Module: AutoDesk Inventor for Problem Solving

Review: Middle-Out Design Strategy, Breaking Out Components, Workplanes, Mirroring,

DO THE WORK DESCRIBED AND PUT THIS INPUT FILE IN YOUR ELECTRONIC PORTFOLIO

Be Sure That You Are Prepared for the Exam… Accordingly…

CONTENTS

1 Expectations -- The Student Will:

#	Expectation	Self-Assessment: How Did I Do?
1	Use a design methodology which will result in the creation of a distinct shop drawing for each part of a product that is cut from stock.	
2		
3		

1 Fundamentals: Unpacking Some Key 3D CAD Learning (Review)

1.1 Inventor is Feature-Based

1) We can use Inventor to "synthesize" (build or create) a part / product one feature at a time. For example if you were to design a box, you could start by sketching and then extruding the bottom of the box (because it sits on the living room floor.) The bottom or floor of the box would thus be your "base feature". Then on a new sketch on the top surface of the box floor, you could sketch one of the sides of the box and then another and another

2) As you add features to the box, watch the Inventor model browser window -- the new features and the sketches on which they are based will be added to the model browser tree
3) You add one feature to the first feature and so on -- every new feature has one or more relationships with existing features:
   a) If a new feature is to be part of a molded product... the new feature is effectively part of the same solid object – so use a "Join" operation when extruding this new feature
   b) If a new feature will be an independently-made part that is later fastened to the base feature with screws for example, it will be a "New Solid" in the Inventor 3D extrusion process
4) Modify the features in various ways (eg Fillet, Chamfer)
5) Output .idw (drawing files) for each part – take the hard copy drawing files to the shop

1.2 Inventor is Parametric

1) New design features reference (ie "refer to") existing design features -- that is, a new feature can only be added in a strict relationship to existing features in the model (or in relationship to a "work feature" as explained in the "Work Planes" lesson)
2) Dimensional relationships are all given unique names in the Inventor database, so they can be changed at any time -- right-click on a dimension to display the name and the current value of a dimension
3) These relationships (dimensions, angles etc.) are all "***parameters***" of the model design -- a parameter can be changed at any time and the entire model will then be updated by Inventor
4) You can address some of your product design intent by creating User-Defined mathematical relations -- this involves creating equations that relate parameters to one another
5) You can easily output variants of your design simply by changing the values of one or two parameters and saving the file under a new name – eg variant_long.ipt and variant_short.ipt. You will be doing this in the Critical Challenge below.
6) Refer also to the "Dimensioning Strategy" in the "Design Strategy" lesson

1.3 Think About the Optimum Approach / Design Strategy – Then Make a Decision

1.3.1 Bottom-Up Design

As in the Backyard Planter design situation (6_Inventor_2014_Assembly_Bottom-Up.doc), you may:
1) design parts as distinct .ipt files
2) assemble the parts together in an assembly (.iam) file using assembly constraints or joints
3) print the .idw drawing files for all distinct parts as well as for the assembly
4) take the idw printouts to the shop, make the parts and assemble the parts together

This strategy has great value when multiple instances of a few distinct parts are assembled together into a final product.

1.3.2 Single Massive Solid Design – Using Join to Extrude Features

If you are:
1) designing something to be made in a mould or
2) intending to carve a product out of a single block of wood

You may certainly use a "Join" when extruding a new profile from a sketch that was drawn on the surface of an existing feature. The result will be just a single solid that embodies both pieces that were joined together. The drawing .idw file will treat the product as a single solid piece.

Recall that you had used this strategy in the 4_Inventor_2014_WorkPlanes.doc assignment – and in that assignment you were to do some critical thinking about the validity of that strategy when making such a tool in the woodshop.

1.3.3 Is there a Better Way? – Extrude: New Solid – Middle-Out Design

Ask yourself – and then look for a way to do this in Inventor…

1) Will it be easier to design your product in one master .ipt part file using various NEW solids and then propagate the several individual solids to related *.ipt part files in an associated assembly file?
   a) And, yes, there is a way to do this in Inventor:
      i) **Tip: When consuming sketches by extrusion, use "New Solid" instead of "Join". Close the Extrude dialog for each new solid, sharing the sketch if necessary. Then, when finished modelling, use Make Components to generate the ipt file for each Solid as well as the assembly .iam file.)**
2) This "Middle-Out" design strategy makes good sense if your product is a number of distinct parts that are fabricated independently and then fastened together (eg using screws, nails, glue) at assembly time.

1.3.4 Design Strategy Decision

For your shop design / build projects, you will need to investigate and decide – up front -- whether you should use a single massive solid, bottom-up or a top-down strategy – or some hybrid strategy in between such as the so-called "Middle-Out".

You will also need to decide whether or not you will be working in the shop in metric units or in imperial. For holes that you will need to drill, have your list handy of drill sizes. Your shop is likely to have more imperial size drills than metric size drills. Set your 3D CAD environment accordingly.

In your project research report, write a new section called "Design Strategy" -- and document your decision-making accordingly. Once you are satisfied that your strategy will lead you to efficiently creating what you will need to take into the shop – ie shop drawings -- for efficient and accurate fabrication, you can start modelling in 3D CAD.

CAUTION: DO NOT DELETE ANY OF THE STANDARD TEMPLATES.

2 Project Initialization Procedure (A Review)

Refer to 6_Inventor_2014_Assembly_Bottom-Up.doc for an introduction to and tips on Inventor Project creation and maintenance.

1. Launch Inventor
2. On the Welcome screen, click Configure Default Template. Be sure that:
   a. Measurement Units are mm and
   b. Drawing Standard is ANSI

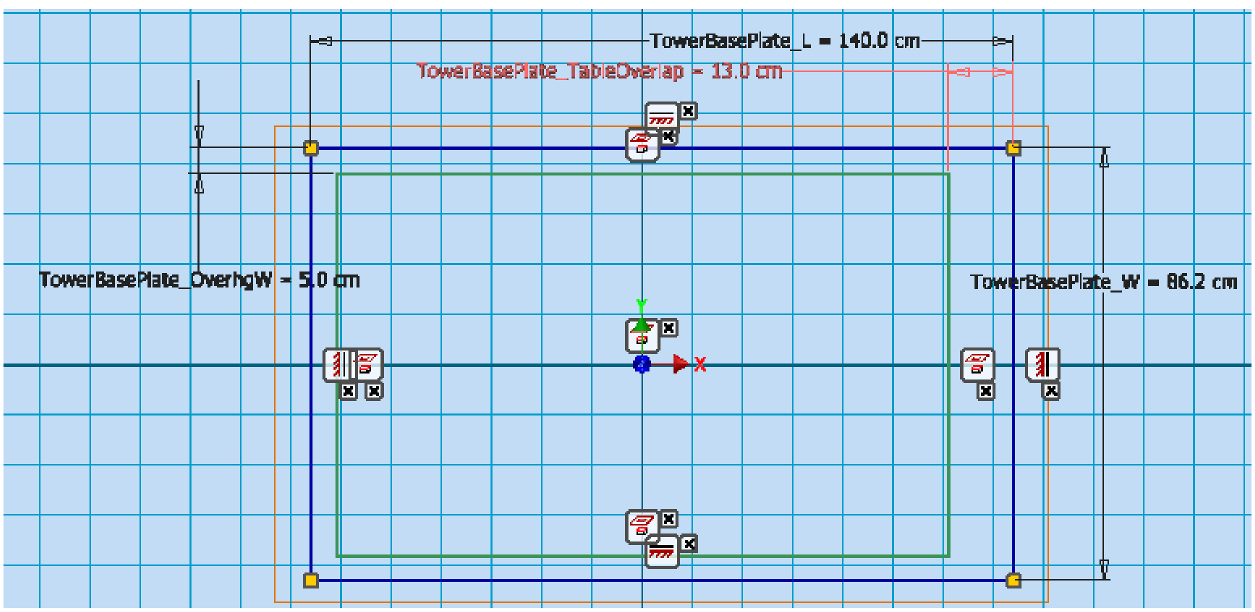
3. After closing the Welcome screen, create a new folder on your USB drive or home drive in which to store your new Inventor Project -- Projects\YourName_MiddleOut_Review
4. In Inventor at the Application icon in top left, select Manage / Projects
5. Create a new Project by clicking New at the bottom of the dialog. In the wizard, select:
   a. SingleUser
   b. Give the new Project a name -- YourName_MiddleOut_Review
   c. Browse to the above new folder in which to place your new Project file YourName_MiddleOut_Review.ipj
   d. At this stage in your learning you will not need to add any libraries – but you will later
   e. Finish
6. In the projects list, double click the new project name to place the check mark beside it and highlight it -- this makes the Project Active such that new files will be defaulted to this new project. (Note: Creating the New Project should result in this happening automatically when you finish the wizard.)
   a. **DO NOT CLICK SAVE.** – Click Done
7. At the Application icon, select New / New from Template:
   a. Start a new Standard.ipt file. Notice that the new file will default to belong to the new project (ipj) that you had just created above. (If your new project is not shown as the Project File, click the “Projects” button to find it.)
   b. You should use your personal template FirstName_LastInitial_PromptConstraint_10cmMajor_template.ipt as long as you had made it correctly). Click on Create to launch the sketching workspace.

2.1 Design Environment Configuration Settings

1) Confirm these design environment configuration settings: Tools/Document Settings
   a. Set units to cm with two decimal place accuracy
   b. Display parameter input as Expression
   c. Display Modeling Dimension as Expression
   d. On Sketches, set the Snap Spacing to 10.0mm for both X and Y and set grid to 1 cm per minor and 10 cm per major grid line
   e. Standard: Material – Wood(Cherry)
   f. Modelling / Make Components Dialog:
      i. Part File Defaults:
         1. Enter your first name as the prefix for Part files (that is, .ipt files which will be created)
         2. For the default template, browse to and select your own PromptConstraint_10cmMajor_template.ipt file
      ii. Assembly File Defaults:
         1. Enter your first name as the prefix for Assembly files (that is, .iam files which will be created)
         2. Be sure that standard.iam is the template that will be used
      iii. Apply / OK

2 Middle-Out Design Methodology Review

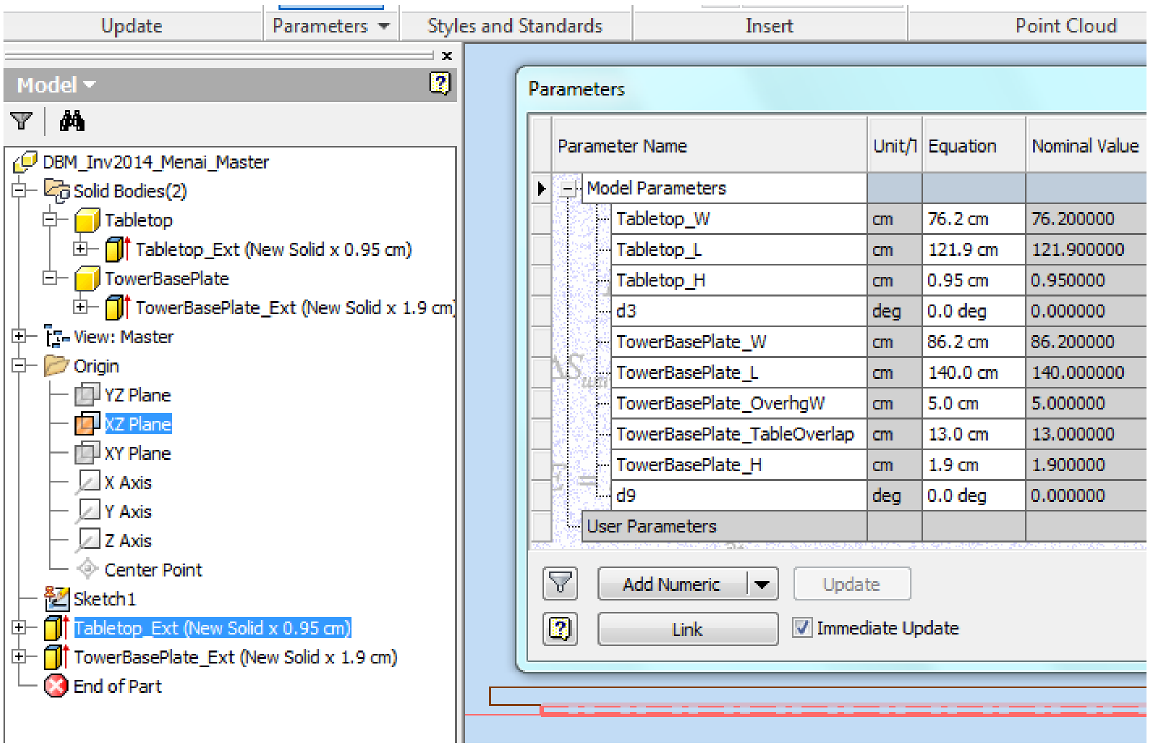
2) In the Model Browser / Origin, review the coordinate axes coordinate planes
   a. You should be looking at the XY plane on the screen as soon as you start a new file (as you had established in your template)
   b. Finish the sketch even though you did not sketch anything
   c. In Model Browser, click on XZ plane, right click, New Sketch
3) Sketch: 2 Point Centre Rectangle where the centre is the origin (where X and Z axes meet) (the origin will be at the centre of the Tabletop)
   a. Dimension the side lengths of the Tabletop in X direction and in Z direction
      i. Tabletop_W=76.2 cm (in Z)
      ii. Tabletop_L=121.9 cm (in X)
      iii. Finish sketch
      iv. Extrude this profile toward you: Tabletop_H= 0.95 cm
4) Rotate the Tabletop and LookAt the upper surface: rightclick: NewSketch
   a. Use Line to sketch four sides of the TowerBasePlate, parallel to the edges of the Tabletop
   b. Dimension the side lengths in X direction and in Z direction
      i. TowerBasePlate_W = 86.2 cm (in Z)
      ii. TowerBasePlate_L = 140.0 cm (in X) Finish Sketch
   c. Position the TowerBasePlate in a relation to the Tabletop
      i. TowerBasePlate_OverhgW = 5.0
      ii. TowerBasePlate_TableOverlap=13.0
      iii. Esc; Show All Degrees of Freedom (should be fully constrained)



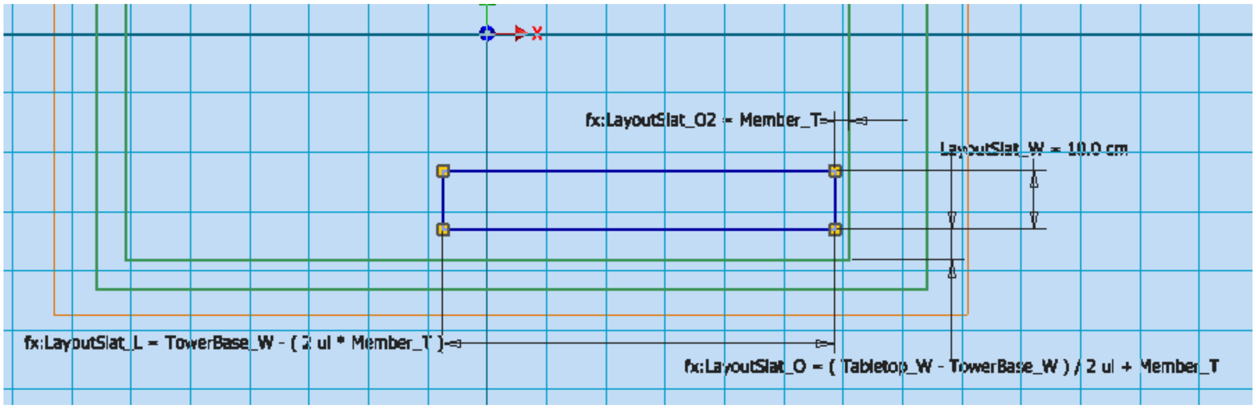
TowerBasePlate Sketch on the XZ Plane

      iv. Finish Sketch
      v. Extrude this new profile upward: TowerBasePlate_H=1.9 cm
5) Manage / Parameters – just to see the Parameters you have created

6) Model Browser / Solid Bodies
   a. Rename Solid1 as Tabletop (based on Tabletop_Ext)
   b. Rename Solid2 as TowerBasePlate (based on TowerBasePlate_Ext)

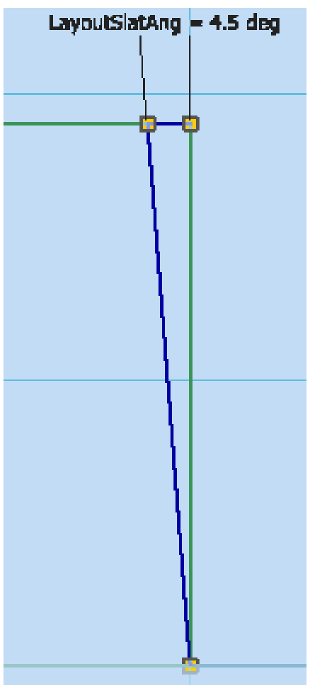


7) Save the part file as YourName_Menai_Master
8) Set User Parameters
   a. TowerBase_W=71.0 (overall width of the tower at the TowerBasePlate)
   b. Member_T = 2.5 (thickness of most of the general structural members of the tower)
9) New Sketch on the top surface of TowerBasePlate
   a. Project Geometry from the four edges of the Tabletop onto this sketch
   b. Sketch what will be a LayoutSlat onto top surface of TowerBasePlate dimensioned as shown
      i. LayoutSlat_O = (Tabletop_W-TowerBase_W)/2 + Member_T (side offset)
      ii. LayoutSlat_O2 = Member_T (offset from the end of the table leading to row of tables)
      iii. LayoutSlat_W=10.0
      iv. LayoutSlat_L= TowerBase_W-(2*Member_T)
      v. Finish Sketch



vi. Extrude
    1. LayoutSlat_T=Member_T
    2. Rename the new solid as LayoutSlat

10) New Sketch on the long edge of LayoutSlat where it sits near the middle of the table length

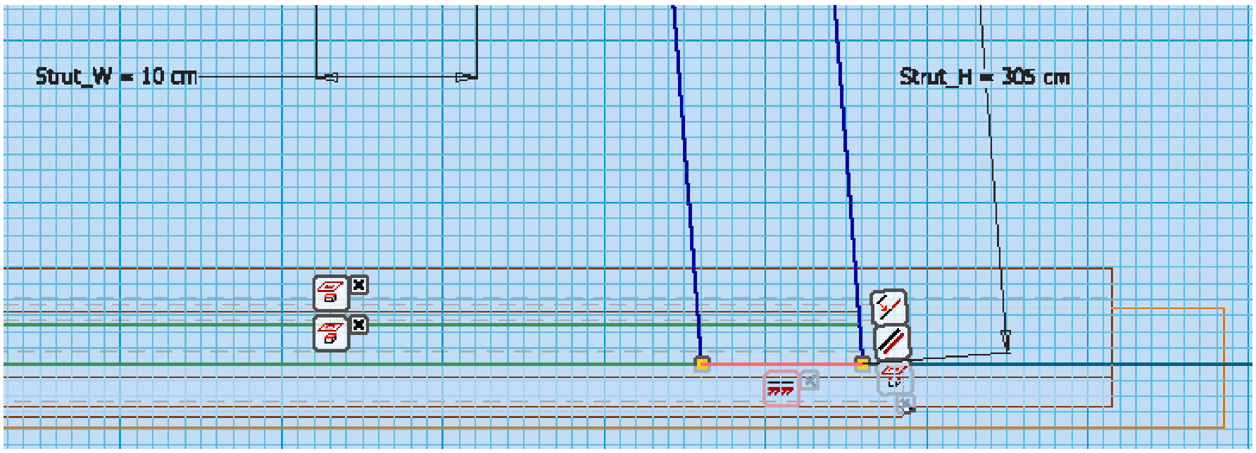


a. LayoutSlatAng=4.5 deg at one end of LayoutSlat
b. Finish Sketch
c. Extrude this wedge shaped piece through all by cutting (determines the angle of the tower-tapering toward the peak of the tower)

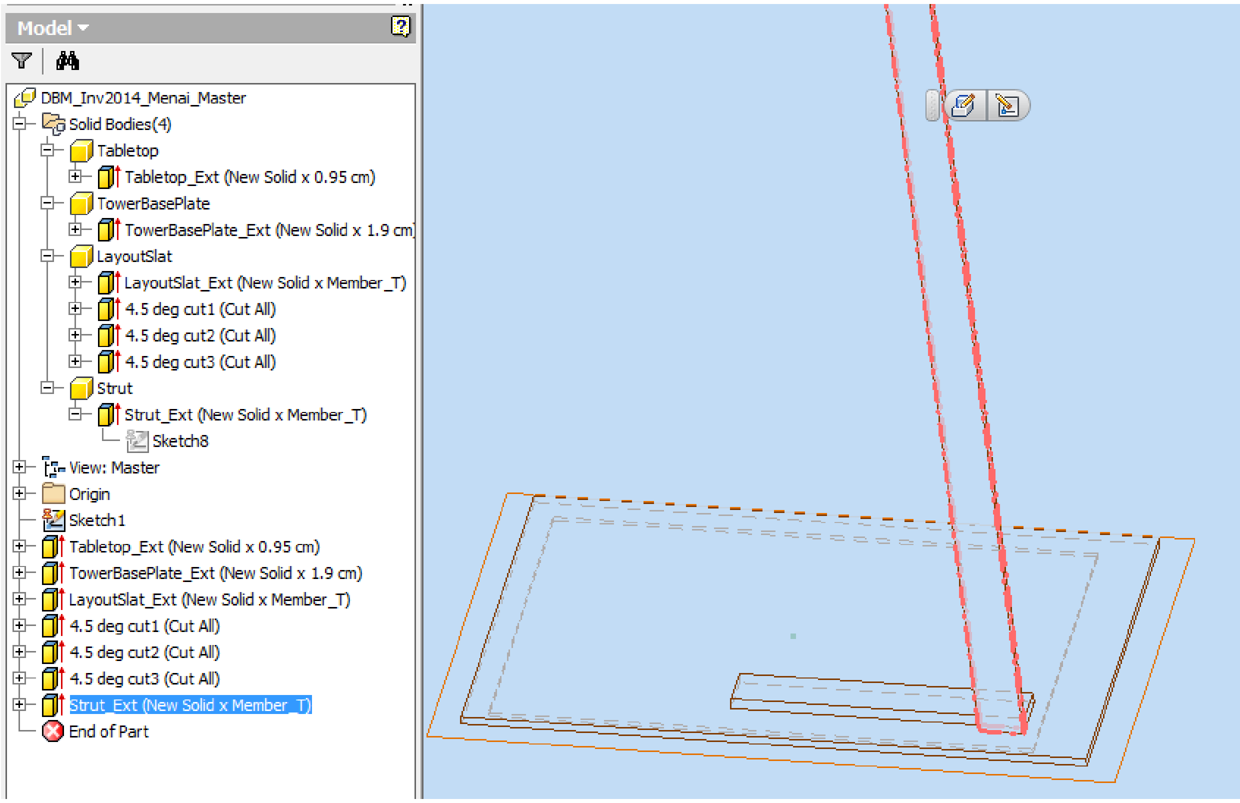
11) New Sketch on the other end face of LayoutSlat nearest the neighbouring table in the row of tables
a. Sketch another wedge to cut at the same angle of 4.5 deg for the tower struts
b. Finish Sketch and extrude the wedge by cutting through all

12) New Sketch on the inside long face of LayoutSlat nearest the neighbouring table (this face is still perpendicular to the table)
a. Sketch another wedge to cut at the same angle of 4.5 deg for the tower struts
b. Finish Sketch and extrude the wedge by cutting through all

13) Three faces of the LayOutSlat now have faces sloping at 4.5 deg to help position the tower which tapers toward the top
14) New Sketch on the sloping long edge of LayoutSlat
    a. Strut_H = 305
    b. Strut_W = 10



    c. Finish sketch
    d. Note: The small wedge at the bottom of the strut caused by the 4.5^{o} angle needs to be "filled in" by joining another extrusion (StrutBot_4_5Extr).
    e. Extrude by New Solid
        i. Strut_T=Member_T

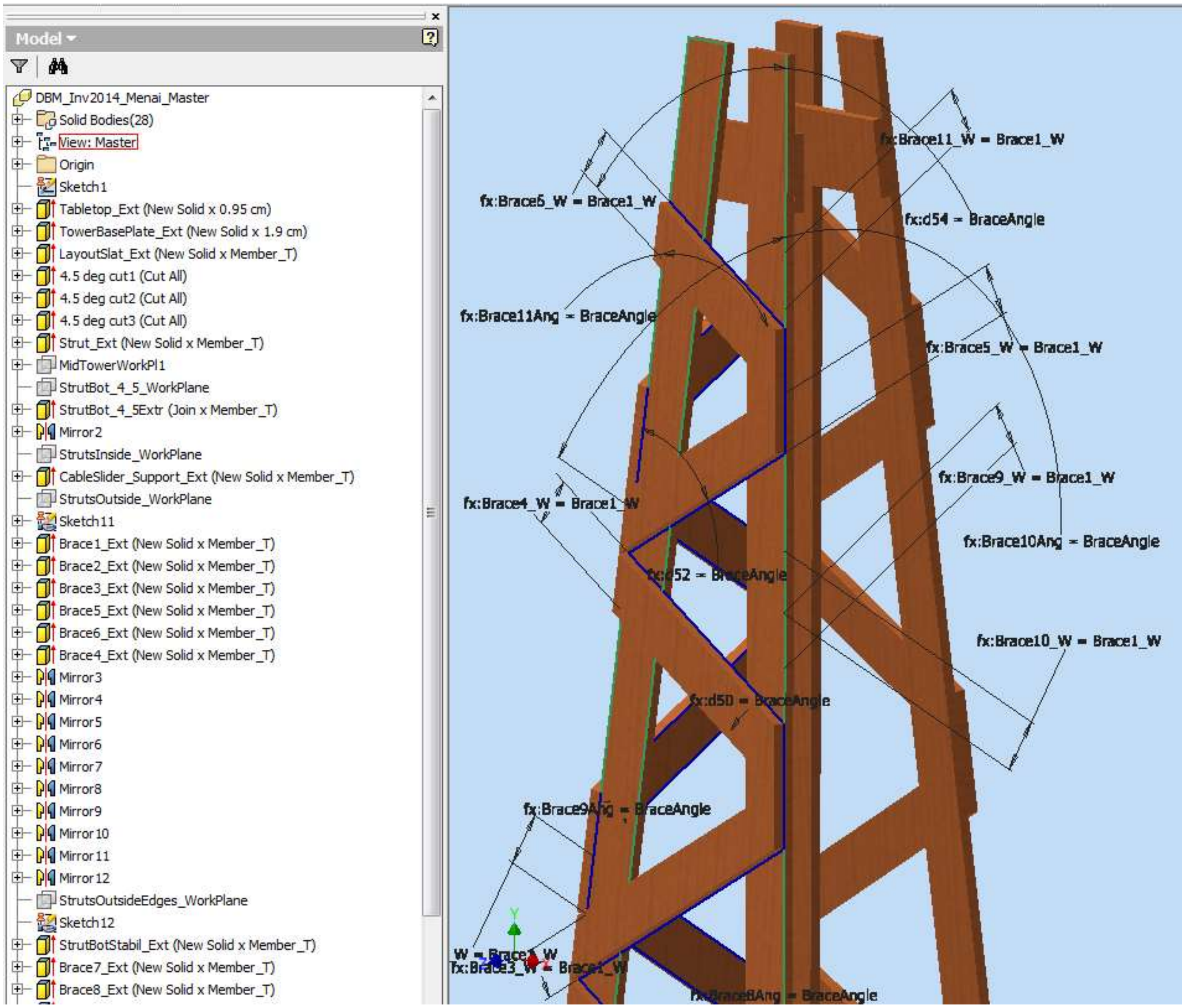


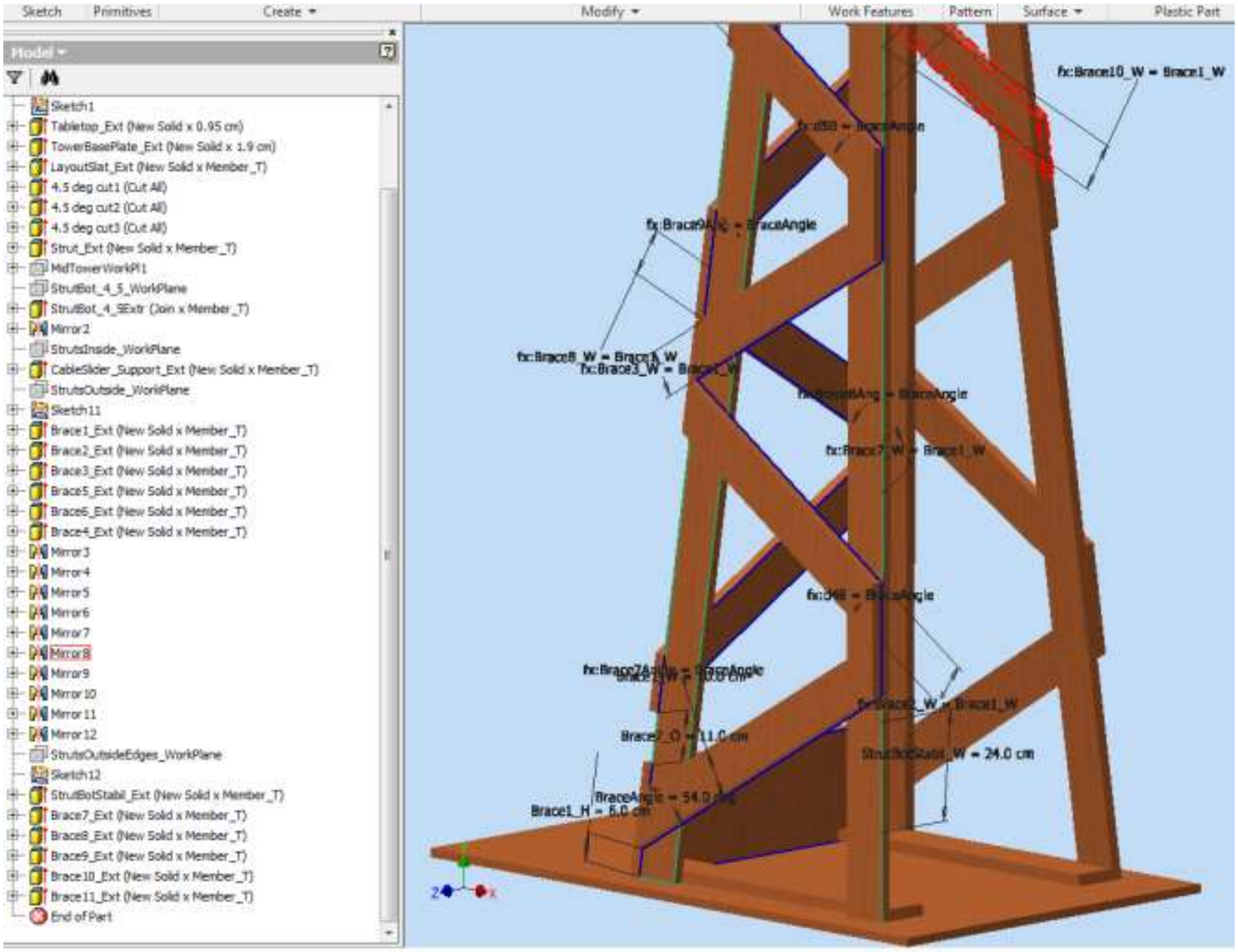
Model
DBM_Inv2014_Menai_Master
Solid Bodies(4)
Tabletop
Tabletop_Ext (New Solid x 0.95 cm)
TowerBasePlate
TowerBasePlate_Ext (New Solid x 1.9 cm)
LayoutSlat
LayoutSlat_Ext (New Solid x Member_T)
4.5 deg cut1 (Cut All)
4.5 deg cut2 (Cut All)
4.5 deg cut3 (Cut All)
Strut
Strut_Ext (New Solid x Member_T)
Sketch8
View: Master
Origin
Sketch1
Tabletop_Ext (New Solid x 0.95 cm)
TowerBasePlate_Ext (New Solid x 1.9 cm)
LayoutSlat_Ext (New Solid x Member_T)
4.5 deg cut1 (Cut All)
4.5 deg cut2 (Cut All)
4.5 deg cut3 (Cut All)
Strut_Ext (New Solid x Member_T)
End of Part

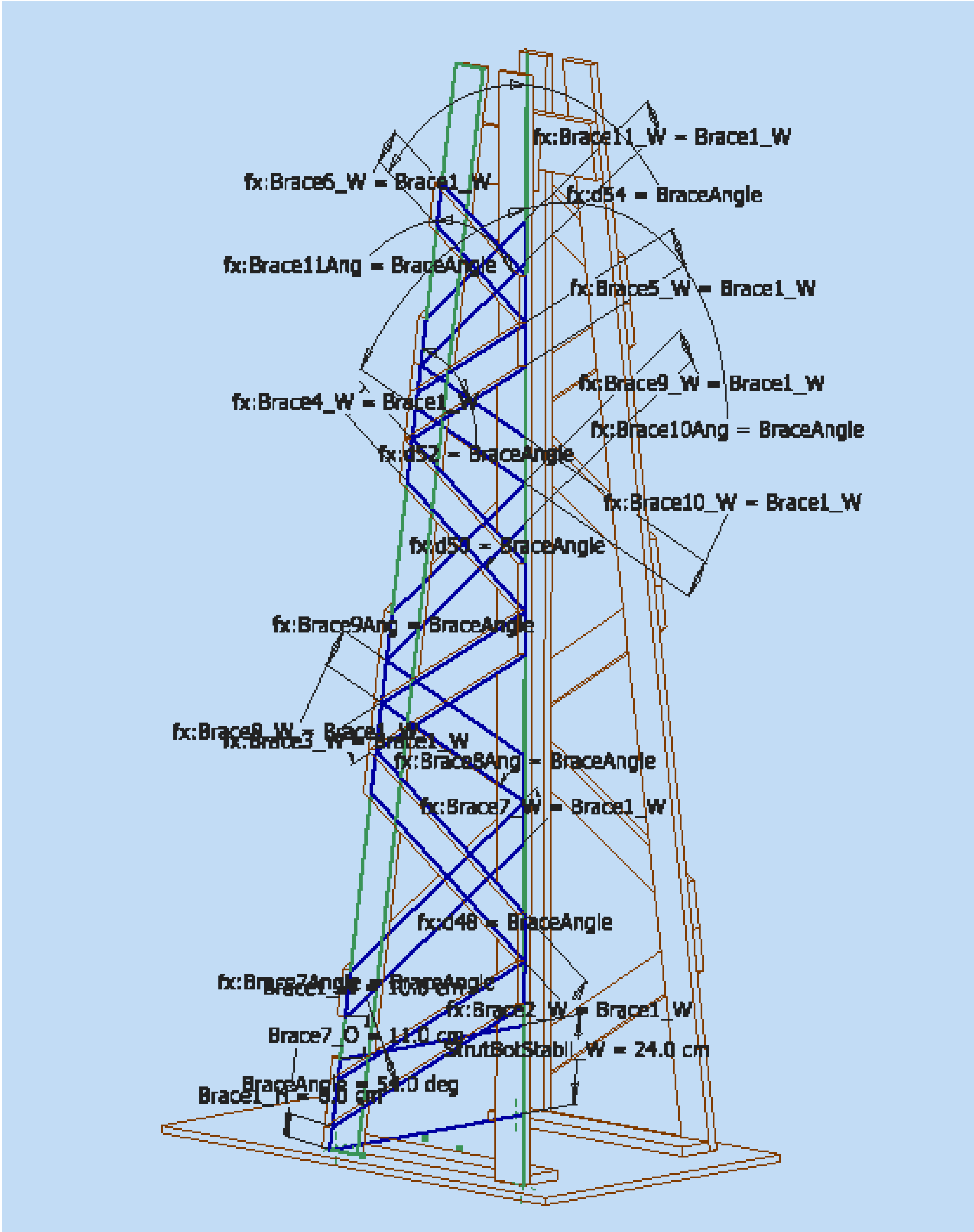
Parameter Name	Unit/T	Equation	Nominal Value
Model Parameters			
Tabletop_W	cm	76.2 cm	76.200000
Tabletop_L	cm	121.9 cm	121.900000
Tabletop_H	cm	0.95 cm	0.950000
d3	deg	0.0 deg	0.000000
TowerBasePlate_W	cm	86.2 cm	86.200000
TowerBasePlate_L	cm	140.0 cm	140.000000
TowerBasePlate_OverhgW	cm	5.0 cm	5.000000
TowerBasePlate_TableOverlap	cm	13.0 cm	13.000000
TowerBasePlate_H	cm	1.9 cm	1.900000
d9	deg	0.0 deg	0.000000
LayoutSlat_O	cm	(Tabletop_W - TowerBase_W) / 2 ul + Member_T	5.100000
LayoutSlat_W	cm	10.0 cm	10.000000
LayoutSlat_O2	cm	Member_T	2.500000
LayoutSlat_L	cm	TowerBase_W - (2 ul * Member_T)	66.000000
LayoutSlat_H	cm	Member_T	2.500000
d15	deg	0.0 deg	0.000000
LayoutSlatAng	deg	4.5 deg	4.500000
d18	deg	LayoutSlatAng	4.500000
d20	deg	0.0 deg	0.000000
d21	deg	LayoutSlatAng	4.500000
d23	deg	0.0 deg	0.000000
d24	deg	LayoutSlatAng	4.500000
d26	deg	0.0 deg	0.000000
Strut_W	cm	10 cm	10.000000
Strut_H	cm	305 cm	305.000000
Strut_T	cm	Member_T	2.500000
d30	deg	0.0 deg	0.000000
User Parameters			
TowerBase_W	cm	71 cm	71.000000
Member_T	cm	2.5 cm	2.500000

15) NOTE: A few of the parameter values above are subject to change (eg Strut_H – depending on the information we will hopefully receive regarding the height of the tapestry panel image).
16) Set up a workplane: MidTowerWorkPl1
    a. Mirror the Strut across it (New Solid)
17) Set up another workplane: StrutsInside_WorkPlane (inside the tower)
    a. Sketch and extrude the New Solid: CableSlider_Support_Ext (this may need to be moved up or down a little depending on the final decision regarding the viewable height of the image in the 3 Tapestry rows)
18) Set up another workplane: StrutsOutside_WorkPlane
    a. Sketch all of the 6 braces – fully constrained; keep a consistent angle to the Struts: Extrude by same thickness as all other Members (Member_T)
19) Mirror the entire pair of Struts with Braces across the XY plane
20) Set up another workplane: StrutsOutsideEdges_WorkPlane

a. Sketch and extrude the StrutBotStabil and the braces on the end face of the Tower (ie on the end of the bridge)




Sketch
Primitives
Create
Modify
Work Features
Pattern
Surface
Plastic Part
Model
Sketch1
Tabletop_Ext (New Solid x 0.95 cm)
TowerBasePlate_Ext (New Solid x 1.9 cm)
LayoutSlat_Ext (New Solid x Member_T)
4.5 deg cut1 (Cut All)
4.5 deg cut2 (Cut All)
4.5 deg cut3 (Cut All)
Strut_Ext (New Solid x Member_T)
MidTowerWorkPl1
StrutBot_4_5_WorkPlane
StrutBot_4_5Extr (Join x Member_T)
Mirror2
StrutsInside_WorkPlane
CableSlider_Support_Ext (New Solid x Member_T)
StrutsOutside_WorkPlane
Sketch11
Brace1_Ext (New Solid x Member_T)
Brace2_Ext (New Solid x Member_T)
Brace3_Ext (New Solid x Member_T)
Brace5_Ext (New Solid x Member_T)
Brace6_Ext (New Solid x Member_T)
Brace4_Ext (New Solid x Member_T)
Mirror3
Mirror4
Mirror5
Mirror6
Mirror7
Mirror8
Mirror9
Mirror10
Mirror11
Mirror12
StrutsOutsideEdges_WorkPlane
Sketch12
StrutBotStabil_Ext (New Solid x Member_T)
Brace7_Ext (New Solid x Member_T)
Brace8_Ext (New Solid x Member_T)
Brace9_Ext (New Solid x Member_T)
Brace10_Ext (New Solid x Member_T)
Brace11_Ext (New Solid x Member_T)
End of Part
fx:Brace10_W = Brace1_W
fx:Brace8_W = Brace1_W
fx:Brace7_W = Brace1_W
fx:Brace2_W = Brace1_W
StrutBotStabil_W = 24.0 cm
Brace7_O = 11.0 cm
BraceAngle = 54.0 deg
Brace1_H = 5.0 cm


fx:Brace11_W = Brace1_W
fx:Brace6_W = Brace1_W
fx:d54 = BraceAngle
fx:Brace11Ang = BraceAngle
fx:Brace5_W = Brace1_W
fx:Brace9_W = Brace1_W
fx:Brace4_W = Brace1_W
fx:Brace10Ang = BraceAngle
fx:d52 = BraceAngle
fx:Brace10_W = Brace1_W
fx:d53 = BraceAngle
fx:Brace9Ang = BraceAngle
fx:Brace3_W = Brace1_W
fx:Brace8Ang = BraceAngle
fx:Brace7_W = Brace1_W
fx:d48 = BraceAngle
fx:Brace2_W = Brace1_W
Brace7_O = 11.0 cm
StrutBotStabil_W = 24.0 cm
BraceAngle = 54.0 deg
Brace1_H = 5.0 cm

Parameter Name	Unit/T	Equation	Nominal Value
Model Parameters			
Tabletop_W	cm	76.2 cm	76.200000
Tabletop_L	cm	121.9 cm	121.900000
Tabletop_H	cm	0.95 cm	0.950000
d3	deg	0.0 deg	0.000000
TowerBasePlate_W	cm	86.2 cm	86.200000
TowerBasePlate_L	cm	140.0 cm	140.000000
TowerBasePlate_OverhgW	cm	5.0 cm	5.000000
TowerBasePlate_TableOverlap	cm	13.0 cm	13.000000
TowerBasePlate_H	cm	1.9 cm	1.900000
d9	deg	0.0 deg	0.000000
LayoutSlat_O	cm	(Tabletop_W - TowerBase_W) / 2 ul + Member_T	5.100000
LayoutSlat_W	cm	10.0 cm	10.000000
LayoutSlat_O2	cm	Member_T	2.500000
LayoutSlat_L	cm	TowerBase_W	71.000000
LayoutSlat_H	cm	Member_T	2.500000
d15	deg	0.0 deg	0.000000
LayoutSlatAng	deg	4.5 deg	4.500000
d18	deg	LayoutSlatAng	4.500000
d20	deg	0.0 deg	0.000000
d21	deg	LayoutSlatAng	4.500000
d23	deg	0.0 deg	0.000000
d24	deg	LayoutSlatAng	4.500000
d26	deg	0.0 deg	0.000000
Strut_T	cm	Member_T	2.500000
d30	deg	0.0 deg	0.000000
Strut_H	cm	Tower_H - (CableSlider_Dia / 2 ul)	287.050000
Strut_W	cm	10.0 cm	10.000000
d33	deg	0 deg	0.000000
MidTower_WkPl1_O	cm	-(LayoutSlat_O2 + (LayoutSlat_L / 2 ul))	-38.000000
d35	deg	4.5 deg	4.500000
d36	cm	Member_T	2.500000
d37	deg	0.0 deg	0.000000
CableSlider_Support_O	cm	15.0 cm	15.000000
CableSlider_Support_H	cm	15.0 cm	15.000000
d40	cm	Member_T	2.500000
d41	deg	0.0 deg	0.000000
Brace1_H	cm	6.0 cm	6.000000
Brace1_W	cm	10.0 cm	10.000000
Brace3_W	cm	Brace1_W	10.000000

Brace3_W	cm	Brace1_W	10.000000
d48	deg	BraceAngle	54.000000
Brace2_W	cm	Brace1_W	10.000000
d50	deg	BraceAngle	54.000000
Brace4_W	cm	Brace1_W	10.000000
d52	deg	BraceAngle	54.000000
Brace5_W	cm	Brace1_W	10.000000
d54	deg	BraceAngle	54.000000
Brace6_W	cm	Brace1_W	10.000000
BraceAngle	deg	54.0 deg	54.000000
d57	cm	Member_T	2.500000
d58	deg	0.0 deg	0.000000
d59	cm	Member_T	2.500000
d60	deg	0.0 deg	0.000000
d61	cm	Member_T	2.500000
d62	deg	0.0 deg	0.000000
d63	cm	Member_T	2.500000
d64	deg	0.0 deg	0.000000
d65	cm	Member_T	2.500000
d66	deg	0.0 deg	0.000000
d67	cm	Member_T	2.500000
d68	deg	0.0 deg	0.000000
StrutBotStabil_W	cm	24.0 cm	24.000000
Brace7Angle	deg	BraceAngle	54.000000
Brace7_W	cm	Brace1_W	10.000000
Brace7_O	cm	11.0 cm	11.000000
Brace8Ang	deg	BraceAngle	54.000000
Brace8_W	cm	Brace1_W	10.000000
Brace9Ang	deg	BraceAngle	54.000000
Brace9_W	cm	Brace1_W	10.000000
Brace10Ang	deg	BraceAngle	54.000000
Brace10_W	cm	Brace1_W	10.000000
Brace11Ang	deg	BraceAngle	54.000000
Brace11_W	cm	Brace1_W	10.000000
d81	cm	Member_T	2.500000
d82	deg	0.0 deg	0.000000
d83	cm	Member_T	2.500000
d84	deg	0.0 deg	0.000000
d85	cm	Member_T	2.500000
d86	deg	0.0 deg	0.000000
d87	cm	Member_T	2.500000
d88	deg	0.0 deg	0.000000
d89	cm	Member_T	2.500000
d90	deg	0.0 deg	0.000000

d90	deg	0.0 deg	0.000000
d91	cm	Member_T	2.500000
d92	deg	0.0 deg	0.000000
User Parameters			
TowerBase_W	cm	71 cm	71.000000
Member_T	cm	2.5 cm	2.500000
PanelImage_H	cm	54.0 cm	54.000000
Tower_H	cm	290.0 cm	290.000000
CableSlider_Dia	cm	5.9 cm	5.900000

3 Safety Reminder

Keep in mind the very important requirement that the Tower be "stable". Use serious critical thinking to dig deep into what this means and how to achieve this.

4 Inputs / Knowledge / Understanding I Still Need For This Module

Give each issue a number for future reference:

NOTE: In the feedback, the Peer Assessor must "make the student think" – not give the student the answer! Be sure to include comments justifying the mark that you are giving.

Assessor's Name and Additional Notes: