Presentation Script

Series 1 Show 2

IV. Using ModelSmart3D

V. Extra for Experts

Slide 1

This series of slide shows was prepared for parents and teachers of technology students.
It provides a method to introduce students to structural engineering design.
Structural considerations for the design and construction of trusses for model bridges is discussed and an example is presented.

Contact:
Robert A. Wolf III, P.E.
5800 One Perkins Pl Dr - STE 10 D
Baton Rouge, LA 70808
(225) 769-3728
www.pre-engineering.com

Slide 2

Using ModelSmart3D
We'll use ModelSmart3D to analyze a model and interpret the results.

You've heard it is said that smart people learn from their mistakes.
Smarter people learn from someone else's mistakes.

We’ll end this section with a summary of the topics (some learned by others the hard way) that must be considered before testing a model.

Extra for Experts
More information for those trying to design very light bridges.

Slide 3

This is our goal:
To create a computer model that we can use to help predict the load carrying capability of our model bridge.

Slide 4

In order to construct a three dimensional model we must add a third, “Z” dimension to our coordinate system.

Joints must now be defined, to the computer, using x,y, and z. J#(X,Y,Z) (an ordered triple)

3D supports restrict motion of a joint in any or all of six ways.
Rotation about the three axes or translation along or parallel to the three axes.

Members need not lie in plane.

Forces can now point in the “+” or “-”, X,Y and Z directions.

This slide shows the computer modeling tools (symbols) we’ll use to describe our model to the computer.

Slide 5

Here is an example 3d model using some of the 3D symbols.

Joints – Sometime called nodes. These are used to define member end points, load locations, and support locations.
(Joints are not pins or hinges and the presents of a joint does not necessarily mean that you plan to cut the member at that location. A joint is an informational location within the model)

Members – A physical elements that connects joints to form the structure.
(Members will have material properties – balsa,bass,density…
Members will also have a shape – a cross sectional size.
A members length is determined automatically from the locations of its end points.)

Force Vector – Graphical representation of the applied load.

Supports – Engineering symbols that represent the attachments of the model to the outside world
Ie. The bridge abutments, table top etc.

For a model bridge the most appropriate is the hinge roller combination.

Think of how the structure behaves in relation to the tabletop.
The model is not glued to the support surface.
Rotation is not prevented.
Slippage will occur but the model will not slip totally off of the surface.

Slide 6

This is the procedure we will use to convert our 2d computer model sketch into a 3d computer model in ModelSmart3D:

1. Appropriately size our WorkSpace to make sure the program runs efficiently.
2. Use the 2d Computer Model Sketch to enter the main truss.

(We will enter joint#1 at a (X=3,Y=4,Z=8) on the XZ GuidePlane.
Then, move the origin on the 3D coordinate system to joint#1 so that the coordinate
system in ModelSmart3d matches our sketch.)

3. Duplicate The main truss.
4. Connect the two trusses.
5. Then add some additional bracing and joint reinforcements.

Slide 7

Double click the ModelSmart3D icon to startup the program.
(The program starts with the latest "saved" preferences.)
The size of the **WorkSpace** effects the speed of the built-in 3D CAD system (SpaceGuide).

Slide 8

SpaceGuide runs much more efficiently when the WorkSpace size is close to the model size.

Set the Xmax,Ymax and Zmax as shown above.
Click “OK”.

(Clicking save before “OK” will cause the program to startup with the new WorkSpace sizes.)

Slide 9

Press “F1” to center the WorkSpace.
Press the “Down Arrow” a couple of times to move the WorkSpace to a comfortable position.

Slide 10

The navigation scheme used is “object-observer”.
Use the following keys to move the observer:
Up-down arrow keys move the observer closer and farther away.
The left-right arrow keys move the observer around the object.
The “+” and “-” keys make the observer taller or shorter.

I clicked the XY and the YZ GuidePlane buttons in the toolbar to hide them.

The program runs faster if it only has to search the one GuidePlane for your cursor location.

I also clicked the “+” key a few times to get a better view of the XZ GuidePlane.

Slide 11

Members are connected joint to joint.
Therefore we need to put joints in the model before creating a member.

To add joints, click the add joint toolbar button (fourth from the left)
to put the program in the add joint mode.

Move the cursor to where you want to place the joint (watch the coordinates in the upper left corner of the screen).
Place your first joint at (X=3,Y=4,Z=8).

Click the left mouse button to place the first joint.

Slide 12

Next, let’s move the origin of the 3D Cartesian coordinate system so that the programs coordinate system will match the coordinate system we used for the **computer model sketch** from the previous slide show.

Slide 13

I placed 4 more joints from the bottom chord of the main truss on our sketch.

An Aside:
After you have placed the first joint you can select the “Joints|Properties” option from the menu and click on a joint to open the joint properties dialog and from there enter the joints by coordinates if you wish. You do have to place the first joint with the cursor.

Slide 14

Here I’ve turned off the ZX GuidePlane and turned on the XY GuidePlane.

We need to move the **XY GuidePlane** forward so that the 5 existing joints lie in the plane.
This is so we can enter the remainder of the joints from the main truss into the program.

Select “**Guides|XY GuidePlane|Move Plane to Joint**”.

Slide 15

Then click on any joint that want to be in that plane.
The plane moves forward to the joint you selected.

Now we are ready to enter the remainder of the joints in the main truss.

But first, a handy navigation trick.

Slide 16

Here is a useful feature.
The “Focus” option.

Click the toolbar button in the far upper left.
Then click the middle joint in the group of five.
This joint becomes the center of the window and the observer focuses on this joint.

Use the “/” to move directly into the joint and use the “*” key to move away.

Slide 17

Use the movement keys (arrow keys) to align the **XY GuidePlane** as shown.

Slide 18

I've added the top 3 joints as shown.

Now to add members for the first truss.

Click the **Add member** toolbar button. This puts the program into the add member mode.

Place the cursor over the joint where you want to start your member. Click and hold the left mouse button down. Move the cursor to the joint where you want to end your member and let go of the mouse button.

You have just created your first member.

Slide 19

Do not run a single member across joints without letting the computer know that you want those joints connected to the member. The way you tell the computer to connect joints to a member is by adding members joint to joint.

(That is, you stop and start the member at each joint.)

The GuidePlanes are not required when adding members. Turn them off to speed up the process.

Slide 20

After finishing the member in the first truss I hid the **XY GuidePlane.**
I also put the **XZ GuidePlane** back in the WorkSpace.
Place one joint behind the main truss where you want to locate the other truss.

(Place the joint 3 inches behind the right end of the truss.)

We'll use a trick to place the other truss.

Slide 21

Select all joints in the main truss **EXCEPT** the one that corresponds with the new joint added to the other truss.
You will not need to replicate this joint and if you try it the program will not allow the command to be executed.)

(You might want to note this in the manual)

Slide 22

Remove the **XZ GuidePlane** and show the **XY GuidePlane**.

Use the navigation keys so that you can see behind the first truss. As shown.

Select the "**Guides|XY GuidePlane|Replicate joint(s) and move plane to joint**" menu option.

Slide 23

Presto.
Now you have all the joints you need in the second truss.

Slide 24

Put the program back into the **add members** mode and add the members to the second truss.
(The GuidePlanes will not be needed when adding members, so remove them from the WorkSpace.)

Slide 25

Cancel the Joints selections by clicking the toolbar button as shown.

Connect the two trusses to one another at each truss node point (joint).

Slide 26

Select the "**Supports|Universal Hinge**" menu option and place the two supports over the two joints at the left ends of the trusses.

Select the "**Supports|X Roller**" menu option and place the two supports over the two joints at the right ends of the trusses.

Now we need to add some load.

Slide 27

First set the default load.

Select the “**Loads|Set Default Force…**” menu option.

For the **Force Y** enter **–10**.

Click the **translated** button so that the load is attached by its tail.

Click OK.

Select the “**Loads|Add/Change force Y**” menu option.

Click on the joints at the midspan of each truss.

Slide 28

Let’s save our work.

Select the **“File|Save As…”** menu option.

Type “**Filename01.3dd**”.

Click OK.

Now we can test our model!!!!!!!!!!!!!!

Select the **“Analysis|Run the Analysis!”** menu option.

An Aside:

Students should save their work under a new file name before the start of each days work.

This way you will be able to trace their progress.

Slide 29

Select “**Analysis|Run the Analysis!”** menu option the analyze the model with the default analysis options.

The red members failed.

Why did the members fail?

Now for a short discussion about **actual forces** vs. **ultimate forces**.

Slide 30

Lets back up a little.
(The above sketch is not our model. I’m just using it for this short discussion .)

The program analyzes your structure and calculates the (**actual**) internal forces in the members due to the applied external force(s).

Then it compare these actual forces to the **ultimate** force that each member can take.

Slide 31

(Since our truss is composed of triangles, our members are subjected to mainly axial forces – tension or compression.)

Ind Addition, since our model is composed of triangles our members might fail in one of the three mode shown above.

(If you design a model that lacks complete triangulation there will probably also be bending forces involved.)

Possible internal modes of failure for truss members:

1. Very short compression members can crush.
2. Long compression members buckle.
3. Long and short tension members can pull apart.

Slide 32

This is the most important equation in engineering.
This is the essence of engineering design. It is an inequality.

I could have called it the member design equation because
what we do is look at each member.
and compare the force in the member due to the applied load versus the force that the member can withstand.

We use an analysis tool (ModelSmart3D) to find the actual force caused in the member due to the applied load – traffic, load block etc.
Then compare this to what we think the member can carry- (ModelSmart3D does this for you too).

****

An Aside:
In real design engineers want the actual force to be safely smaller than the ultimate force so we divide the ultimate force by a safety factor (sometimes 2 or higher – usually given by the applicable structural code)
Engineers use an equation more like this:
Actual Force less than or equal to the Allowable Force

Where the Allowable force = Ultimate force/Safety factor.

You could do something similar.
Tell the student to design for a total load of 20 lbs with a safety factor of 2 and test their structure with 40 lbs.

Slide 33

Here's an example of how to interpret the member results.
Select the "**Analysis|Member Results…"** menu option.

Click member #1 as shown in the slide.
Look under **Actual Forces**.
Member #1 has 6.24 lbs of compressive force in it (note the "C" by the member #).

(You could also tell that the member is in compression by noting that Axial1 is positive and by the sign convention positive points toward the end of the members at the members starting joint. Therefore the member is in compression.)

Now look under **Ultimate Forces**.
Member #1 can take an ultimate force of 5.62 lbs in compression.
Since 6.24 is not less than 5.62, the member fails.
Look at the Mode Ratio – the member is about 11% over stressed.
Look at the Interaction Ratio (A-M) it is about the same because there is very little bending in the member.

Slide 34

How can we keep the members from failing?

Use bigger members. Not longer. Chunkier.

A chunkier cross-section.

Let’s change the shape of the members that failed to **3/16x3/16**.

Select the “**Members|Set Default Material and Shape**” menu option.
Use the drop down box to change the default shape to **3/16”x3/16**”.
Leave the material set to **BalsaD2** which is a medium grade balsa.
(We will work with material grades later.)

Slide 35

Select the “**Members|Change Shape**” menu option.

Change all the red members to the new default shape by clicking on each one.

Slide 36

It worked!

That is, the computer model worked.
You should be thinking:
How will this compare with the actual model that I will build?

Your “computer model” in an ideal model.

The program assumes that you will build your model exactly the way it was entered.
In fact, your real model will probably not be constructed so exactly.
It may lean slightly to one side.
The test load might be placed slightly to one side.

These effects can cause your model to want to tilt to one side as it is tested.

Also when compression forces are created in the top chords of the trusses, there is a tendency for the top chords to want to buckle together to the side (laterally buckle).

The program can consider lateral loads – the problem is you are not testing you model with any lateral loads.

You should!

You should apply a lateral load of somewhere between 2-5% of the max internal force in the top chord of both trusses.

Slide 37

Let’s check the force in the top chord.

Select the “**Analysis|Member Results**…” menu option then click on the top chord member as shown in the slide.
(Your member number might not match the member number in my “Member Results Dialog”. This will depend on which order you numbered your members when you built your model.)

The top chord of the truss has an internal force of 3.75 pounds.
(Since there is only one panel to each side of the center span and since the truss is symmetrical about the center span the chord force of 3.75 pound is the maximum internal top chord force.)

There are to trusses. Therefore we need to apply a lateral force of .05 x 2 x3.75 = .375 pounds

Let’s use .4 pounds.

Here’s a neat trick.
You can use the “**Joints|Properties**” dialog to change and/or add loads to the joints.

Select the “**Joints|Properties**” menu option. Click on the joint shown in the slide.

Input a “Z” force of .4 lbs “translated” as shown in the slide.

Let’s test it again.

Slide 38

The model seems to have enough lateral stiffness.

It would be a good idea to add some additional miscellaneous bracing to prevent the lateral movement from becoming excessive and to keep the top chord from buckling as one member.

Slide 39

You should provide the additional bracing shown in gold.

(The reasons for adding this extra bracing is beyond the scope of this introductory example. If you wish to know more about the subject of lateral buckling and effective length, see the "Extra for Experts" section.)

Next, let's consider a situation in which we want to design the bridge to accommodate a load block.

Slide 40

Let's think about how the load, from a load block, gets into the model.

In relation to the members in the truss the load block is infinitely rigid.

Slide 41

Think about how the structure behaves under load.
The truss members moves away from the load block as the load is applied.
The load application point moves out to the edge of the load block.

A concentrated load should be placed where the supporting members touch the edge of the load block or at the truss joint supporting these members.

If there is little distance to the truss node just put the forces at the joints.
(Otherwise, you will have to create joints in the members and place them there.)

Slide 42

I moved the joints in the bottom chord to accommodate the load block.

I placed $1/4^{th}$ of the total weight to be hung on the load block on the joints shown in the slide.

Slide 43

Another problem is getting the load into the other parts of the structure without having a local failure.

It would be better to use a strut in bearing on the bottom chord instead of relying on the shear at the end of the member.

Also, add some gussets on each side to lesson the chance that the bottom chord will be pulled off of the trusses by the load.

Slide 44

Gusset Plates

Wood gussets can be used with the gain running parallel to load.
You can also use strips of Hat buckram.
(Hat buckram can be purchased from fabric stores. It is used in making lady's hats.)

Slide 45

Analyze your bridges with some lateral load.

Bridge are not built with perfectly true geometry and loads are sometimes placed less than perfectly.

Provide some additional miscellaneous bracing to limit lateral movement.

Use gussets at tension connections especially near the point of loading.

Use gussets at tension or lap members at all tension connections.

Adding an extra joint in a member that is not needed will cause the computer to use an incorrect buckling length when calculating the ultimate force in compression.

Remember your goals:

Create a mathematical model that accurately reflects the behavior of real model you plan to build.

Use the mathematical model to design a light structure that supports the load.

Slide 46

ModelSmart3D assumes that each joints is laterally supported.
It uses the distance between joints as the effective buckling length for a member in compression.

The member “A” is in compression. The program is currently using the default buckling length defined by its end points to be 3”. The buckling mode for this length would appear as buckling up or down and taking on a shape similar to curve “A”.

There is another way this member could buckle.

It could buckle (to the side) along with the next member (Member "B") to it into a shape similar to curve "B". And this is probably what would happen first.

To more closely model this you must change the effective length of both members to 6". View the next slide to see how to do this.

Slide 47

Select the "**View|Advanced Options**" menu option to turn on advanced options.
Select the "**Members|Properties**…" menu option.
Click the member shown in yellow on the slide.
Change the Effective Length (YY) to 6 and check UD (for user defined).
Click The "Apply" button then the "OK" button.
Change the effective length (YY) for each of the members in orange.

Slide 48

No.

That would just make all of the members buckle at the same time!

Slide 49

In addition to adding the cross member, you must add two additional members to form a horizontal truss in the top of the bridge.

As mentioned before, to keep the top chord from buckling you should be able to support a lateral load equal to about 2-5% of the maximum internal force in the top chords.

The force in the top chord of one of the trusses is 3.75 pounds.
Therefore, we should apply a lateral force of .4 pounds (3.75 x 2 x .05).

Slide 50

It worked.
Had it not worked, you would probably of had to add the other bracing show in slide 39.

Slide 51

If your model cannot support the suggested lateral load, you may want to consider adding one of more “Portal Frames”.
You could add one at each vertical truss member.