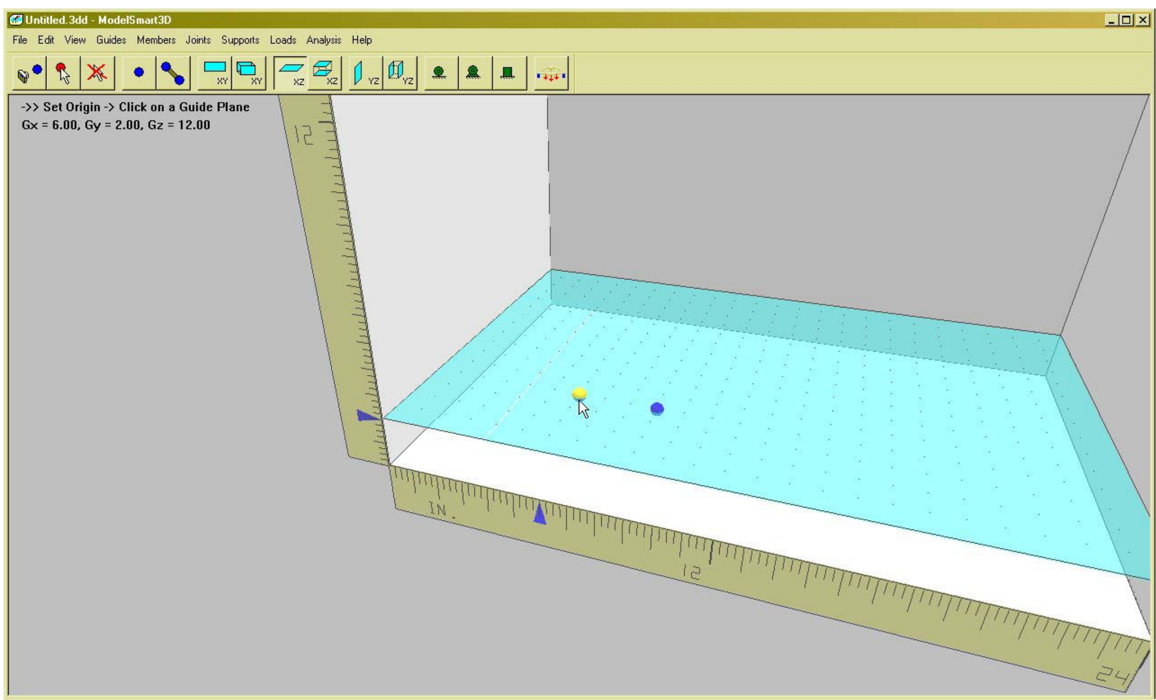
Quickly Left	Toward Model	Quickly Right
Move Observer Left	No Movement	Move Observer Right
Quickly Left	Away From Model	Quickly Right

"F1" return the observer default.

Here's another handy feature – "**Move Origin**".

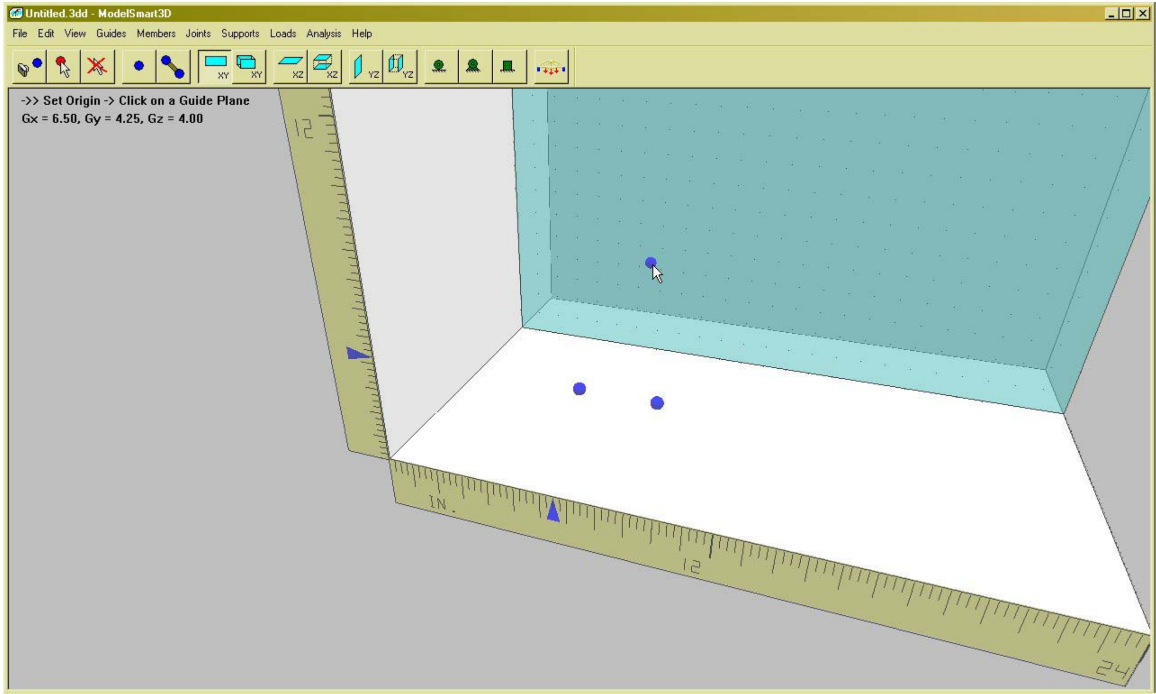


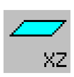
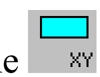
Select the "Guides|Move Origin" menu option.



Click the joint at the left.
Now the coordinates reported are relative to this new location.
.

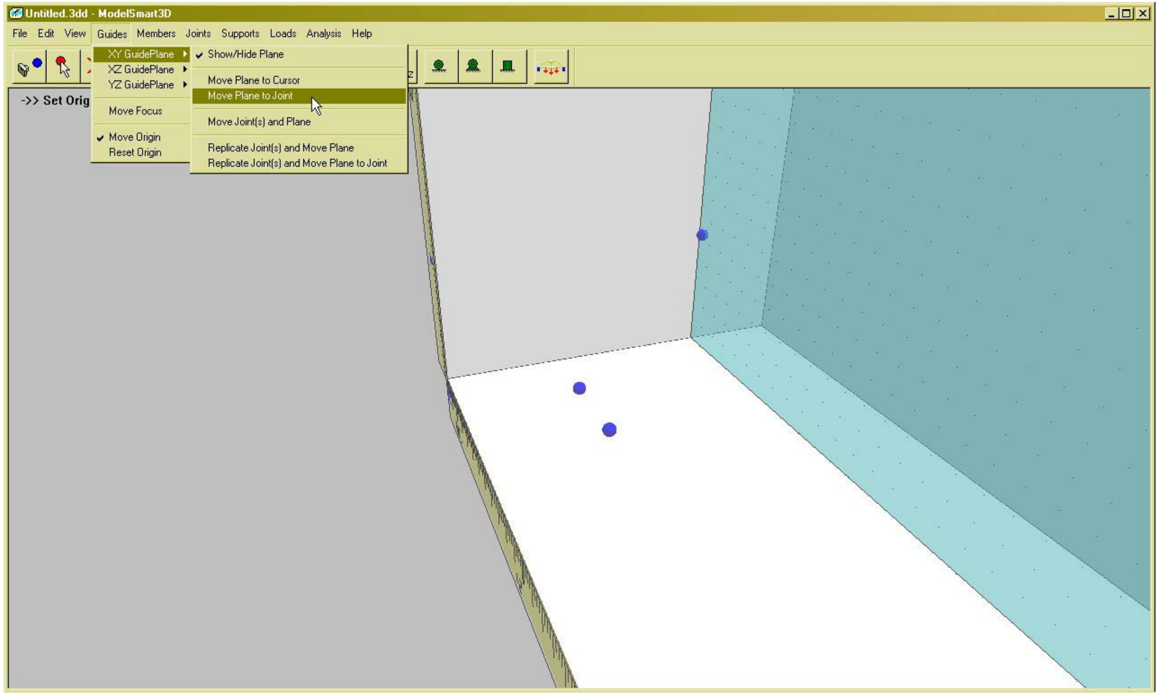
Let's add the top joints to the front side of the tower.



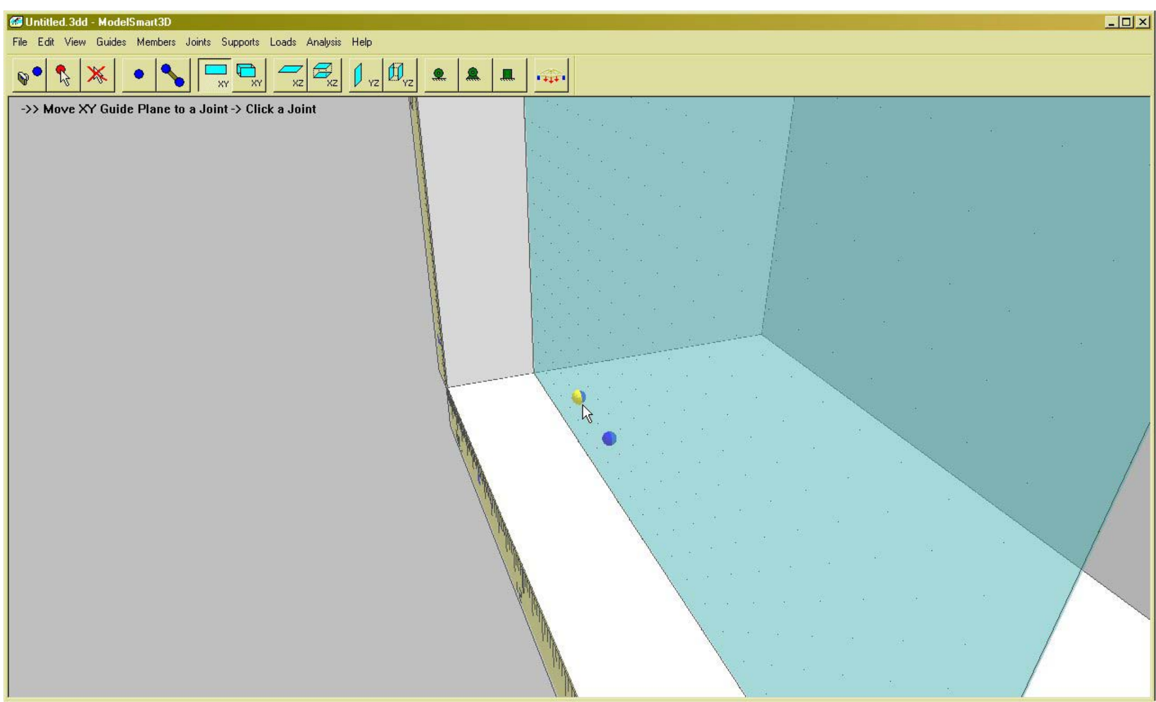
Click the  toolbar button to turn off the XZ GuidePlane and click the  toolbar button to turn on the XY GuidePlane.

The cursor will now be confined to moving in the XY plane.
But inorder to work on the fron side of the tower, we need to move the XY GuidePlane forward until it includes the first two joints.

Use the arrow keys to rotate the XY GuidePlane to the position shown below:

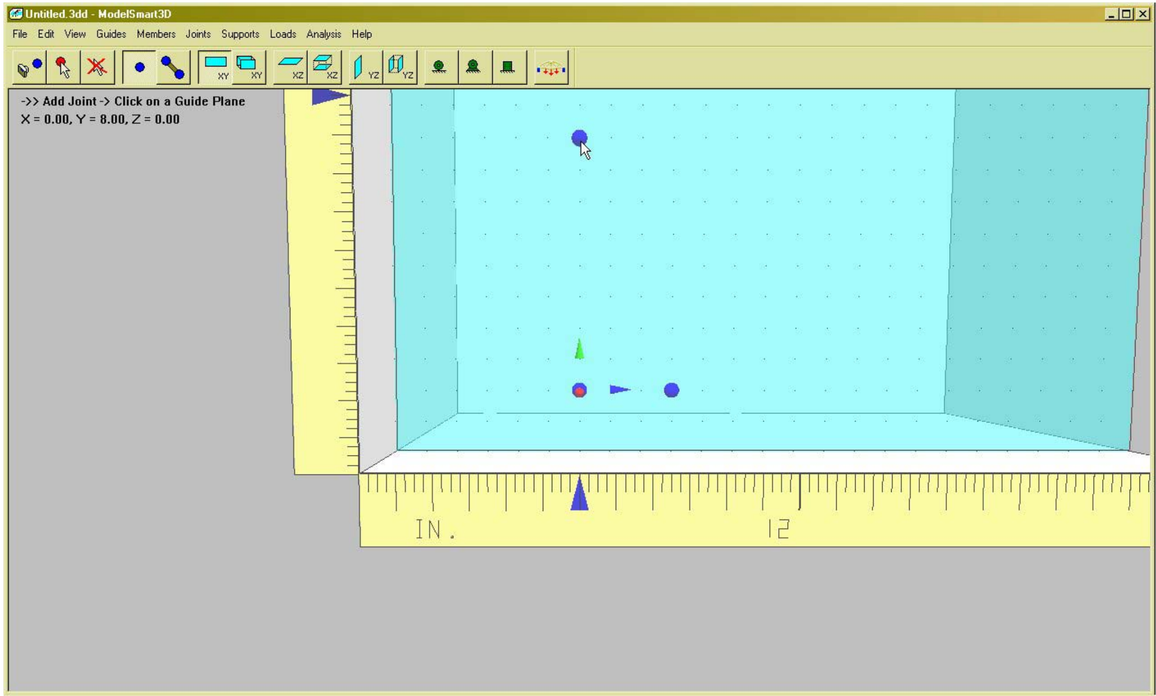


Select the "Guides|XY GuidePlane|Move to Cursor" menu option.
Move your mouse cursor until it points at either previously placed joint and left click it.

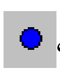


The XY GuidePlane snaps to this new location.

Let’s add a joint 8” above the joint presently at coordinate [0,0,0]. (Remember we moved the origin to the location of our first joint.)



Use the arrow keys to rotate the WorkSpace and the ”-“ key on the numeric keypad to lower the observer’s eye level.

Click the  “Add Joint” toolbar button and move the cursor to the coordinate [0,0,0]:

X=0.0, Y=8.0, Z=0.0

Click the left mouse button to place the third joint.
Move the mouse cursor to point at the coordinate:

X=3.0, Y=8.0, Z=0.0

Click the left mouse button to place the fourth joint.

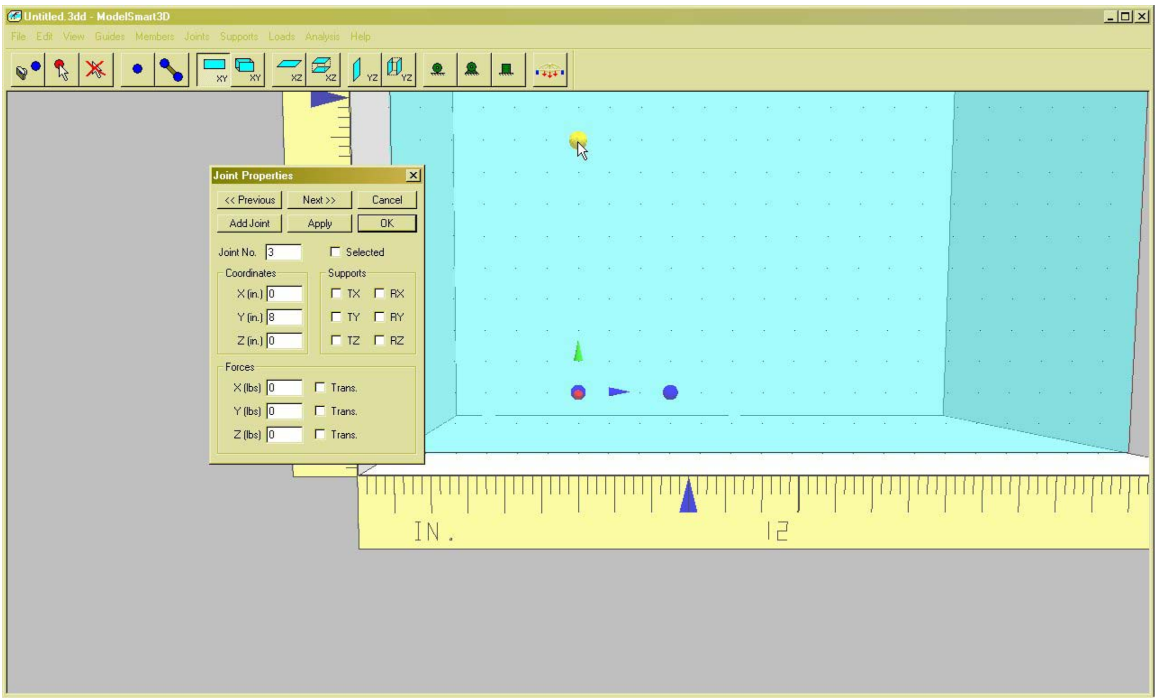
Note:
The blue ball representing the joint may appear to move erratically. This is because it does not move in relation to the pixels on your computer screen; It moves when you hit a target with the, continuously firing, mouse pointer. The targets are a grid dot, the point halfway between grid dots, and the points between the half points. Point the mouse cursor at one of these locations and the blue dot will move to it. This targeted moving method is required for drawing in 3D space.

Here's another way to place joints – using the "**Joint Properties…**" **dialog**.

Let's delete joint number 4 (the one we just placed) and re-add it another way.

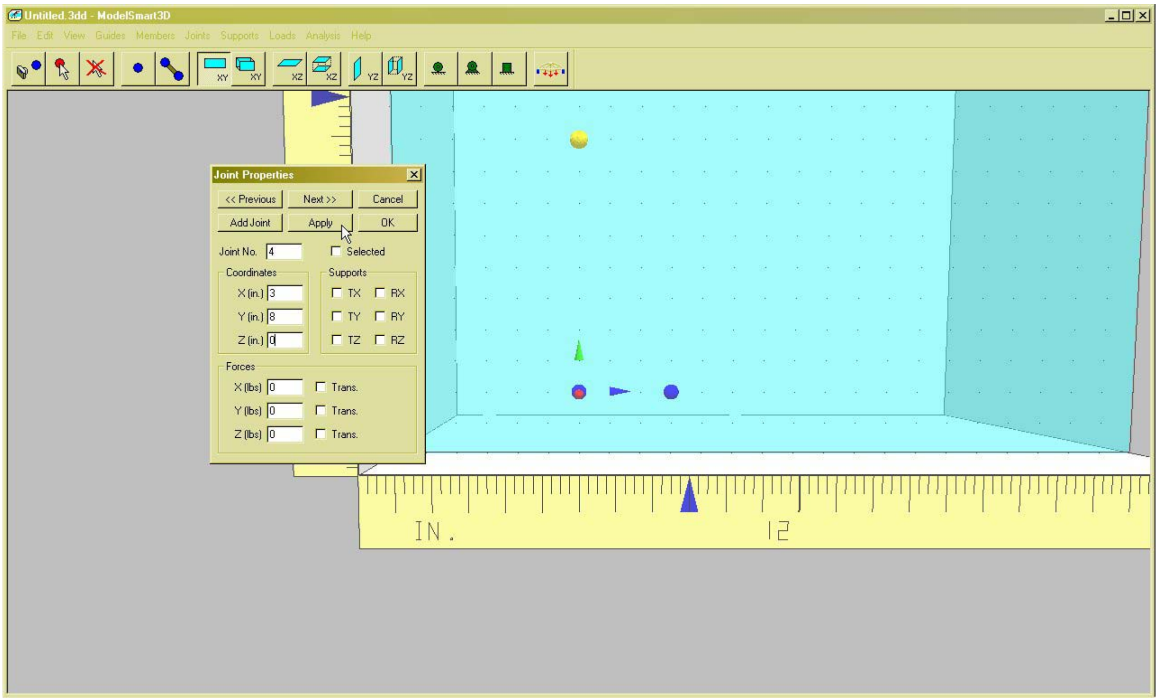
Select the "Joints|Delete" menu option. Then, click joint four to delete it.

Now, select the "Joints|Properties…" menu option and click the third joint



This brings up the "Joint Properties" dialog. Note the coordinates of joint number three.

Click the “Add Joint” button.



Now, change the coordinates for the new joint (#4) to:

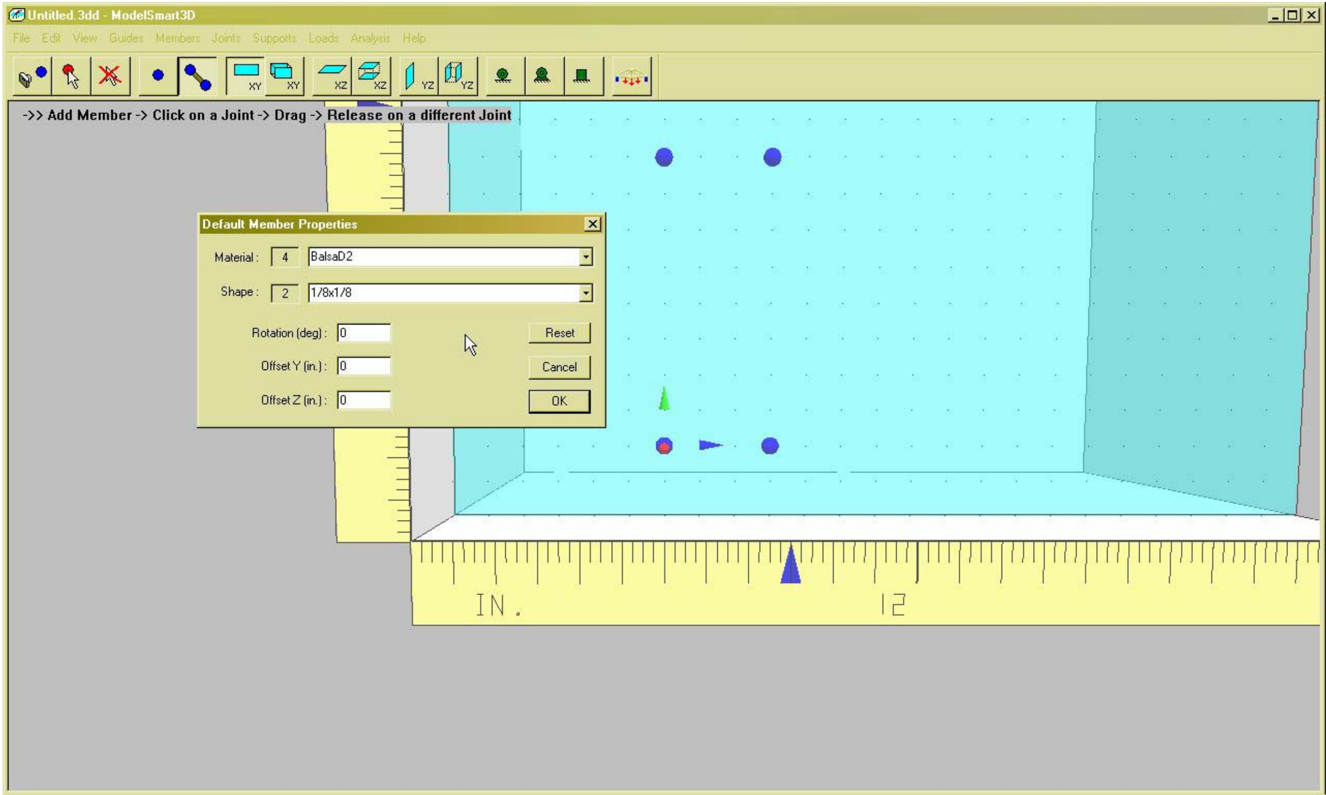
X=3
Y=8
Z=0

Click the “Apply” button and a joint will appear at the requested location.

(If you don’t change the default coordinates you will end up with a joint located at the intersection of the three GuidePlanes.)

Ready to add some members?

Click the “OK” button to close the “Joint Properties” dialog.



Adding Members

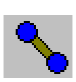
Select the “Members|Set Default Material and Shape” menu option. This will open the “Default Member Properties” dialog.

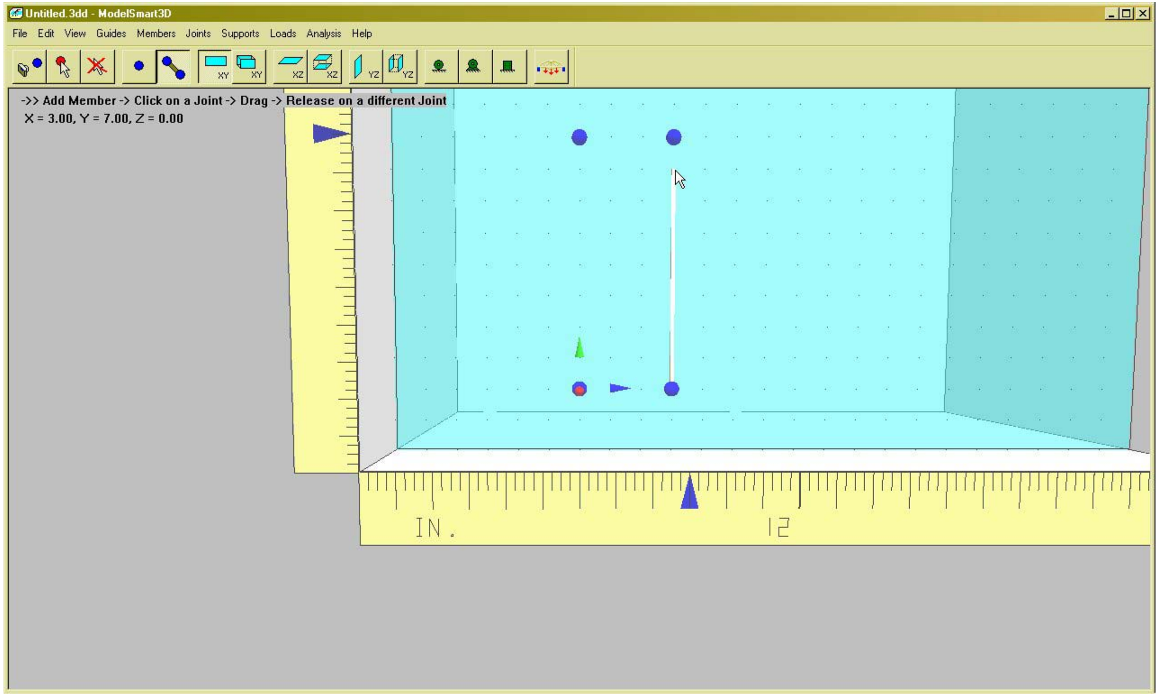
For now, accept the standard member defaults of :

Material = BalsaD2
Size = 1/8”x1/8”

If yours is not set to these defaults, please change them using the pull down arrows. The Rotation, Offset Y & Offset Z should all be set to zero.

Click the “OK” button to close the dialog.

Click the  toolbar button to put the program in the “Add Member” mode.



Move the mouse cursor to point at the lower right joint (it will turn yellow).
Click **and hold down** the left mouse button.

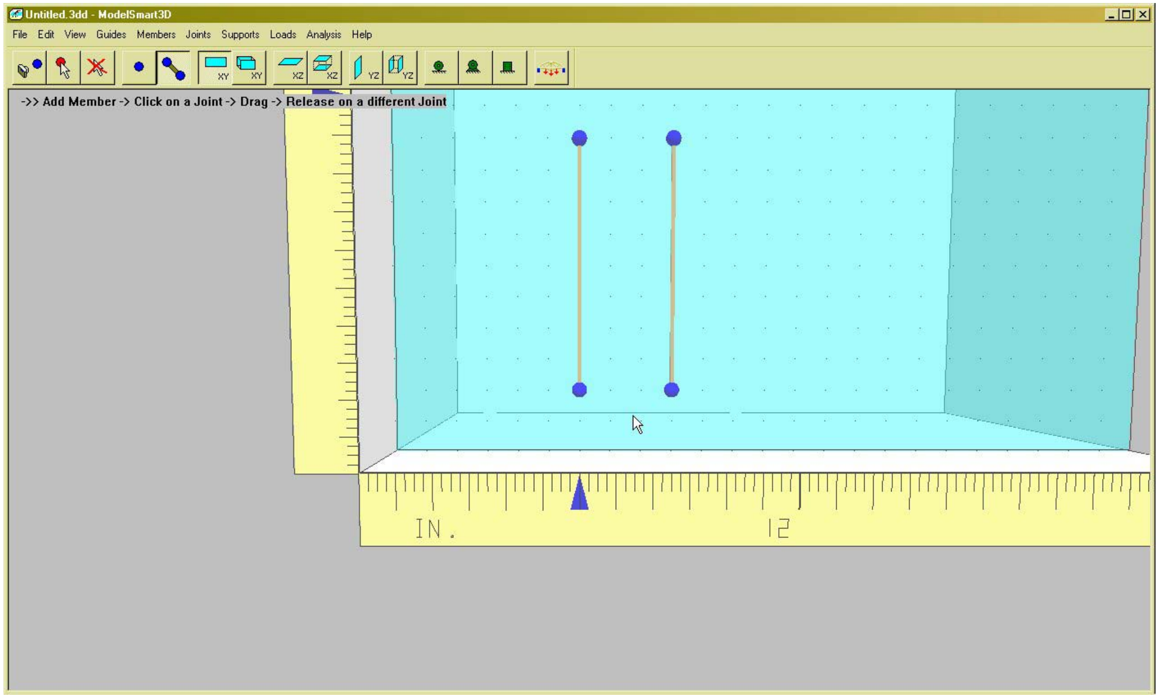
While still holding down the left mouse button, move the mouse cursor to point at the upper right joint (it will turn yellow).

Release the left mouse button.

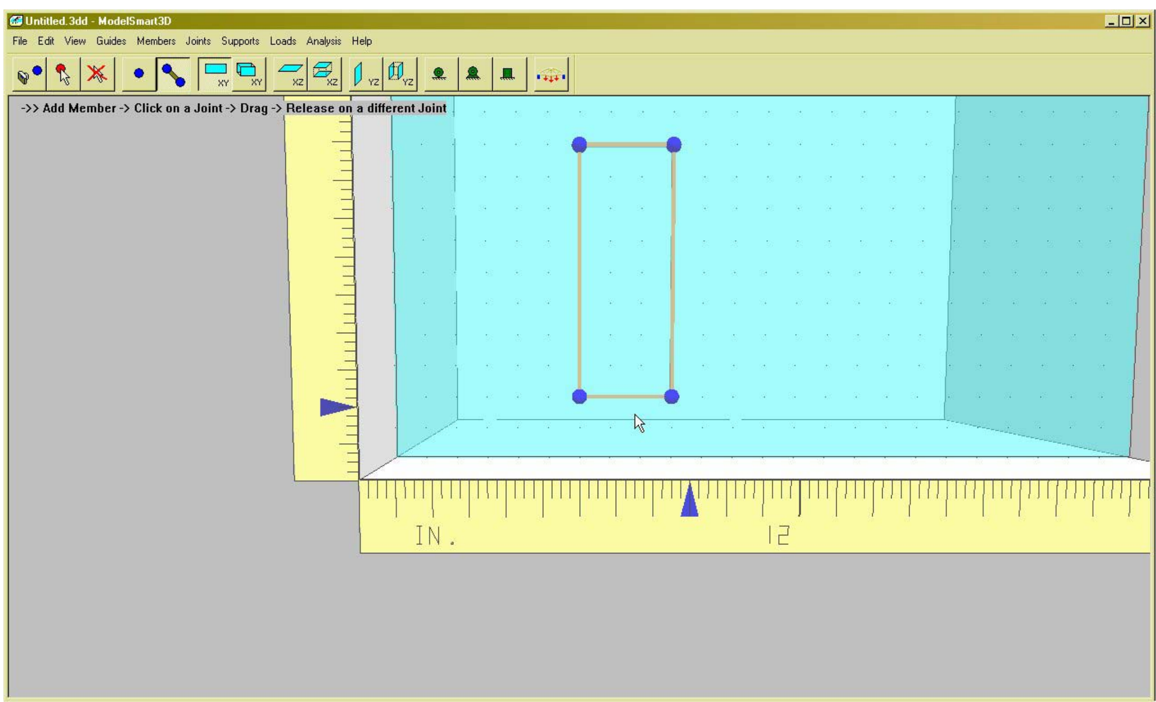
You have just added your first member.

Add a member between the lower and upper joints on the left using the same procedure.

You now have two vertical members (columns) in your tower.
Let's connect them to each other at the top and bottom.



Add two more members.
Add one connecting the top two joints and another connecting the bottom two joints.



Normally, you will want to add a bracing system in your tower to brace your columns against buckling.

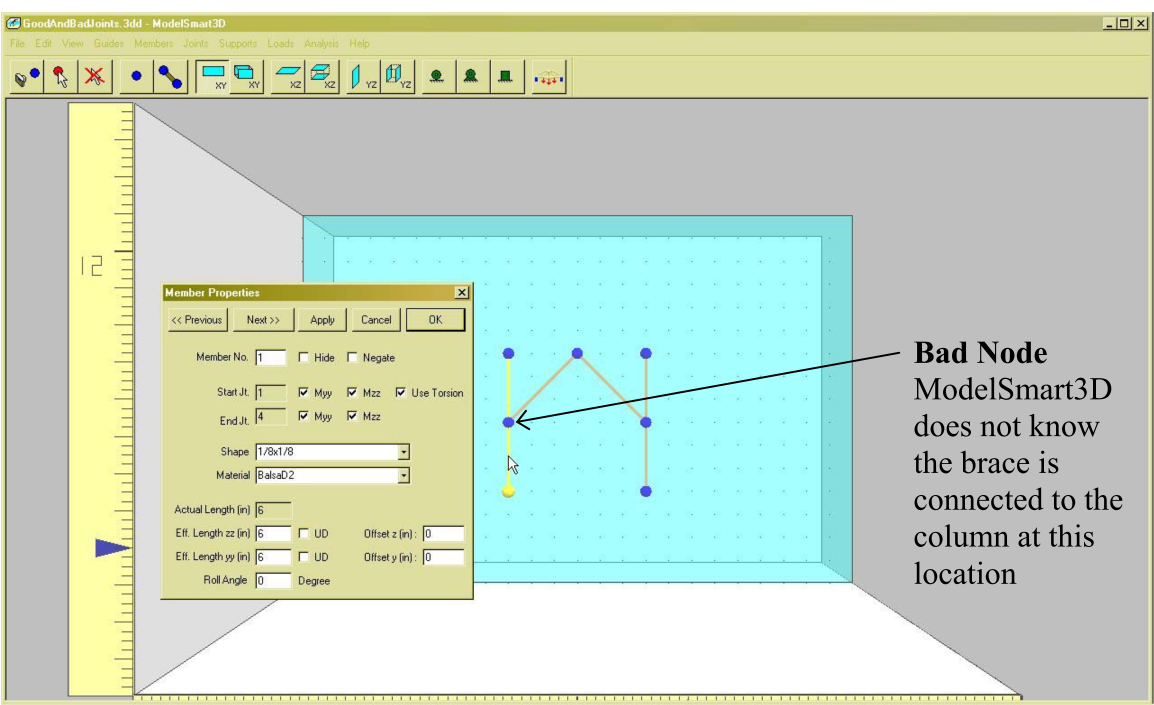
ModelSmart3D model description rule:

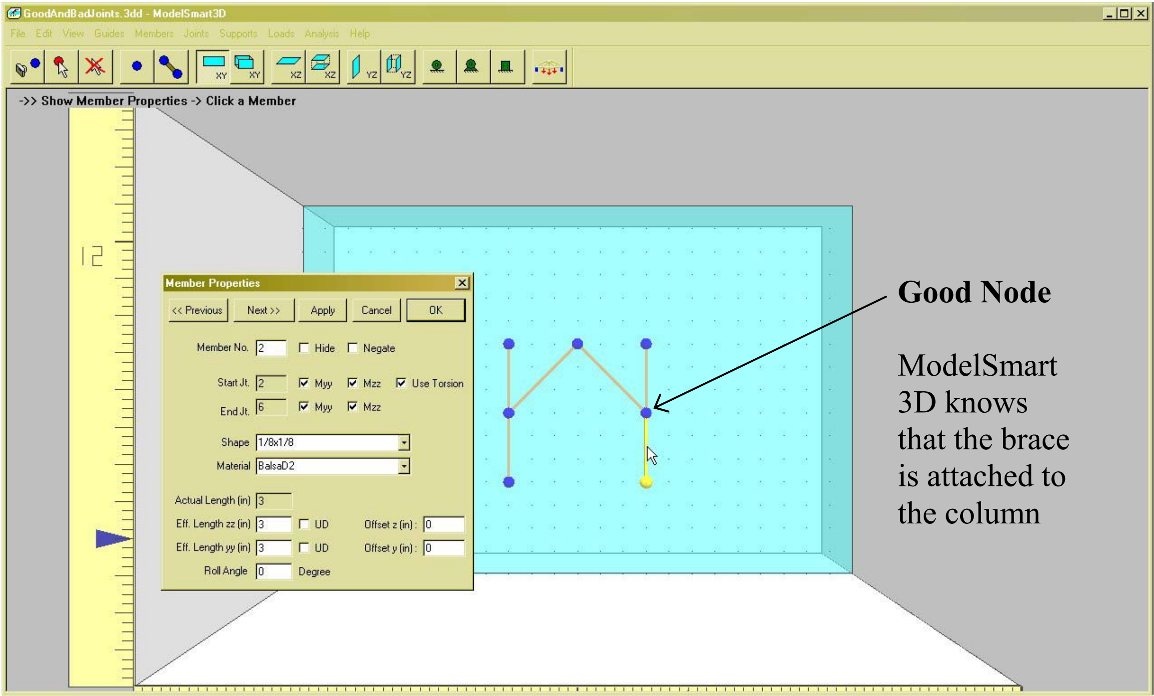
Never draw a member through a joint
without starting and stopping it at that joint.

ModelSmart3D requires a joint at the beginning and end of each structural element in the model. This does not mean that you will necessarily cut the real member at each joint.

Don't think of joints as connections. Think of joints, or joint locations, as informational points of interest – nodes - locations where we want to convey information to the program about the model. Joints (or nodes – you can use these terms interchangeably) are locations where we tell the program that members have a common connection, but also where we restrict movement (add supports to the external world) and add external forces on the model.

By default, ModelSmart3D will assume, for analysis purposes, that members are either continuous (one piece) or glued at joints (nodes) unless you change this default using an advanced member option (member releases). Here's an example:





Good Node

ModelSmart 3D knows that the brace is attached to the column

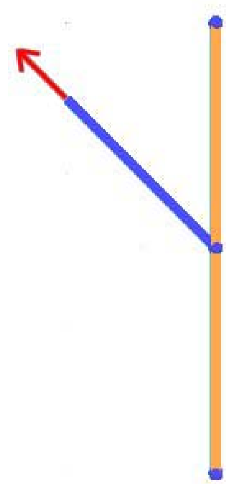
By default, vertical member on the left is analyzed as a continuous member with a brace glued at the location shown.

This might be a good place to point out the **modeler's enemy** – the tension connection.

The weakest connection in your model is a tension connection. The illustration at the left shows a bracing member glued to the side of a continuous vertical member.

The force in the member is tension. The member is trying to pull away from the vertical member. Unless this connection is reinforced by a gusset or the brace member is lapped on the side of the vertical member, the glue at the end of that brace will pull the face grain right off the vertical. A tension force, perpendicular to grain, is the weakest direction of loading for wood.

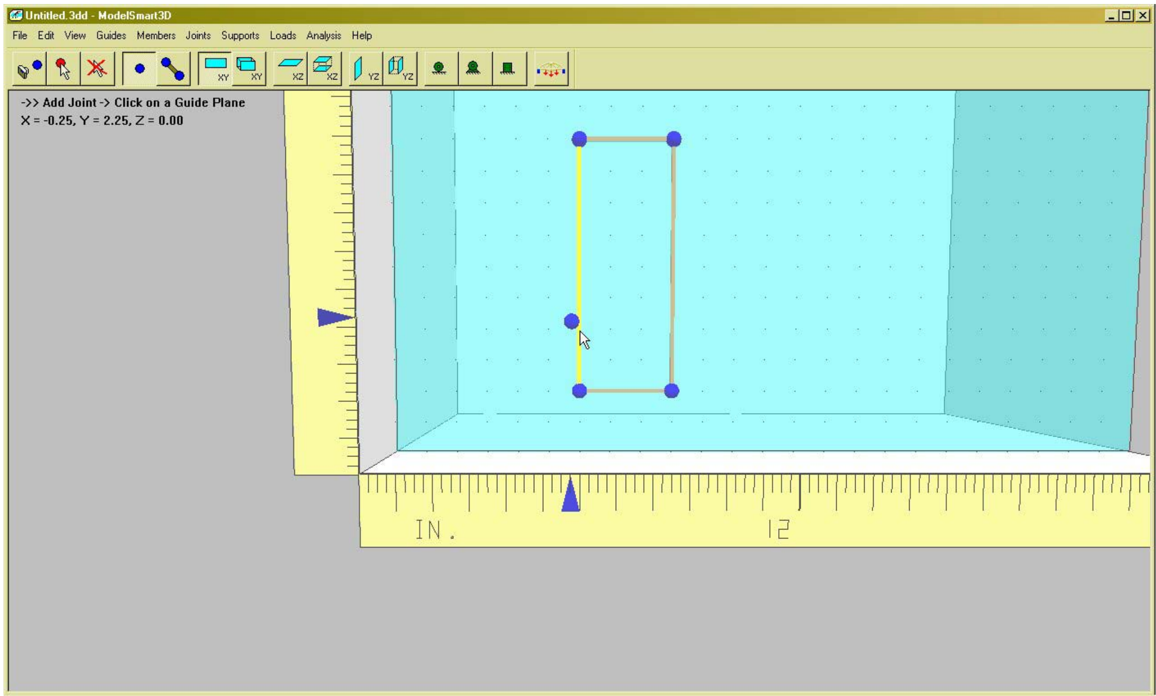
An Asside: To obtain this drawing I used the "Joints|Size|Small" menu option to reduce the size of the joint. This has no effect on the analysis. I also used "Members|Color|Paint Member" to color members.



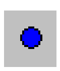
Tension Connection

Let's assume you have decided to brace your columns at the third points.

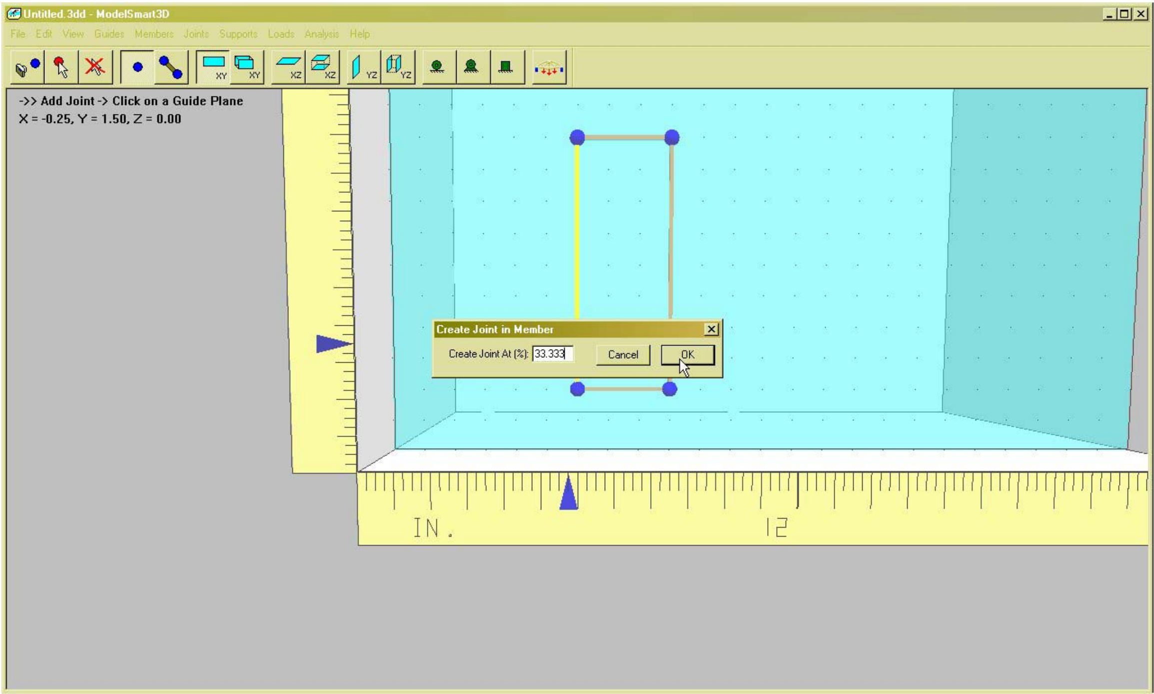
An easy way to accomplish this, correctly, is to use the "Create Joint in Member" dialog.



Select Joints|Use Create Joint in Member" menu option.

Now, click the  "Add Joint" button and point the mouse cursor near the bottom of the column on the left (the column turns yellow). Click the left mouse button.

The "Create Joint in Member" dialog pops up.



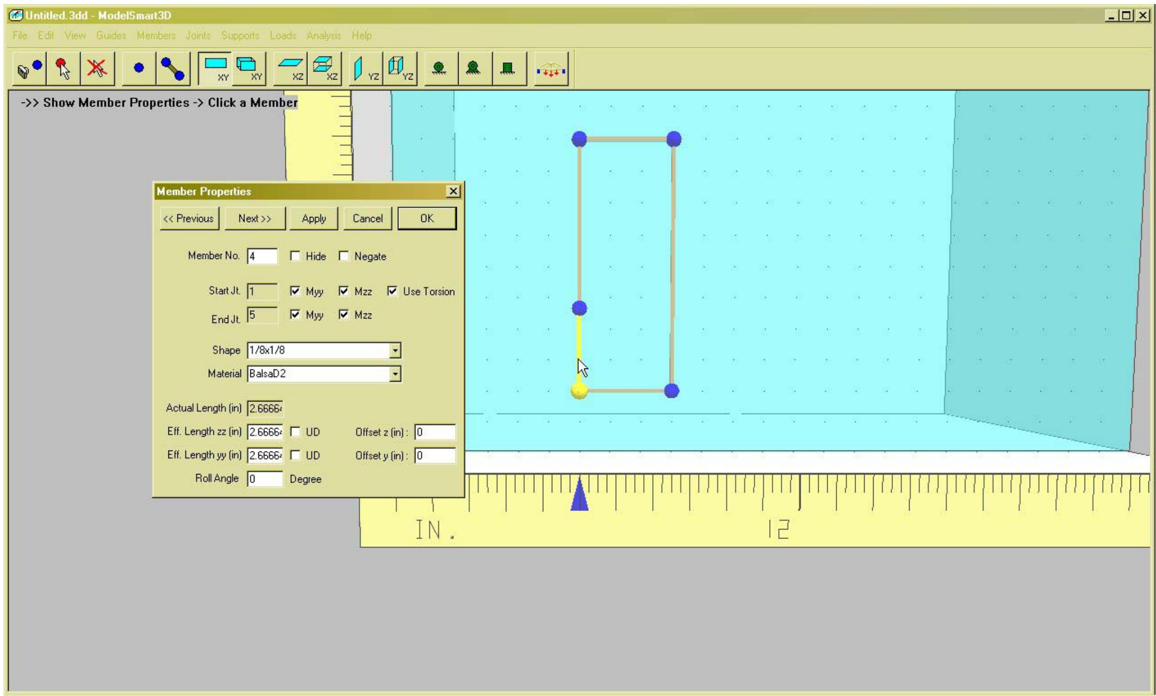
Enter "33.333". This represents, in percent of column length, the distance from the bottom of the column to where you would like to insert a joint (node).

Click "OK".

(We are adding a node to give us a location were we can connect a brace to the column.)

Let's investigate a little. Select the "Members|Properties..." menu option.

Click the lower part of the column on the left. Now click its upper part. It appears to be in two pieces. Click the "X" in the upper right of the dialog to close it.



Remember: This does not mean that you should actually cut your column here at this new node. This is a necessary modeling convention that insures the program will understand that all members at the node (joint) are connected. Internally, the analysis engine will treat the column as continuous and the brace(s) that connect into this location as glued.

Try it again. Put a joint 1/3 of the way up from the bottom of the right column.

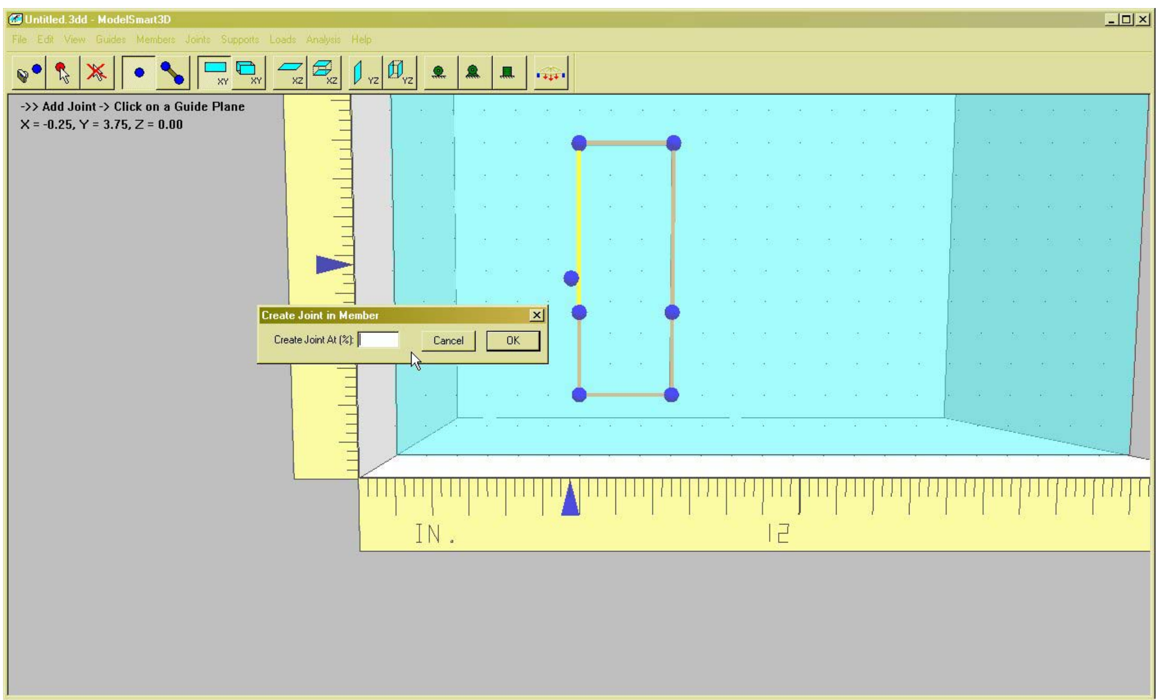
We need to think a little before adding the next joint in the column.

What would you enter into the dialog to place a joint 2/3 of the way from the bottom of the original column on the left?

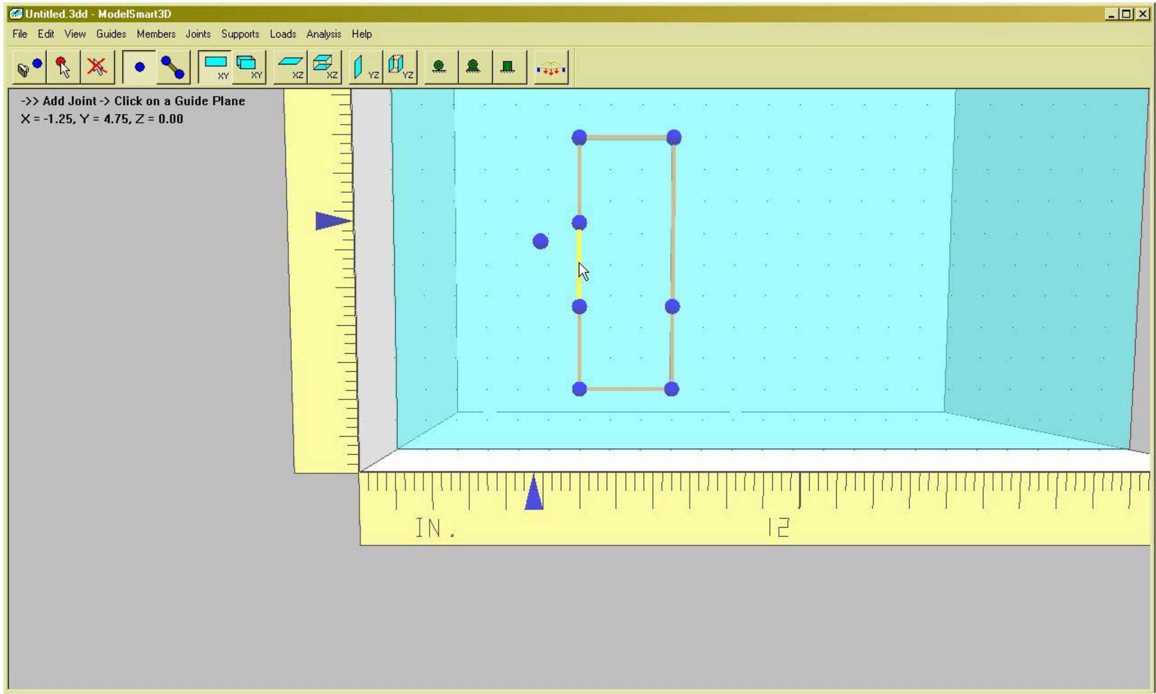
Remember: We wanted joints at the third points along the original column length.

Hint: You subtracted 1/3 from the column length when you added the first joint. Therefore, 2/3 of its length remains.

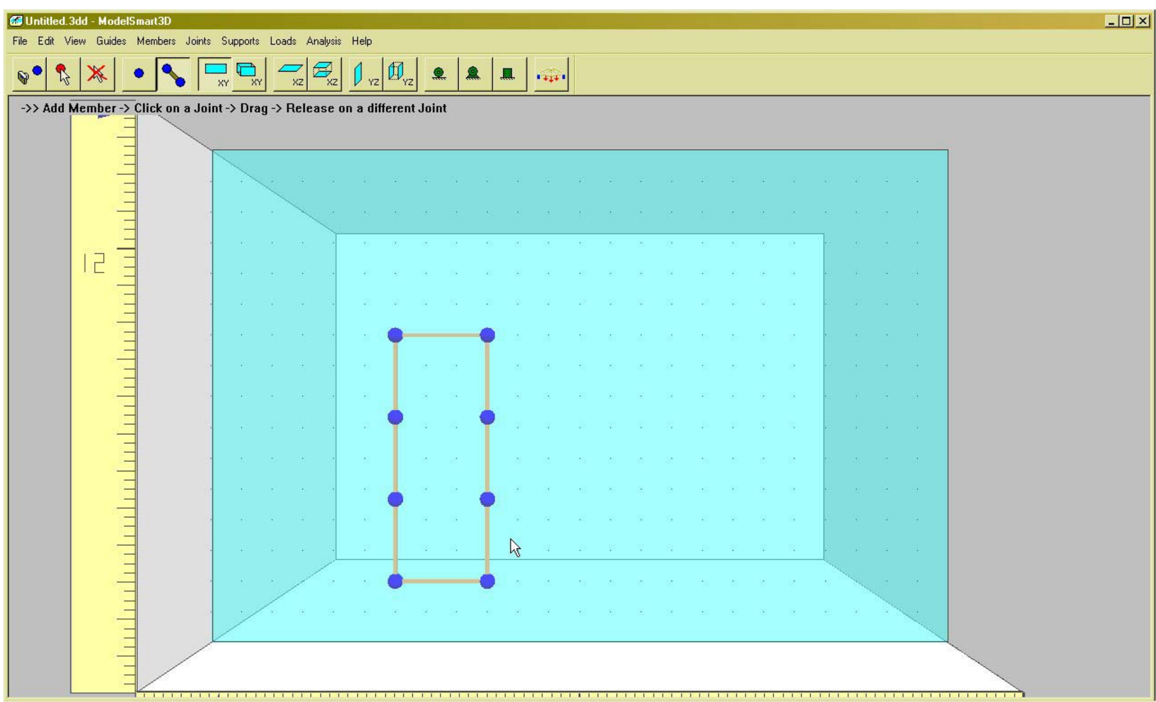
If you divide the 2/3 segment in half (by 2) you will get two 1/3 segments. That's what we want.



Enter "50.0" into the dialog and click "OK"

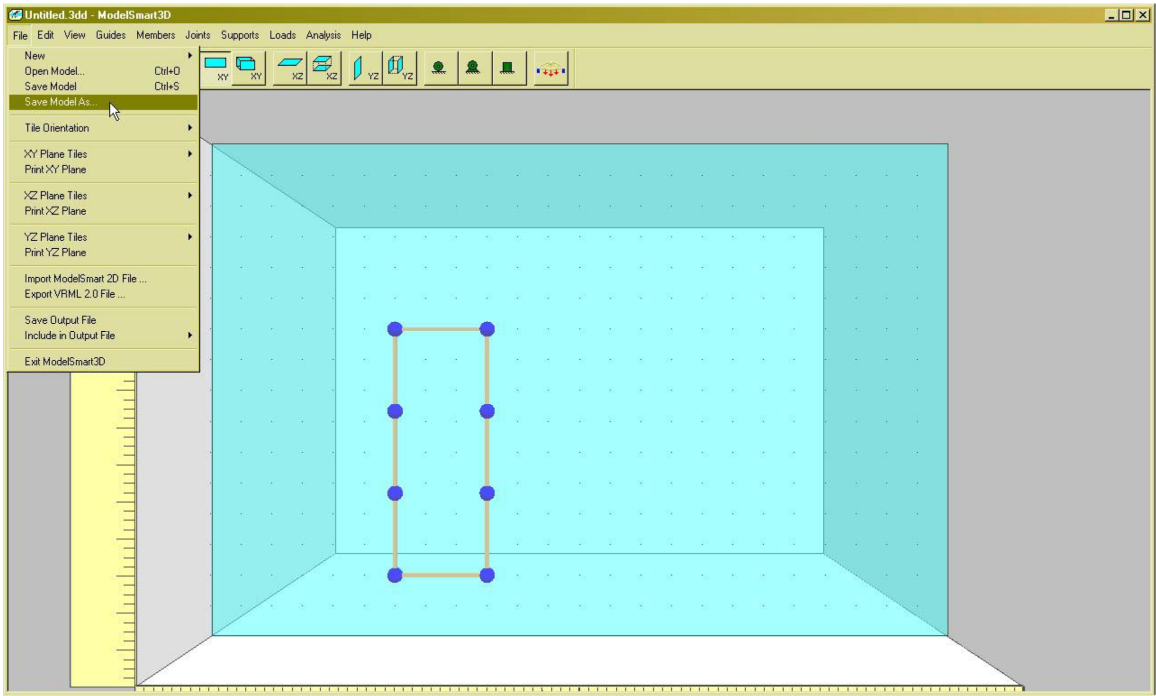


Now, do the same thing to the column at the right

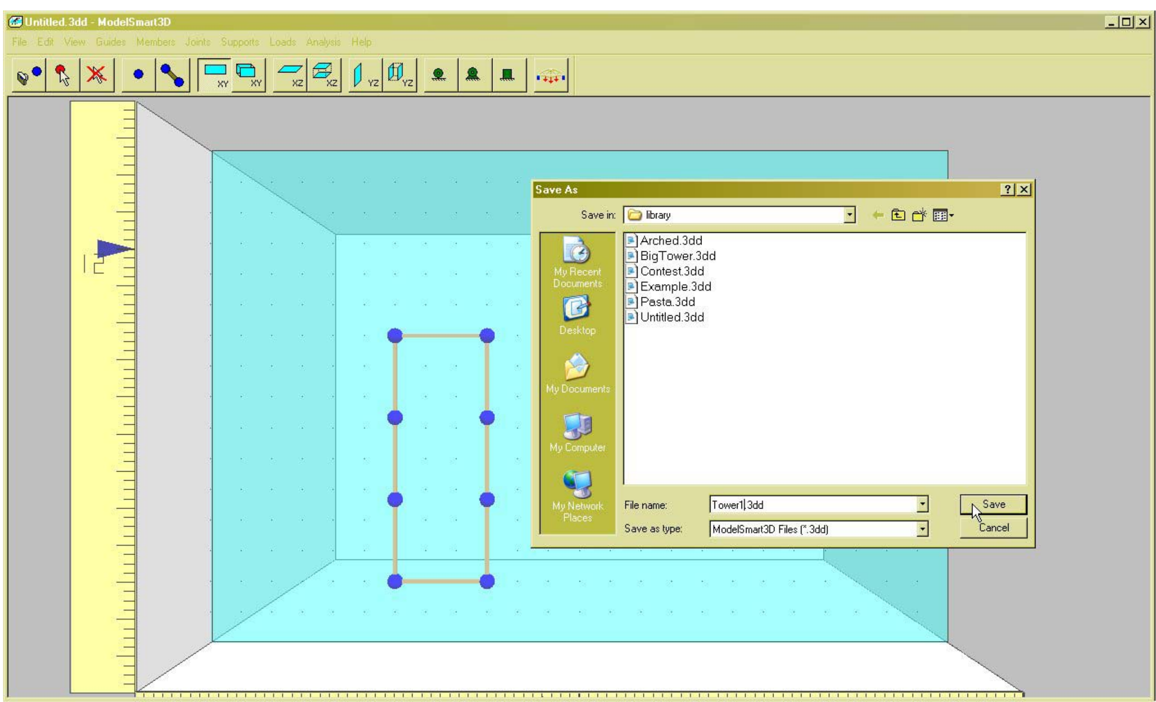


We now have nodes at the third points on both columns. Each column is composed of 3 continuously connected vertical structural elements

Let's save all of our hard work.

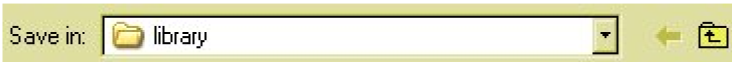


Select the "File|Save ModelAs…" menu option.



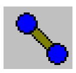
By default, ModelSmart3D saves the model data file in its "Library" sub folder.

It's sometimes necessary to navigate to either the ModelSmart3D sub folder or other folder using Window's navigation tools:

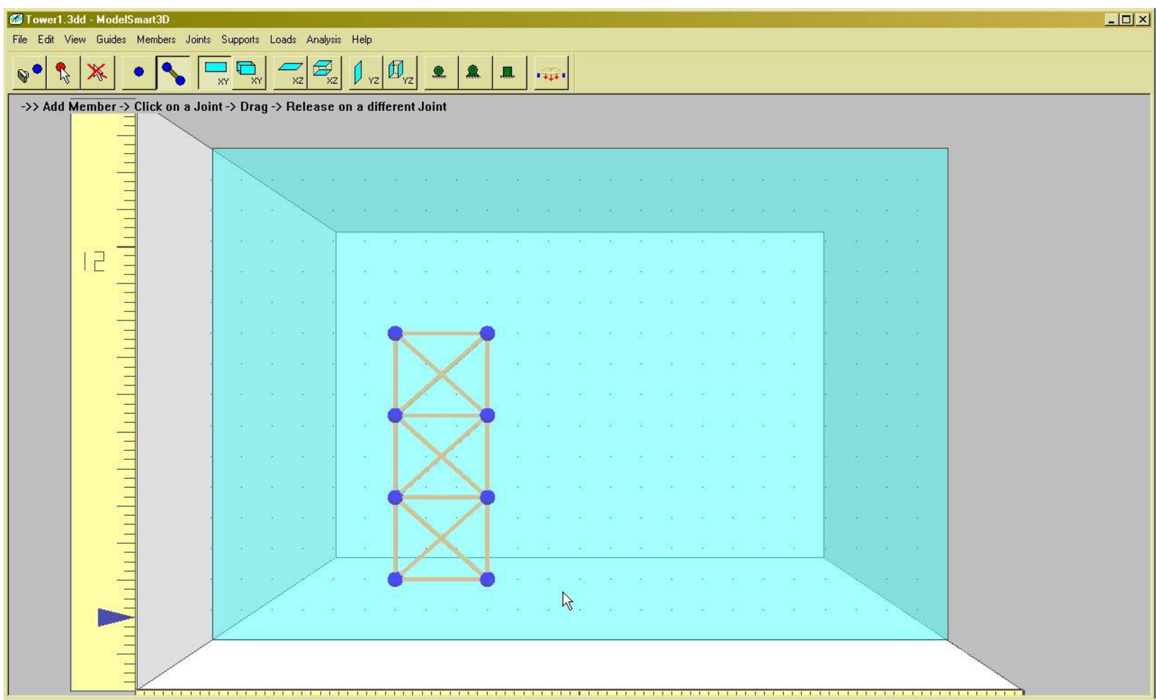


When you have located where you would like to save your file, give it a name (such as "Tower1") and click the "Save" button.

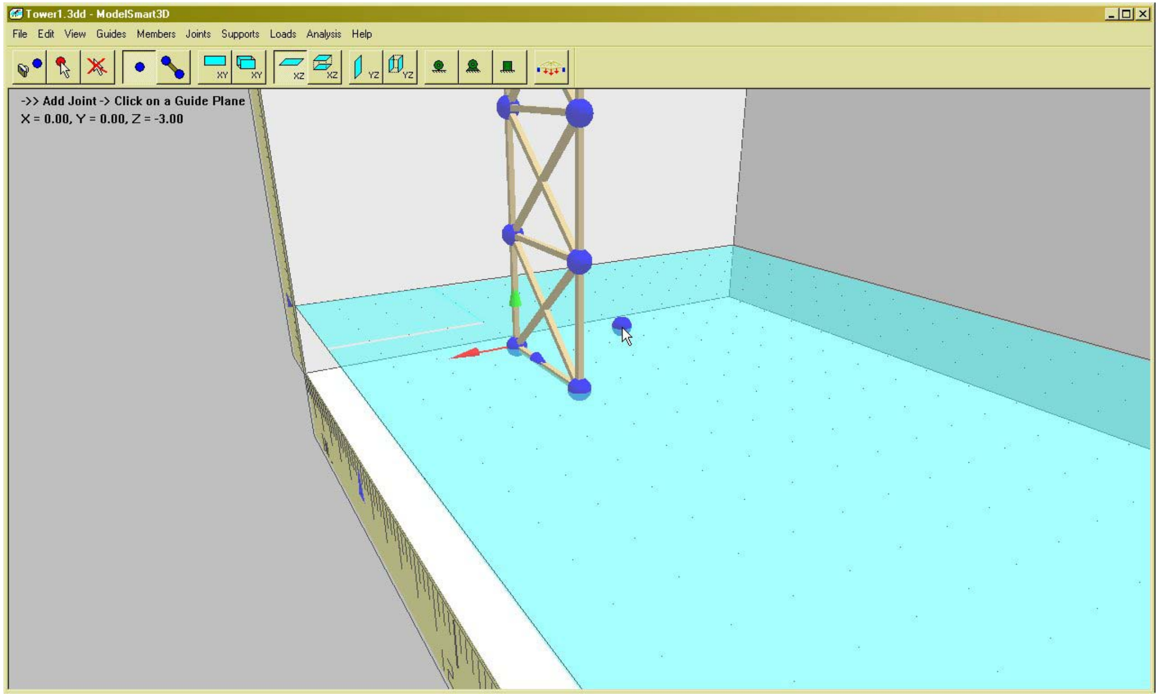
Time to add the bracing members.

Click the  toolbar button to put the program in the "Add Member" mode.

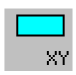
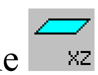
Construct the members shown below using the method previously outlined.



To save your file again, use the "File|Save Model" menu option. Save often!
I like to use the "File|Save As…" before I start making design changes to make it easier to go back to previous version of the original design. You might want to save various design versions as "Tower1", "Tower2", "Tower3"…etc.

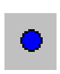


Replicating Joints

Click the  toolbar button to turn off the XY GuidePlane. Click the  toolbar button to turn on the XZ GuidePlane.

Use the "Move Focus" feature and the "Navigation" keys to obtain the orientation of the WorkSpace shown above.

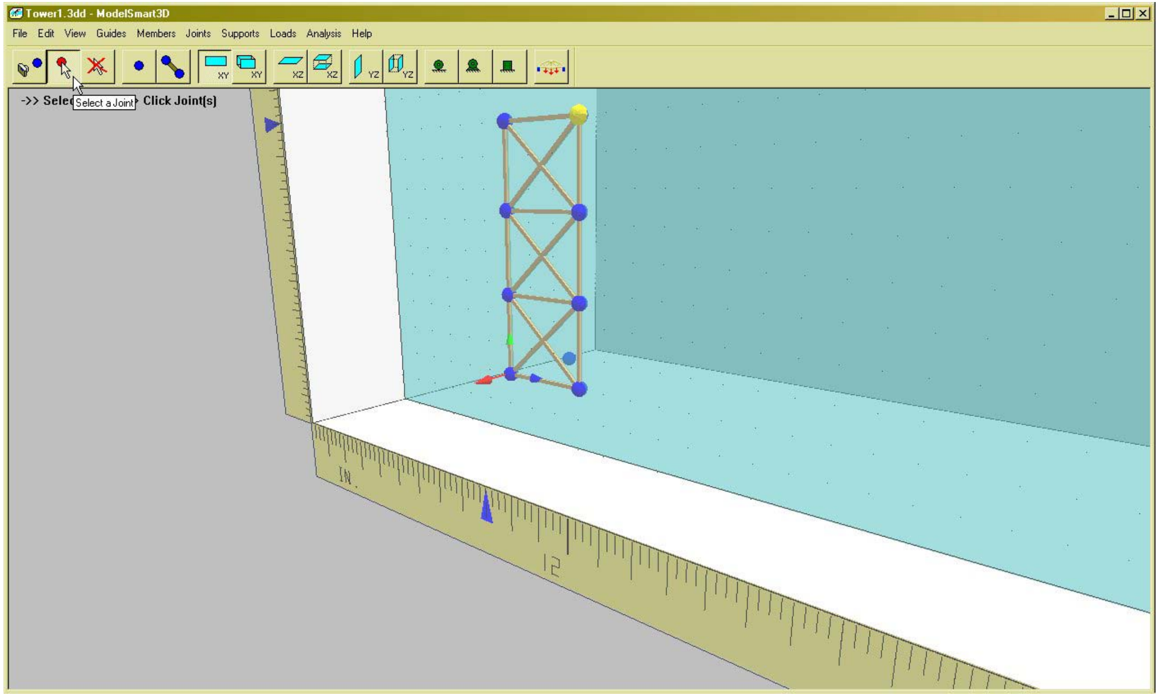
We are preparing to use another handy feature to duplicate the joints in the front side of the tower and at the same time move those joints -3" in the Z direction to the new back side of the tower. (A negative 3" means the negative Z direction – backward.)

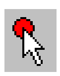
Click the  toolbar button to put the program in the "Add Joint" mode.

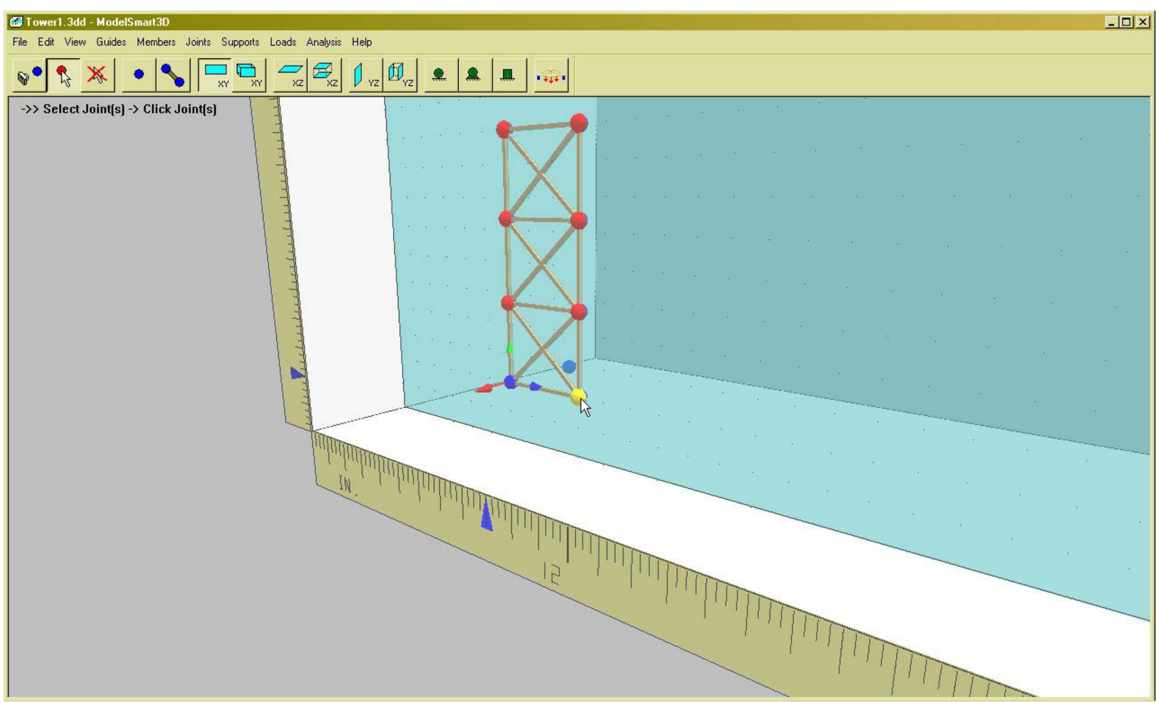
Move the mouse cursor until it points at the coordinate:

X=0.0, Y=0.0, Z=-3.0

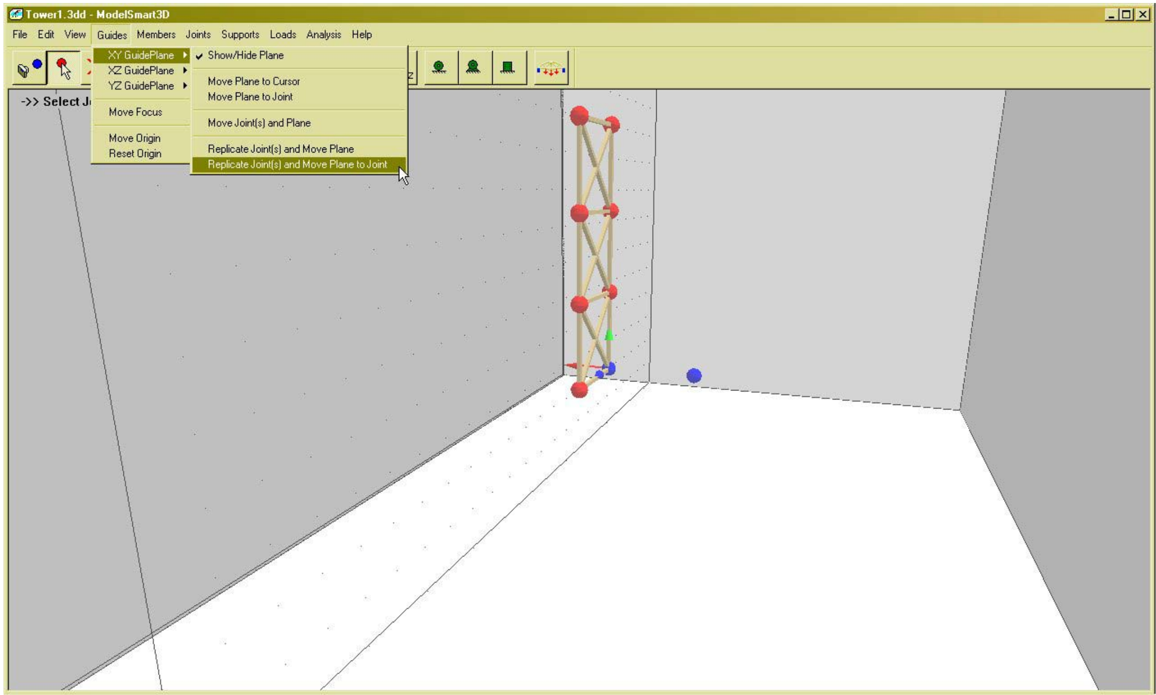
Place the joint.



Click the  toolbar button to put the program in the "Select Joints" mode.

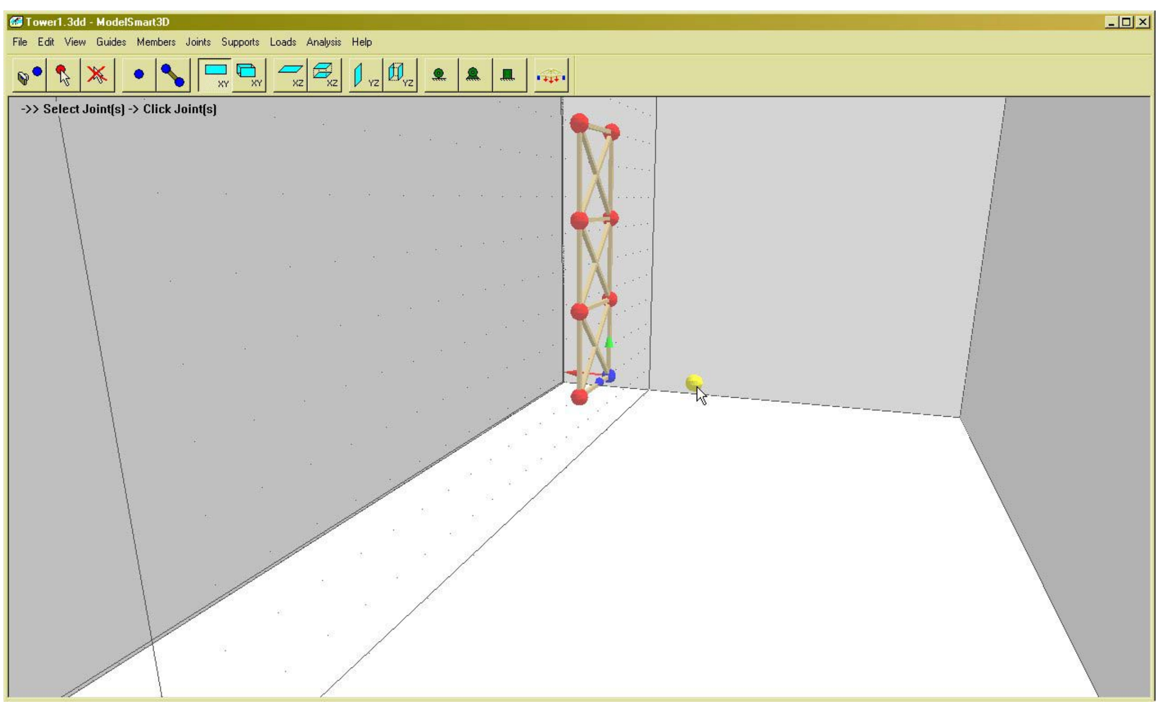


Select, left clicking, each joint in the front side **except for the joint directly in front of the one that you placed at Z=-3.0.**

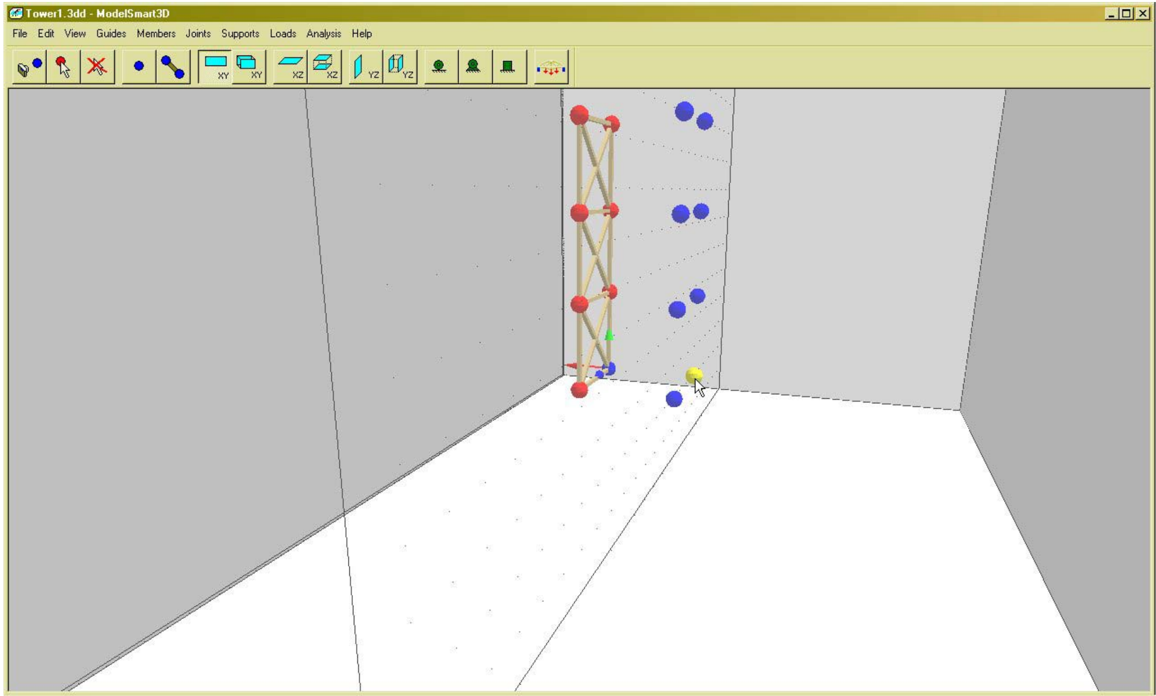


Use the arrow keys to rotate the WorkSpace around until you can see the back of the XY GuidePlane and the YZ Base plane.

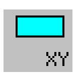
Select the "Guides|XY GuidePlane|Replicate Joints and Move Plane to Joint" menu option.

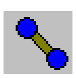


Left click the joint located at Z=-3.0".



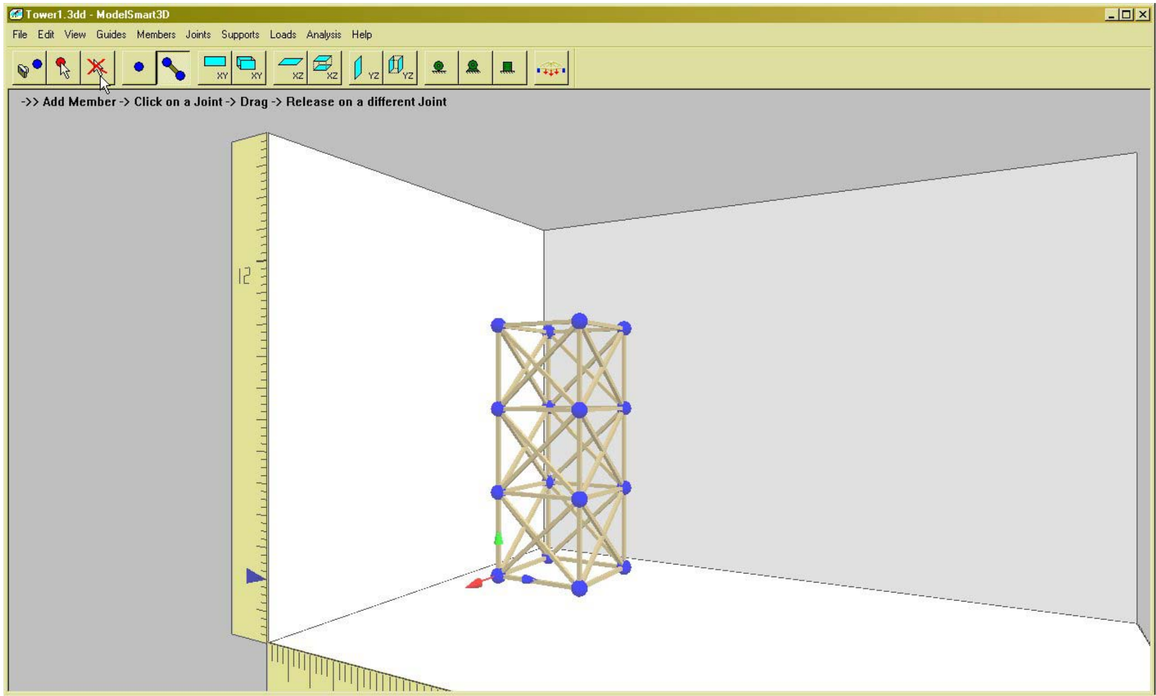
All of the joints for the back side of the tower have now been generated.

Use the  toolbar button to toggle the XY GuidePlane off – no guide plane will be needed to add the remaining members. The program runs much faster without the presents of unneeded GuidePlanes.

Click the  toolbar button. Use the arrow keys to re-orient the Workspace.

If you make a mistake while adding members, you can use the “Members|Delete” menu option to delete misplaced members.

If you are unsure of what got added, use the “Members”Properties..” menu option to take a look.



Click the "View|Show Origin" menu option to hide the origin arrows.

We now have a complete preliminary structure floating in space – there are no attachments to the outside world, no supports.

Let's add some supports.

Supports are modifications to joints (nodes) that restrict movement of that location in certain translational and rotational directions.

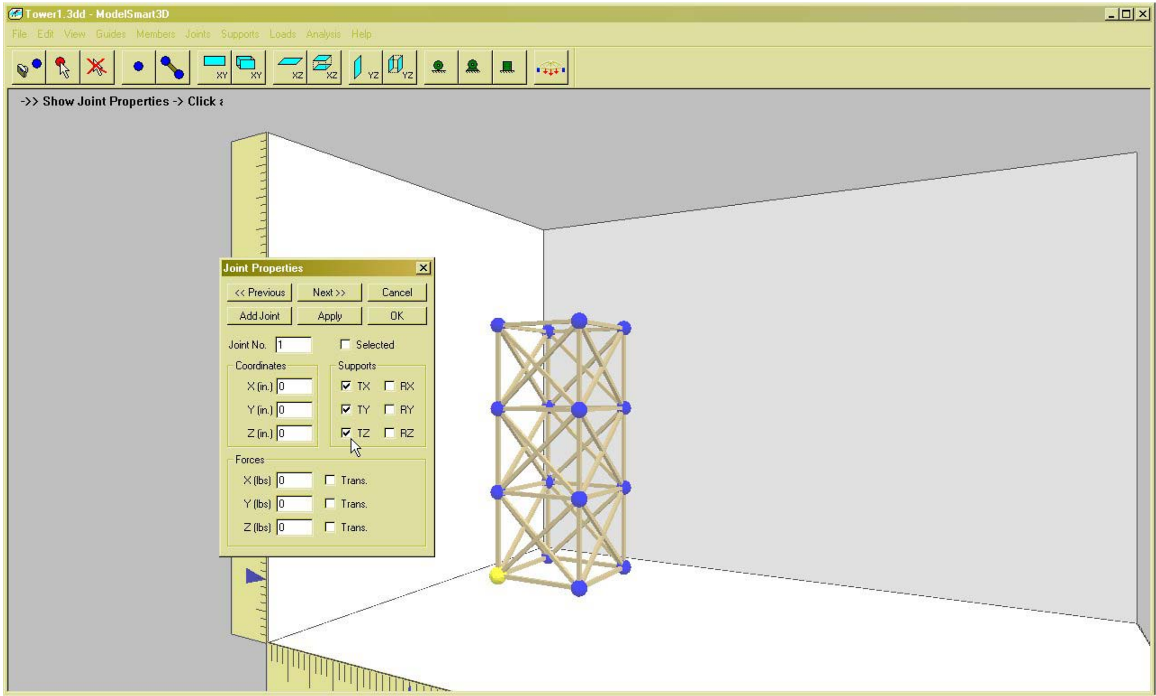
The bottom vertical legs of your tower will be resting on some flat surface and the loading apparatus will be applying a vertical downward force. Therefore, unless the surface is very slick, friction should hold the legs in place.

To simulate this type of joint restraint (support) the "Universal Hinge" is appropriate to use.

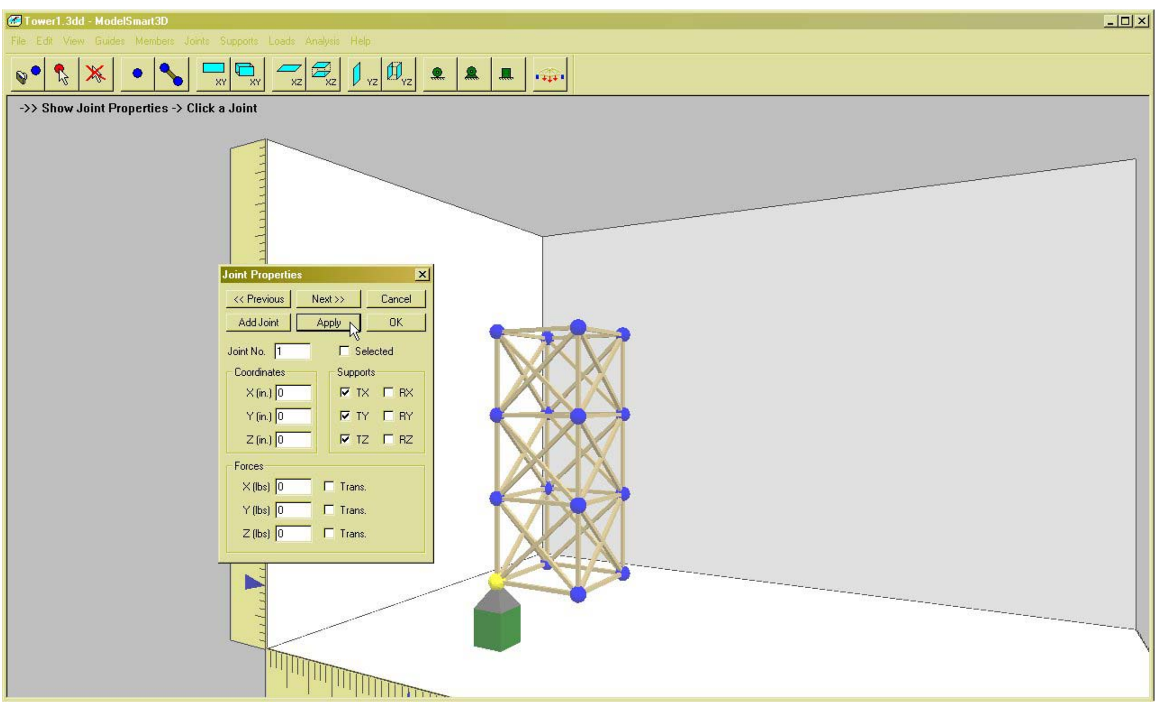
Note: The "universal hinge" is just an engineering symbol (like a math symbol) that represents the fact that we have chosen to restrain translational movement of the joints at the base of the legs in the X,Y, & Z directions.

Try this. Instead of selecting the "Supports|Universal Hinge" menu option then clicking the joints at the base of the legs, let's use the "Joint Properties" dialog and directly set the joint restraints.

Select the “Joints|Properties…” menu option. Click on the joint at [0,0,0].



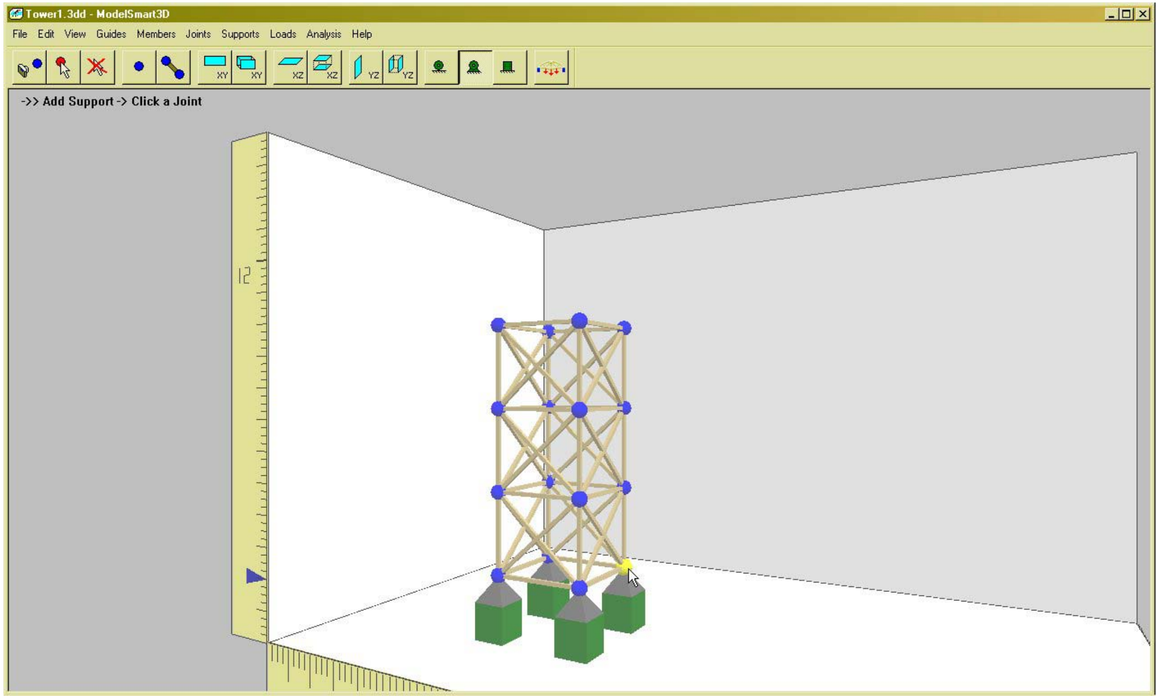
In the dialog, check the boxes for TX, TY, & TZ. This means that you wish to restrain translational movement of this joint in the X, Y, & Z directions. Click “Apply”.



A “Universal Hinge” symbol pops in place!

That was the long way. Close the "Joint Dialog" box.

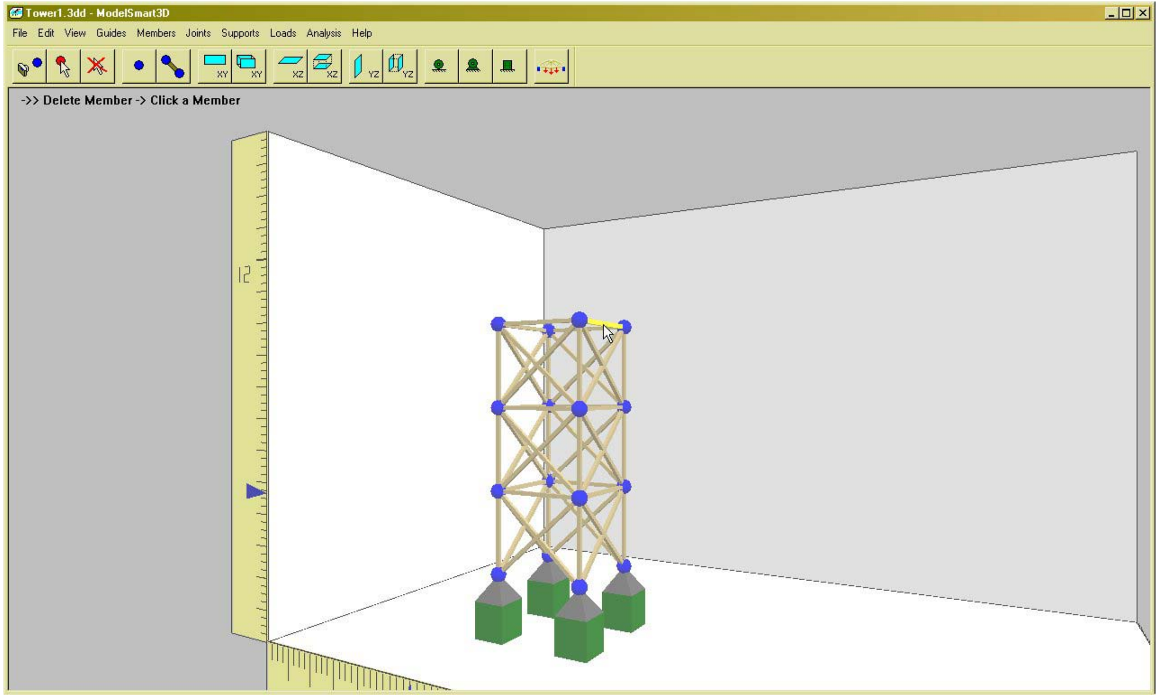
Select the "Supports|Universal Hinge" menu option and left click on the joints at the bottom of the other three legs.



We have supports!

You might want to do this also. If the friction at the base of the tower is sufficient, we might be able to get away with removing the bottom horizontal elements (I guess I could have called them struts, or horizontal braces).

Select the "Members|Delete" menu option. Click on each of the bottom struts, one at a time, to delete them.



You might also be able to get away with removing the top horizontal struts – I'll leave that for you to decide.

Loads! How much and how do we apply them? Should we apply them on the horizontal top struts or at the tops of the columns?

Imagine placing the palm of your hand on the top of the tower and pushing down. What would you feel? The stiff tops of the columns. This is because load (force) is "attracted" to the stiffest elements of a structure. It can be mathematically proven that the rather flexible horizontal struts (when load perpendicular to their axis) will yield in bending much more that the relatively stiff columns will deform in compression.

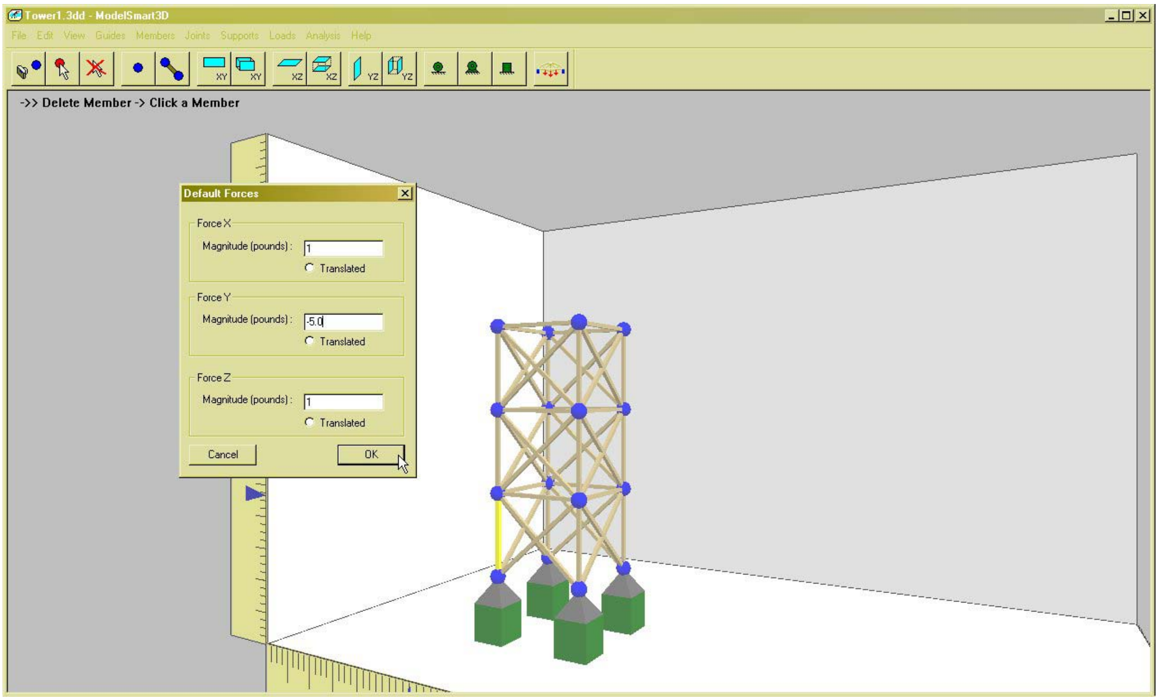
Furthermore the ¾"thick load block, which can be thought of as infinitely stiff in relation to the relative flimsy horizontal struts, cannot deform enough to follow the downward bending to the struts.

What this all means is, put the load at the tops of the columns.

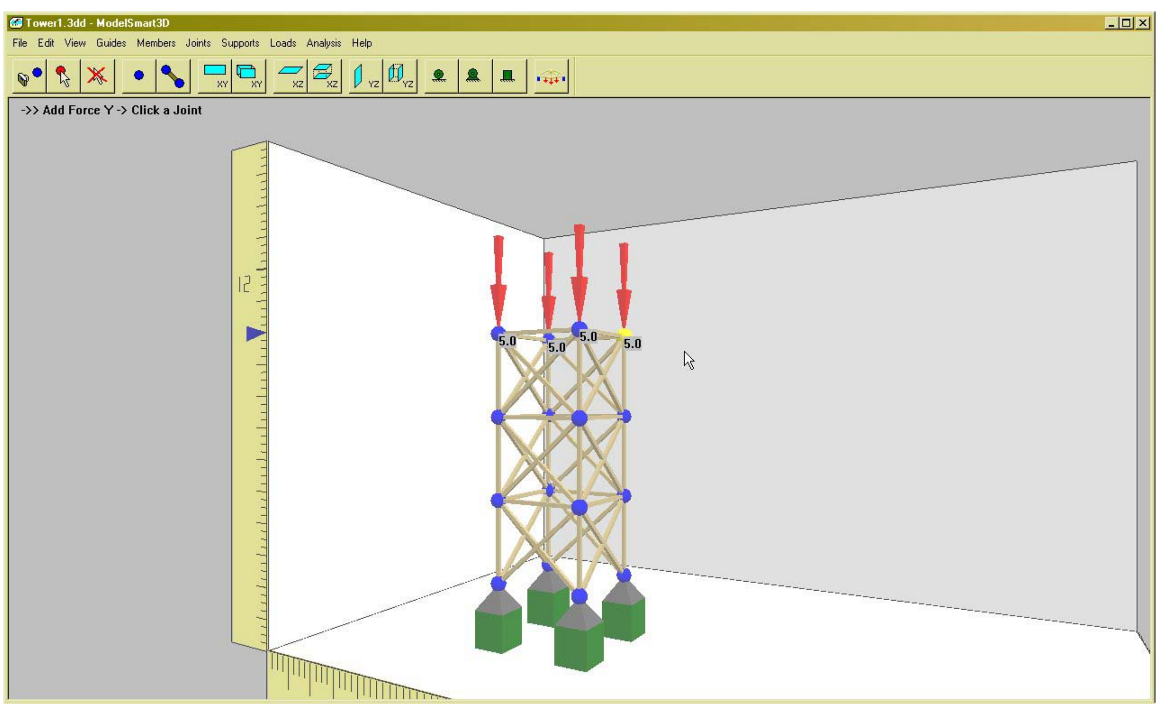
But how much load?

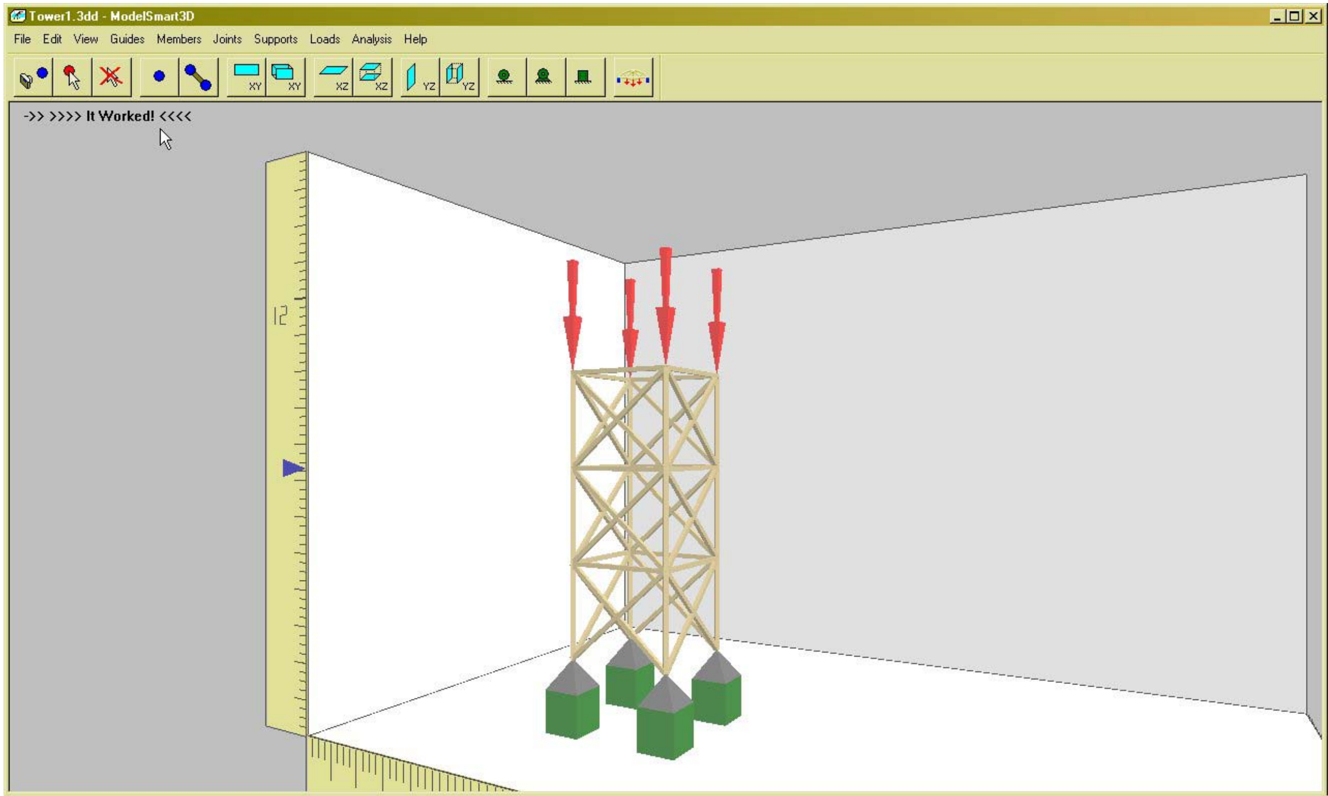
The problem statement calls for a 10 pound working load with a safety factor of 2.0. This means we don't want the tower to fail even if it has as much as 20 pounds on it.

Place 5.0 pounds (20 punds/4 columns) at the top of each column pointing down. Select the "Loads|Set Default Force…" menu option.

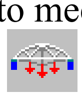


Enter "-5.0" in the box for Force Y (do not check translated). Click "OK".
Select the "Loads|Add/Change Force Y" menu option. Left click each joint at the top of the columns.





That's it. We now have a completed preliminary (not yet optimized or dimensionally adjusted to meet exact specifications for size requirements) model.

Click the  toolbar button or select the "Analysis|Run the Analysis!" menu option to see what happens.

You get a cheer! The text at the upper left of the screen reports, "**It Worked!**".

But how well did it work? Could you have used lighter members, less bracing, lighter bracing?

These are optimization questions and deserve a separate chapter.

You have probably browsed through many of the menu options and features that ModelSmart3D offers and know there are a great many options.

Here's a quick way to find out how much load you tower can currently support.

Select the "Analysis |Analysis Options|Find Breaking Force(s)" menu option.
Re-run the analysis.

The text at the upper left of the screen reports a maximum load of "-103.05 lbs."

That's a downward total load of 103.05 pounds. The problem statement only requires a working load of 10 pounds.

You currently have a safety factor of 103.05/10 = 10.3.

You can definitely save some weight somewhere.

Have fun designing.

See you in the next chapter – **optimization**!