Introduction to Using Autodesk® Simulation Products to Simulate Solar Thermal Loads in Concrete Bridges

Peggy Menzies – Autodesk Inc.

SM1571-L

In this hands-on lab, learn how Autodesk® Inventor® Professional, Autodesk® Simulation CFD, and Autodesk® Simulation Mechanical design tools can empower structural engineers and architects to accurately assess designs for the infrastructure of our growing world. Autodesk Suites are a smorgasbord of design tools, many with unlimited cloud computing capabilities. Creatively exploring various combinations and interoperability functions of Autodesk Simulation tools in nontraditional workflows can have a profound effect on design. This lab provides an opportunity to practice the workflow and techniques in a hands-on environment. A case study will discuss the techniques needed then justify using simulation tools and workflows to accurately predict the thermal profile and resultant stresses through validation against measured bridge data. Attendees will learn how Inventor designs can be analyzed for thermal loads using Simulation CFD and for thermal stresses using Simulation Mechanical.

Learning Objectives

At the end of this class, you will be able to:

- Create an analysis model in Inventor suitable for solar loading
- Set up and simulate solar analyses in Simulation CFD
- Process thermal results in Simulation CFD
- Transfer thermal results to Simulation Mechanical to evaluate stresses

About the Speaker

Dr. Peggy Menzies is currently the Senior Software Quality Assurance Engineering lead for the Computation Fluid Dynamics product line at Autodesk, Inc. Previously, she managed the software and mechanical engineering groups of an aerospace instrument manufacturing facility. She has two patents on instrument designs for military and commercial aircraft. Peggy successfully completed her doctorate from Old Dominion University researching unsteady, transonic flows. During her master's program from the Naval Postgraduate School, she researched FEA structural optimizations. After graduating from the United States Naval Academy with an engineering degree in Naval Architecture, she served as an officer in the United States Navy.

Peggy.Menzies@autodesk.com

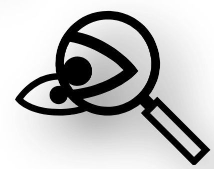
Welcome to Class!!

This class handout was created with the complete novice in mind. Every mouse click and pick is shown with images so you can repeat these steps even after the class is done. Experienced users should be able to skip to the pertinent information. Use the margin icons to help navigate as follows:

Look for this figure in the margin for the steps to be performed in the hands on lab ->

Look for this figure in the margin for tips and key points to remember ->

Look for this figure in the margin for more details or an in depth explanation ->



Stars overlaid on the figures highlight where to click in the dialogs:

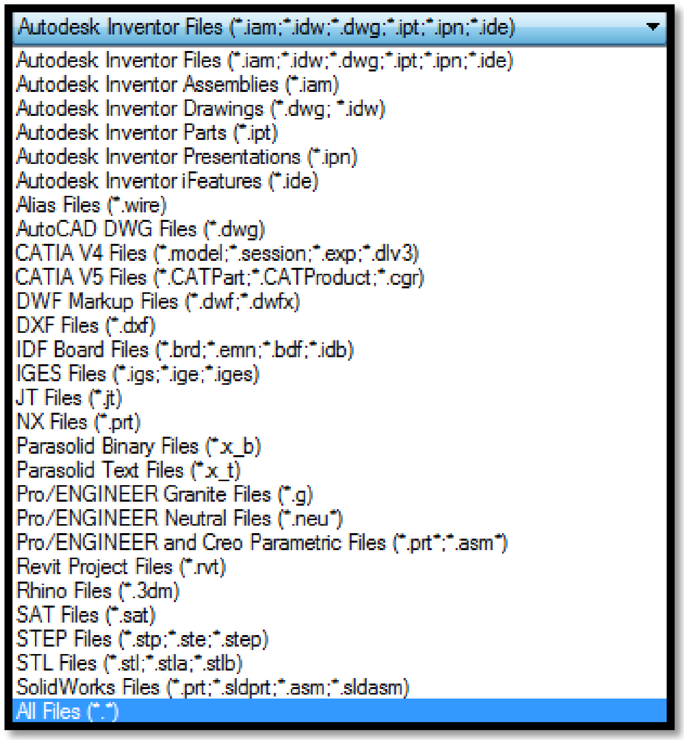
Section 1: Create an analysis model in Inventor suitable for solar loading

By the end of this section, you will understand how to prepare CAD geometry to be used for solar simulation analyses using Autodesk® Inventor® Professional. The methods presented are applicable to most structures exposed to solar thermal loading, not just bridges, though this course is specific to concrete bridges.

Note: Items with the “to do list” figure in the margin will be performed in the class while the remaining items are provided for detailed reference.

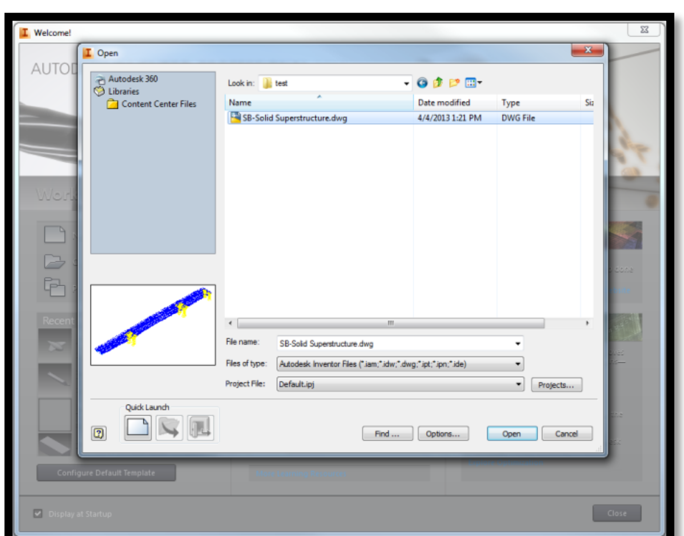
Start with model geometry

Nearly any format of CAD geometry can be used with Autodesk tools to create simulation analysis models. The following is a list of formats easily imported into Inventor Professional:

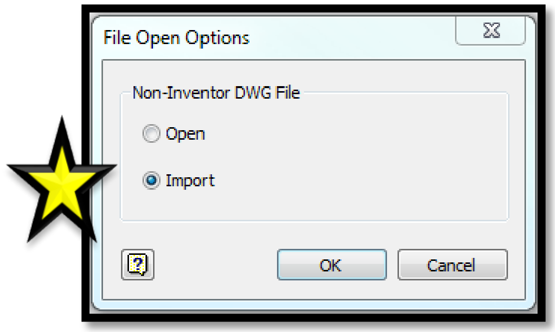


Import AutoCAD geometry as volumes

As an import example, AutoCAD geometry can be opened for use in Inventor Professional. Navigate to the desired geometry file:

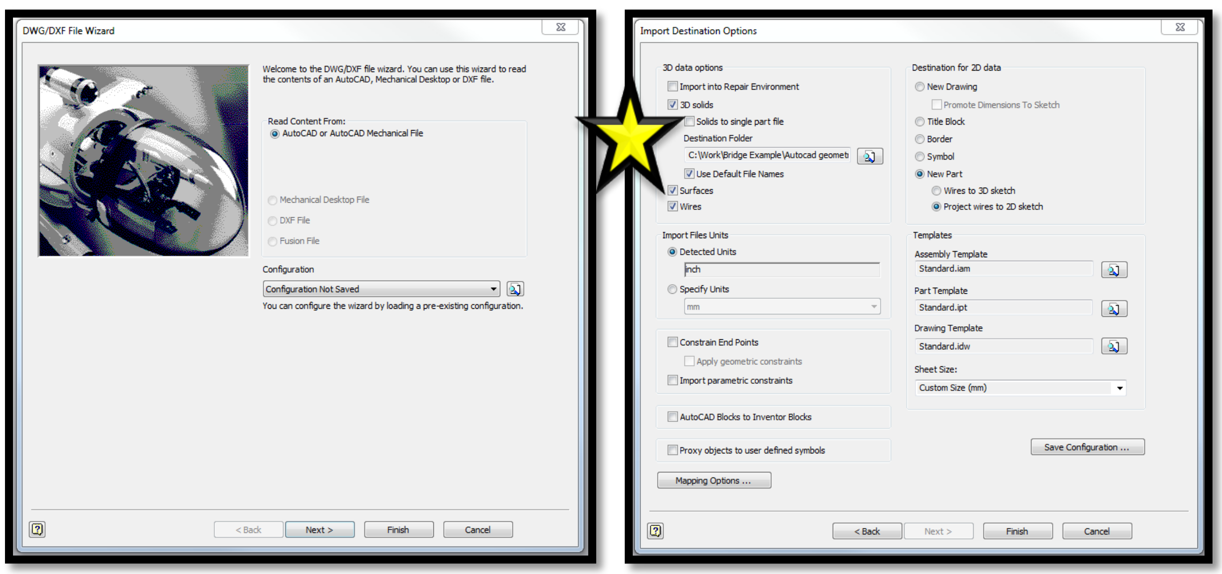


Select Options… then check Import followed by OK:



Then click Open on the main Open screen.

This will open the DWG/DXF File Wizard:



Tip: When importing geometry, be sure to select 3D solids and check for correct "Import Files Units."

If you plan to repeat the process, use Save Configuration … before selecting Finish.

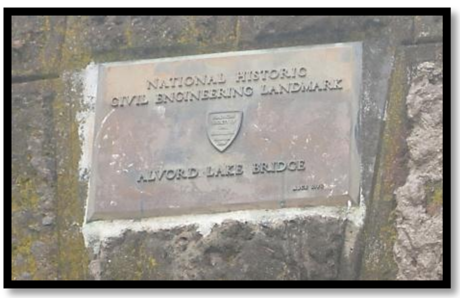
Alternately, a model assembly can be created for analysis directly in Inventor Professional.

Open and explore the example geometry – Bridge Assembly.iam

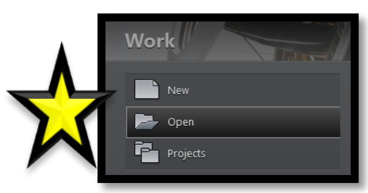
For the hands-on lab, a sample bridge assembly has been created in Inventor Profession to approximate the Alvord Lake Bridge located in Golden Gate Park in San Francisco, CA.

Side note from Wikipedia:

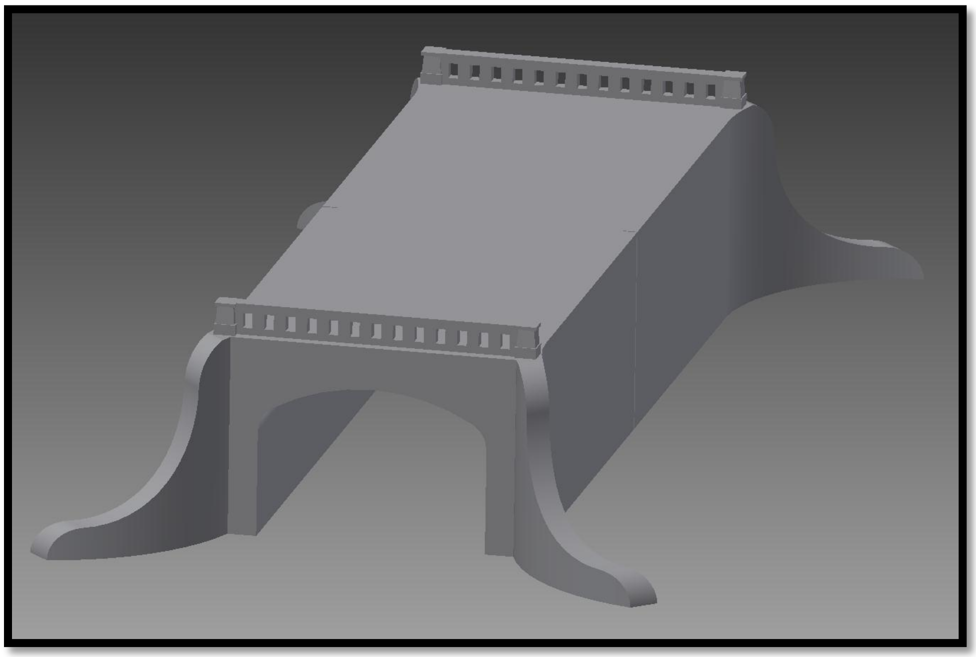
The Alvord Lake Bridge was the first reinforced concrete bridge built in America. It was built in 1889 by Ernest L. Ransome… (he) left San Francisco a few years later, frustrated and bitter at the building community's indifference to concrete construction. Ironically, the city's few reinforced concrete structures, including the Alvord Lake Bridge, survived the 1906 earthquake and fire in remarkable shape, vindicating Ransome's faith in the method. The Alvord Lake Bridge, which arches over a pedestrian entrance to San Francisco's Golden Gate Park, was designated a civil engineering landmark by the American Society of Civil Engineers in the 1970s



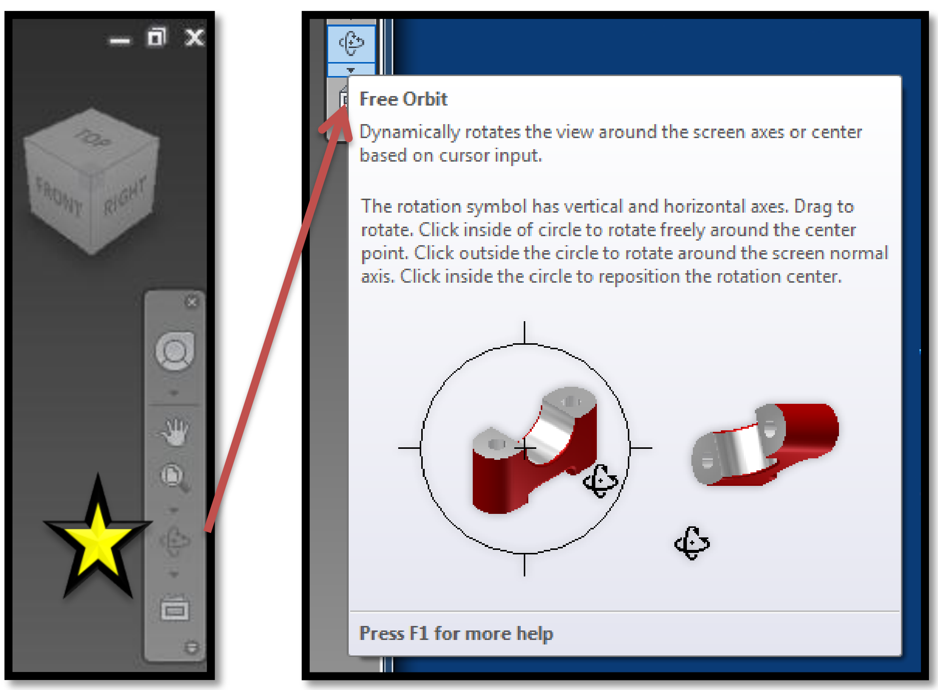
Open Inventor Professional then Select the Open Icon under Work:



Navigate to the "Demonstration\Inventor Geometry" directory, and select "Bridge Assembly.iam." You should see a model similar to the following:

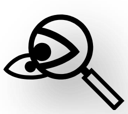


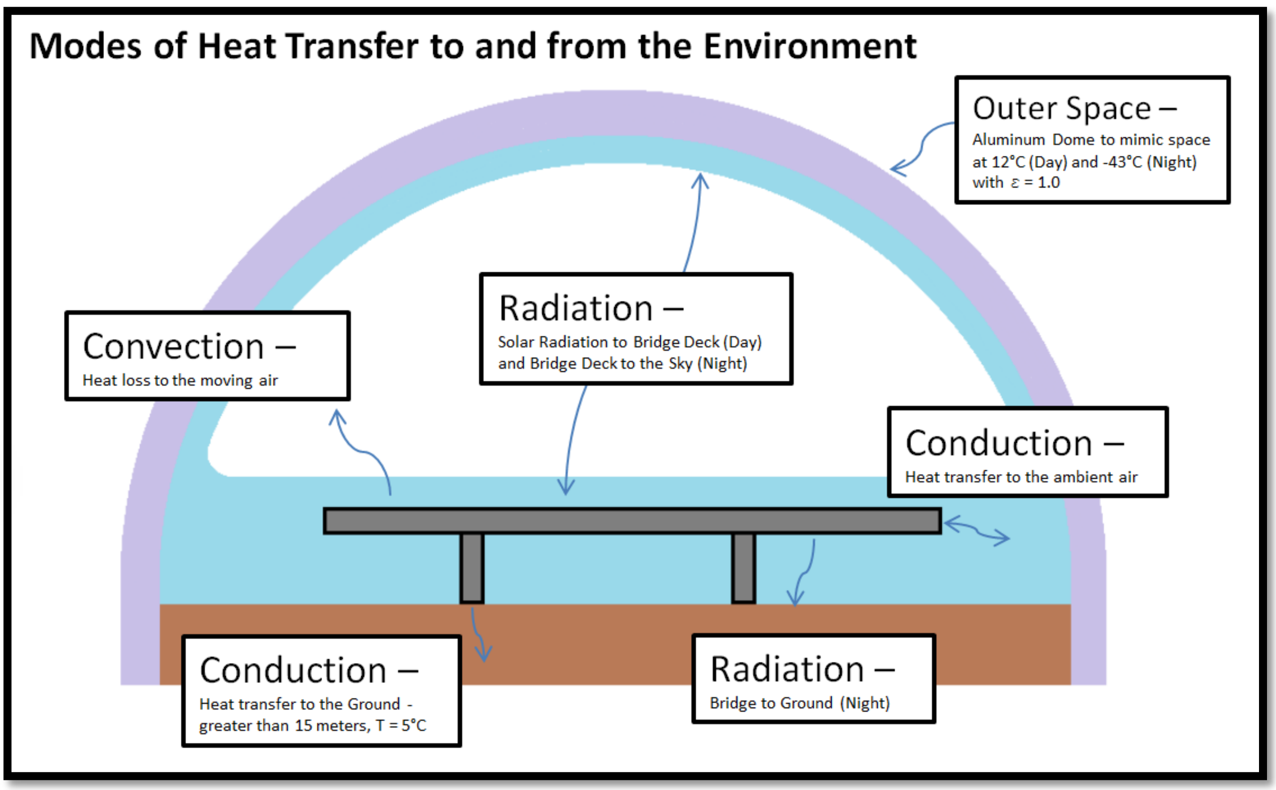
Use Free Orbit from the right hand navigation bar to rotate and inspect the geometry:



Creating analysis assembly – Solar Assembly.iam

To perform a solar thermal analysis in Simulation CFD, environment features need to be created in Inventor to correctly model the heat transfer to and from the environment. In the following schematic, a simplified representation of a bridge is shown under a sky dome to represent the minimal environment features and how they relate to heat transfer:





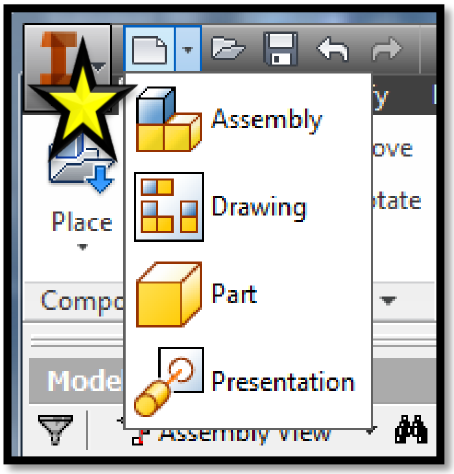
For the lab, the environmental features for the Alvord Lake Bridge have already been created for you. Open “Solar Assembly.iam” from the “Demonstration\Inventor Geometry” directory. You can skip to page 18 when you are ready to continue.

Tip: For your structure, you will want to create a new assembly with the geometry model of your structure as a sub-assembly.

Creating an independent analysis assembly facilitates aligning your structure with global coordinate axes and gives you options to experiment with environment features without altering the model of the actual structure. These next paragraphs show you how.

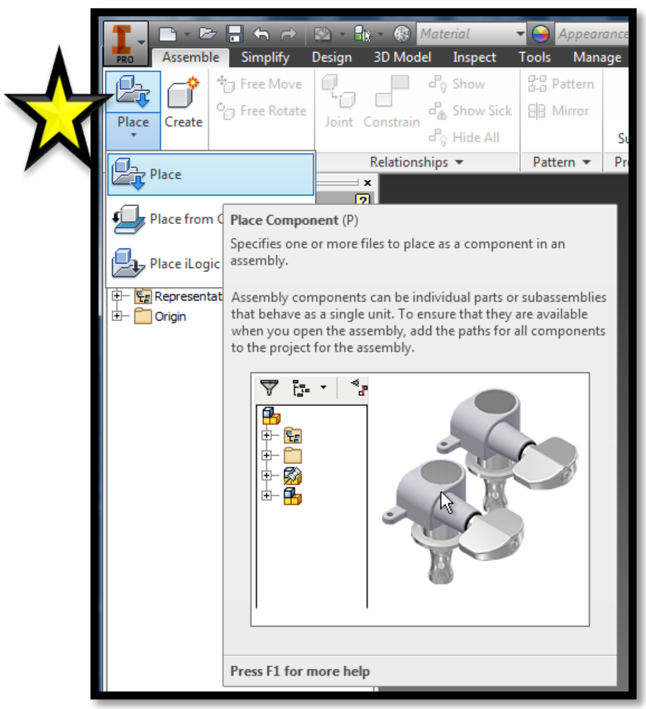
Create an analysis assembly

When you need to create an analysis assembly for your structure, in Inventor, select the New_Assembly icon:

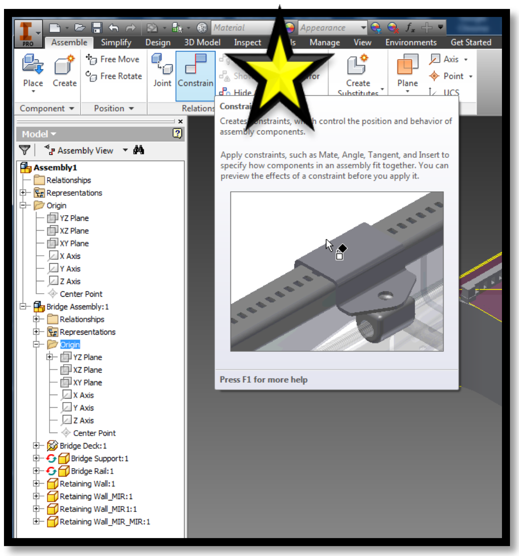


Place bridge assembly

Place the bridge geometry (the bridge assembly) in the analysis assembly by selecting the Place icon, then selecting the bridge assembly:



Be sure to constrain the bridge assembly in the analysis assembly:

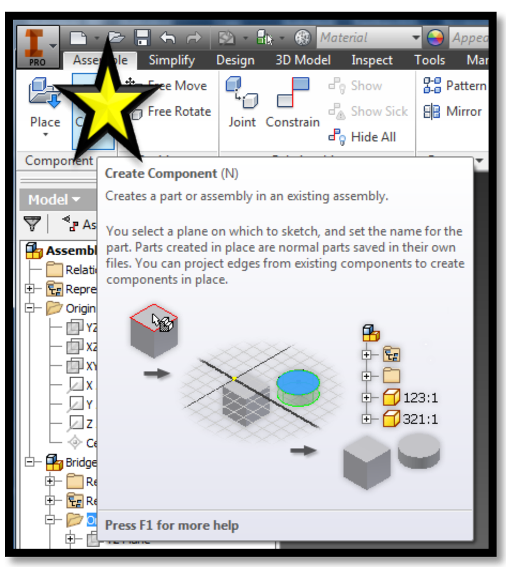


The origin of the analysis assembly will be the reference point for simulations.

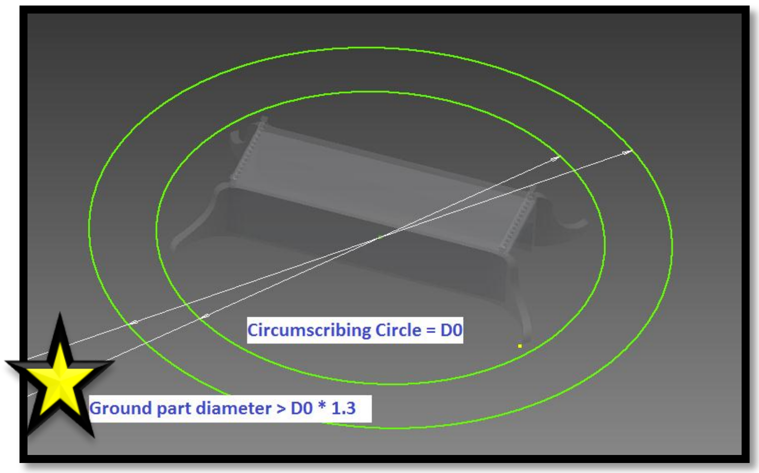
Tip: A good recommendation is to ensure the origin of the analysis assembly is located at the center of the uppermost surface of the bridge deck. Additionally, ensure the compass directions, relative to the bridge, align with a coordinate axis and note which axis represents east and which axis represents the direction of the sky. Note: the Alvord Lake Bridge runs North - South for traffic.

Create the Environmental Features starting with the Ground part

Use the Create icon to create a new component for the analysis assembly:



Tip: A good recommendation is to use either a body of revolution or an extruded circle for the ground part such that the resultant diameter is at least 130% of the diameter of the bridge footprint:

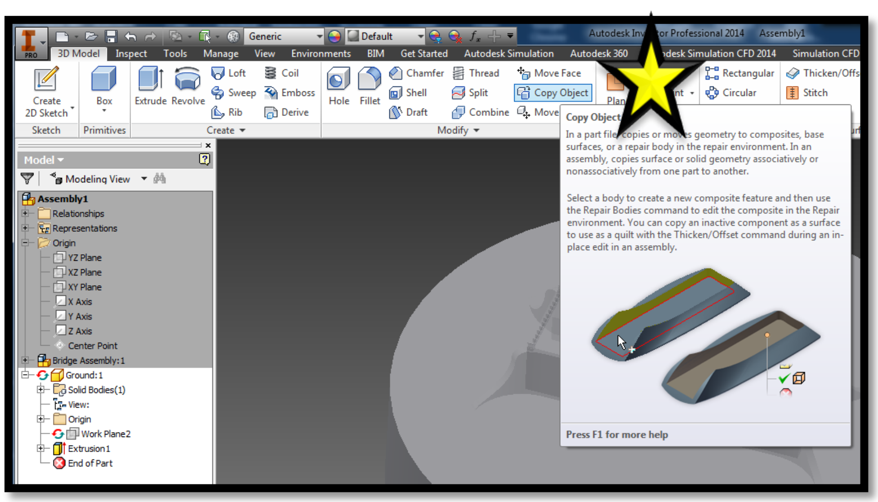


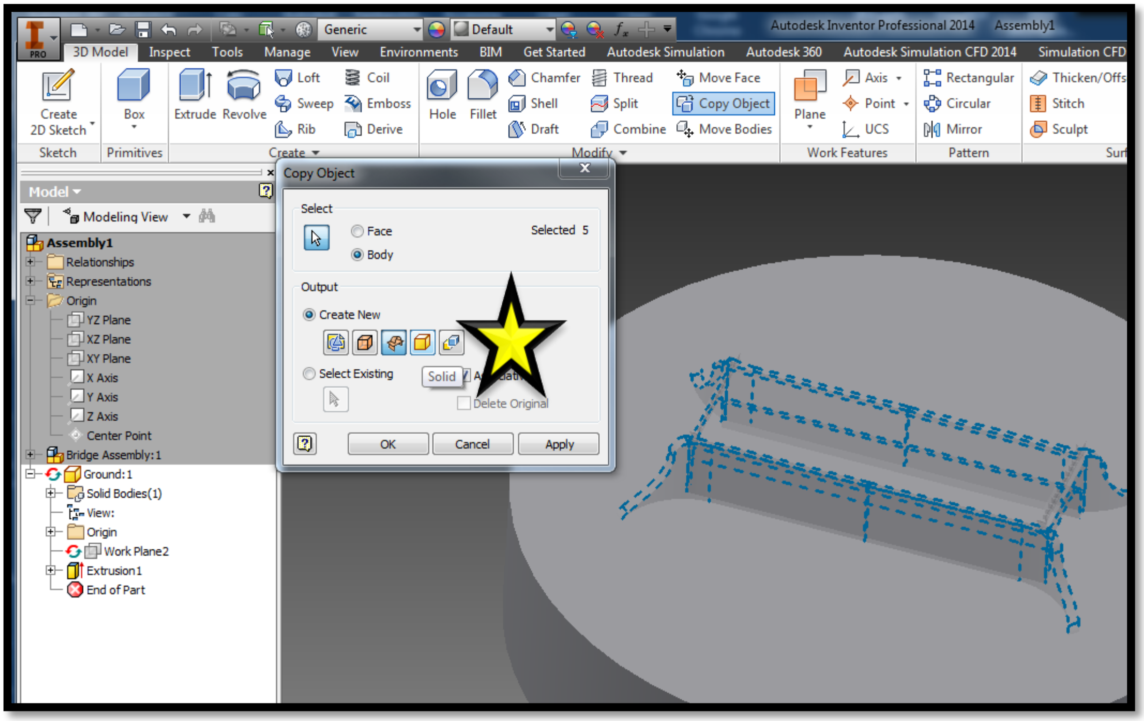
To minimize the effect of ground temperature conditions, the ground part should be sufficiently deep (> 15 meters). At >15 meters, the ground is assumed to have a uniform temperature.

The bridge geometry should be connected to the ground geometry at all supports. If the bridge geometry did extend into the ground, a Boolean operation would be needed to remove the self-intersection of the two parts.

Remove intersection by cutting the ground if necessary

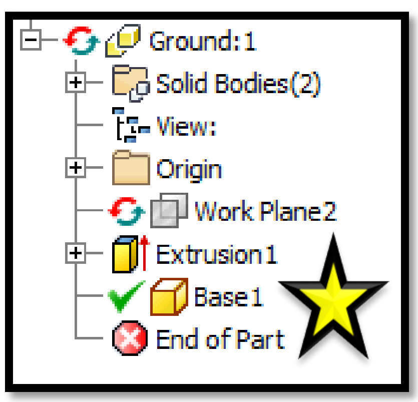
Inventor Professional makes it easy to remove any interference between parts. In this case, since you will want to remove the bridge geometry from the ground part, select to edit the ground part, and then use Copy Object command to make a copy of the bridge geometry parts that intersect in the Ground part:



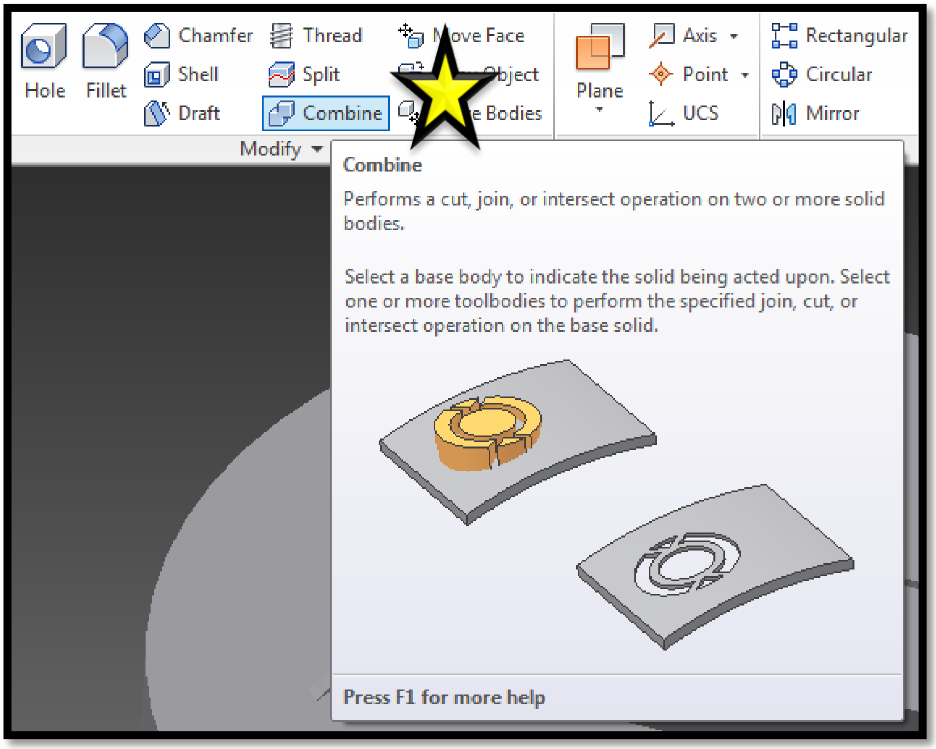


Tip: When removing intersections, be sure to select Body and Create New with the Solid option. All of the intersecting bridge parts can be selected at the same time. 

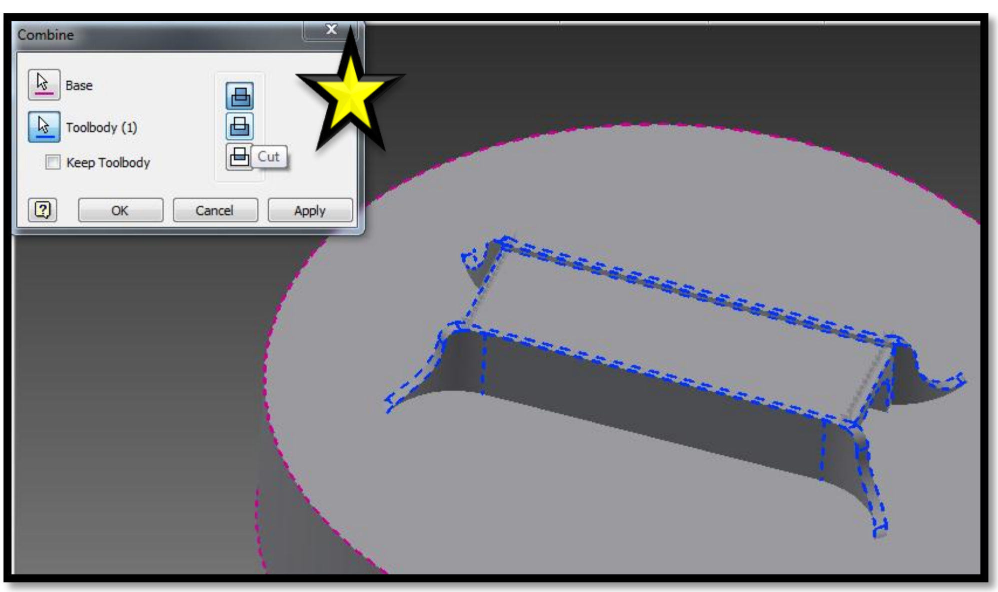
If you have performed this correctly and the ground did intersect with the bridge, the feature tree for the ground part will show an additional Solid Body called Base:



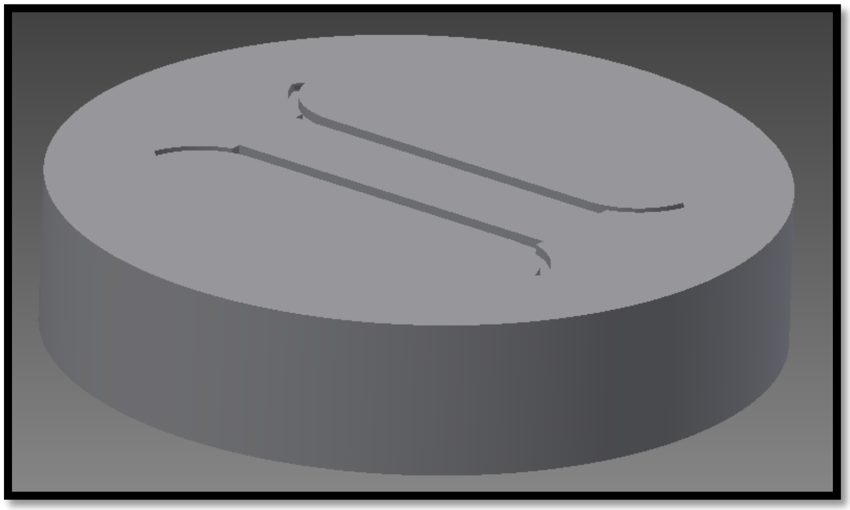
While still editing the ground part, select Combine:



Then select the circular disk (the extrusion or body of revolution of the ground part) as the Base and the copied geometry (the bridge parts) as the Toolbody. Then select cut  on the combine dialog:



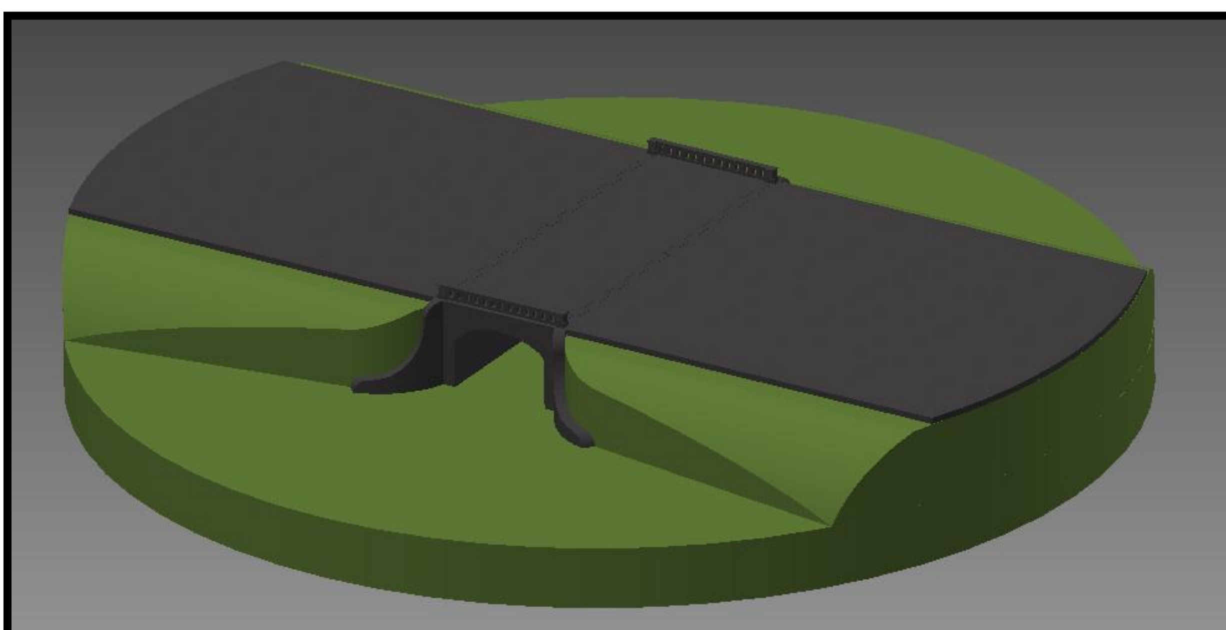
If this has been performed correctly, upon opening the ground part, you will see a cutout where the bridge parts intersect with the ground similar to this:



Note: This is for example only; the demonstration model you will be using in class does not have a ground interference with the base of the bridge.

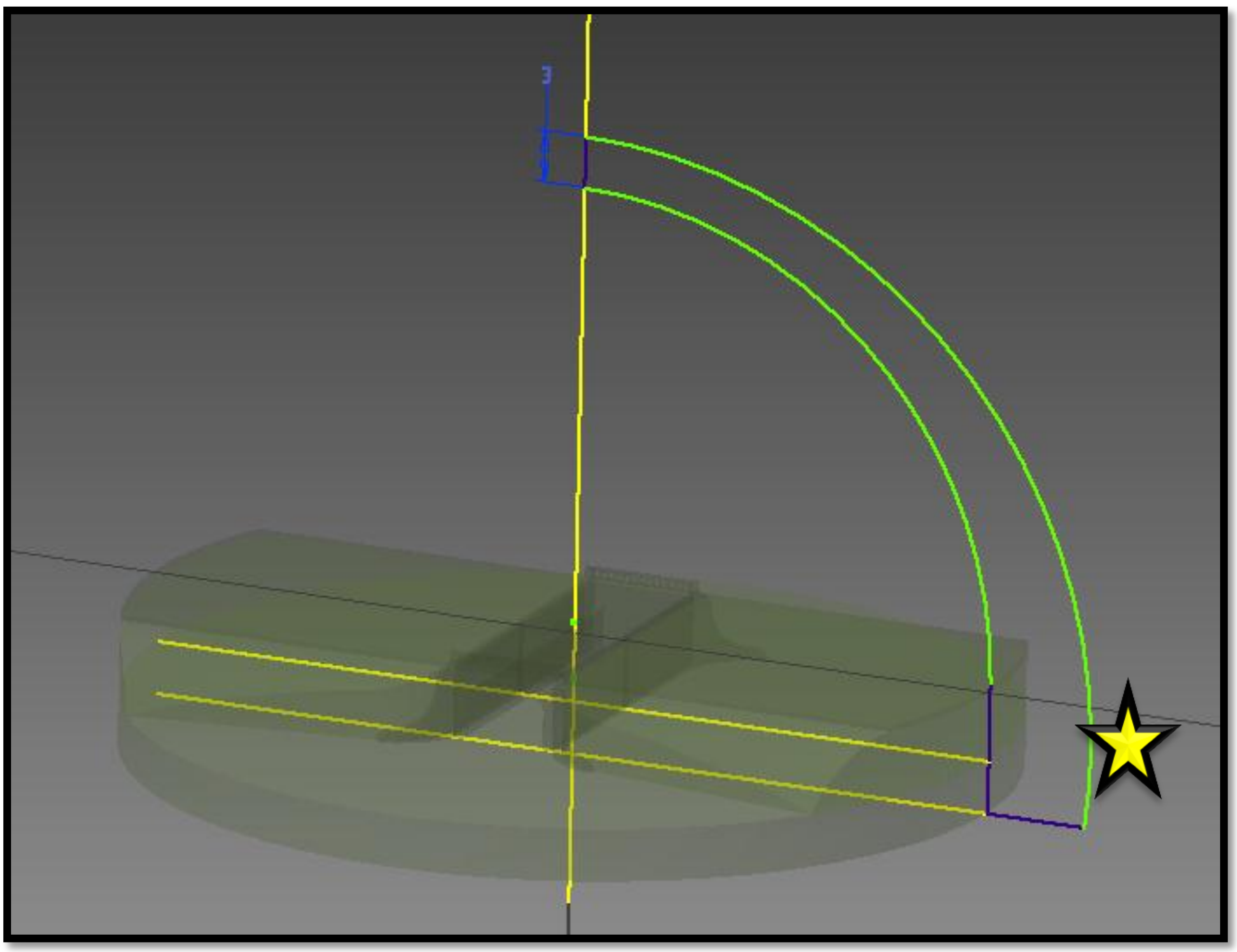
To help visualize the Alvord Lake Bridge in situ, additional ground parts and the road bed were added to the demonstration analysis model. The addition of these parts had little effect on the thermal profile of the bridge but do impact the structural analysis.

The demonstration analysis model should appear as follows:



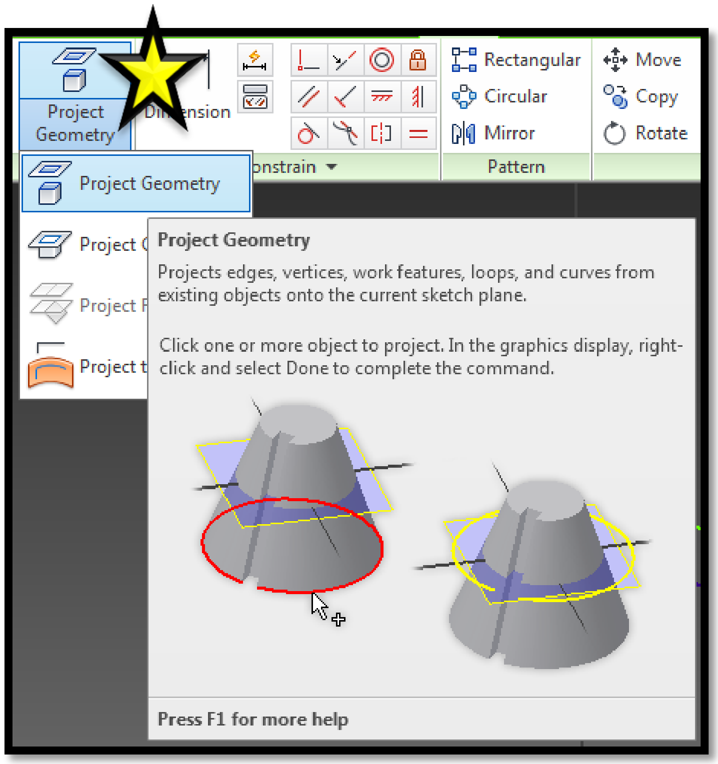
Create the next Environmental Feature – the Sky Dome part

Use the create icon again to create a part to represent the Sky environment. The sky will be represented by a dome and can easily be created as a body of revolution. The dome should be created so it is flush with the bottom and sides of the ground and should be a couple of meters thick for ease of meshing in the analysis. The sketch used for the example problem follows:

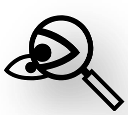


Note: The dome will not be a single arch but a straight section (along the ground part which can extend up to the height of the bridge deck) and an arch to complete the dome.

Tip: Whenever possible, be sure to use the Project Geometry Command to ensure the edges of the dome precisely meet the edges and bottom surface of the ground:



Create the conceptual Environmental Feature – the Sky Void part

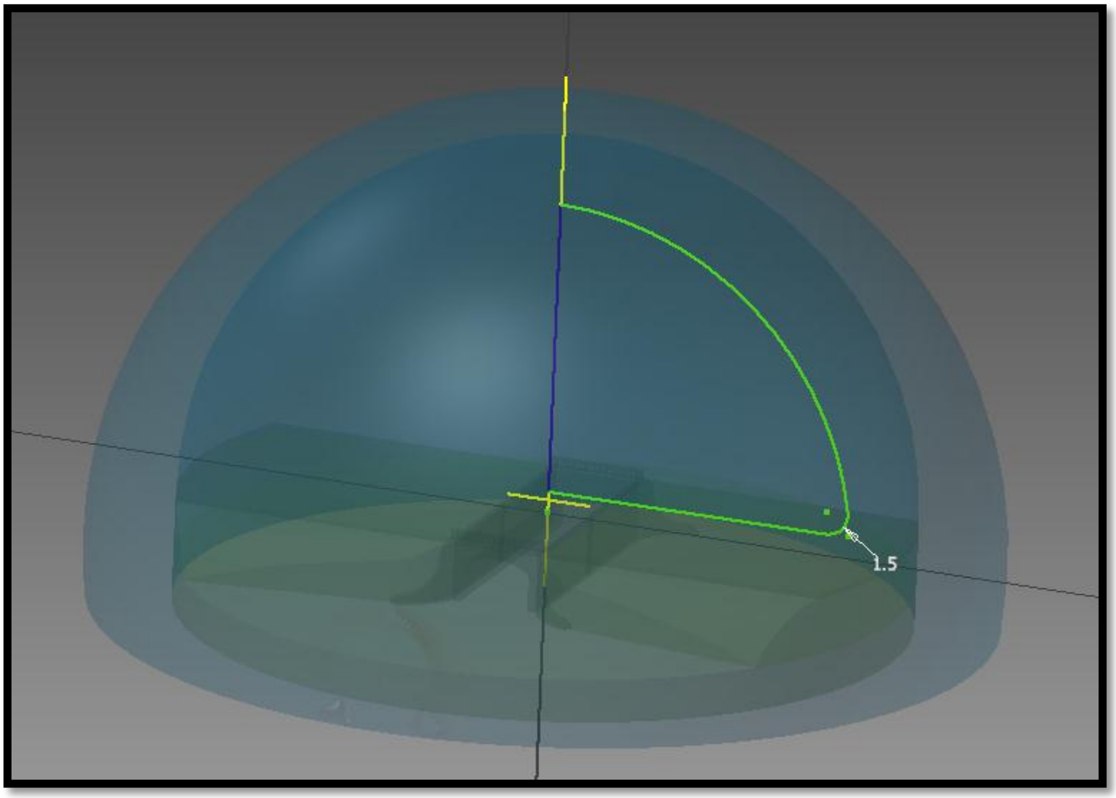


In Simulation CFD, space completely enclosed by a combination of parts, will create an additional part – a void fill. Therefore, the space between the Sky Dome and Ground part that is not occupied by the Bridge assembly will become this separate part when the model is launched into CFD. We are leveraging this function to surround the structure with Air – a part you normally do not create in CAD.

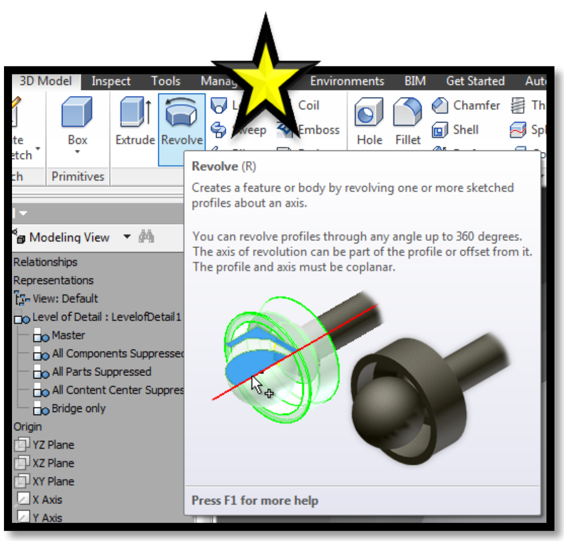
Our next effort will be to create a conceptual part to make a hole in the sky. With the hole, we can approximate the natural environment from the ground to the sky without modeling all the way up to the atmosphere. This conceptual part will minimize the mesh of the simulation analysis (reducing computational effort) and minimize solving effort by approximating natural convection in the environment with a surface boundary condition. The sky void should be created so it is completely between the geometry and the Sky dome part.

Tip: A good recommendation is to leave a reasonable gap (~3 meters) between the Sky Void outer surface and the Sky Dome inner surface. This will facilitate more uniform meshing. I also recommend add a substantial Fillet to the outer diameter edge. Solar radiation calculations can produce strange reflections on sharp corners. Since the Sky Void part is not physical, it is best to avoid such a corner.

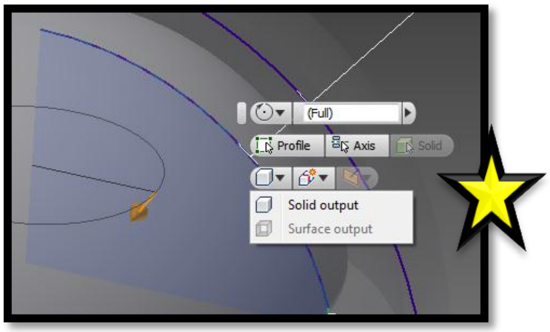
Use the create icon to create a new part - Sky Void.   Then create a sketch similar to the following:

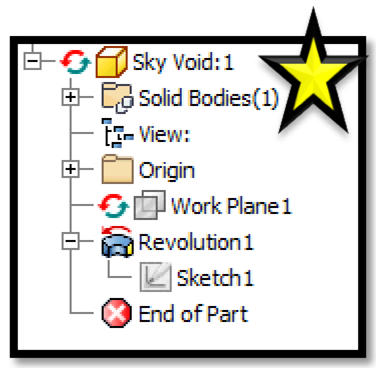


Use the Revolve icon to create the body of revolution, selecting the profile and axis for a Full revolution:



Be sure a solid body is created and not just a surface part – you can check this in the context menu to ensure the Solid output icon is selected or in the feature tree upon completion to see Solid Bodies (1) listed.

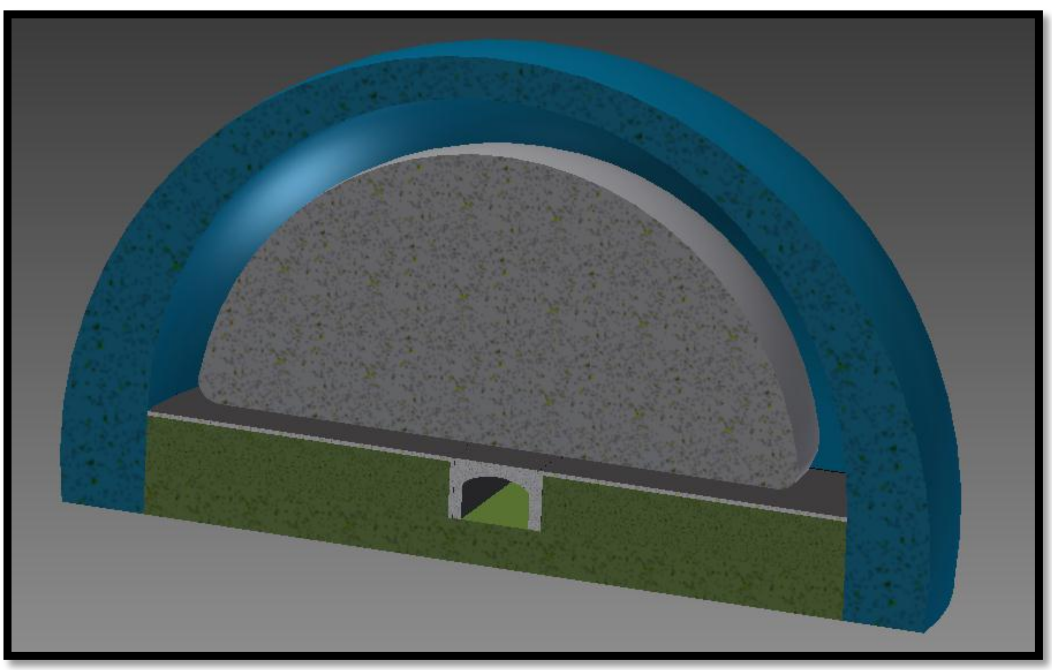




Return from editing the Sky Void part.

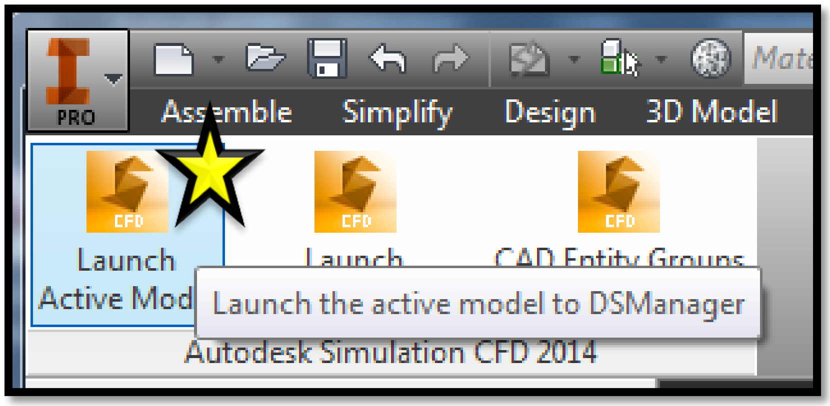
Create any other Additional Environmental Features

Depending on your structure's specific placement, you may need to add additional environmental features to account for uneven terrain, water features, or connecting structures. The Alvord Lake Bridge is actually an underpass for pedestrians to enter Golden Gate Park without crossing traffic. As such, the bridge was built into a slight rise in the ground. For aesthetics in the demonstration model, the raised ground and connecting roadbed have been added to the simulation model. In a Half Section View, the demonstration bridge looks like this in Inventor:



Launch Active Model

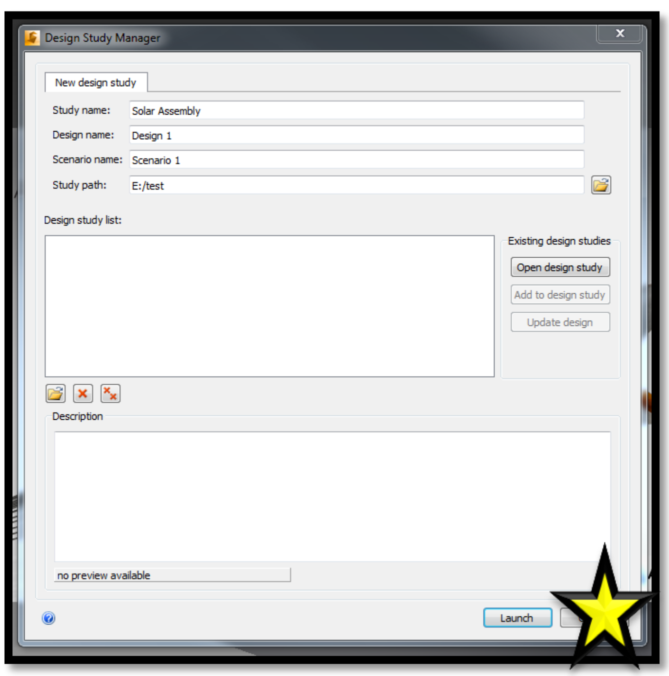
You are now ready to launch your analysis assembly into Simulation CFD for determining the Solar Thermal loads on your structure. Locate and select the Autodesk Simulation CFD 2014 tab in the Ribbon, then select Launch Active Model:



Note: To use the instructor completed analysis assembly, locate and open "Solar Assembly.iam" from the "Demonstration/Inventor Geometry" directory in Inventor Professional to continue with the exercise.

Click OK to continue.

Check the names and paths given for the "New design study" and proceed with Launch:

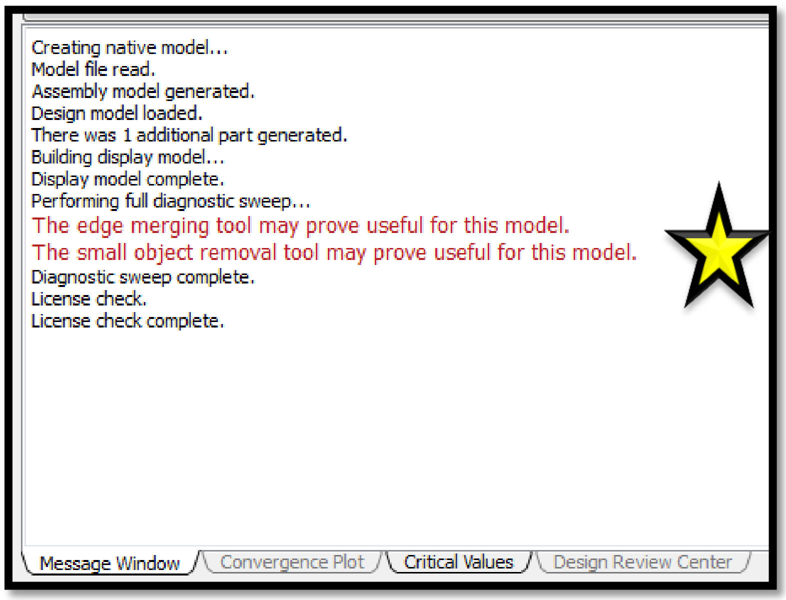


Section 2: Set up and simulate solar analyses in Simulation CFD

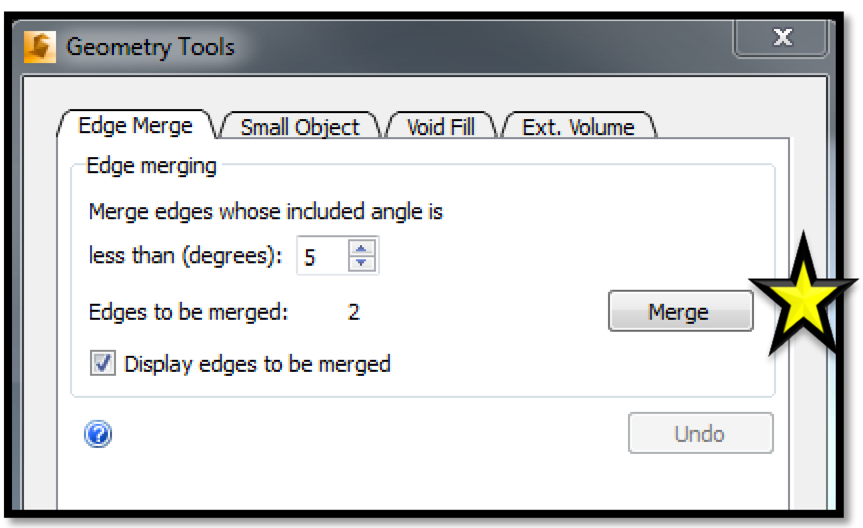
By the end of this section, you will know how to setup and solve for the time dependent temperature profile of your structure in Autodesk® Simulation CFD.

Navigation and Essential UI Skills

Upon launching from Inventor into Simulation CFD, red text may appear in the Message Window:



Along with the Geometry Tools dialog:



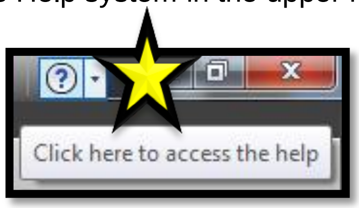
Select Merge in the dialog when Edge Merge is Active, since this will connect tangent edges for optimal computational performance. Then close the dialog.

Tip: Typically, structural geometry is of a size and proportion to prompt the user for Small Object removal in Geometry Tools. If tiny objects are found in the geometry, change to the Small Object tab and the small entities will be highlighted with red arrows in the display. The remove button will remove them from the analysis model to improve computational performance. For very large geometry, a good recommendation is to increase the Tolerance slider to the maximum (all the way to the right) then select remove.

In the training model, there are no small objects to remove. You can now close this dialog.

Default Navigation

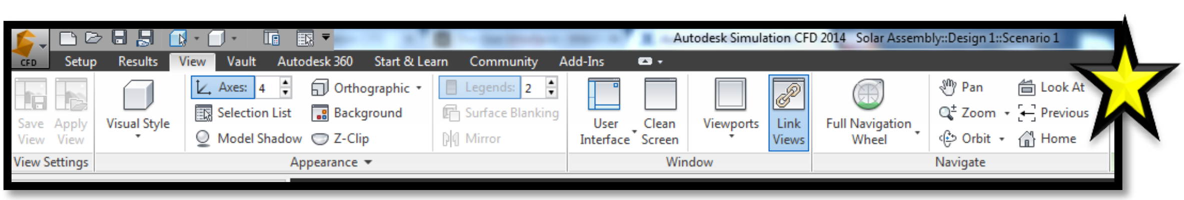
Information on model navigation and setup is provided for the novice user or as review. For complete details consult the online Help system in the upper right hand corner:



The navigation can be customized to personal preferences but for the purposes of this lab, the default settings will be used as follows:

Display Action	ADSK CFD Mouse Command
Wheel Zoom	Scroll
Rotate	Shift + MMB
Pan	MMB
Select/deselect	LMB
Rubberband select	LMB drag
Blank/hide	Ctrl + MMB
Show all	Ctrl + MMB off model

Additional navigation aides are available under the View tab:

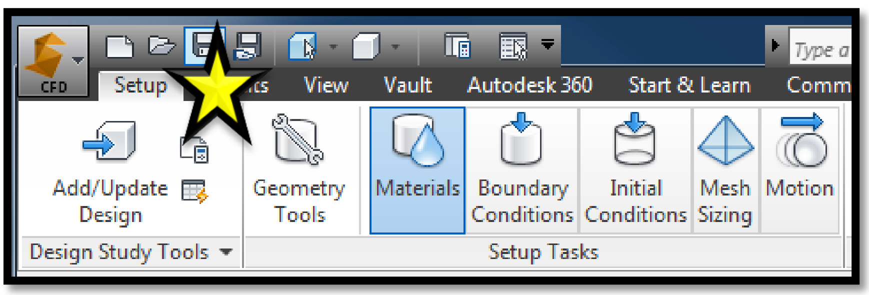


Navigation Practice

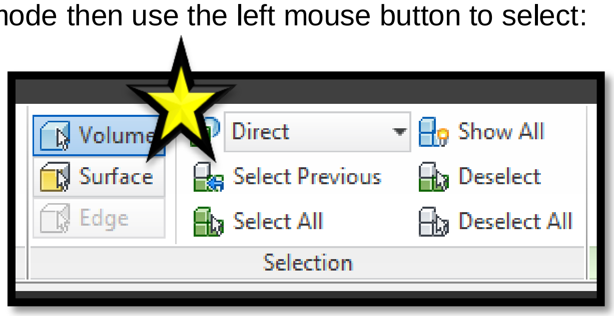
Take time to ensure you can zoom and rotate with the mouse or scroll wheel. Alternately, select the View tab for more navigation options or use the View Cube and Navigation tool bar in the display.

Selection Practice

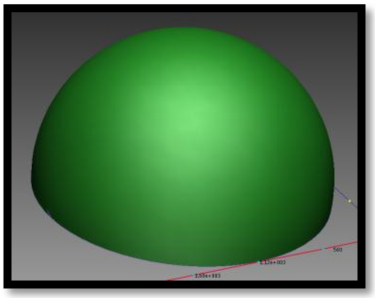
This next step will help you understand how to select items to apply analysis settings. Ensure the Setup tab is selected:



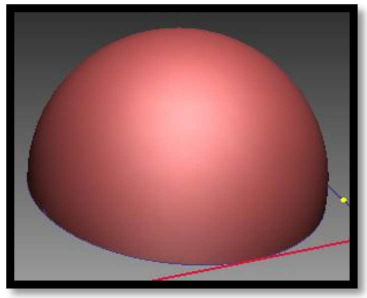
To select items for setup assignment, you will need to be able to select and deselect entities. Observe the Selection mode then use the left mouse button to select:



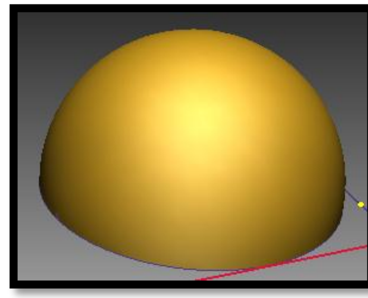
When volume and direct is selected in the Ribbon, a left mouse button click on an object in the display will select the volume (think geometry part) and once selected will turn that volume red:



Green – ready to select



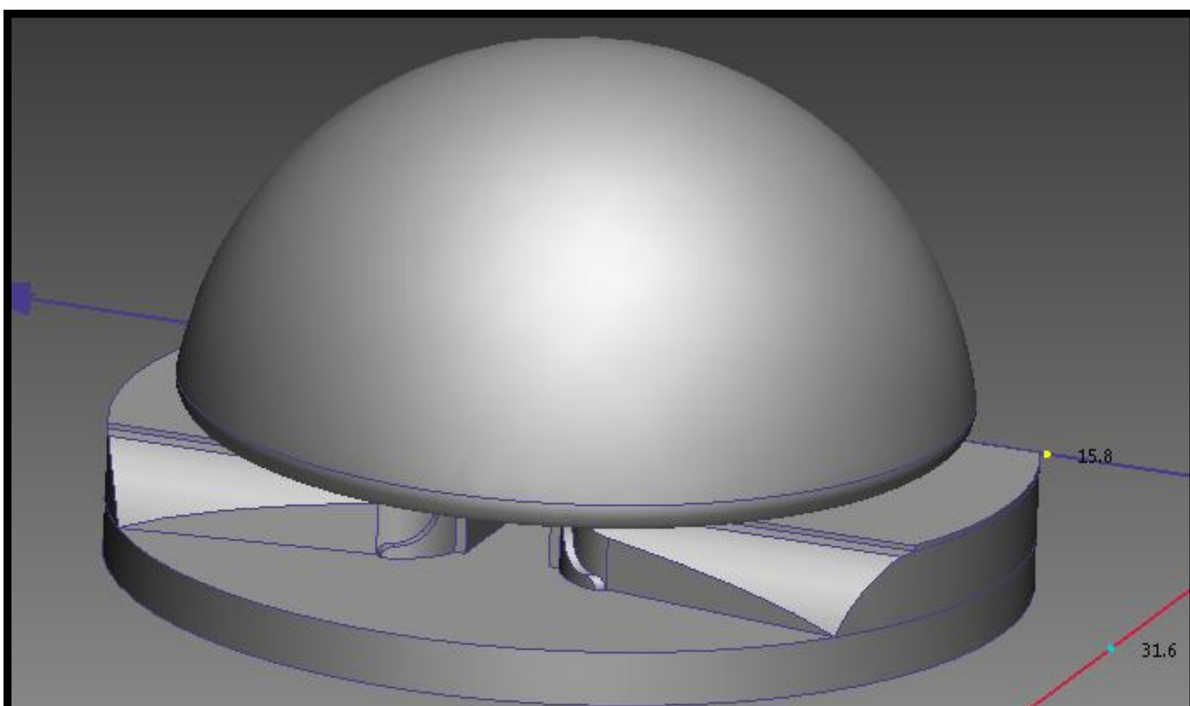
Red – Selected



Yellow – about to Deselect

Blanking and Hiding Practice

Finally, practice blanking or hiding geometry to see the inside of the analysis model. Move the mouse over the geometry you wish to hide, hold the control key, and then press the middle mouse button. In Volume selection mode, you will now blank the outside volume. Change to Surface selection mode to blank a surface only. Blanking one volume and two surfaces should leave you with a view similar to the following image:



To unhide the geometry, simply hold the control key and press the left mouse button anywhere off the model in the display window.

Apply Settings for a Solar Simulation Analysis

For the purpose of the hands-on lab, we will be analyzing the Alvord Lake Bridge on July 4th, 2013. We want to determine the temperature profile of the bridge throughout the day to be able to evaluate the peak thermal gradient which results in peak thermal stresses.

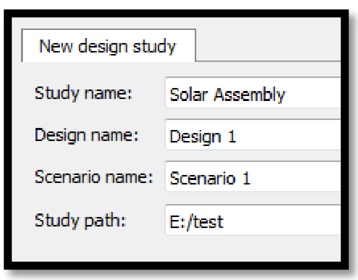
The following steps can be considered a recipe to follow for setting up many Solar Thermal analyses. In addition to the specific steps to complete the hands-on exercise, explanations are provided to apply the setting methodology to other analyses of interest.

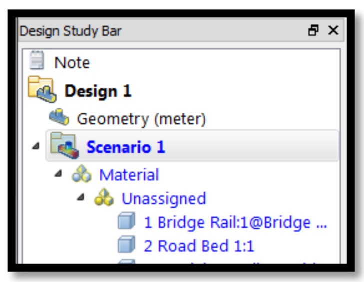
In general, think of the setup steps as telling the simulation tool what you know about the environment.

Name the Design and Scenario

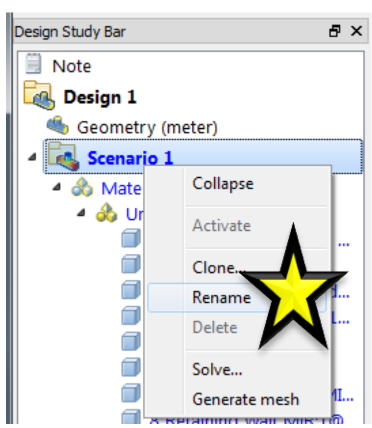
By now, you have your model in the Simulation CFD user interface and should be a little more comfortable navigating with the model.

If you did not change from default names when launching from Inventor, the analysis Design Study Bar will show Design 1 and Scenario 1 for the Design and Scenario names, respectively.





You might need to change the design and scenario names in cases where you are evaluating multiple designs in the same study or when a given design has multiple environmental conditions such as different seasons. The design and scenario names can easily be changed by right selecting on the name in the tree, selecting Rename and typing in a more sensible name:



Let's rename the scenario to July4th since that is the day we will be evaluating for this example.

Check Geometry units

How embarrassing and costly are unit system errors?

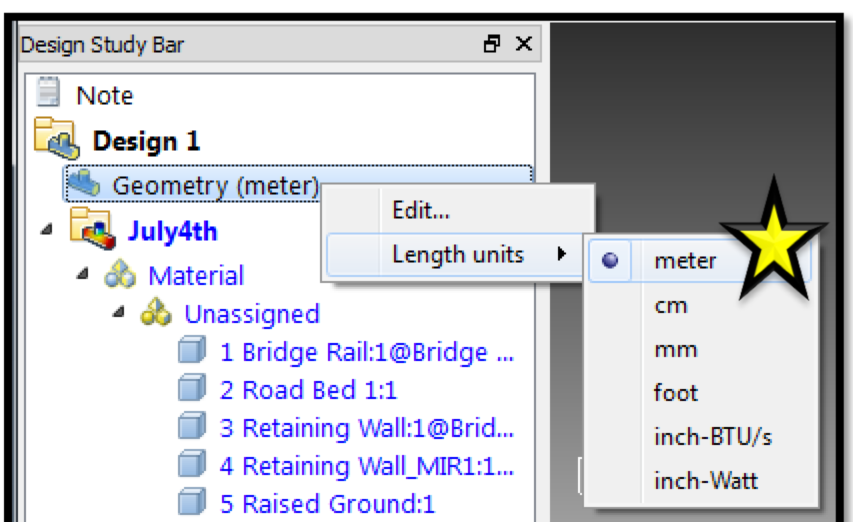
From a CNN website:

> *(In 1999) NASA lost a $125 million Mars orbiter because a Lockheed Martin engineering team used English units of measurement while the agency's team used the more conventional metric system for a key spacecraft operation, according to a review finding released Thursday.*

Tip: Always double check the analysis units for the Design. Use the Axis markings in combination with the unit system listed for the Geometry to ensure the model is the correct size. A right click on Geometry_Length units can help correct unit systems.

For our demonstration analysis, the Inventor parts were created in meters, and we will work with Watts to measure power. As a check, the "gumdrop" shape should be ~ 36 meters high.

Should you need to change length units, Right select on the word Geometry in the Design Study Bar. Pull down to Length units for the desired units.



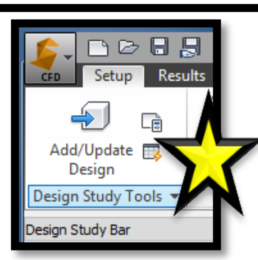
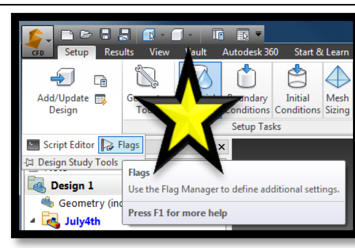
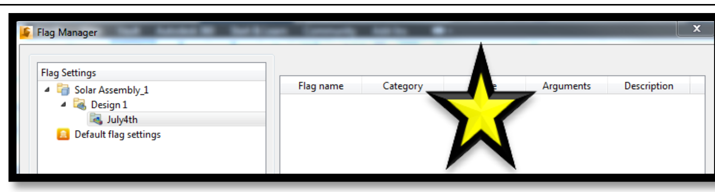
Correct the Solar Flux for the environment location

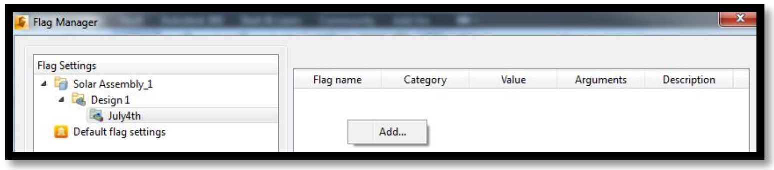
With the help of the internet, find the Insolation for the environment of interest (I looked up the Solar Power Map for San Francisco and found an average value of 4.6 kWh/m^2/day). Factor in how many hours of daylight exist per day for the time of year and determine the Solar Heat Flux value. For the 4th of July in San Francisco this year, there was approximately 13.5 hours of daylight.

For our example analysis, the calculation should look like this:

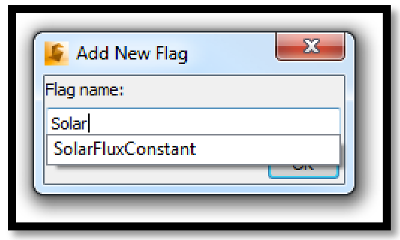
$$\text{Solar Heat Flux} = \frac{4.6\ \text{kWh/m}^2\text{/day} * 1000\ \text{W/kW}}{13.5\ \text{hours of sunlight/day}} = 340\ \text{Watts/m}^2$$

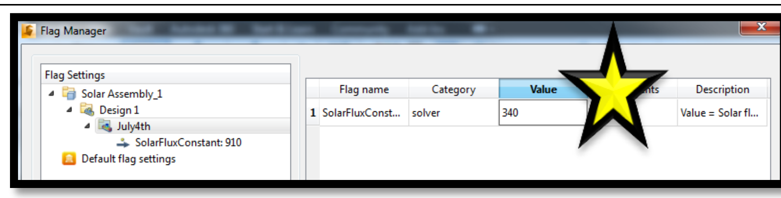
This value needs to be entered as a flag setting in the user interface.

Expand the Design Study Tools section in the Ribbon	
Select Flags	
Right Select in the Right hand panel of the Flag Manager	

<table>
<tr>
<td>Select Add</td>
<td>


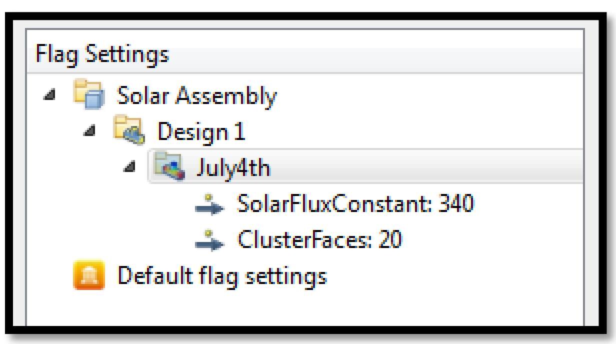
</td>
</tr>
<tr>
<td>Type Solar in the Flag name field

Select SolarFluxConstant and OK</td>
<td>


</td>
</tr>
<tr>
<td>Change the Value to 340 by clicking on 910 in the Right hand panel and typing 340</td>
<td>


</td>
</tr>
</table>

For the hands on lab, we will also use an additional flag setting to help speed the radiation calculations. Repeat the above steps to add ClusterFaces to the flags. Then set the value to 20. Select apply, then close the Flag Manager dialog. Your flag setting should appear as follows:

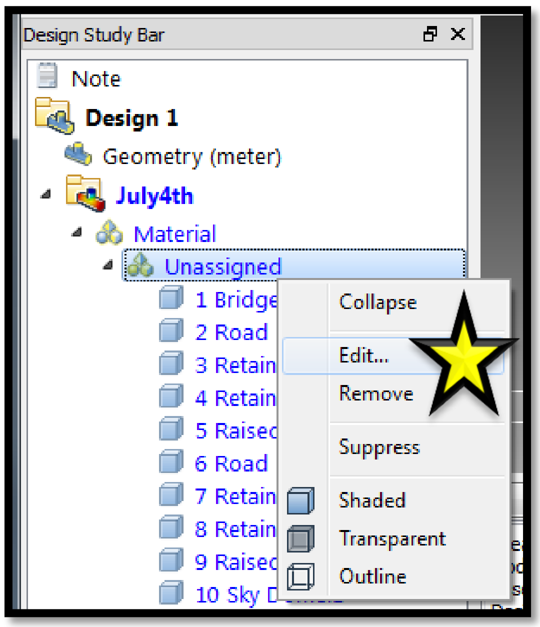


For a typical simulation with a more detailed model and mesh, this ClusterFaces flag is normally not needed. Once the flags are added, close the Flag Manger dialog.

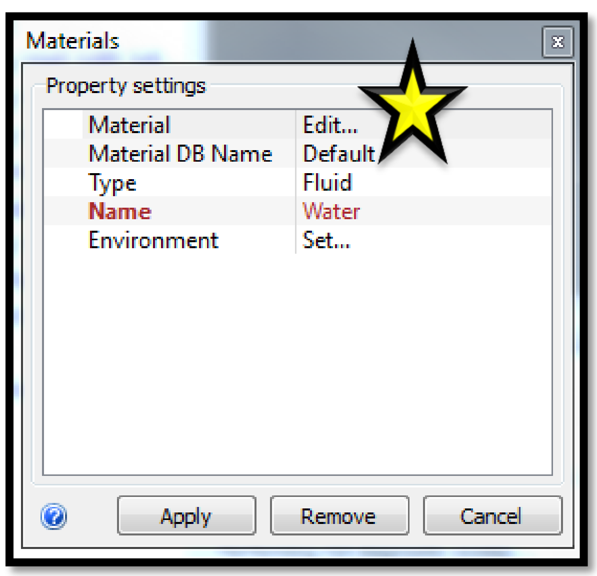
Add the demonstration material database

For the demonstration, a special material database has already been created for you. This will ensure everyone uses the same material properties for the demonstration. For more information regarding material databases visit the online Wiki help.

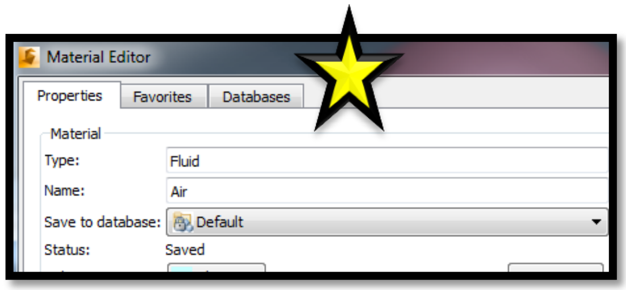
To see the custom materials created for this class, you will need to add the material database to the Material Editor. Right click on the word Unassigned in the Design Study Bar then select Edit…:



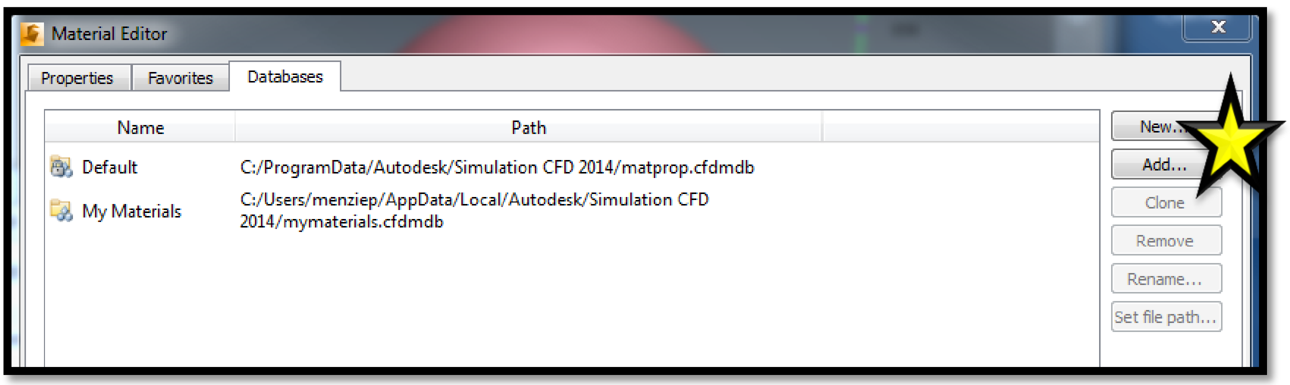
Next, select Edit in the Material dialog:



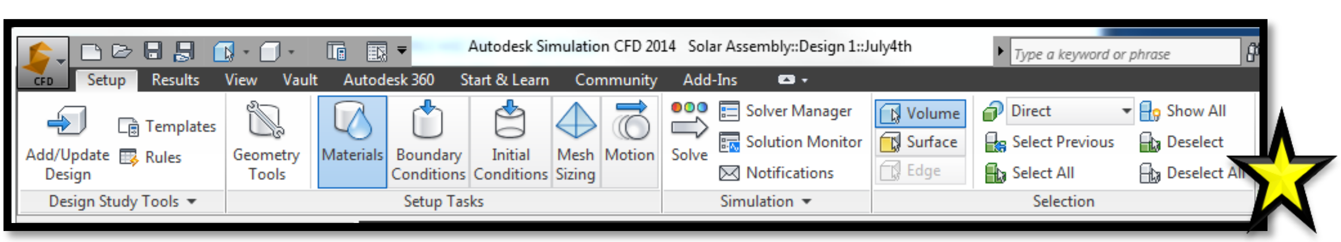
And proceed to the Databases tab in the Material Editor dialog:



Select Add…

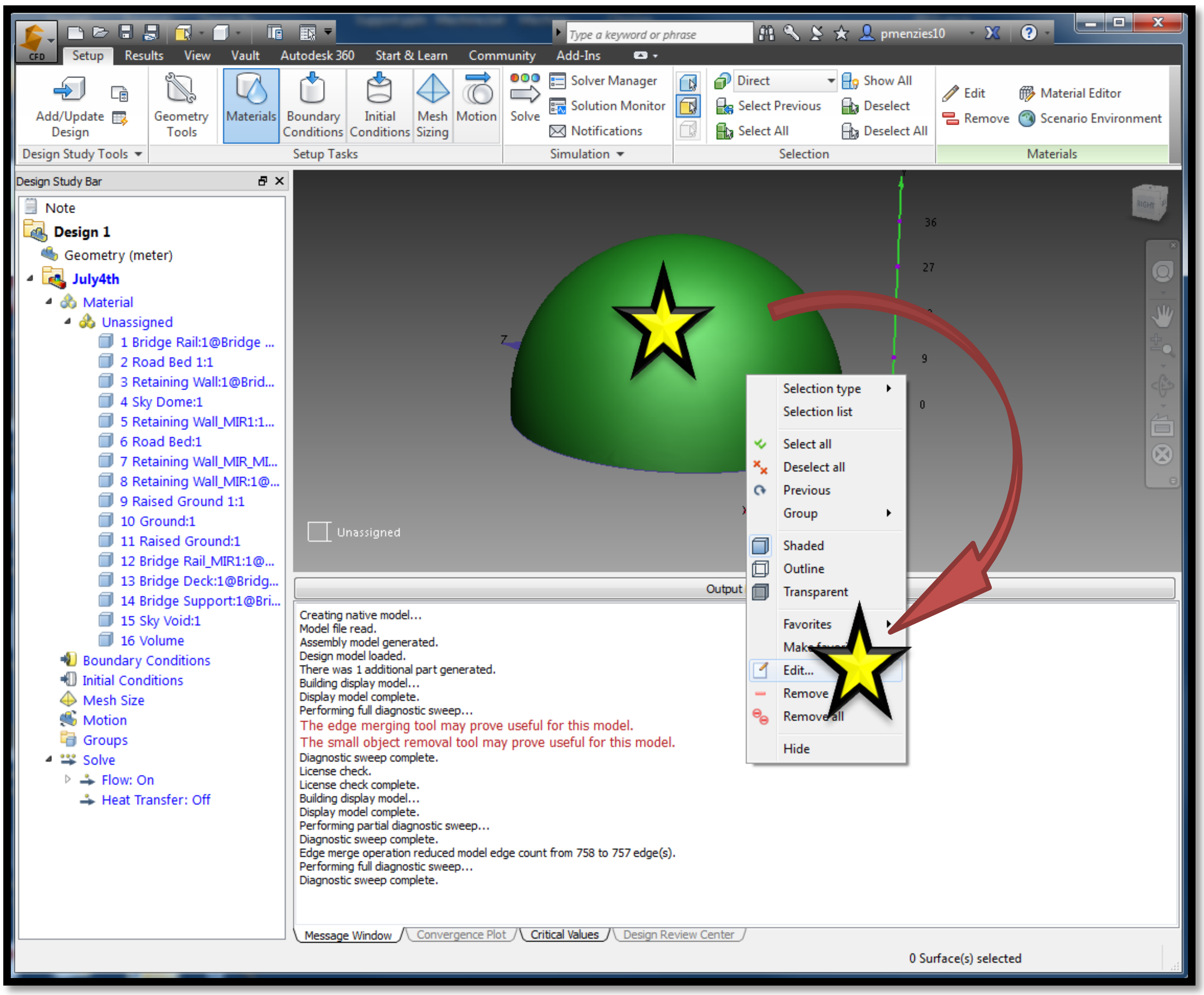


Then navigate to the "Demonstration/Datafiles" directory and select "Demo Materials.cfdmdb" and then click Open. You can now close the Material Editor dialog and cancel the Materials dialog. To make sure nothing has been selected by mistake, please click on Deselect All in the ribbon:



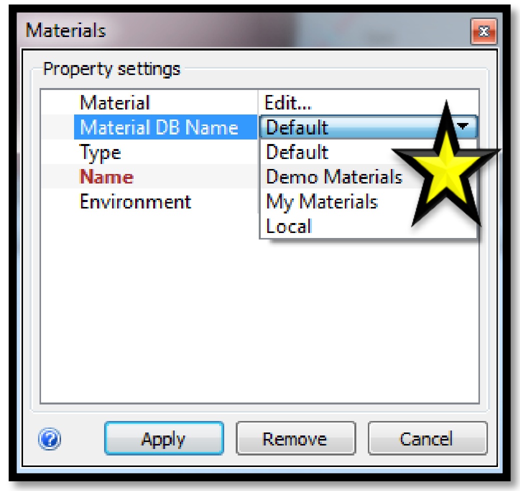
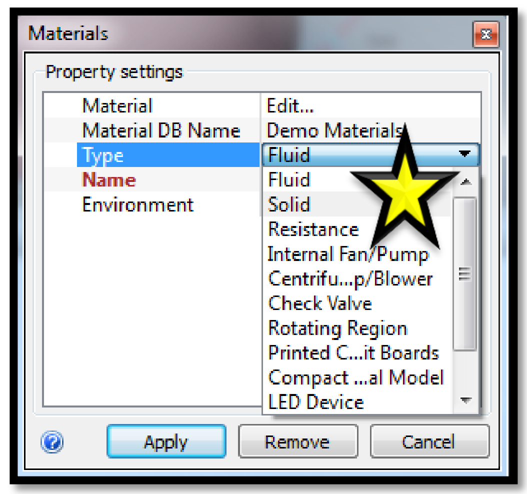
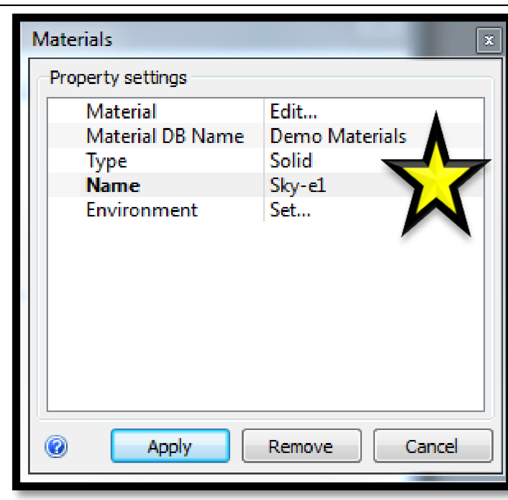
Assign Materials to the geometry parts

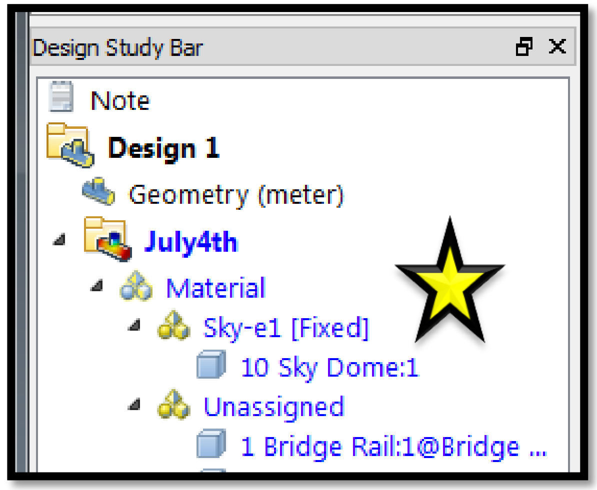
For item listed under Material in the Design Study Bar, we will assign a material to let the simulation tool know what the various parts are made of and their thermal properties. In general, you will mouse over a volume in the display window (the volume will turn green), then right select and pick Edit from the context menu. You will know you have selected the correct volume when it turns red at this point.



Sky Dome -> Sky-e1

Starting from the outside of the model to the inside, you should first apply a material to the outside dome which represents outer space. Mouse over the dome, right select, select edit in the context menu, then you will make the following selections in the Materials dialog:

<table>
<tr>
<td>Use the Material DB Name pull down to select the Demo Materials Database</td>
<td>
</td>
</tr>
<tr>
<td>Change Type to Solid</td>
<td>
</td>
</tr>
<tr>
<td>Ensure Sky-e1 shows for Name</td>
<td>
</td>
</tr>
</table>

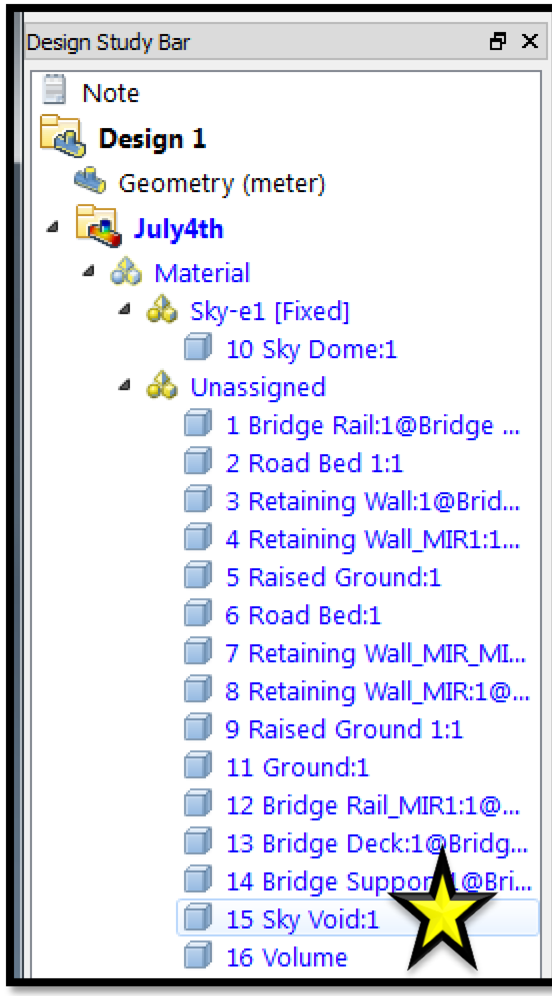
<table>
<tr><td>Select Apply</td><td>
</td></tr>
<tr><td>Verify correct material assignment in the Design Study Bar</td><td>
</td></tr>
</table>

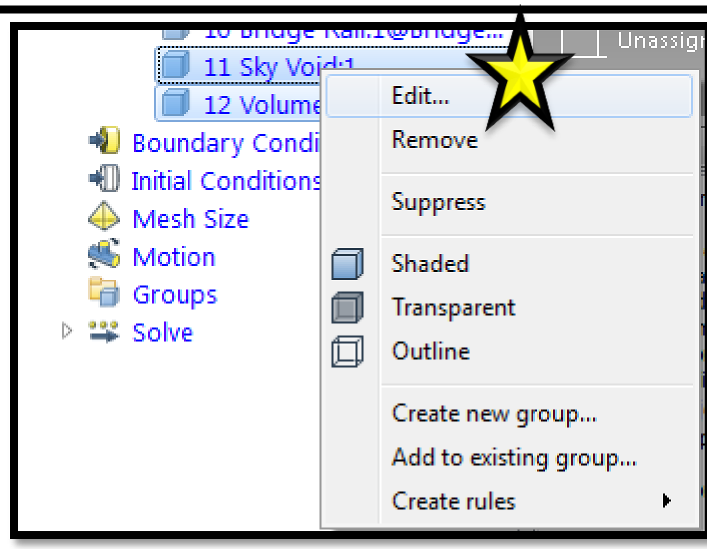
The purpose of the Sky Dome is to mimic outer space in the simulation. The Demo database contains Aluminum from the default database since it is fairly conductive but was customized with an emissivity set to 1 to represent a black body. All radiation to outer space will be absorbed.

Sky Void and Volume –> Air-conductive

Now that the Sky Dome is assigned to Sky-e1 material, blank the Dome (hold the ctrl key and middle mouse select on the dome) to see the created volume inside the dome. This volume will represent the ambient environment (the air) that surrounds the bridge.

We will use a different strategy to apply materials to these volumes so you can chose your favorite method going forward. This method uses the Design Study Bar. Ultimately, we want the Sky Void and the Void Fill materials to be Air-conductive.

Select the Unassigned volume "Sky Void:1" in the Design Study Bar (use the left mouse button)	
Hold the Control Key and Select "Volume" in the Design Study Bar (again use the left mouse button)	

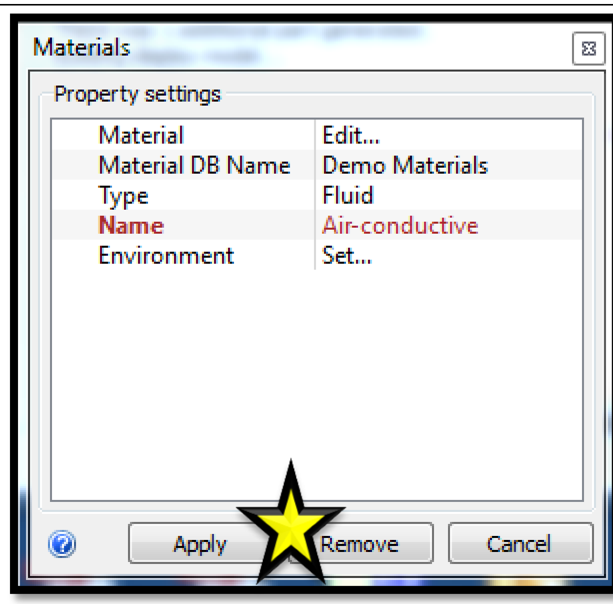
<table>
<tr>
<td>Right Select on one of the volume names and select Edit… from the context menu</td>
<td>
</td>
</tr>
<tr>
<td>Ensure:

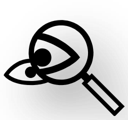
Material DB Name is set to Demo Materials

Type is set to Fluid

Name is set to Air-conductive

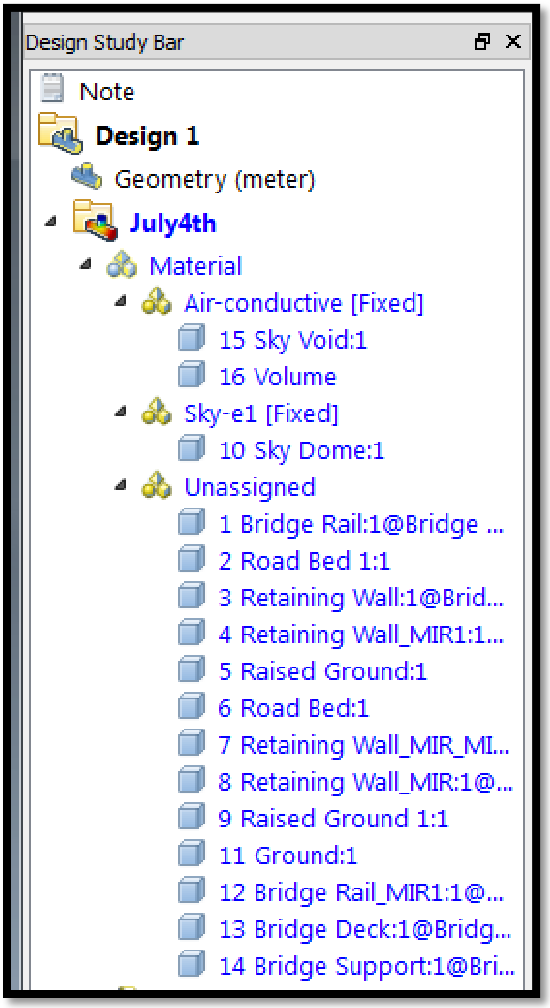
in the Material dialog then

Select Apply</td>
<td>
</td>
</tr>
</table>



The custom Air-conductive material is a copy of the default database air material with the conductivity increased 100 times to account for the increased heat transfer due to natural convection. Basically, the air surrounding a structure is never still which increases heat transfer. For computational performance, we will only use the CFD tool to solve the heat transfer during the simulation not the flow field. Without a moving flow field, the additional heat transfer (due to natural convection currents) will be approximated with an increase in thermal conductivity.

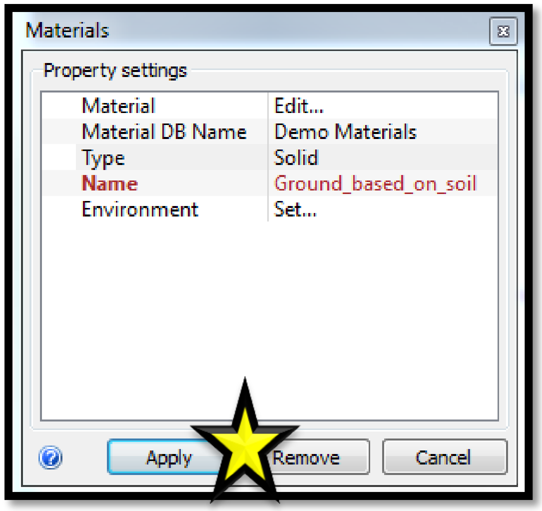
Your Design Study Bar should be filling under the Material's section like this:



Note: there are two volumes listed under Air-conductive and one volume listed under Sky-e1.

Ground, Raised Ground, and Raised Ground 1-> Ground_based_on_soil

You can choose either strategy for assigning materials for the Ground volumes. With further blanking, the three ground volumes should be visible in the display or they can easily be selected from the Design Study Bar. Once selected, edit and on the Materials dialog ensure the following settings are visible before selecting apply:



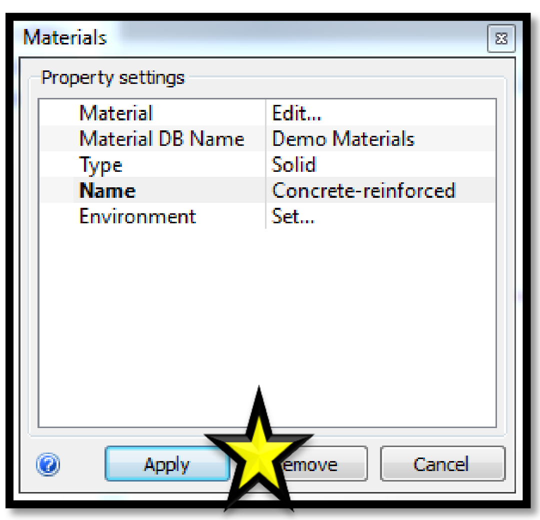
The custom ground material is based on the default database for Soil (Sandy) type in the default database. Adjust the thermal properties including emissivity (=1-albedo) for your particular situation as appropriate for best accuracy.

Remaining Bridge and Road Bed volumes-> Concrete-reinforced

The remaining volumes are all part of the bridge or concrete road bed and will be assigned the custom solid material, “Concrete-reinforced.”

Tip: By applying materials to all the easily selected parts first, you can select on the word “Unassigned” in the Design Study Bar to edit all the remaining volumes to apply Concrete-reinforced. Saving the most numerous parts for material assignment for last, reduces the number of clicks to apply the materials.

On the Materials dialog ensure the following settings are visible before selecting apply:

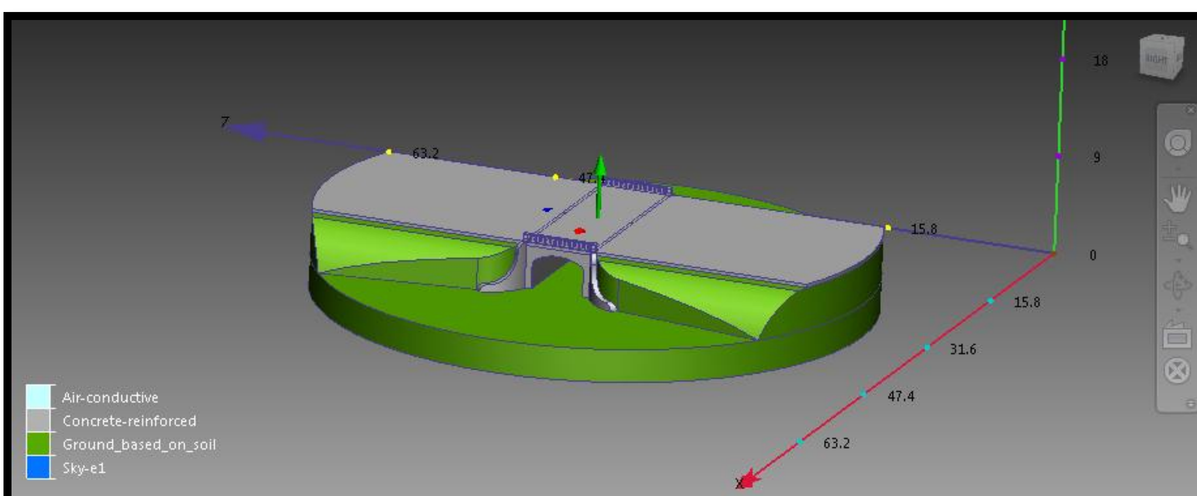


If you are interested, you can review the concrete material properties by selecting edit to view the Material Editor. Note the emissivity of 0.85. There is much discussion on the reflectivity of concrete depending on age and weathering.

Check Material Assignments

It is always wise to double check your material settings before moving on. The easiest check is to ensure the Material "Unassigned" does not exist in the Design Study Bar. You should see the following by volume number:

If you blank through the model, you will be able to see various material colors applied in the display to match the legend:



Assign Boundary Conditions to geometry parts

In this step, we will assign what we know to the boundaries of the analysis. Recall this will solve for the temperature profile in the concrete structure as a function of time. We will start simulating at midnight to analyze the 24 hours of the 4th of July.

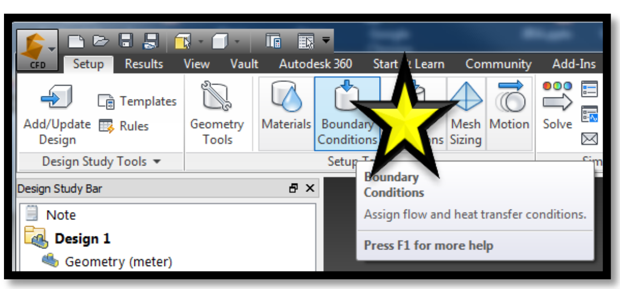
First we will need to model outer space with respect to radiation. The boundary definition can be represented with two temperatures -> 12°C during daylight hours and -43°C at night. Using an internet search for July 4th in San Francisco, we can determine that sunrise occurred at 5:53 am (we'll approximate this as 5:45am) and sunset occurred at 8:35pm (we'll approximate this as 8:30pm). We will want this information in a comma separated value (.csv) file format to provide a piecewise linear representation for the boundary condition. Tabulated, the data is as follows:

Iteration	Temperature °C	Simulation Elapsed Seconds	Comment
0	-43,	0	Midnight – the sun is down
22	-43,	19800	5:30 am start transition to sun up
24	12,	21600	6:00 am – the sun is up
81	12,	72900	8:15 pm start the transition to sun down
83	-43,	74700	8:45 pm – the sun is down
96	-43,	86400	Midnight – the sun is down

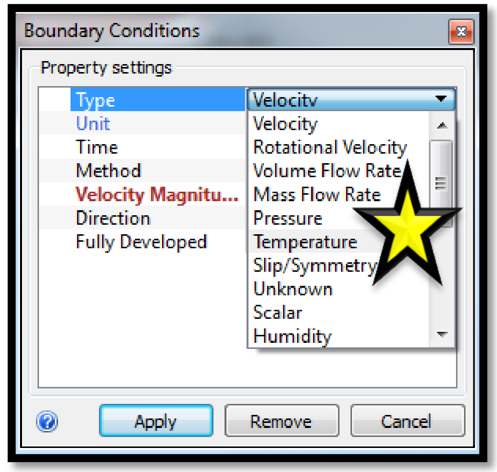
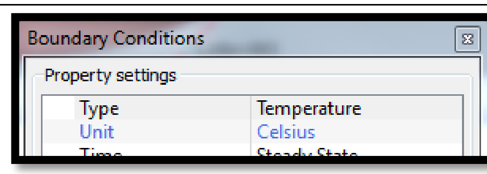
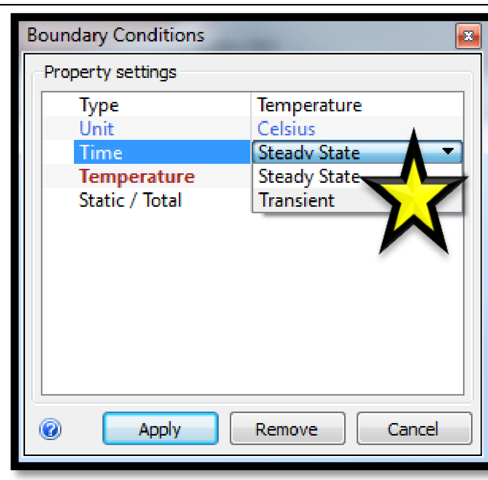
The values in green are needed in the .csv formatted file. This has been saved for you in the "Demonstration\Datafiles" directory, "Sky temperature.csv."

Sky Dome Exterior Surface -> Transient Temperature Boundary Condition

To ensure geometry parts are not blanked, hold the control key and select the middle mouse button in the background of the display window. Then select Boundary Conditions in the ribbon to proceed to the next phase of analysis setup:

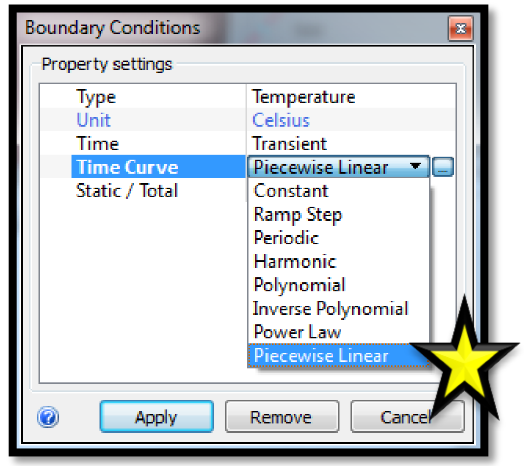


Similar to selecting items for material assignments, you will select surfaces for Boundary Condition assignments. Mouse over the dome, right select, select edit in the context menu, then you will make the following selections in the Boundary Conditions dialog:

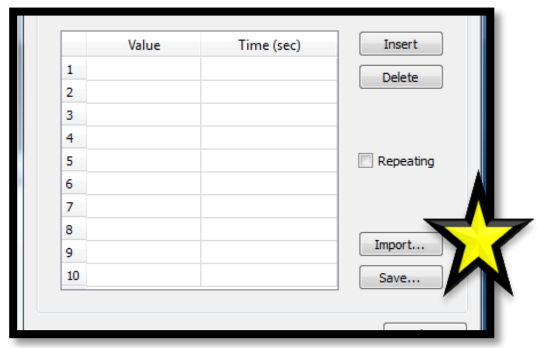
Use the Type pull down to select Temperature for the type of boundary condition	
Use the default unit system for the analysis units Celsius	
Change Time to Transient since the analysis will solve as a function of time	

Select the Time Curve edit box 

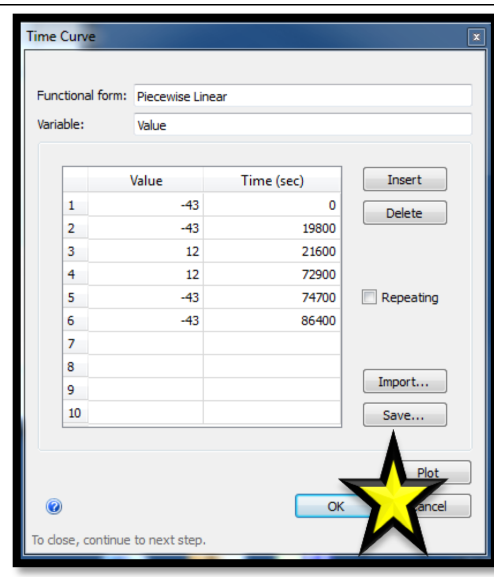
And use the pull down to select Piecewise Linear which opens the Time Curve dialog



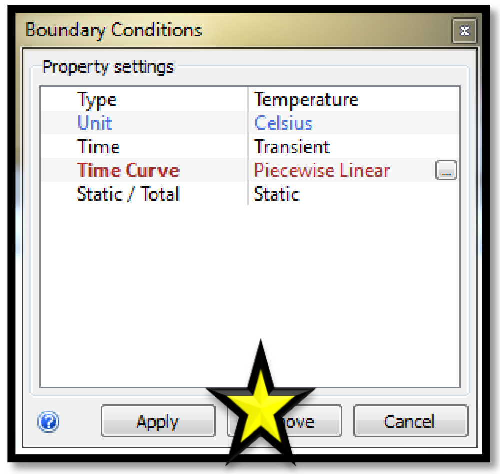
Select the Import… button



Navigate to the “Demonstration\Datafiles” directory to select Sky temperature.csv

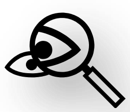


Select OK (or plot if you would like to see the function in a graph), then Apply



You should now see a teal blue stripe on the dome and an entry in the Design Study Bar for the Temperature Condition.

Ground Exterior Surface -> Steady State Temperature Boundary Condition



Deep enough in the Earth, the temperature is considered to be fairly constant at 5°C. You will apply a temperature boundary condition on the exterior surface of the ground to mimic this consistent temperature.

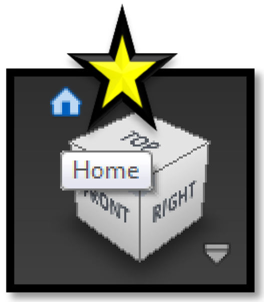
Navigate the model in order to see the bottom flat surface – the bottom of the ground. You can use orbit, the view cube and select the bottom face, or hold shift with the middle mouse button. You will want to select the circular surface (not the annulus) to apply the temperature boundary condition. As with the Sky Dome surface, move the mouse of the surface until it turns green, then right select to Edit… .

Ensure the following settings are in the Boundary Condition dialog before selecting Apply:

Type	Temperature
Unit	Celsius
Time	Steady State
Temperature	5
Static/Total	Static

Sky Void Surface -> Transient Film Coefficient Boundary Condition

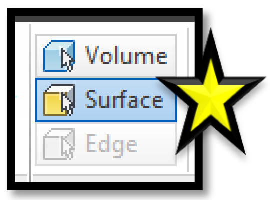
Recall, the Sky Void will actually be a void in the analysis model; therefore there is an exterior surface in the middle of the model. To see this surface, we will first restore the view to the Home view by selecting the House on the View Cube.



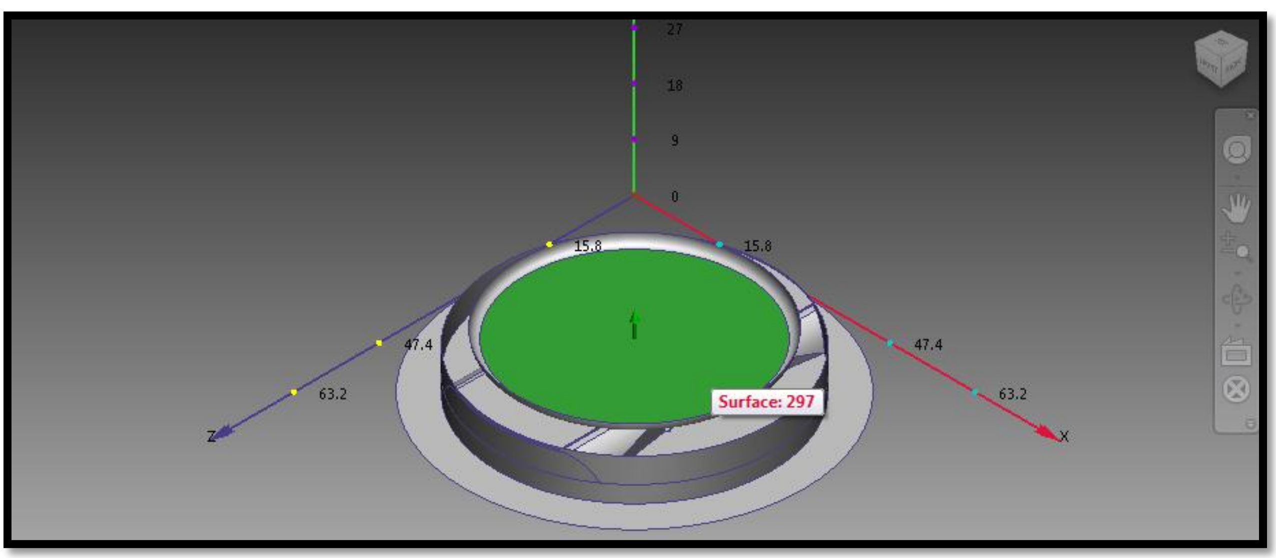
.

Hold the middle mouse button and select in the display window but off the model.

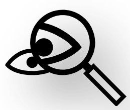
Now mouse over the dome, hold the control key then right select on the dome surface. This will blank the surface and not the volume (as before) because we are now in Surface selection mode.



The next surface you see is the inside of the Sky Dome – the interface between the sky and the ambient air. Blank this surface too and the next dome like surface (the surface of the Sky Void volume). The display of your model should now look like this with the mouse hovering over the Sky Void lower surface:

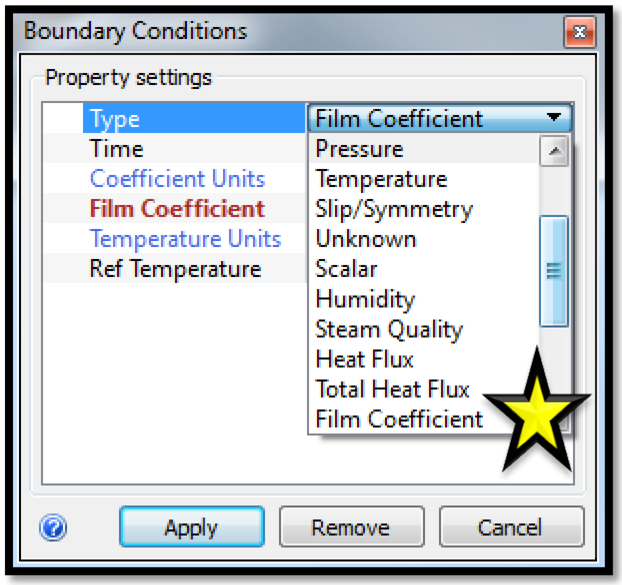


Similar to the Sky Dome temperature, this boundary condition will also be a transient boundary condition with a table of data to import. This condition mimics the convective cooling of the structure in relation to the ambient temperature conditions. Neglecting wind, the cooling

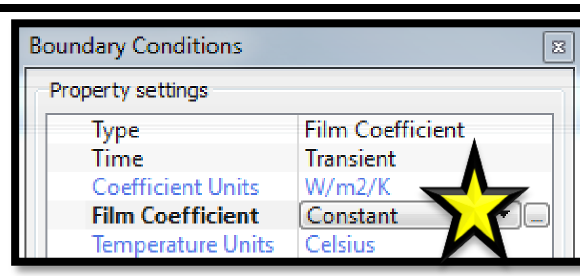


coefficient can be considered = 100 Watts/ (m^2 °K). The reference temperature will be varying with the ambient air temperature as reported for July 4th this year. The high temperature for the day was recorded as 25°C with a low of 15°C. This will vary as a sinusoid with the peak high temperature occurring at 4pm and the minimum temperature occurring at 4am. A second comma separated value table has been created for the exercise with these constraints.

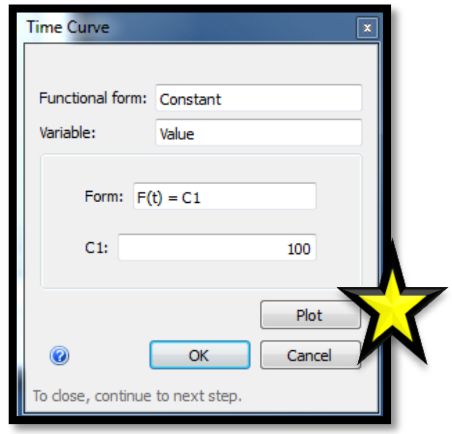
Right select and edit the circular surface and select Type = Film Coefficient. You will need to use the side scroll bar to find Film Coefficient near the bottom of the list.



Then select Time = Transient and Coefficient Units = W/m2/K. For this example we will assume the worst case for thermal loading; when there is no wind.

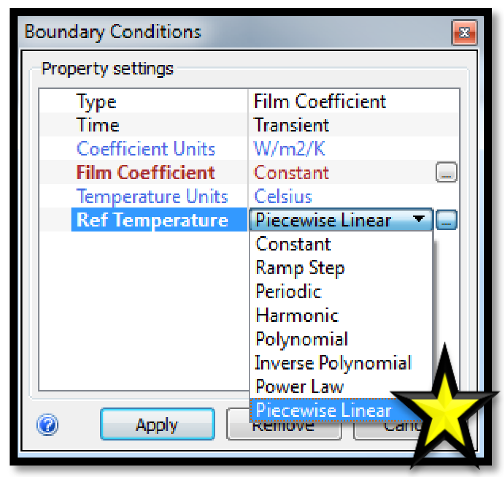
Film Coefficient = Constant (Select the edit box  if the Time Curve dialog does not appear)	

Enter 100 for C1:

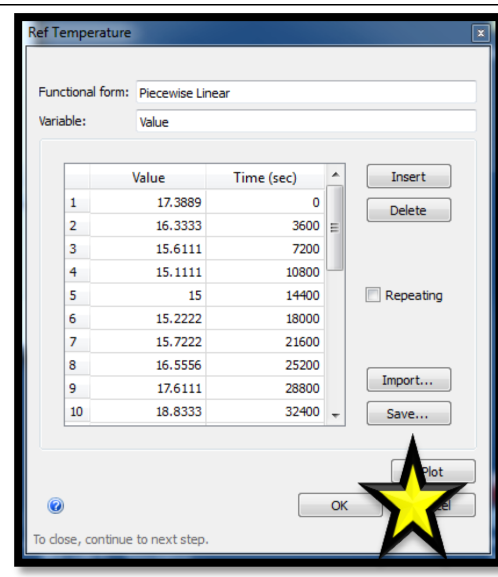


Ensure Temperature Units = Celsius

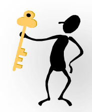
Then for Ref Temperature, select the edit box to select Piecewise Linear



In the Ref Temperature dialog, select import then navigate to the "Demonstration\Datafiles" directory to select "Hourly temperature.csv"

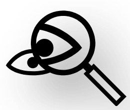


Select OK, then Apply on the Boundary Conditions dialog. Look for a green stripe on the surface when complete.



Tip: If you need to account for a strong wind for your analysis, the film coefficient constant can also be used in table form where the coefficient is a function of the constant plus a factor times the wind speed.

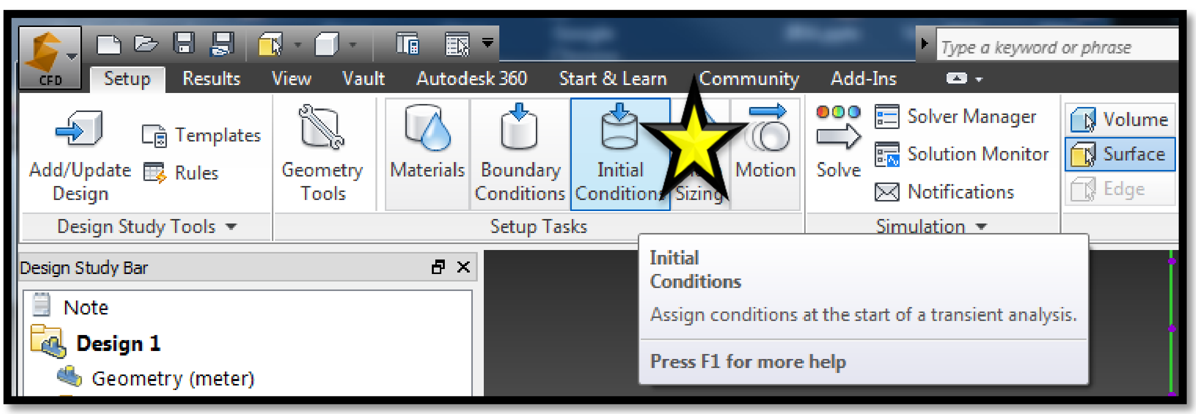
Assign Initial Conditions to geometry parts



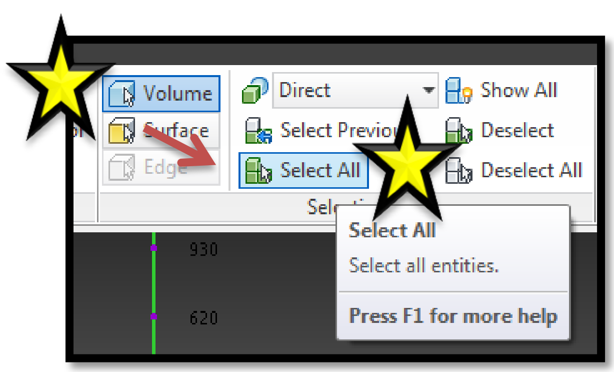
With time dependent analyses, when the simulation analysis starts it is useful to pre-assign the temperature to all parts of the analysis since temperature profiles would exist in all the parts in the actual environment. In this case, we start the analysis a significant number of hours prior to our time of interest to build the correct temperature profile as it would build in the actual environment.

We will use the ambient temperature at midnight when the analysis starts as the initial temperature condition applied to all volumes – from the “Hourly temperature.csv “ file, the first line of data contains the temperature of **17.3889°C**.

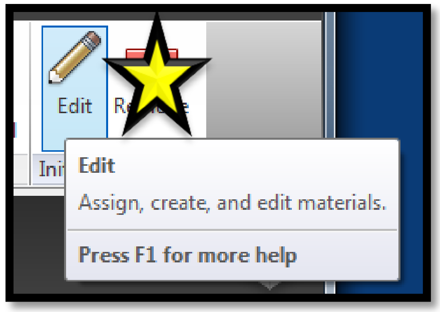
To apply the Initial Condition, proceed to Initial Conditions in the ribbon:



Ensure the Volume selection mode is selected, and then use Select All:

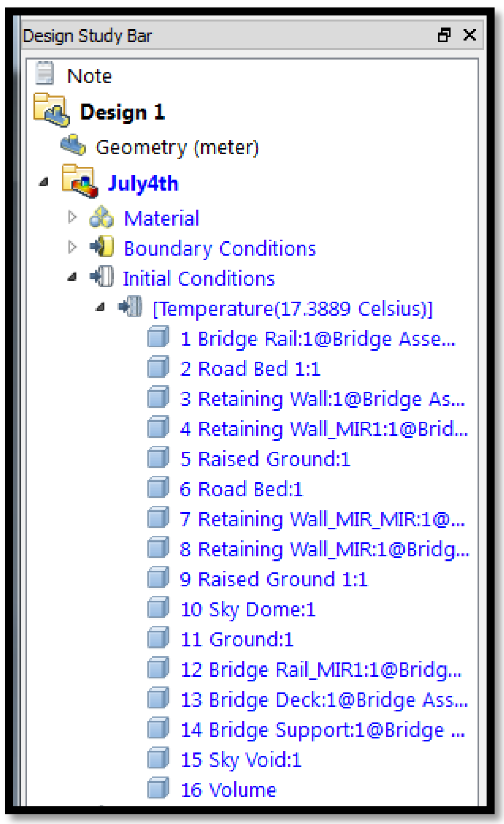


Next, select Edit from the Ribbon:



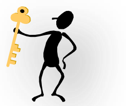
Ensure Type is set to Temperature, Units to Celsius and leave the Re-initialize check box unchecked. Key in 17.3889 for Temperature and leave the Static/Total as Static. Click Apply.

Check to ensure the settings are correct in the Design Study Bar:



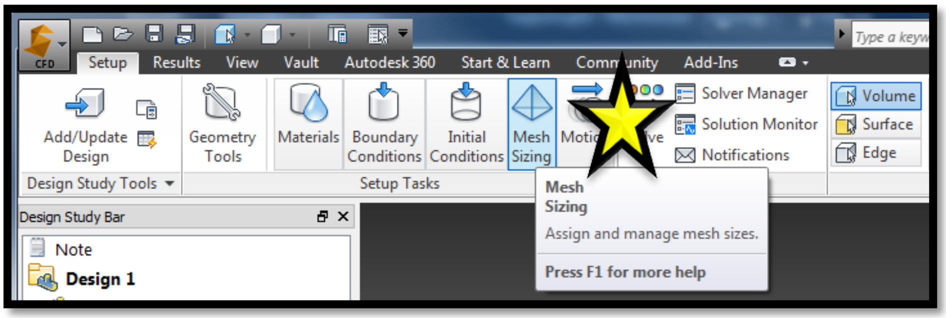
Apply Mesh Settings to Geometry parts

Apply simple mesh settings to achieve the best quality mesh for typical geometry. There are three tasks to do for mesh settings – suppress geometry not to be meshed, turn off “enhancement layers” since this is not a fluid analysis, and adjust the geometry refinement length to account for high aspect ratio geometry.



Tip: Small adjustments, geometry tool model simplifications, eliminating enhancement for heat transfer only analyses, and mesh suppressed voids, will reduce the mesh and increase computational performance – especially helpful for a series of analyses.

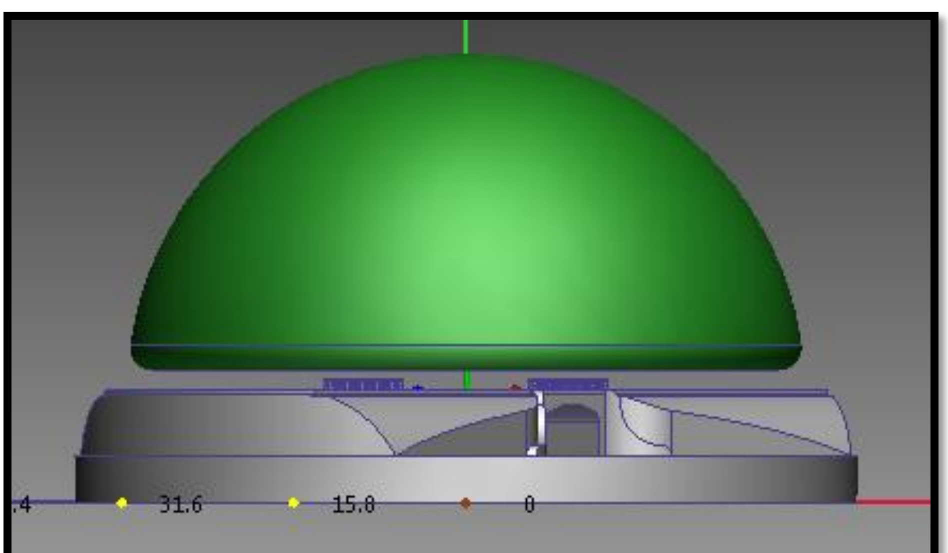
To apply mesh settings, proceed to Mesh Sizing in the ribbon:



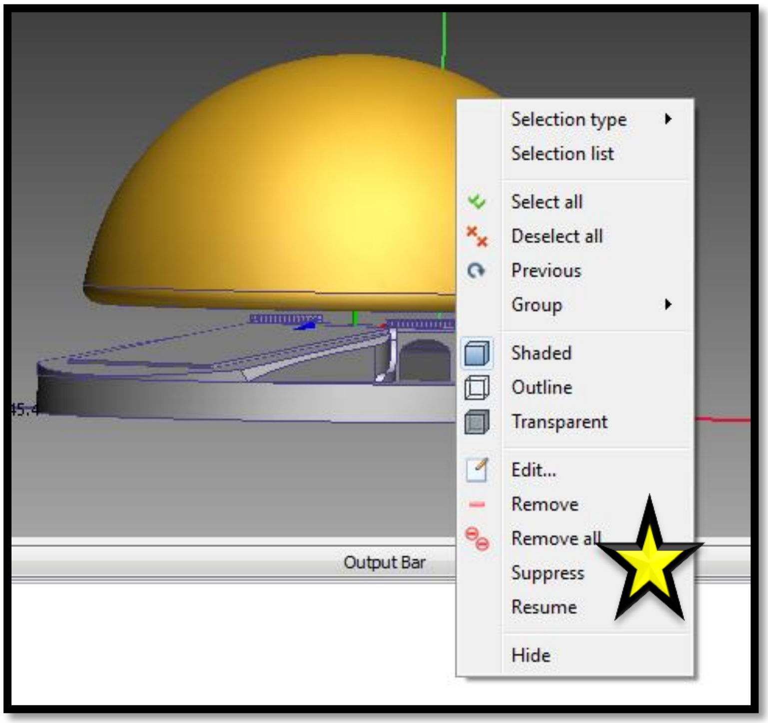
Mesh suppression

Recall the Sky Void part. This part was created to make a void in the simulation analysis. As such we will suppress this part to prevent it from meshing in the analysis model.

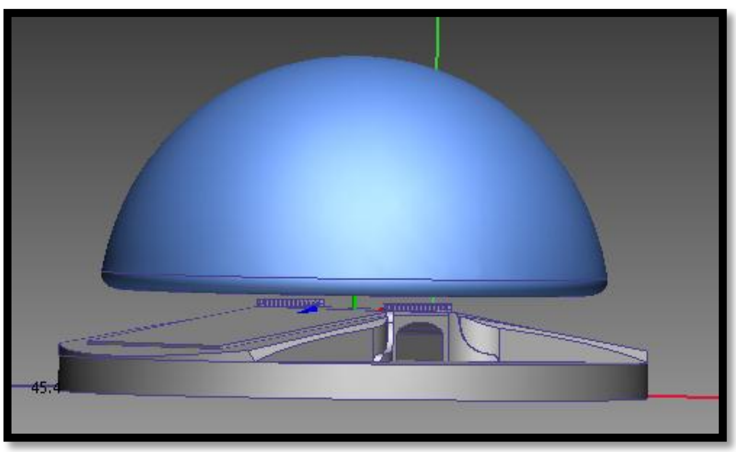
Undo any blanking in the display by holding the control key and selecting the middle mouse button in the display window. Ensure Volume selection mode is still selected in the ribbon (see Initial Conditions section for images). Hold the control key and select with the middle mouse until the Sky Void part is visible above the bridge. If it looks like an alien spacecraft hovering over the bridge, you have correctly blanked the correct volumes:



Hover the mouse over the spaceship shape, then right select to find suppress in the context menu:



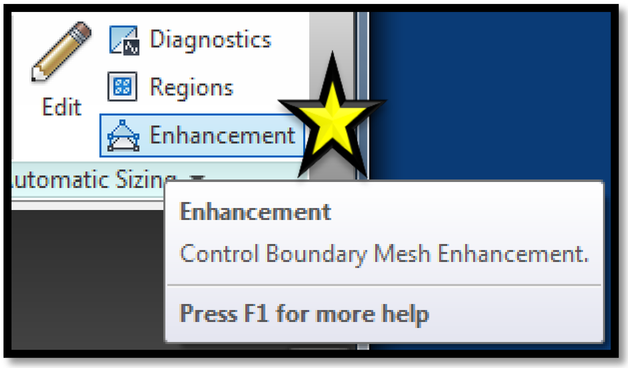
The part in the display will turn blue when suppressed:



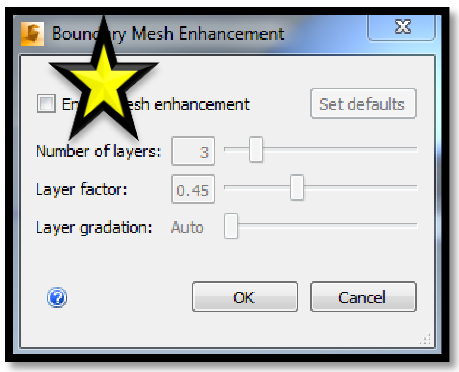
Disable or Turn off mesh enhancement

When solving for fluid flow, in addition to heat transfer, it is desirable to place extra mesh along all the solid-fluid interfaces. For this heat transfer only analysis, there is no need for those extra elements so you should disable the enhancement.

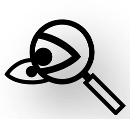
In the ribbon, find Enhancement and select:



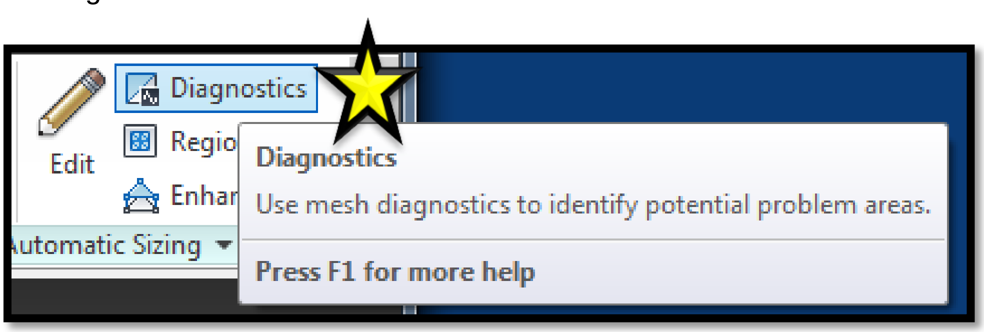
On the Boundary Mesh Enhancement dialog, uncheck the Enable mesh enhancement check box:



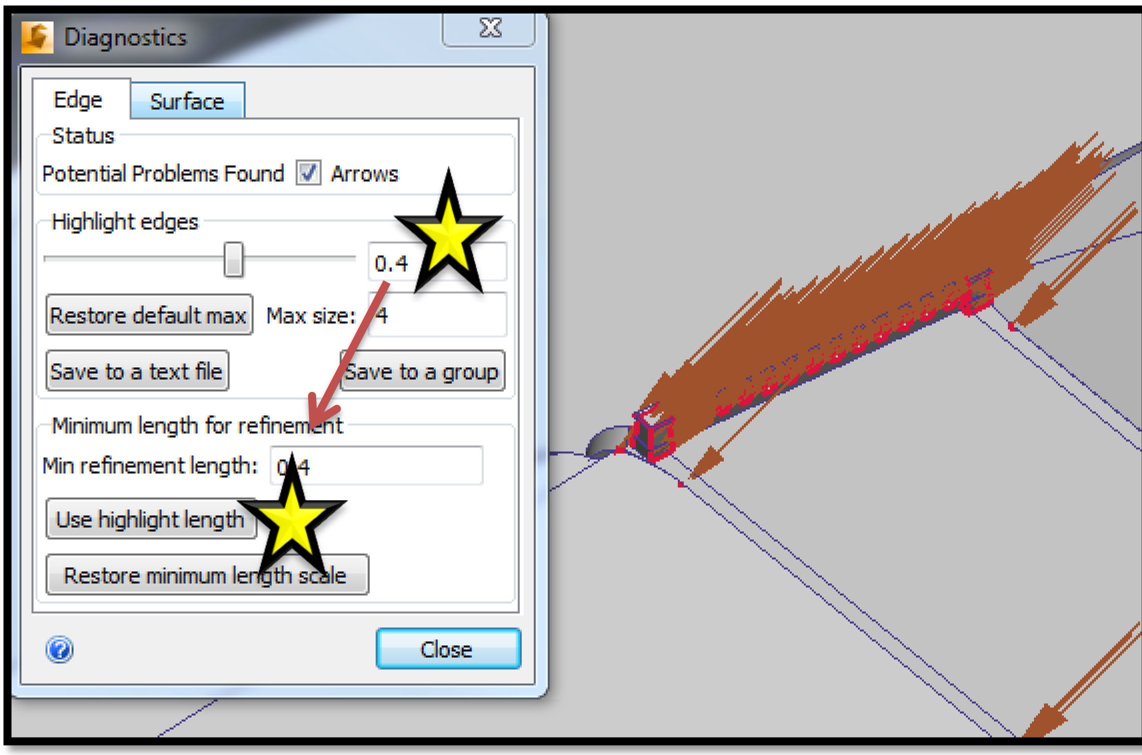
Adjust the minimum length scale to mesh high aspect ratio geometry



Most structures subject to solar thermal loads are considered high aspect ratio geometry – that is, overall very large geometry with some tiny features. The automatic sizing within CFD needs to understand what level of resolution you need for your results. To adjust the minimum length scale, select Diagnostics from the ribbon:



In the Diagnostics dialog, move the slider to the left to the desired value or if edges are highlighted, until edges of significant length are no longer highlighted. Then select Use highlight length:

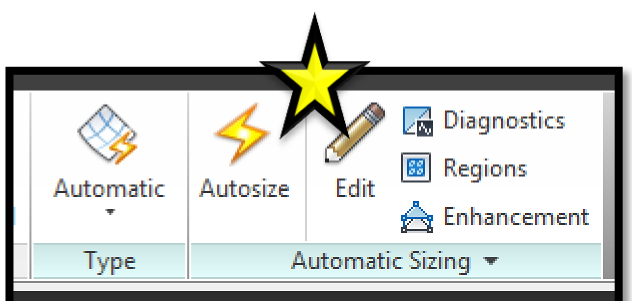


For the demonstration analysis, this step is unnecessary and can result in larger mesh sizes.

Tip: If your bridge deck is relatively thin to the overall length, you will want a minimum length scale to be less than half the deck thickness at the thinnest point.

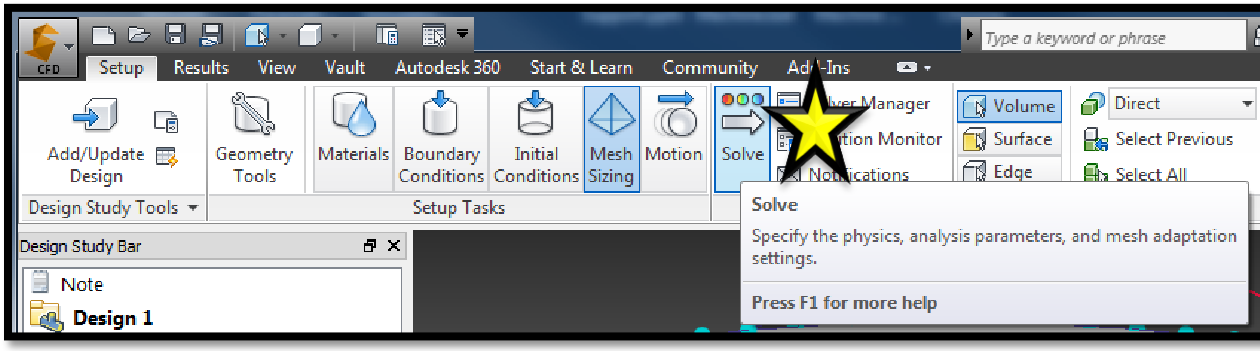
If a red text warning: "Check model surface diagnostics…" had appeared in the Message Window when launching from CAD, you should consider adjusting the minimum refinement length in Diagnostics.

Last, select the Autosize lightning bolt from the ribbon. Light Cyan dots will now cover the edges of the model.



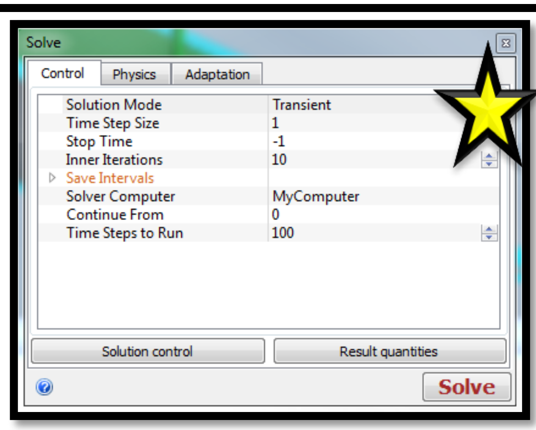
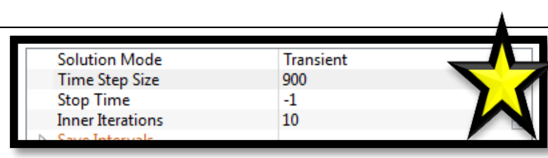
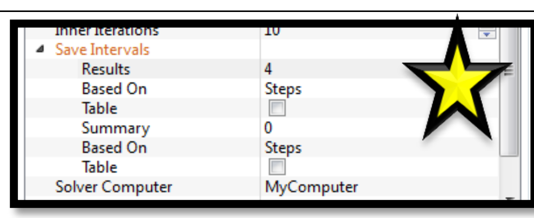
Last setup task – Select Physics and Solve

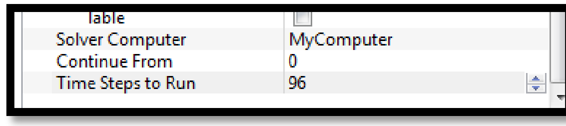
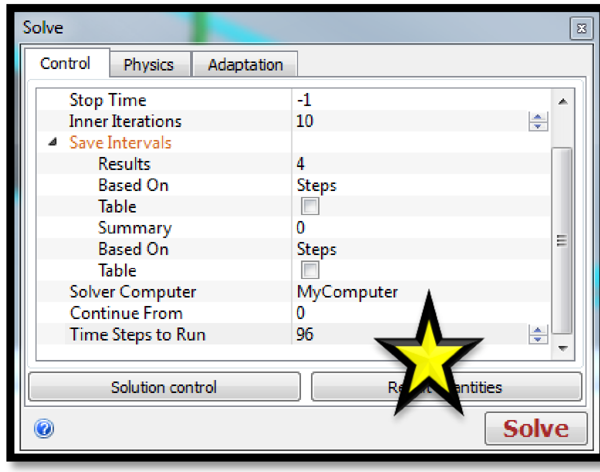
Are we there yet? - Just one more task. You will need to let the simulation tool know what you want to solve and how. Choose Solve from the ribbon to select the Control and Physics for the analysis:



Control Tab

On the solve dialog, ensure the Control tab is selected. The control tab sets how the analysis will run. For this transient analysis, there are more settings to ensure the analysis will solve as a function of time. Note, all simulation time is in seconds. To setup the control section, follow the table of steps:

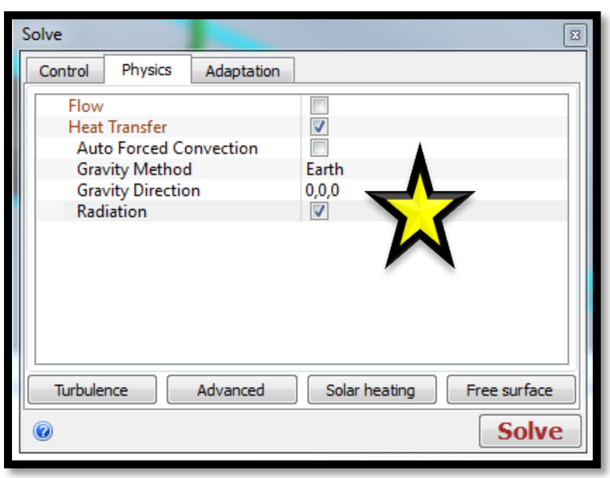
Solution Mode = Transient	
Time Step Size = 900 (this is equivalent to every 15 minutes)	
Expand Save Intervals and set Results = 4 based on steps (this is equivalent to once per hour)	

Solver Computer = MyComputer	
Time Steps to Run = 96	

DO NOT select Solve yet – you have not setup the Solar Radiation.

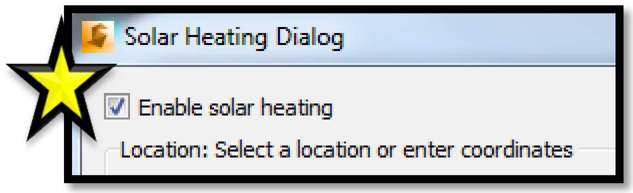
Physics Tab

Switch to the Physics tab. You will want to turn Flow off by unchecking the box and turn Heat Transfer on by checking the Heat transfer box. Once the Heat Transfer checkbox is selected, a Radiation check box will be available and you should check that too. It should look like this:

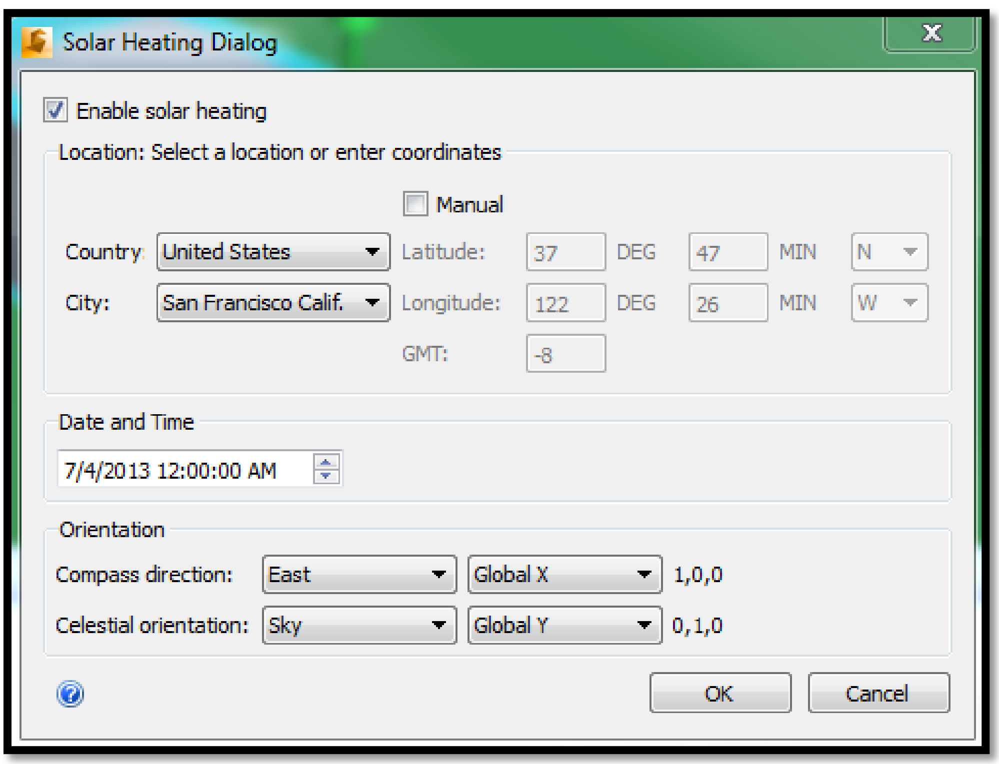


Solar Heating Button

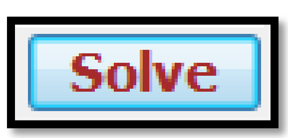
Select the now active Solar heating button and on the Solar Heating Dialog, check the box to Enable solar heating:



In this dialog, you will specific your analysis location (San Francisco, California), the Date and Time (07/04/2013 12:00:00 AM), and the compass directions (East = Global X and Sky = global Y). When complete, it will look like this:



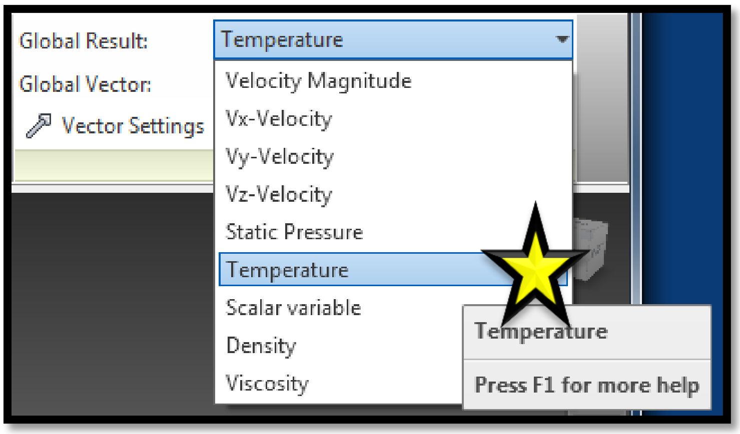
Click ok on the Solar Heating Dialog. Now, you can click on the Solve button:



If you have any problems with your model, you can open “Solar Assembly_support.cfdst” in the “Demonstration\CFD Setup\Solar Assembly” directory and proceed straight to Solve.

Analysis Run-Time

At this point, you should see text scrolling in the message window indicating the progress of the analysis. Once the mesh has been built and the analysis starts to solve, the display will shift to show results of the model and change to the Convergence Plot tab. The default results are for Velocity which will be 0 at all times for this analysis, so all surfaces should appear blue. To see the thermal results as the analysis proceeds - Select Temperature for Global Result in the ribbon:

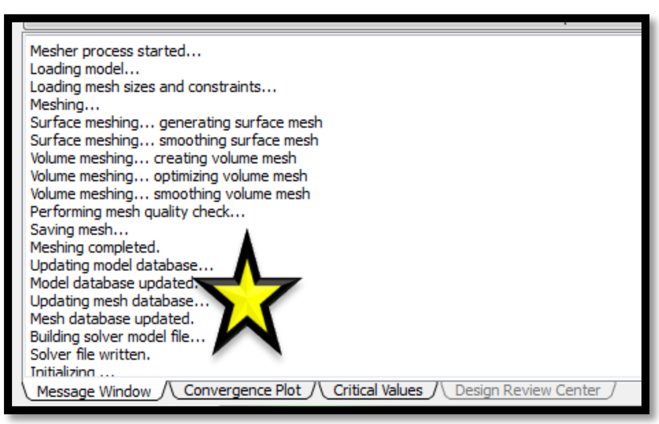


Don't forget to blank the Sky Dome and surrounding Air so you can see the bridge and the ground.

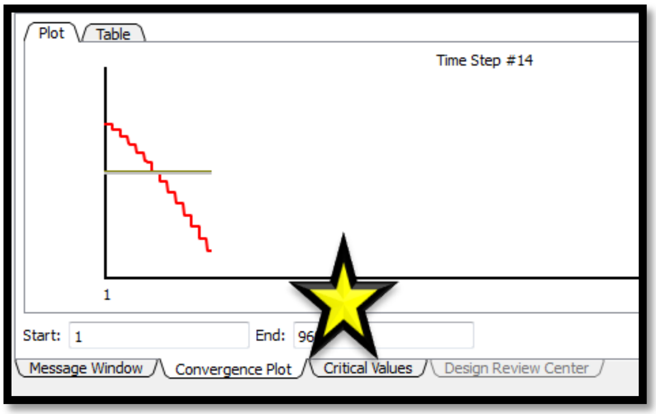
Things to look for while running an analysis

Tip: While the analysis is running, it is a good practice to view the solution as it develops to double check for reasonableness. If temperatures are unrealistic, stop the analysis and double check analysis settings.

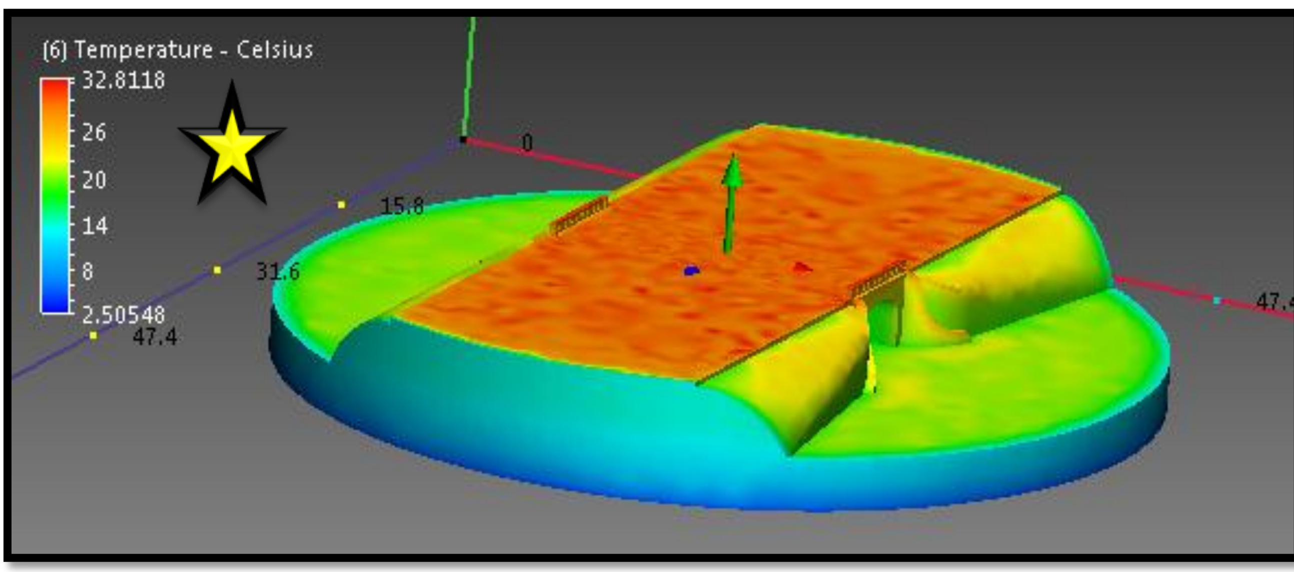
Review the Message Window information. If there are errors in setup, the message window may give an indication of what is wrong:



Next review the convergence plot. If the analysis is unbounded – typically indicating a non-physical boundary condition, the convergence plot will show signs of divergence. For the demo model, the convergence plot should appear something like this:



Last, spot check the temperature extremes in the legend and in the display. If there are temperatures in the legend far exceeding the boundary conditions, the solution is likely to diverge:



The temperature at the edge of the ground will not be representative of the actual temperature since it is non-physical. In the simulation model, the Earth is flat and the Sky meets the ground – hardly reality. If the temperature from the edge of the Earth appears to be overly influencing the structure temperatures, you should make your Sky dome larger.

Do the rest of the temperatures seem reasonable? Then Yippee! Your analysis is running.

Section 3: Process thermal results in Simulation CFD

In this section, validation results obtained using this technique will compare the simulation results with measurement data from an actual bridge instrumented with thermistors. Instructions will follow for creating the datasets to show you how to process the information to aid your designs. By the end of this section, you will be able to visualize pertinent thermal results of your initial simulation analysis.

Validation Analysis Results

In 2007 the I-35W Mississippi River Bridge collapsed during evening rush hour on the first of August. The bridge has since been replaced with an instrumented concrete bridge known as the St. Anthony Falls Bridge. Thanks to the support of the University of Minnesota, Professor Carol Shield and Graduate Research Assistant Brock Hedegaard, and with permission from the Minnesota Department of Transportation, hourly recorded temperature measurements internal to the bridge were made available to compare the simulation results to actual measurements of a bridge in use today.

For background, research conducted by the Center for Highway Research has concluded daily changes of the environment result in a temperature gradient over the bridge cross section that cause temperature induced stresses.[1] In a Journal of Bridge Engineering article, research indicates temperature induced stresses in concrete bridges can be larger than those induced by vehicle loading.[2] The Center for Highway Research further concludes the thermal response of structures can be highly variable since it is dependent on location, climate, material properties, deck coverings, and cross sectional shape.[1] Typically the largest positive thermal gradients, causing the largest induced stresses, occur when the intensity of solar radiation is greatest, with little wind or cloud cover, and when the ambient temperature range is large. As a result of the study performed on the St. Anthony Falls Bridge, the design thermal gradients using the AASHTO LRFD Bridge Design Specifications are noted to be not necessarily conservative.[2]

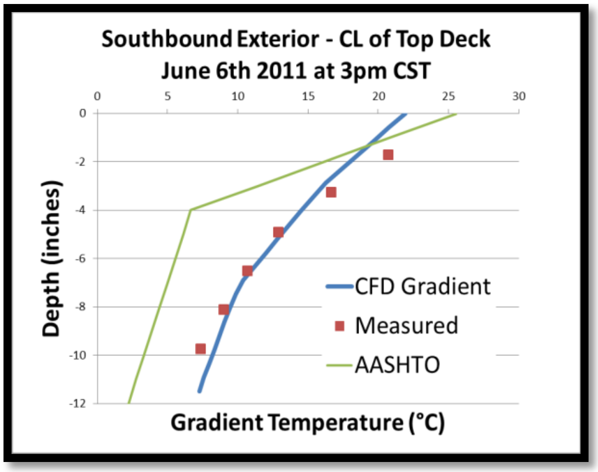
Example Data

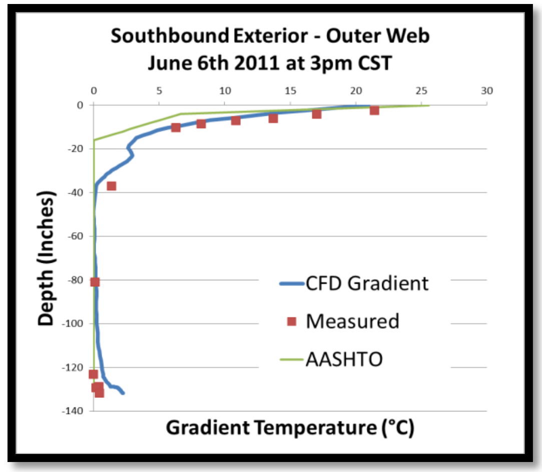
As one example from the validation study: June 6th, 2011 in Minneapolis was clear with little wind. From the measured data, at 3pm CST, maximum positive thermal gradients were observed in the bridge structure with > 21°C temperature differences measured in the southbound lanes. In the following two graphs – one from the centerline of the top deck and the other from the exterior web - the measured data points, the AASHTO LRFD scaled to the top surface temperature, and the results from Simulation CFD can be compared:

---

[1] Thepchatri, Thaksin; Johnson, C. Philip; and Matlock, Hudson; (1977); "Prediction of Temperature and Stresses in Highway Bridges by a Numerical Procedure using Daily Weather Reports;" Report No. FHWA/TX-77-23-1; Center for Highway Research, University of Texas at Austin.

[2] Hedegaard, B.D.; French, C.E.W.; Shield, C.K.; (2012); "Investigation of Thermal Gradient Effects in the I-35W St. Anthony Falls Bridge;" *Journal of Bridge Engineering*; in press.





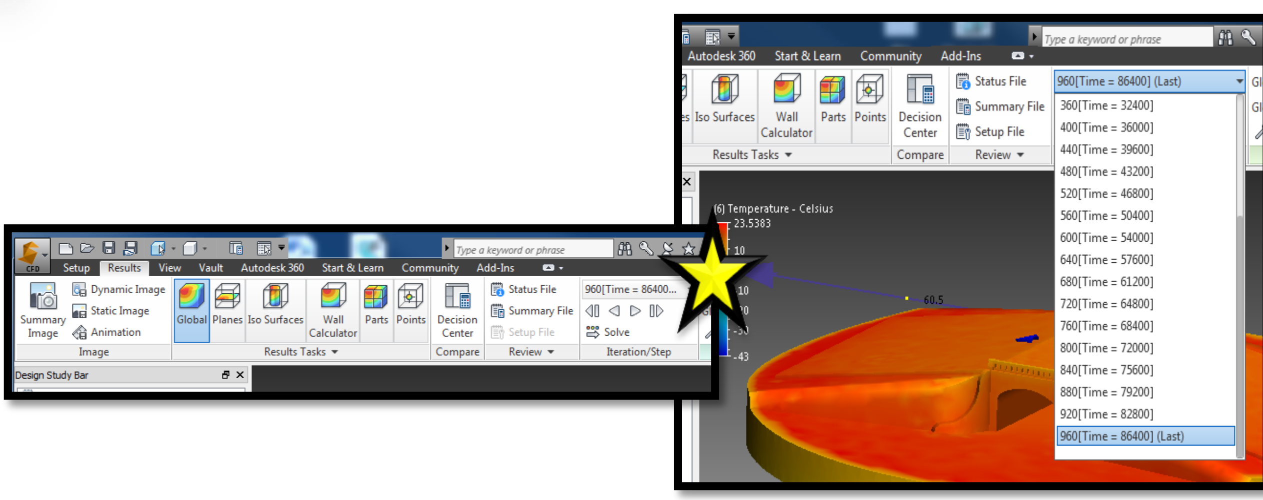
In direct comparison of thermistor recorded hourly temperatures to analysis temperatures, the mean percent error was less than 5% over a 24 hour time increment. In conclusion, following the recipe of this handout, a CFD numerical solar simulation will be accurate to measured data.

If your analysis has not completed yet, you can open a completed version by using file Open and selecting “Solar Assmebly.cfdst” from the “Demonstration\CFD with Results\Solar Assembly” directory.

Post analysis visualization

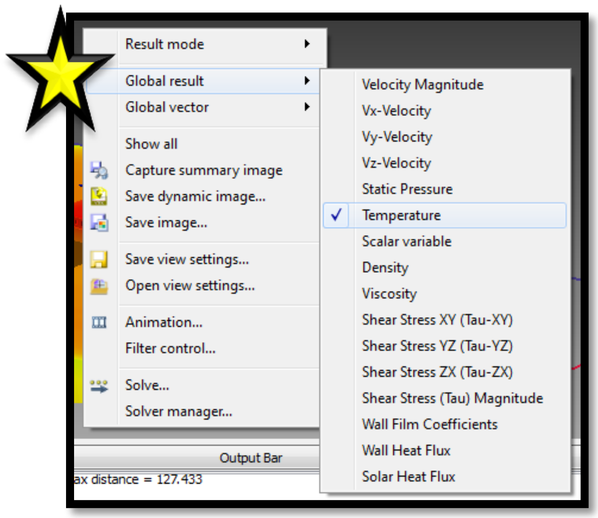
This is a transient analysis – it is run as a function of time – so there is a wealth of data available to help you understand all the thermal profile of your structure at any moment or place. Be sure the Message Window completes with the lines: Analysis Completed Successfully and all the results have loaded into the display window.

Tip: You will know you are working with a completed analysis when the Iteration/Step dialog has multiple results available in the pull down list:

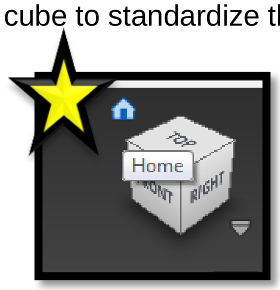


Global Results

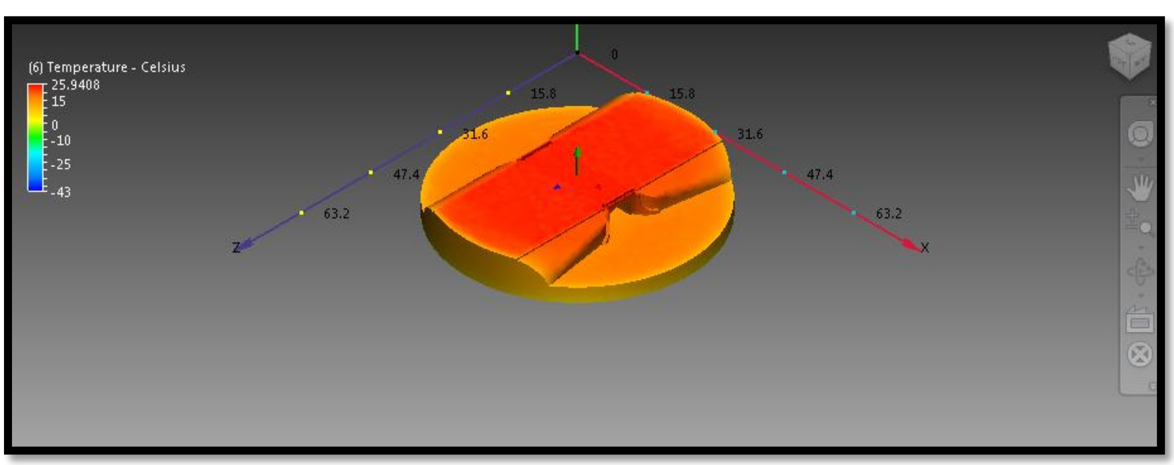
If you haven't already done so, change the Global Result to Temperature. You can use the Ribbon or right select in the background of the display window and chose Global result from the context menu:



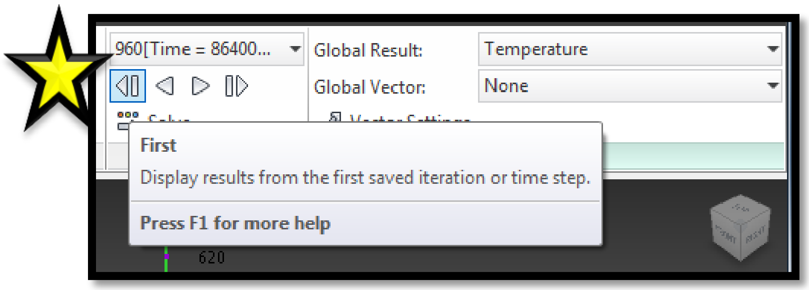
Select the home view from the view cube to standardize the display:



Next, while holding the control key, select objects in the display to blank until you can see the bridge and the ground (you should only need to blank two parts to see this):

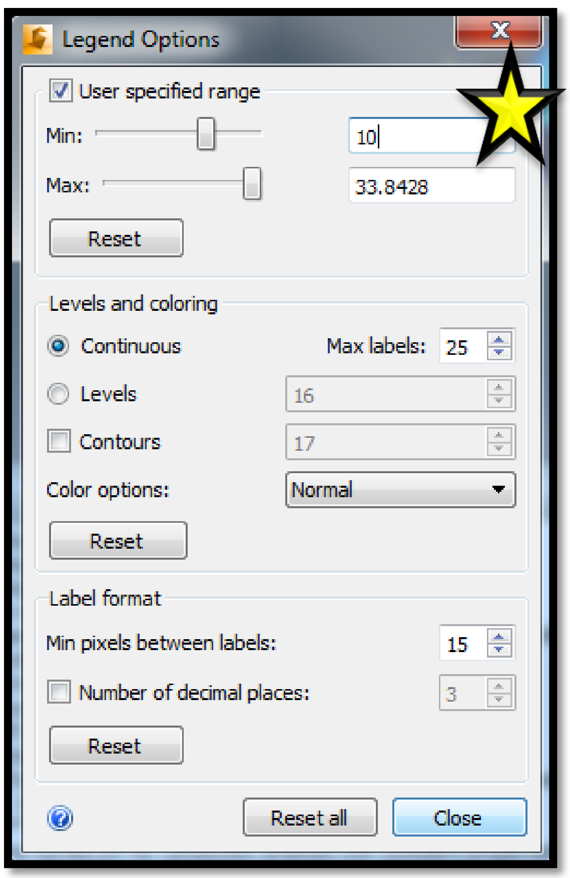


Use the Iteration/Step control in the Ribbon to reset time to the first saved result set:



Tip: Use the next button to increment through all the results sets. This will normalize the Temperature legend for the entire analysis.

We also want to adjust the temperature range of the legend to eliminate the night sky boundary condition. Right select on the legend to Options… and select a minimum temperature of 10°C. The maximum will be approximately 34°C:



If you are more comfortable thinking of the results in other units, then select units from the legend context menu to change them.

Now return to the first step in order to proceed through the steps more slowly. Watch the temperature on the surface of the bridge during the 24 hour day and think about the sun rising in the east (the +X axis). I find it helpful to have a table of seconds to hour conversion handy:

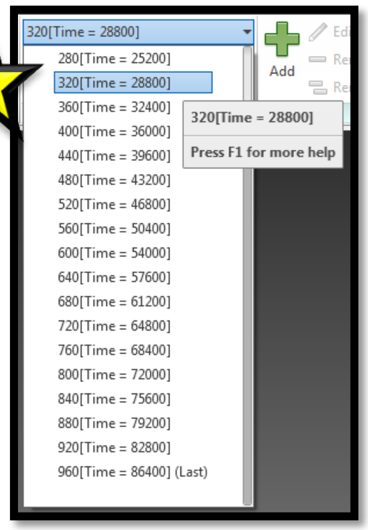
Time Step	Iteration Number	Seconds	Time of Day	Comment
0	0	0	12:00:00 AM	Midnight
4	40	3600	1:00:00 AM	
8	80	7200	2:00:00 AM	
12	120	10800	3:00:00 AM	
16	160	14400	4:00:00 AM	
20	200	18000	5:00:00 AM	
24	240	21600	6:00:00 AM	Sunrise
28	280	25200	7:00:00 AM	
32	320	28800	8:00:00 AM	
36	360	32400	9:00:00 AM	
40	400	36000	10:00:00 AM	
44	440	39600	11:00:00 AM	
48	480	43200	12:00:00 PM	Peak Solar Intensity
52	520	46800	1:00:00 PM	
56	560	50400	2:00:00 PM	
60	600	54000	3:00:00 PM	
64	640	57600	4:00:00 PM	Peak Air Temperature
68	680	61200	5:00:00 PM	
72	720	64800	6:00:00 PM	
76	760	68400	7:00:00 PM	
80	800	72000	8:00:00 PM	Sunset
84	840	75600	9:00:00 PM	
88	880	79200	10:00:00 PM	
92	920	82800	11:00:00 PM	
96	960	86400	12:00:00 AM	

Did you notice the east side (+X direction) of the railings started to increase in temperature just past sunrise at 7am? Note: the peak temperature of the bridge deck is closer to the time of the peak air temperature rather than the peak solar intensity.

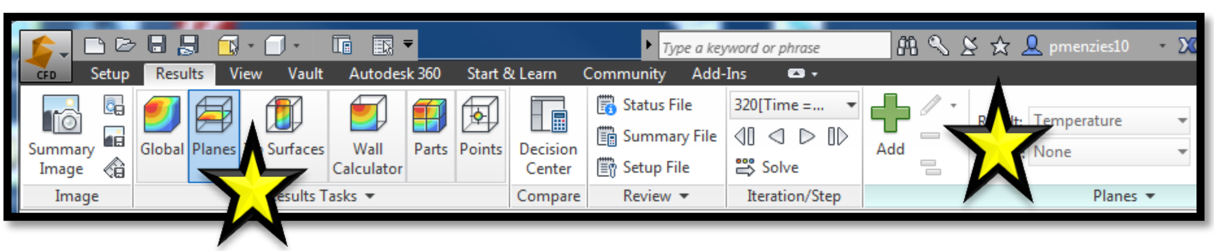
Imagine walking through the pedestrian tunnel of the bridge and finding it a cool shelter on a hot sunny day.

Plane Results

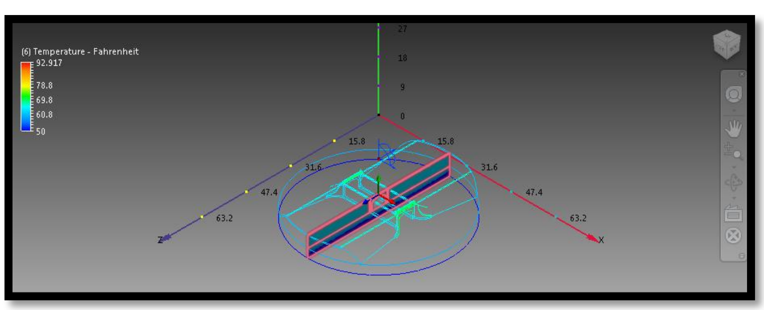
With Plane Results, you can virtually cut into the bridge to explore the temperatures inside the structure. The help system can show you all the possibilities for results processing. In this class, we will explore techniques used to obtain the published results describing positive thermal gradients. So first, pick a time from the Iteration/Step pull down – in this case select: 320 [Time = 28800]:



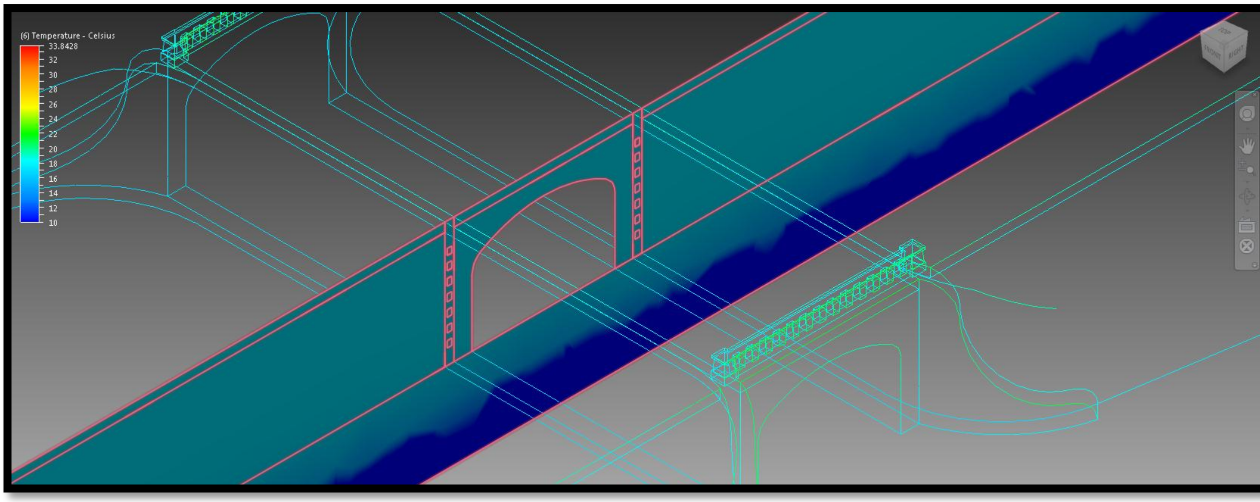
Then proceed to the Planes Results Tasks in the Ribbon and select  :



You view will shift to an outline mode and from Home view should look like this:

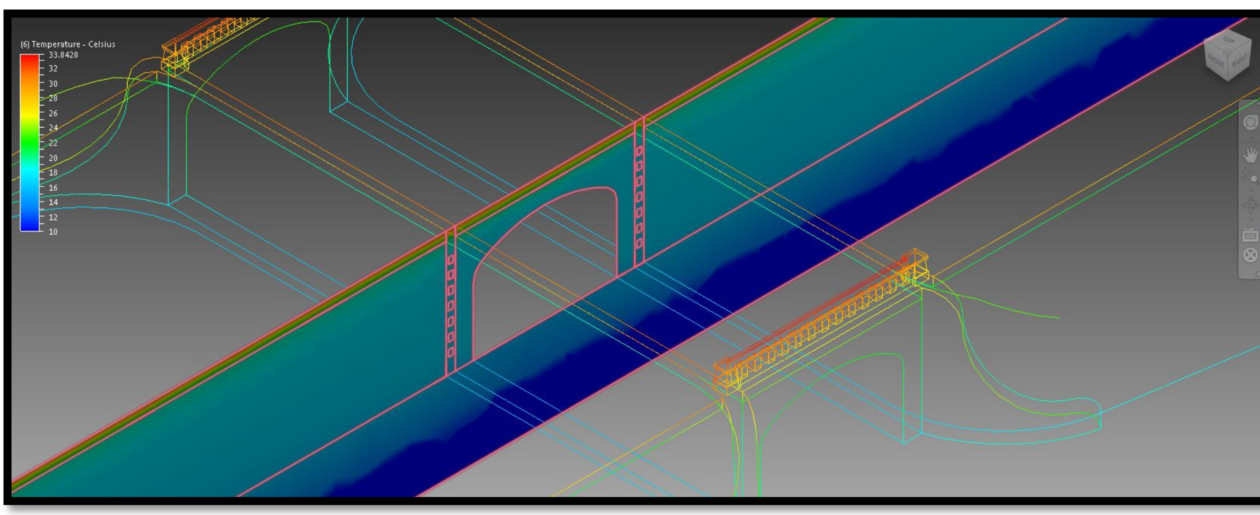


Use the middle mouse scroll wheel to zoom the model in the display to focus on the cross section of the bridge. Notice there is very little difference in the temperature profile of the cross section at this time:



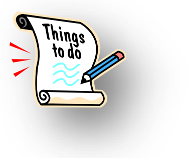
According the Center for Highway Research, the temperature profile through concrete bridges is fairly uniform two hours after sunrise on clear and windless days and nights. Use this as a checkpoint for your own simulations to ensure your setup is not adversely affecting your results.

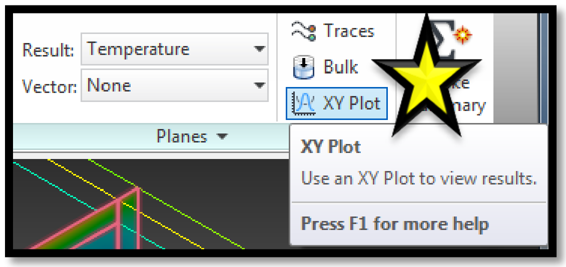
Now select Iteration/Step 640[Time = 57600] after the Sun has passed its zenith and the ambient air temperature is maximized:



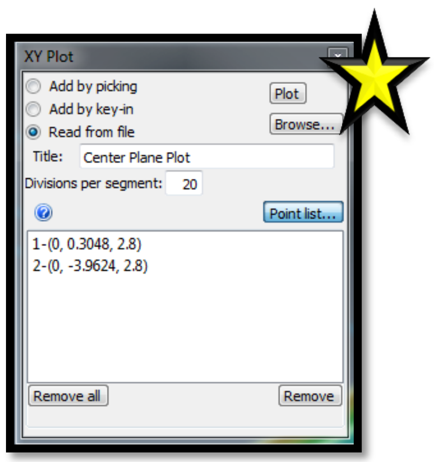
The bridge deck has reached its maximum temperature but due to the low conductivity of the concrete, the rest of the cross section is still relatively cool creating a thermal gradient.

To evaluate actual numbers, you can capture the plane data with an XY Plot. Proceed to the XY Plot button in the Planes task:

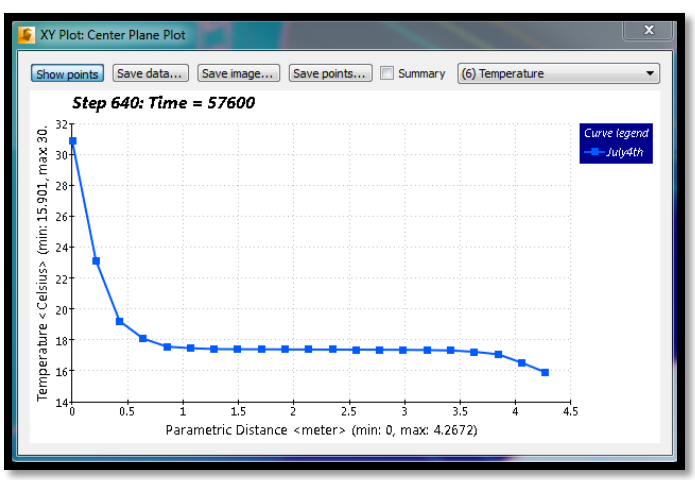




In the XY Plot dialog, you can select to Add points and just left click on the plane in several locations or you can key in x, y, z data points. For consistency during the class, select to Read from file and browse to the "Demonstration\Datafiles" directory to find the file "Center Plane Plot.xyp," then select Plot:



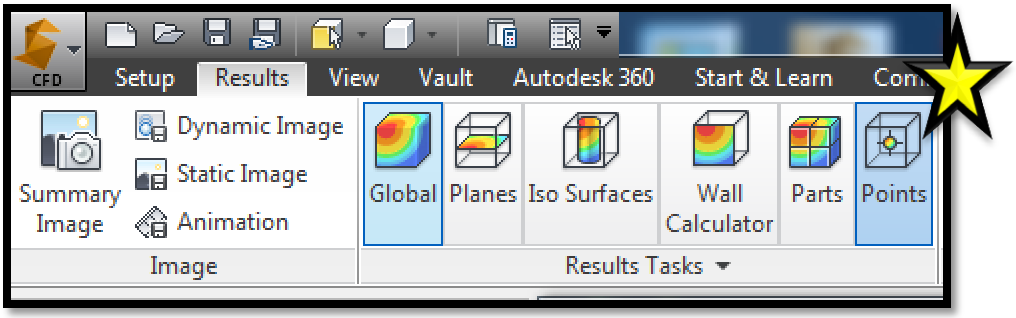
You should then see an XY Plot in a separate dialog:



From the XY Plot dialog, you can manipulate the units, save the data, or save the image of the plot. If you would like to see the temperature data along the same points at a different time, close the XY Plot dialog, use the pull down to select a different Iteration/Step, then repeat the process to read from file to add you data points.

Points Results Task

The XY Plot from Planes provides data for a selected moment in time. To see how the temperature at a specific location of the model varies over time, you can use Points from the Results Tasks:

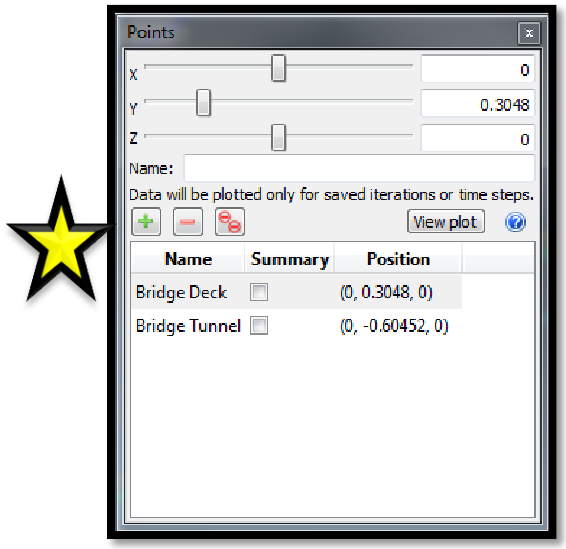


Recall when creating the analysis assembly in Inventor, you chose with care where the model origin was located. As with XY Plot, the values for coordinates are based on the assembly's origin. For this bridge, you will investigate a point on the bridge deck at the center, and a point directly underneath.

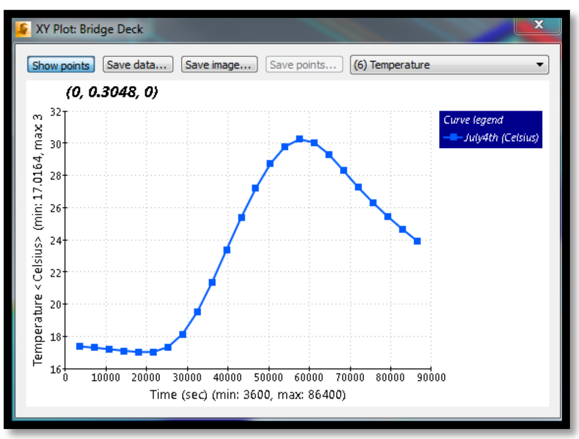
Manipulate the sliders or key in the table values in the Points dialog then select add before going to the next column:

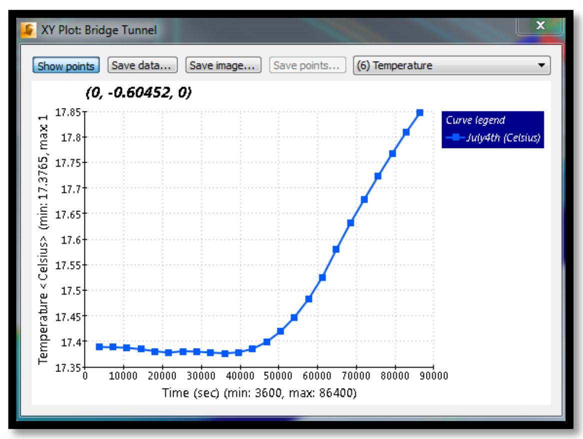
Description	Top Center of Bridge Deck	Top Center of Bridge Tunnel
X	0	0
Y	0.3048	-0.60452
Z	0	0
Name:	Bridge Deck	Bridge Tunnel

For example, the Points Dialog should look like this before selecting the  button:



Add both points in sequence and note their location in the display. Now select one of the points then View Plot to see the temperature of that point as a function of time:





Observe the temperature of the ceiling of the tunnel rises through the day as the heat from the bridge deck slowly dissipates through conduction to the rest of the bridge structure.

There are endless ways to review the data captured in a simulation model. These are only the most basic steps to product the data needed to compare with measured data. Take time to manipulate the planes with the glyphs, add other points of interest, and explore the display options.

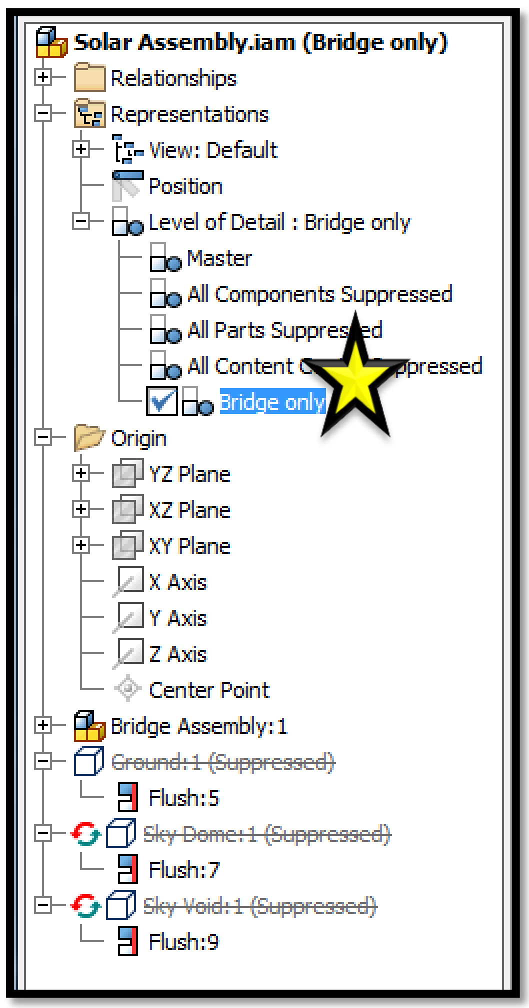
When you are finished, be sure to close the Simulation CFD analysis to enable access in Mechanical.

Section 4: Transfer thermal results to Simulation Mechanical to evaluate stresses

In this last section, you will learn how to import the thermal profile results into Autodesk® Simulation Mechanical to explore and understand the thermal induced stresses in the bridge.

Back to Inventor

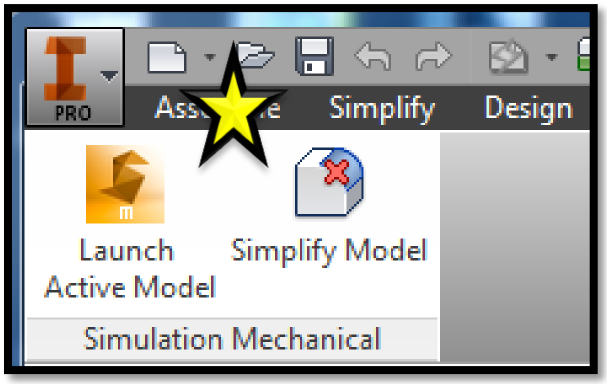
You will return to Inventor Professional with the model still open. For the Mechanical Simulation, you will only study the physical parts of the bridge. A Level of Detail has already been saved called “Bridge only.” Double click on the name to activate the Level of Detail needed.



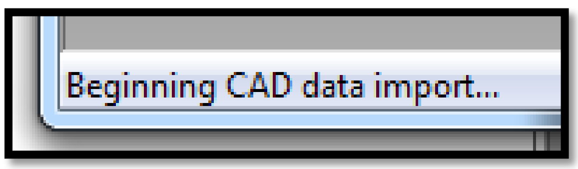
You can see in the Model tree, the environment and simulation parts have been suppressed. They are no longer needed.

Launch Active Model

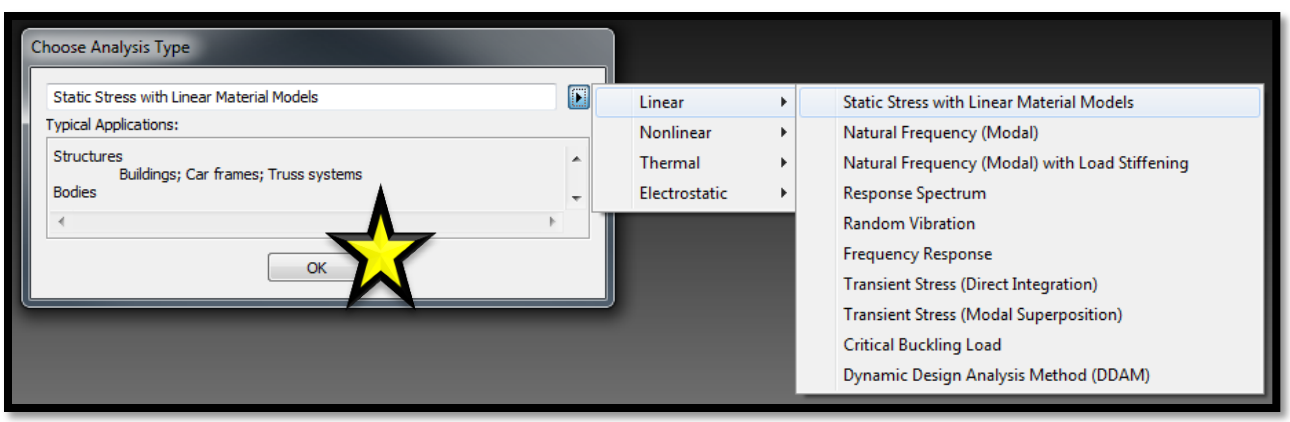
You are now ready to launch your structural model into Simulation Mechanical for determining the thermal induced stressed in your structure. Locate and select eh Autodesk Simulation tab in the Ribbon, then select Launch Active Model:



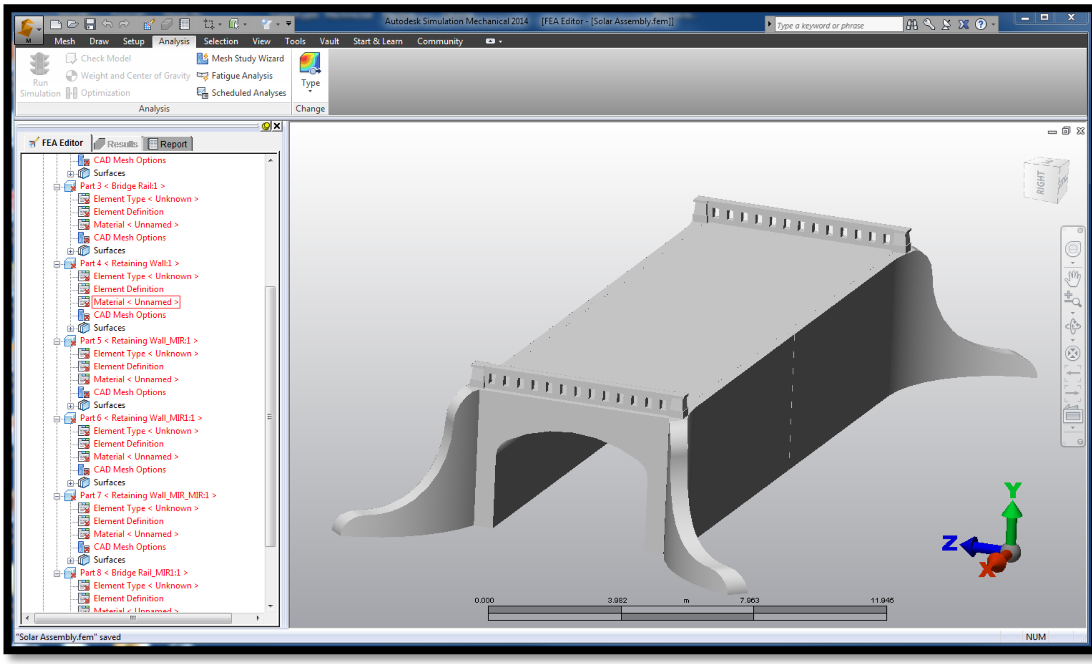
Be patient, this can take a bit of time:



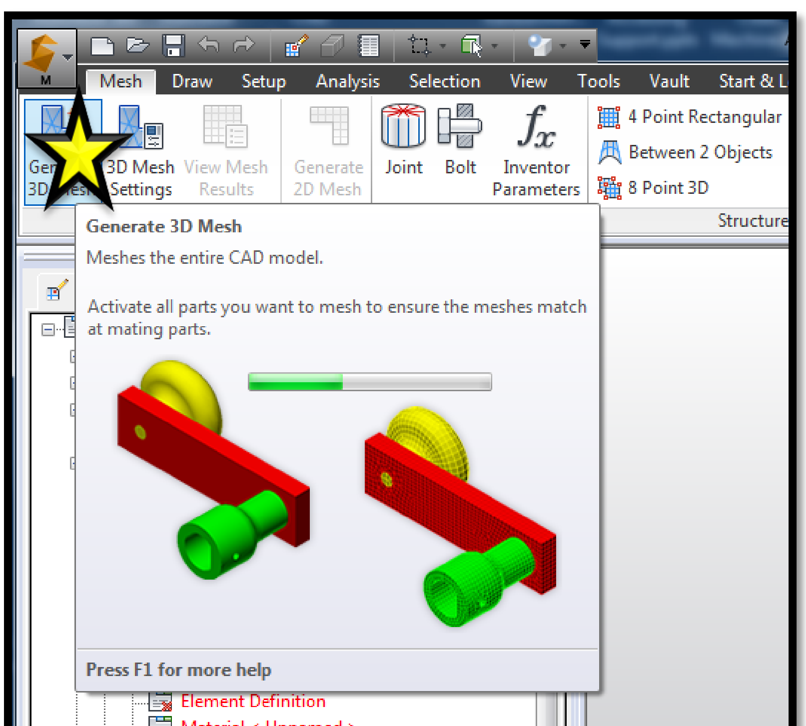
Note, we will not import Work points for the class. Once Autodesk Simulation Mechanical 2014 interface opens, you will be prompted to Choose Analysis Type. Be sure to select Static Stress with Linear Material Models as shown then click OK:



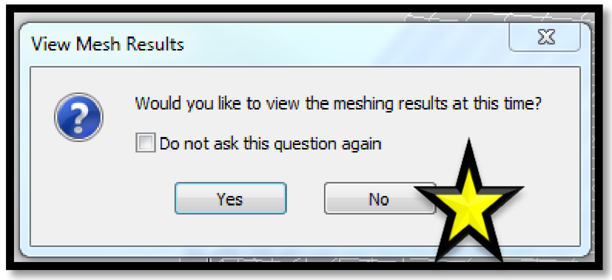
Your model will now appear in the Mechanical display window with the Mesh tab active. It should look like the following:



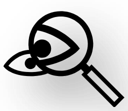
To enable editing, proceed to Generate 3D Mesh, the default settings are fine:



You can select No for viewing the meshing results at this time:



Notice the previously undefined items (red text) in the FEA Editor are now defined.

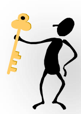


Apply settings for a Static Stress Analysis

For the purpose of the hands-on lab, we will be analyzing the static stresses in Alvord Lake Bridge at the time of peak thermal gradients. The maximum gradient occurred at CFD Time = 57600 or 4pm in the afternoon with nearly a 15 °C gradient. As with the previous simulation, think of the setup steps as telling the simulation tool what you know.

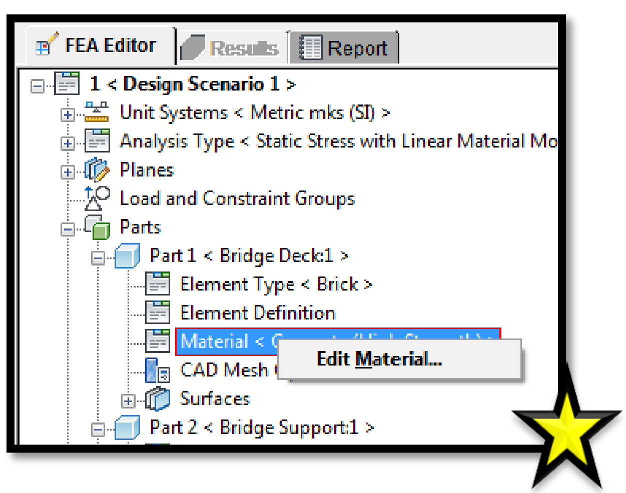
Modify Material Selections

A generic concrete material was applied to the bridge parts in Inventor.

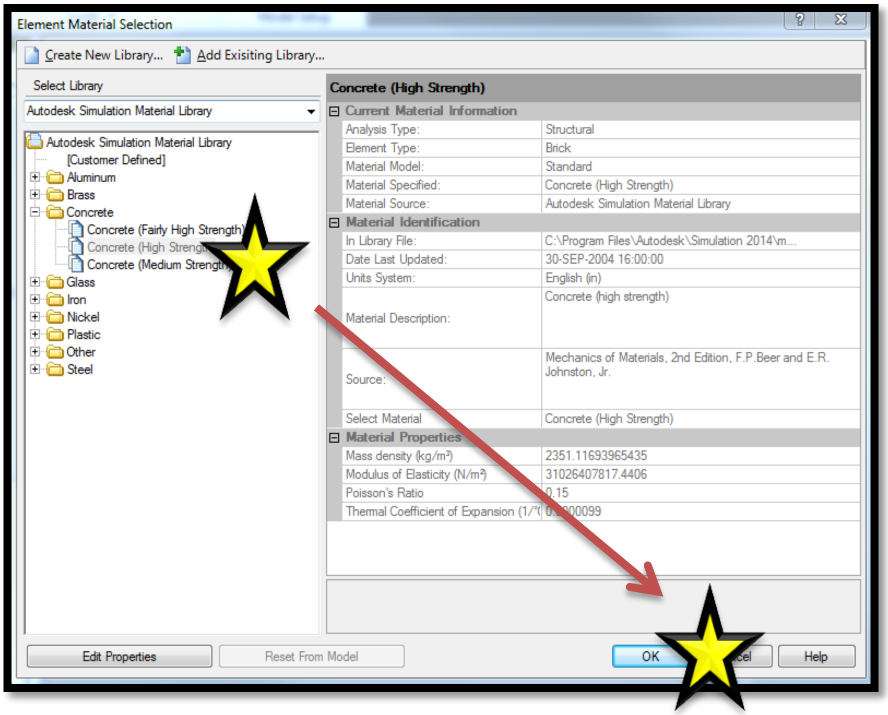


Tip: Once the model is meshed in Mechanical, the material attributes applied in Inventor are applied to the elements for each part.

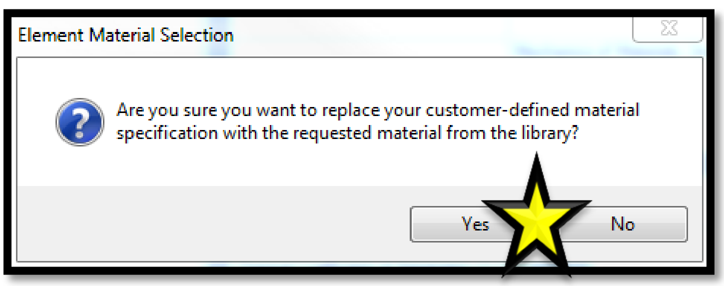
To adjust the materials to better match structural concrete, you will want to edit those material properties. For each part in the FEA Editor, right select on Material < concrete > and select to Edit Material… :



Edit Material… will open an Element Material Selection dialog. From this dialog, you can Edit Properties as necessary to describe the material structurally. For the purpose of the Demo, you will select from the Concrete choices, Concrete (High Strength):



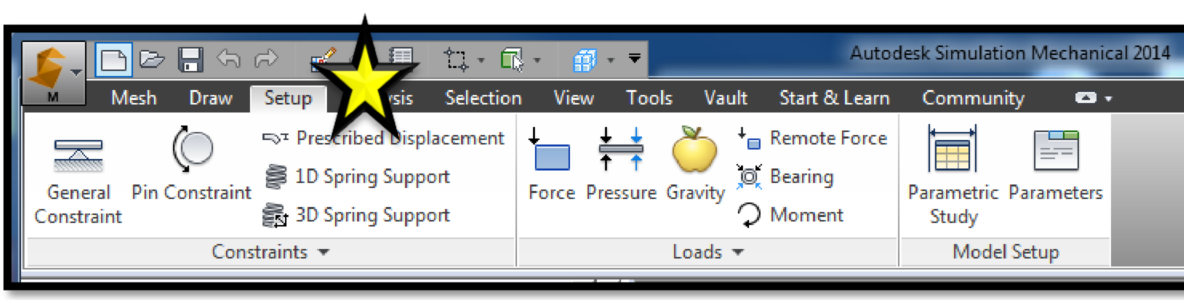
When you select OK, you will be asked if you want to replace the material – click Yes:



You will need to perform this step for each of the 8 parts. Double check the FEA Editor list to be sure the proper materials are selected.

Apply Constraints

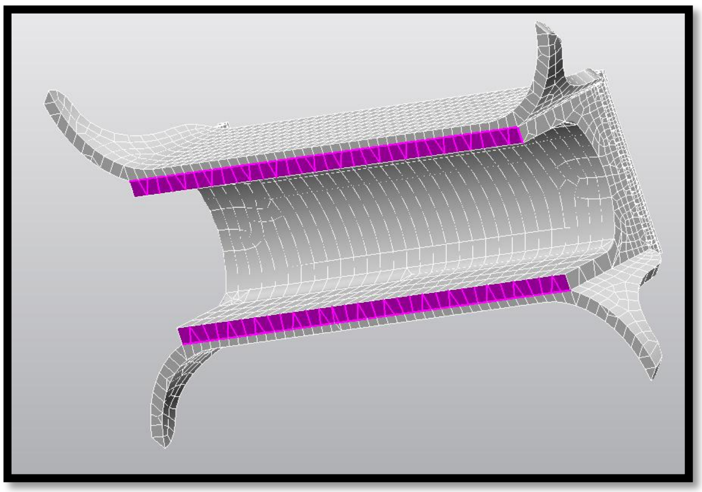
Next, you will shift to the Setup tab in the UI to attach the bridge to the ground:



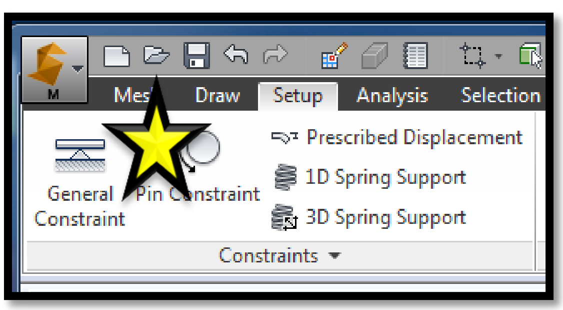
For a more traditional bridge, you might need to apply a variety of Constraints to the piers, expansion joints and to mimic the slide bearings. You would also apply specific loads to mimic the vehicle loading and gravity to account for the weight of the bridge. Surface contact between the parts would also need to be specified.

For the purpose of the hands-on lab, you will fix the bridge to the ground where the concrete touches the ground part with a General Constraint fixed in all 6 degrees of freedom. The ground and the road bed fill the sides of the bridge to the height of the bridge deck, inducing their own loads which you will model as a hydrostatic load.

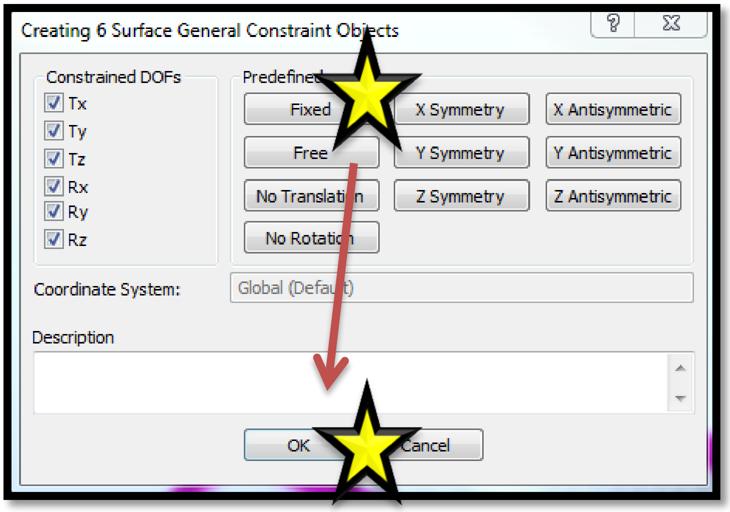
Use the Orbit tool or hold the Alt key and use the middle mouse button to orbit the model in order to observe the bottom surfaces of the bridge. Next, hold the control key and use the left mouse button to select the surfaces. You are interested in the two main surfaces of part 2. The surfaces will highlight fuchsia when selected:



With the surfaces selected, click on General Constraint in the ribbon:



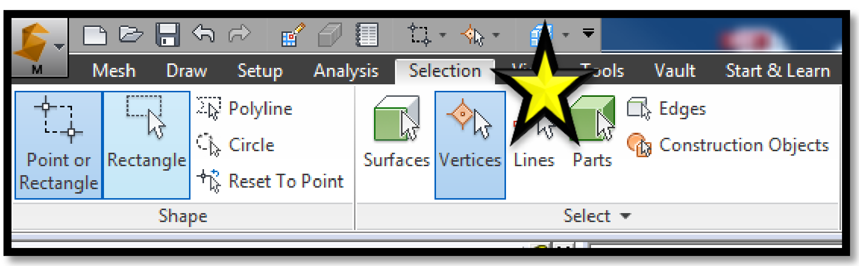
Use the Fixed button in the Predefined section to quickly constrain all 6 degrees of freedom:



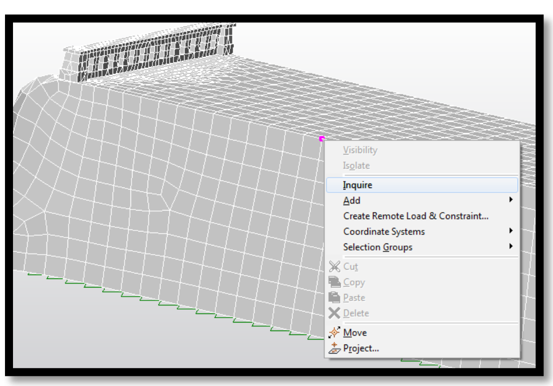
Green triangles at each mesh vertex will indicate the fixed constraint has been applied.

Apply Loads

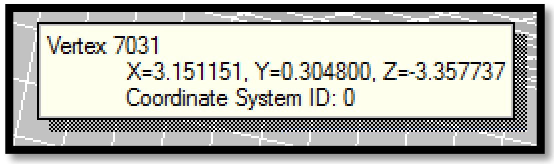
To represent the ground behind the retaining walls, a hydrostatic load will be added to the outer surfaces of the retaining walls to mimic the increase in load due to the depth of the ground. First determine the height of the retaining wall. Change to the Selection tab in the Ribbon and select Vertices:



As you mouse over the walls of the bridge, mesh vertices will highlight yellow. When you have one at the top of the wall, right select (the highlight will turn fuchsia), and select Inquire:



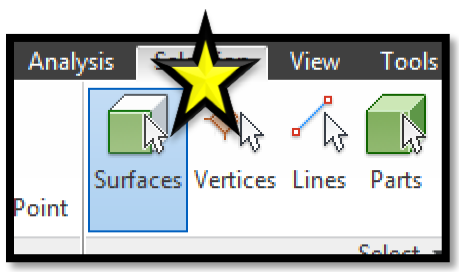
Take note of the Y coordinate:



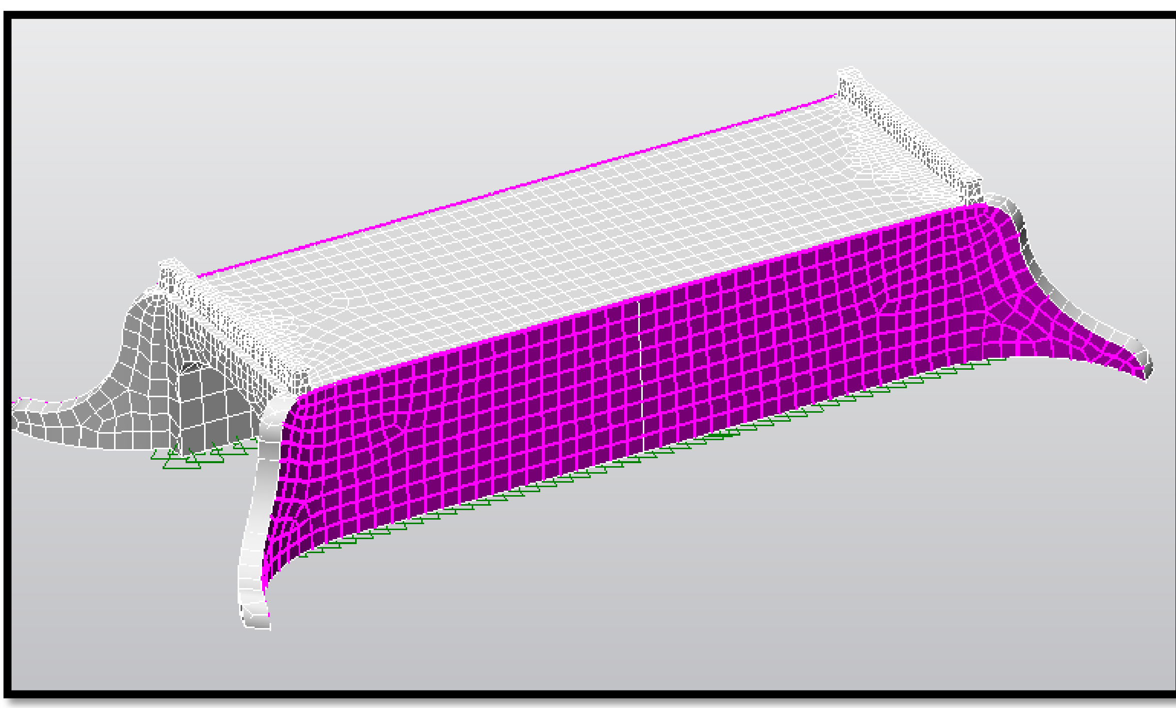
You will need this value to specify the top of the load distribution.

Tip: In vertices selection mode, a right click_Inquire will yield coordinates for the vertex selected.

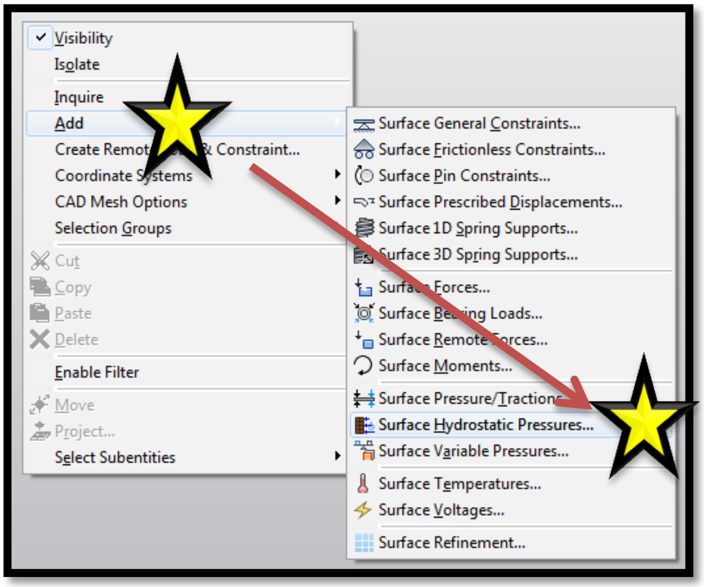
Return back to Surfaces for the selection mode by clicking on the Surfaces icon in the Selection tab:



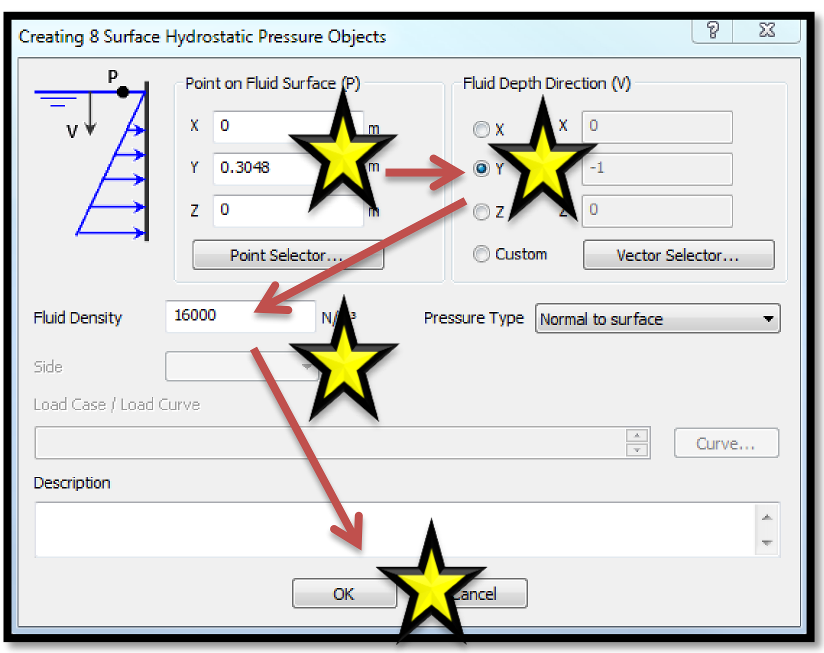
Now, hold the control key while selecting each of the eight external surfaces of the retaining wall. Your model should look like the following when selected correctly:



A right select in the display window (but off of the model) will show the context menu with the option to Add, and then select Surface Hydrostatic Pressures:

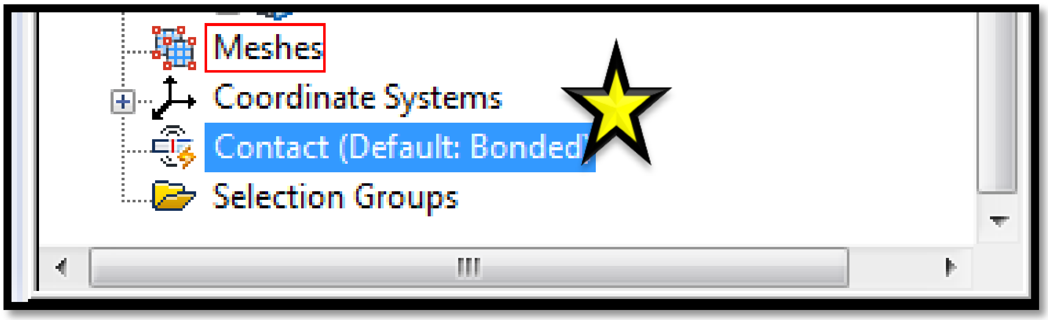


You will manually type in the Y coordinate for the Point on Fluid Surface (P), identify the direction of gravity with the Y radio button for Fluid Depth Direction (V), then apply a Fluid Density that approximates the ground (typical ground is ~16 kN / m^3). Enter the values to look like the following then click OK:

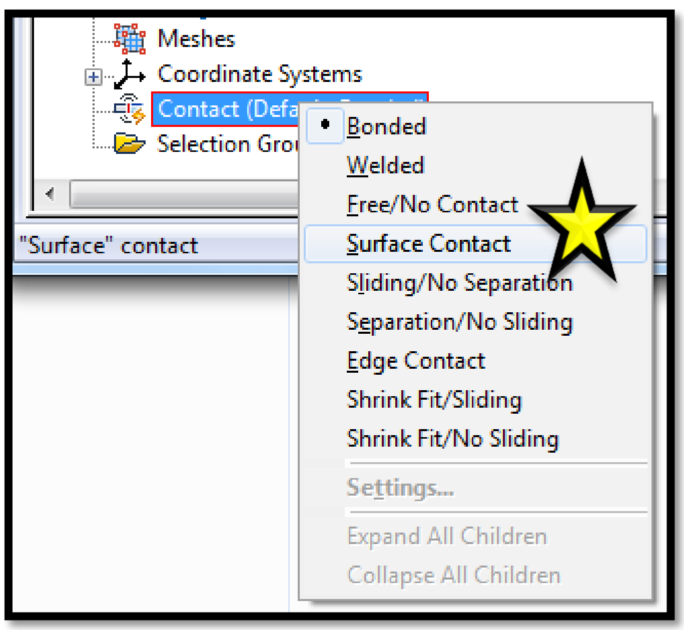


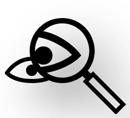
Apply Surface Contacts

Next, you can use the FEA Editor to select the contact between parts. Either scroll to the bottom or collapse the FEA Editor tree until you see the Contact icon and setting:



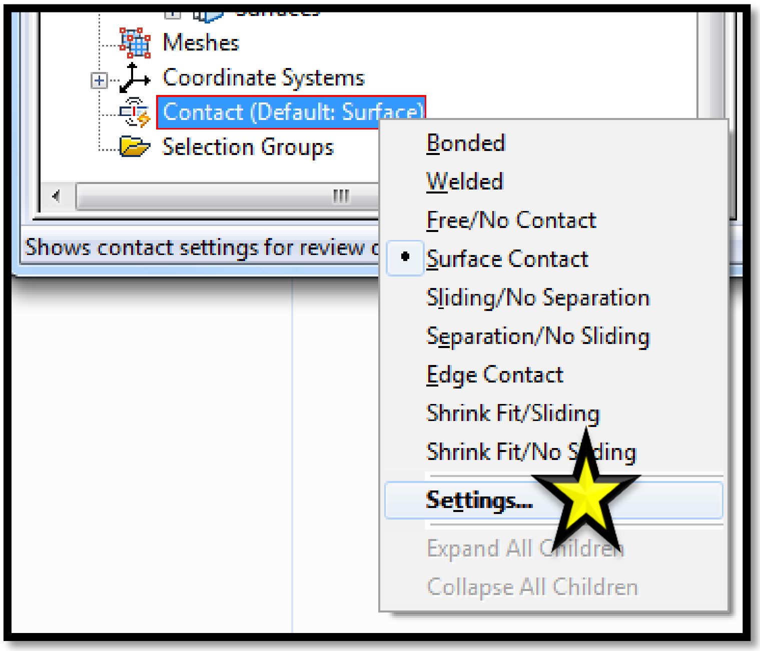
By default, the contact will be bonded, which we will use for the class. If you need to change the contact between surfaces to better match your structure, right select on Contact in the tree and use the context menu to select a different form:



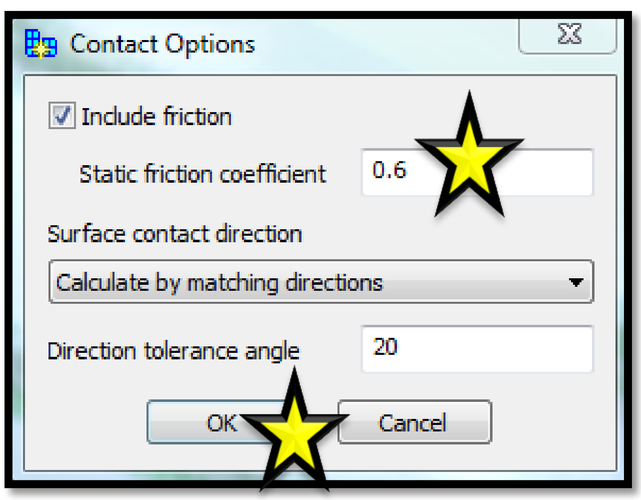


For example, if the majority of the parts touch such that they are free to move relative to each other, you may want to select for Surface Contact. Once Surface Contact is selected, you will then have the ability to select the "Settings…" option to include static friction.

The following images show the mouse selections necessary to include friction for the Surface Contact between parts:

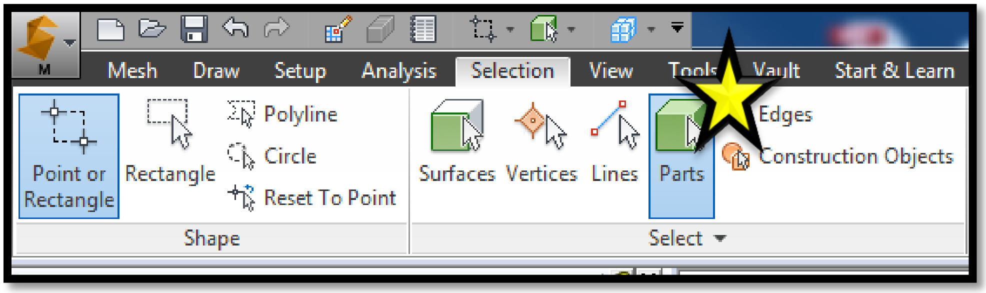


On the Contact Options dialog, check to Include friction and enter a Static Friction coefficient:

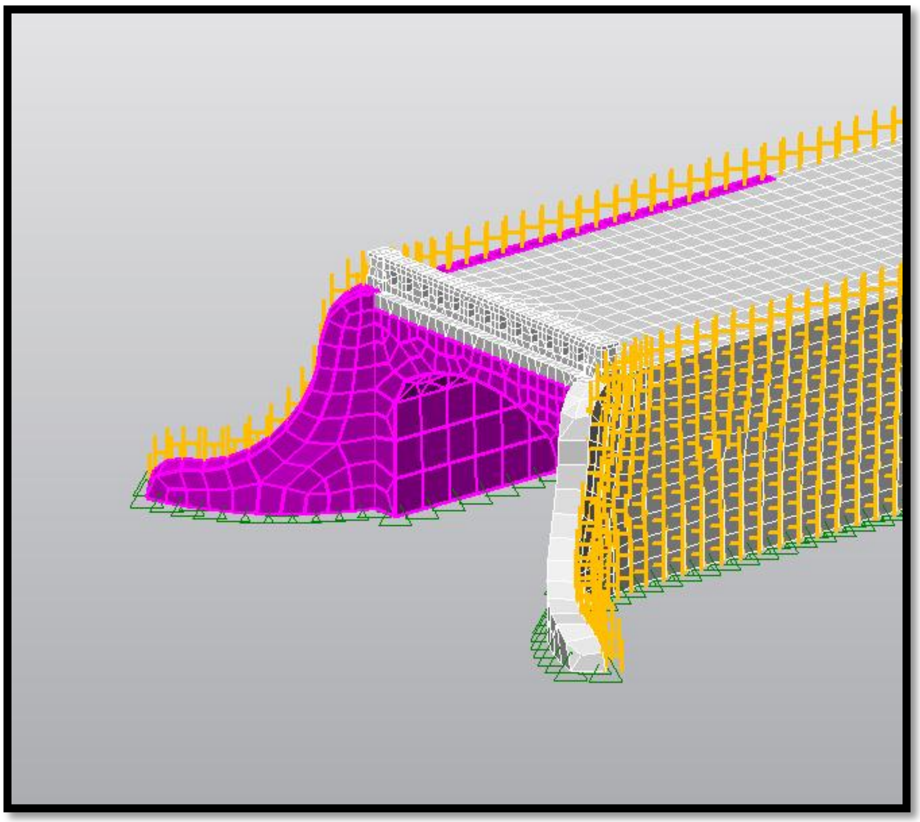


The preceding steps will apply friction to the interface of all touching parts. Should two parts touch with a different type of contact, you can manually select the parts to apply individual contact options to the interface surfaces.

In the Selection tab, change the selection mode to Parts:

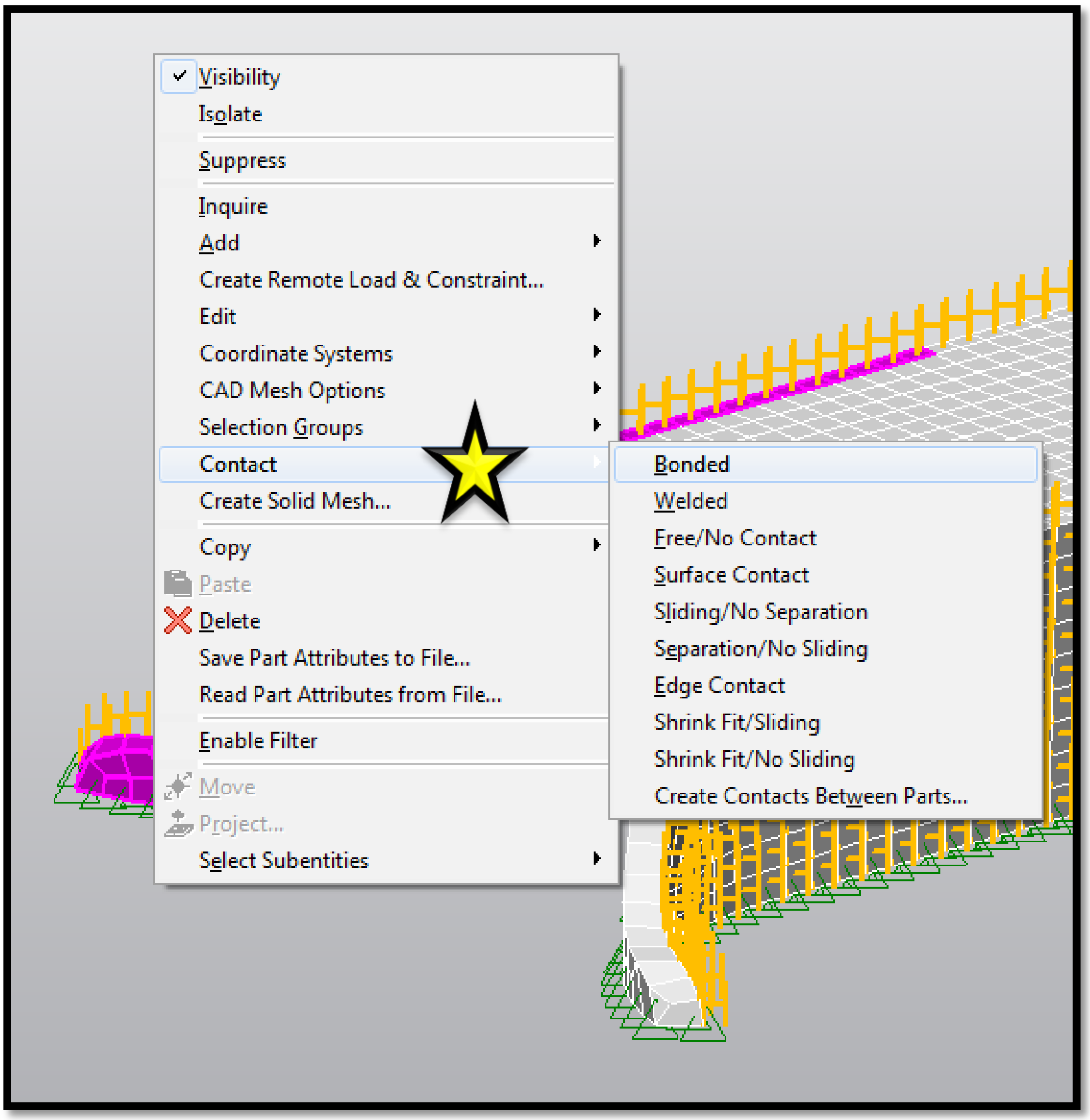


While holding the control key, select on the first part, then select the touching part with a left mouse click on the parts:

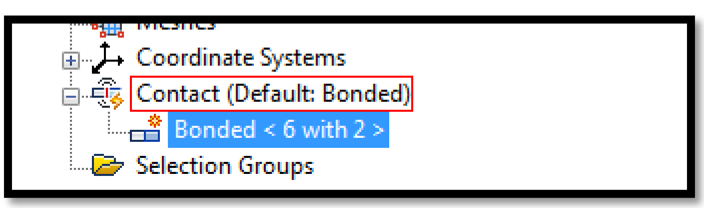


In this image, one of the retaining walls and the main bridge support have been selected as an example. The parts will highlight fuchsia when selected.

Next right select in the display window but off the model to select Contact and choose the appropriate contact method for your application:

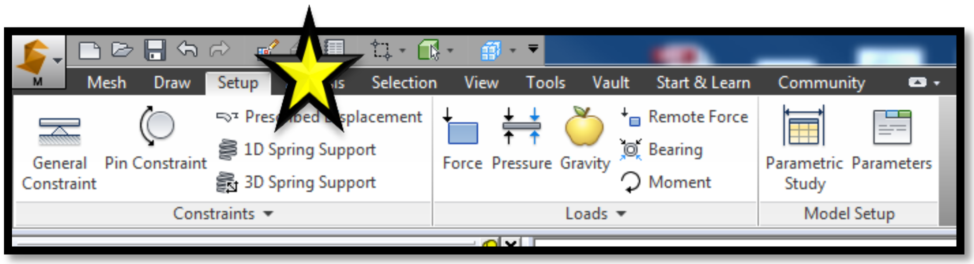


In the FEA Editor, you will see the part numbers highlighted. Hit Enter to apply. Repeat for all surfaces requiring a Contact setting (other than the default previously selected.) The FEA tree will look like this when you are finished:

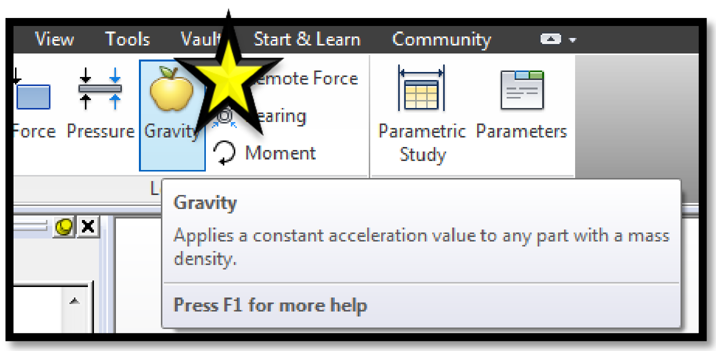


Gravity

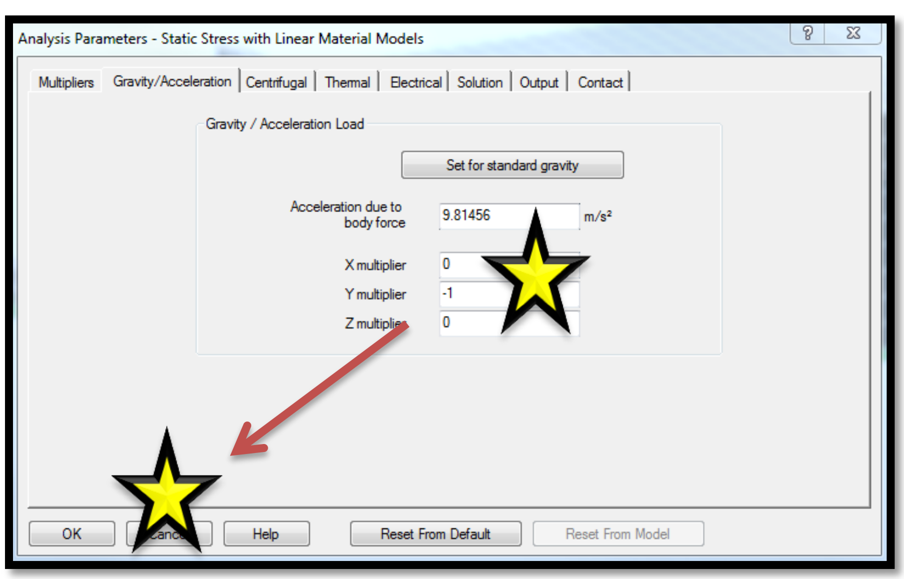
Last, you should apply gravity to match the orientation of the analysis model. Change to Setup tab in the Ribbon:



Select the Apple (Gravity) icon from the ribbon:



If you recall, the Sky was in the +Y direction. Therefore, you will need to set the gravity multipliers to a unit vector in the –Y direction as shown in the following image:



Change the values to match, then click OK or enter to apply.

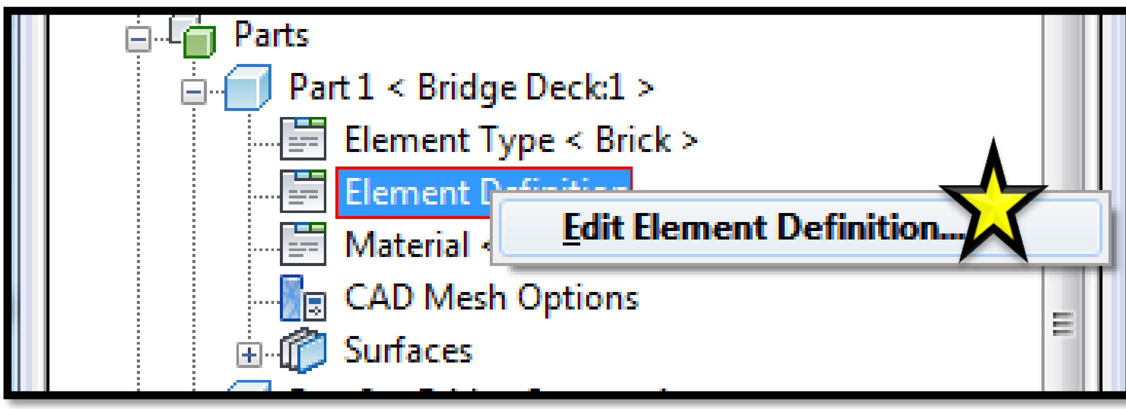
Apply CFD Temperature Results as Thermal Loads

Before analyzing the model, you first need to establish the stress free reference temperature, and then apply the CFD temperature results to the model.

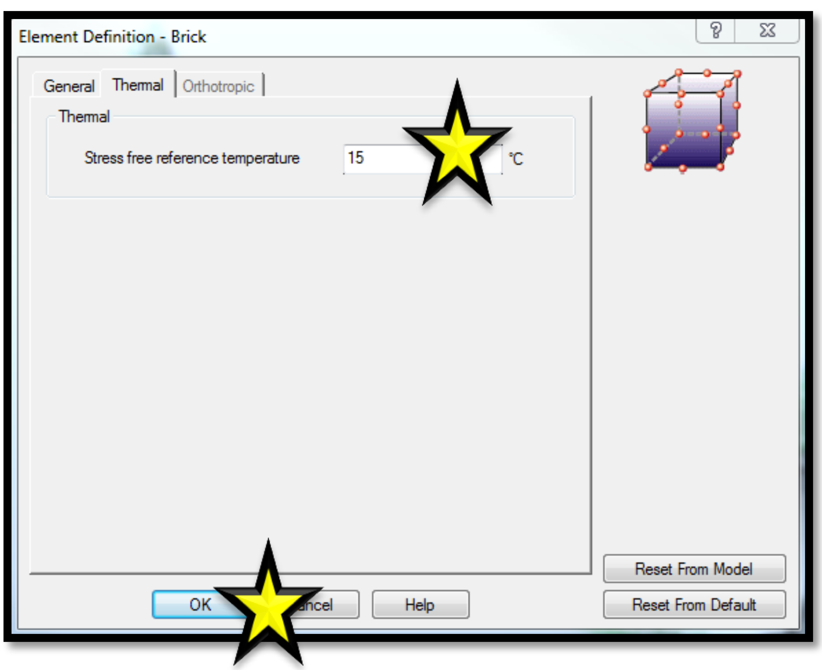
Stress free reference temperature

For each part, you need to set a reference temperature at which the structure is free of thermal strain. For lack of construction information on the Alvord Lake Bridge, you will use 15°C (60°F), considered an optimum pouring temperature for concrete.

Return to the FEA Editor tree and right select the Element Definition for a part:



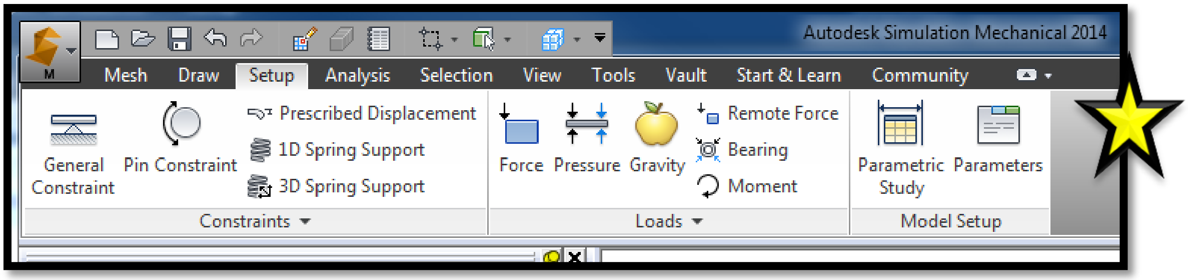
In the Element Definition – Brick dialog, select the Thermal tab and enter 15, then OK:



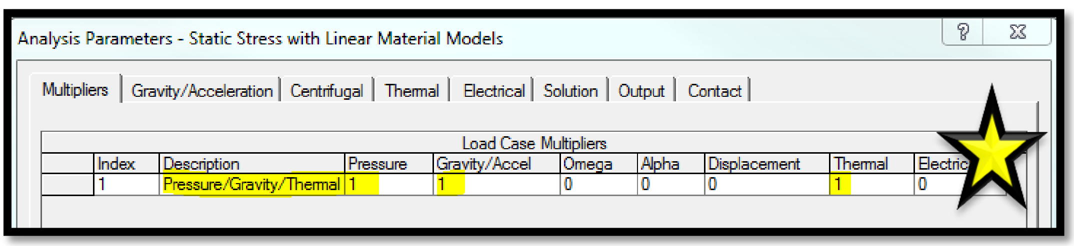
Repeat this for each of the 8 parts in the model.

Read CFD Thermal Results

Still on the Setup tab in the ribbon, select Parameters:



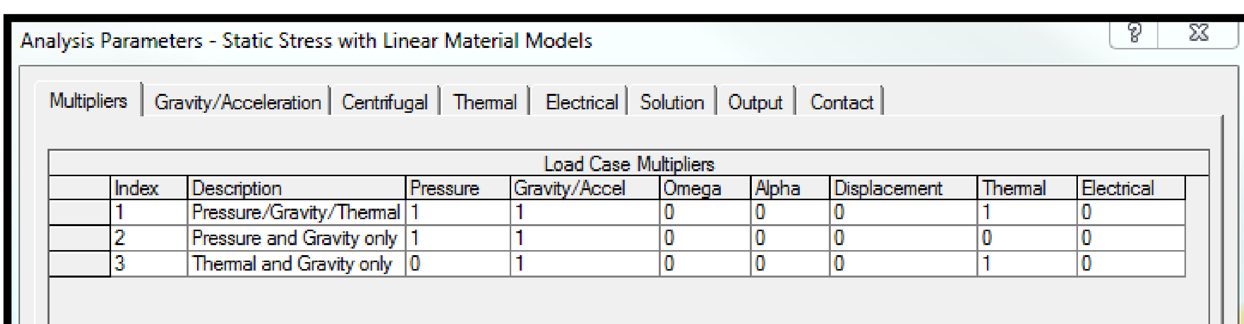
In the Analysis Parameters – Static Stress with Linear Material Models, you first enter a Load Case Description then you need to set the Multipliers. You need a Multiplier for Thermal to use the temperatures as calculated by Simulation CFD without a scaling factor. At the same time, you add the multiplier for Gravity/Accel (if not already done) in order to consider the weight of the concrete. Then remove the default Displacement multiplier since it is not needed for this analysis:



In this case, you can enter a description of “Pressure/Gravity/Thermal Load Case” or anything else to help you recall the case settings.

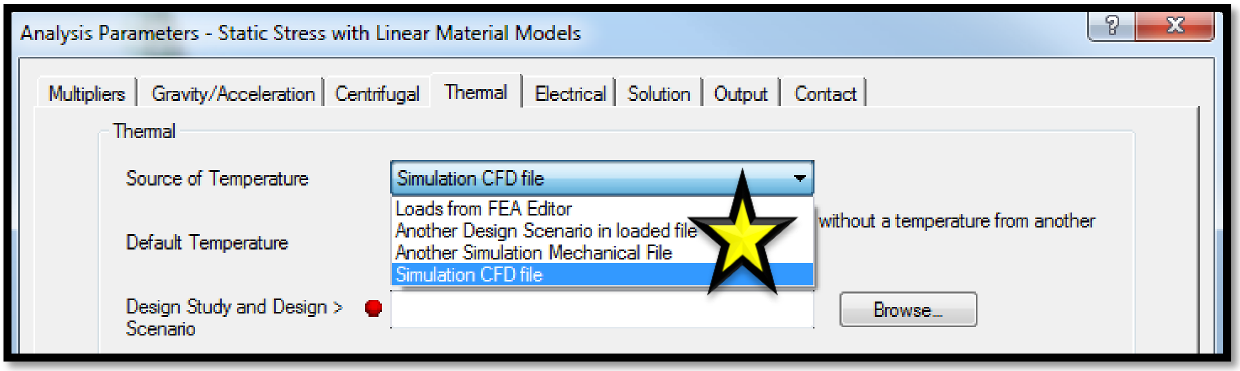
Tip: Using the Add Row button, you can choose to examine isolated loads to determine the influence of the load on the resultant stresses by simply adjusting the Multipliers to turn loads on or off.

For the class, please add the following rows:

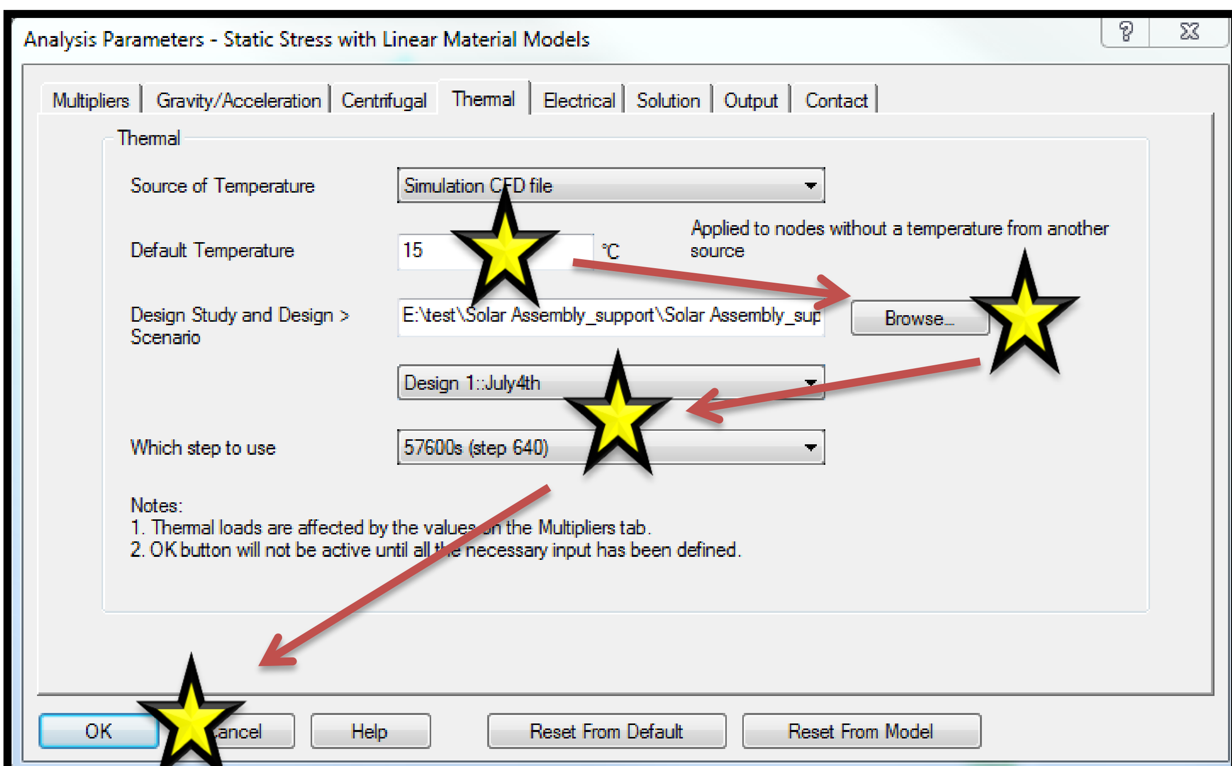


Index	Description	Pressure	Gravity/Accel	Omega	Alpha	Displacement	Thermal	Electrical
1	Pressure/Gravity/Thermal	1	1	0	0	0	1	0
2	Pressure and Gravity only	1	1	0	0	0	0	0
3	Thermal and Gravity only	0	1	0	0	0	1	0

Next, select the Thermal tab to select the CFD thermal results:

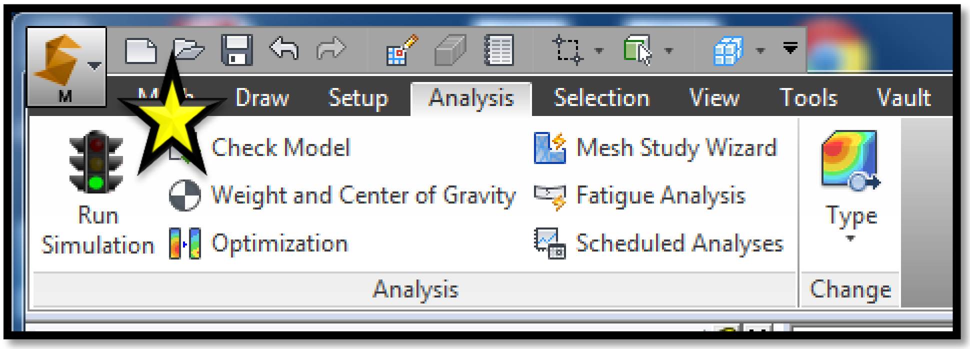


Set a default temperature of 15°C to match the zero stress temperature applied (this is not necessary but a good practice should something not transfer correctly). Use the Browse… button to navigate to “Solar Assembly.cfdst” file created and solved in CFD, then select Design 1::July4th and 57600s (step 640):

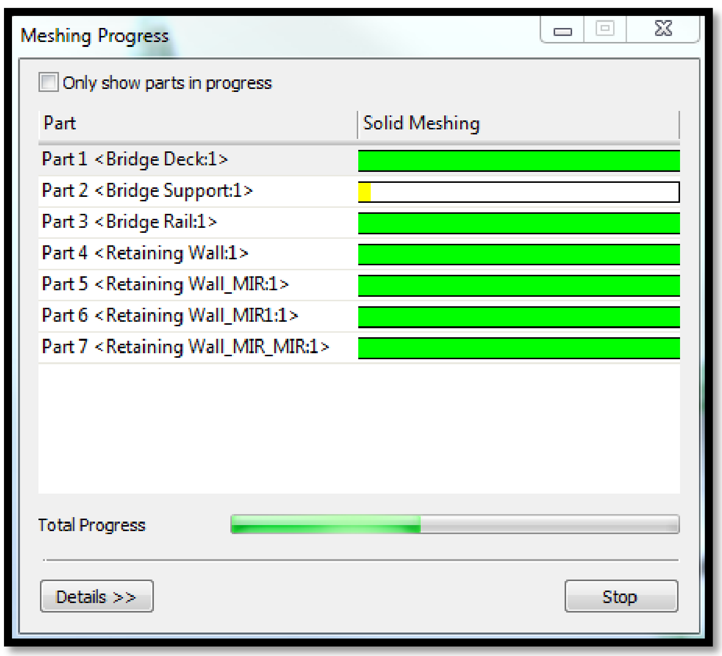


If your analysis did not run, you can use the previously run analysis in the “Demonstration\CFD with Results\Solar Assembly” directory.

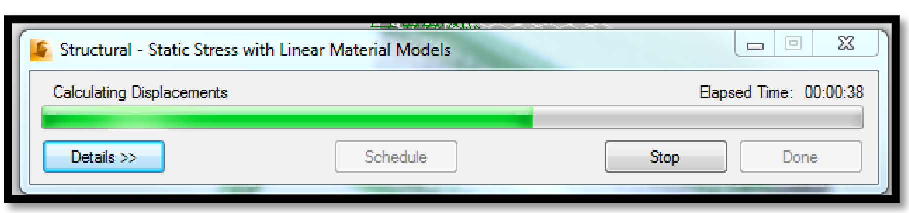
Proceed to the Analysis tab in the ribbon and observe the Run Simulation icon is active. You are ready to run:



The analysis will mesh then solve:

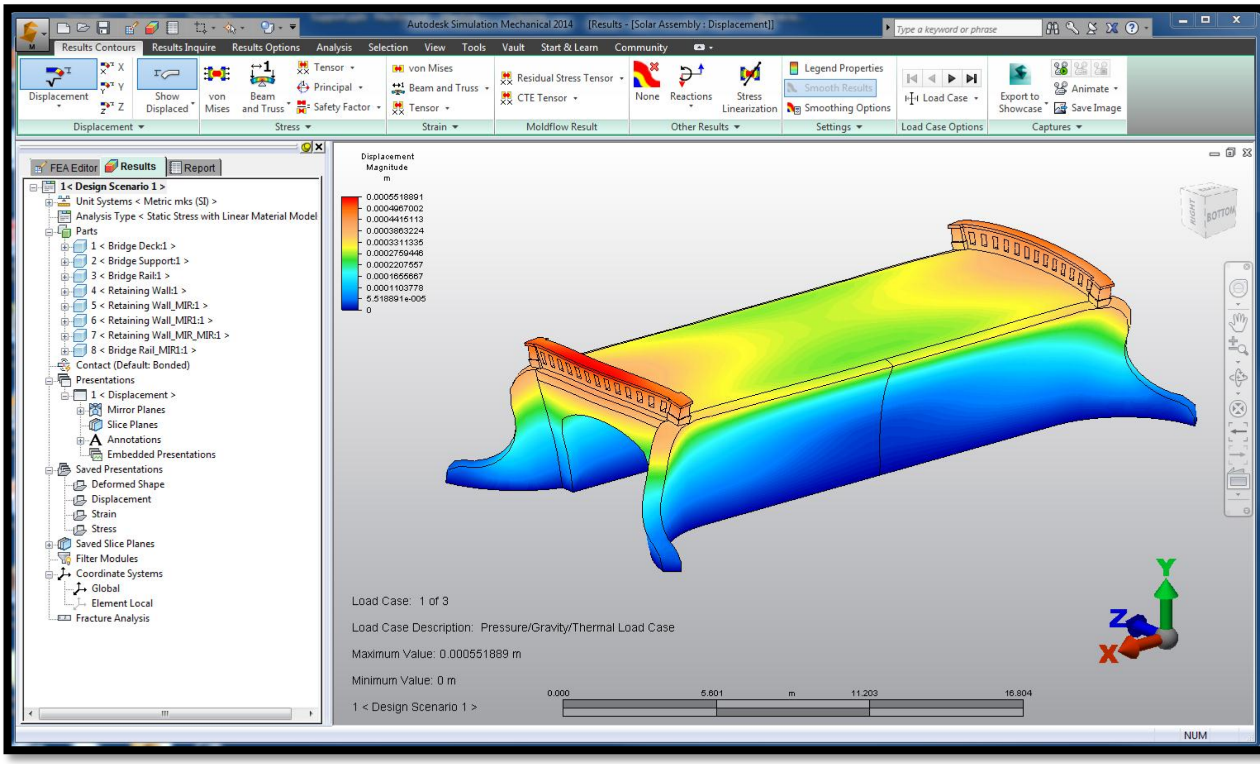


After all meshing bars are green, the analysis progress bar will appear:



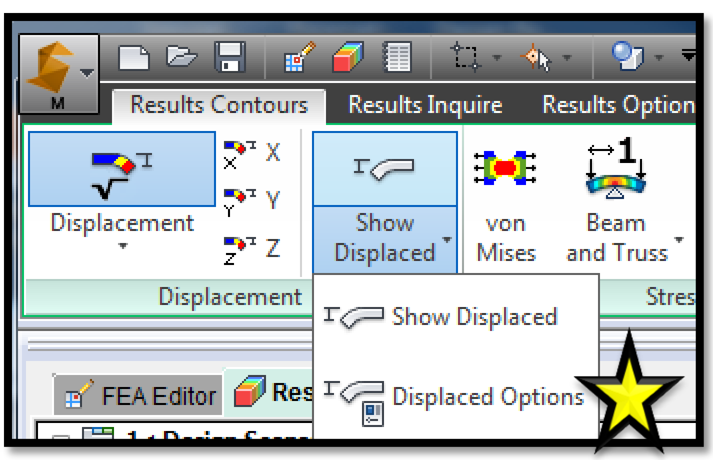
Visualize Results

When the analysis completes, the Results Contours tab will become active and displacements will show in the display:

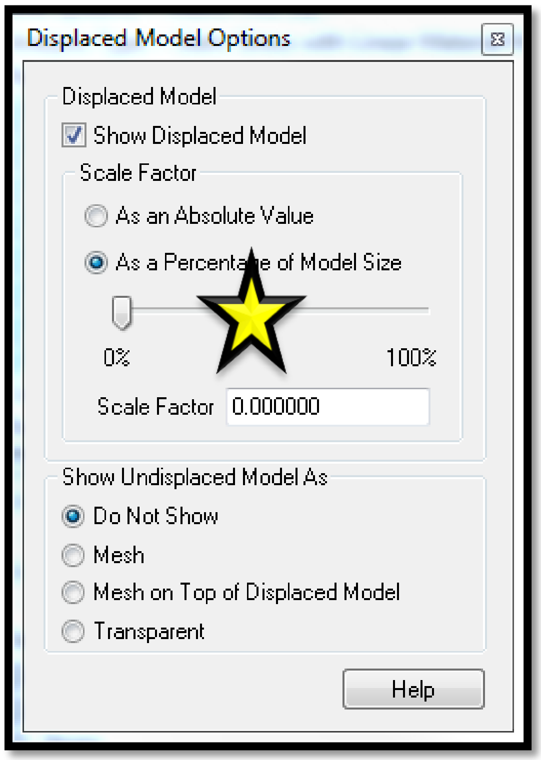


If you have experienced any issues, you can use file_open, to open "Solar Assembly.fem" in the "Demonstration\Mechanical with Results" directory. Then select the Results tab to continue.

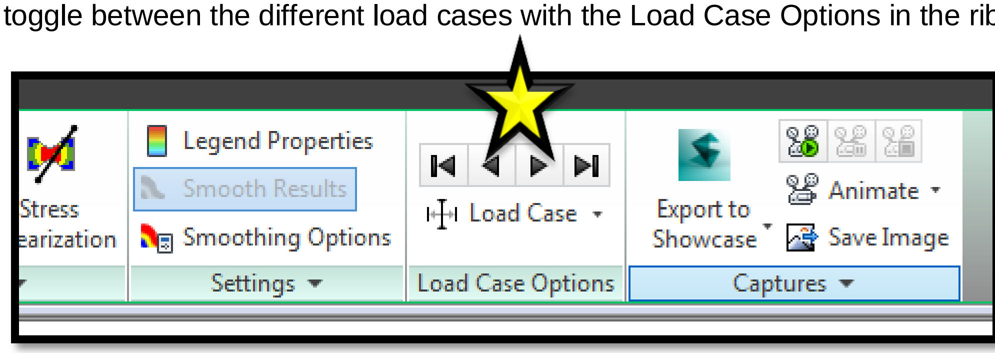
Visualizing results can be fun, but more importantly, can provide insight into how your design will perform. You should experiment with the Displaced Options:



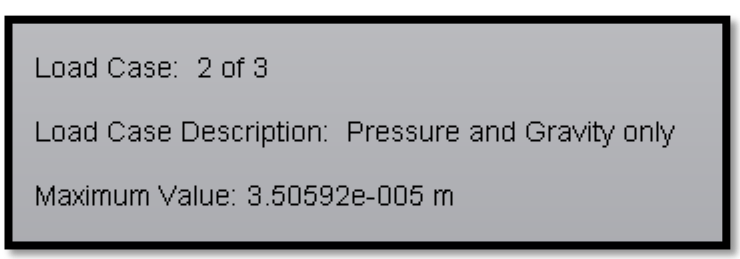
Adjust the slider for the displacement scale factor as a Percentage of Model Size to see which parts of the bridge displace:



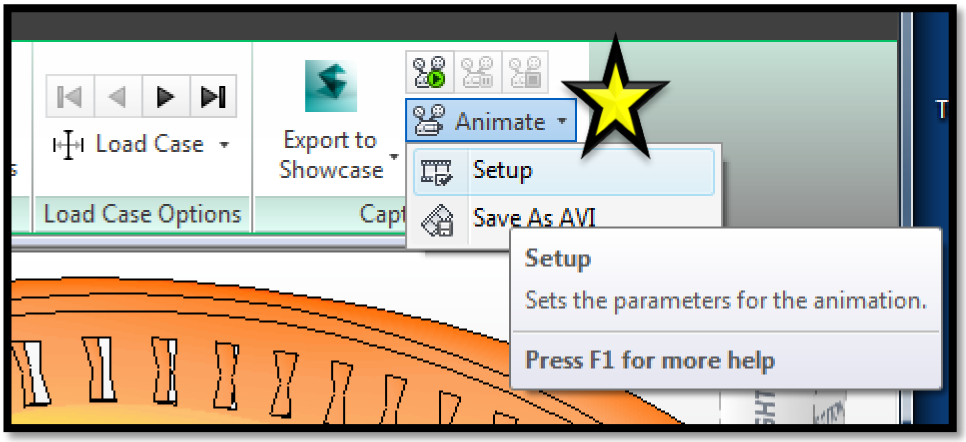
You can toggle between the different load cases with the Load Case Options in the ribbon:



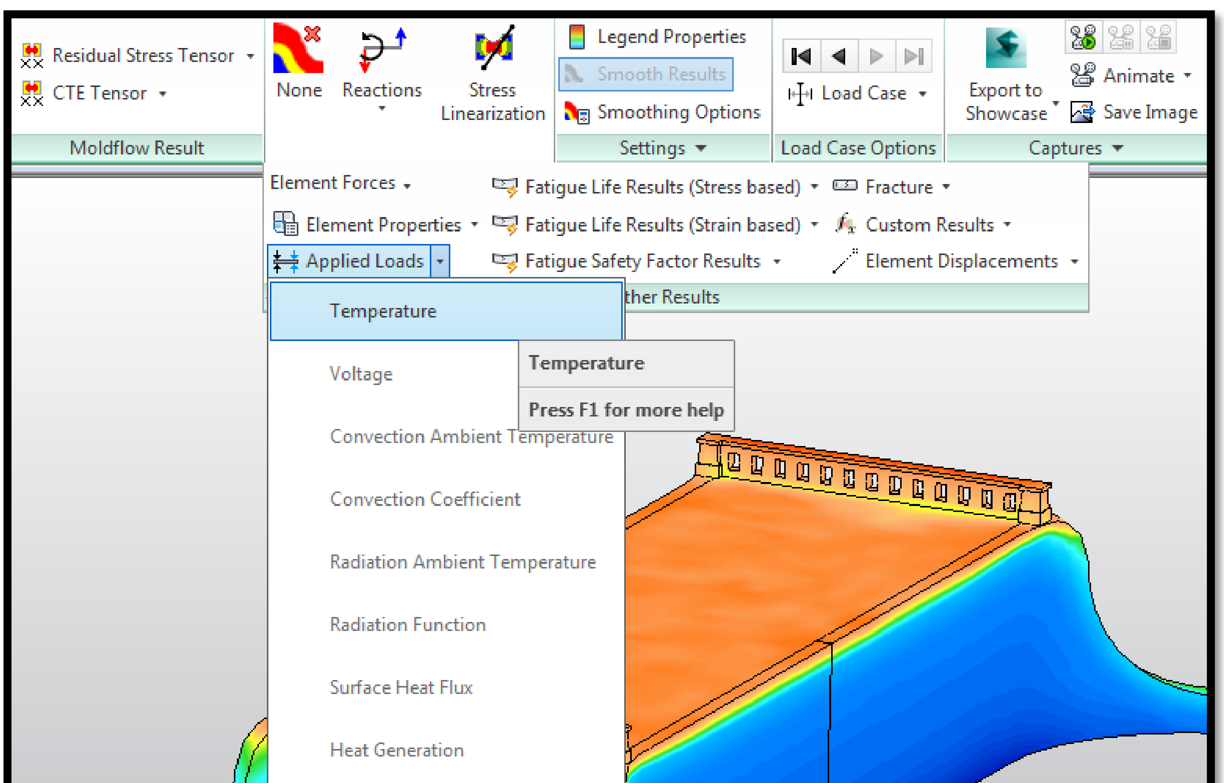
The display will indicate which load case you are viewing:



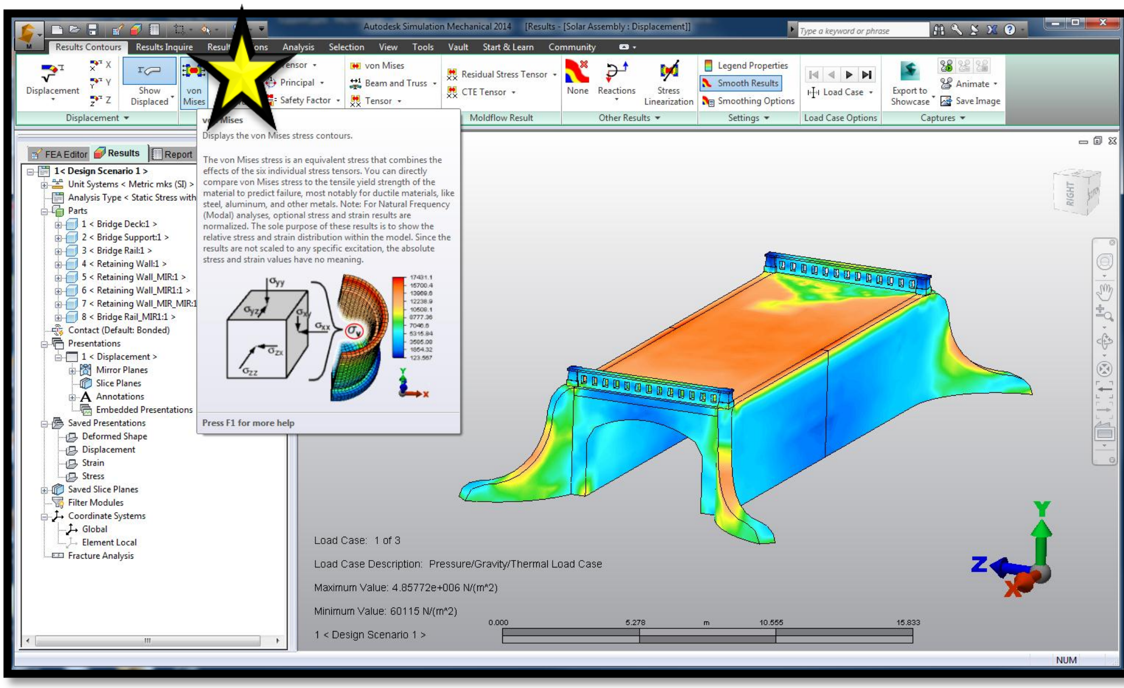
Don’t forget to capture an image or animate your model with scaled displacements. Select Captures in the ribbon and choose to Setup an animation for some visual fun:



Tip: To verify the correct thermal results were transferred from CFD to Mechanical, proceed to the Other Results tab in the ribbon. Select Applied Loads option then Temperature from the context menu. These temperature contours should match the results previously seen in CFD.

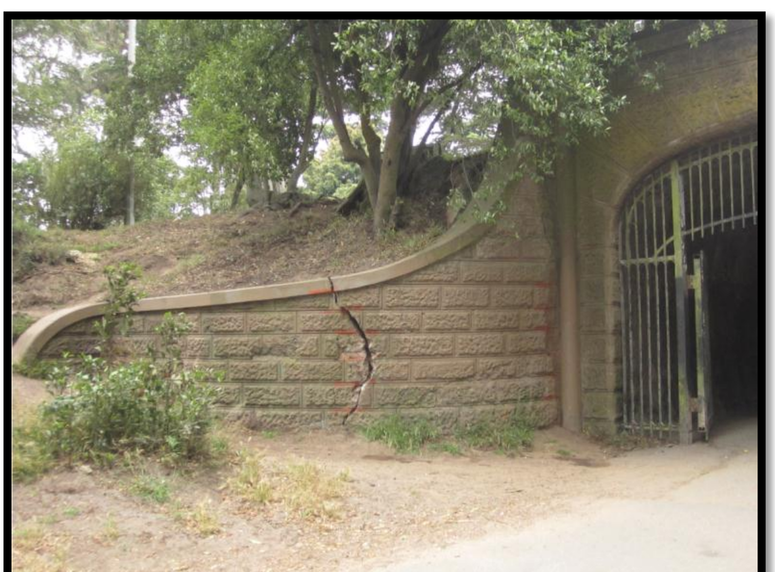


Shift attention now to the Stress tab to display the von Mises stress contours:



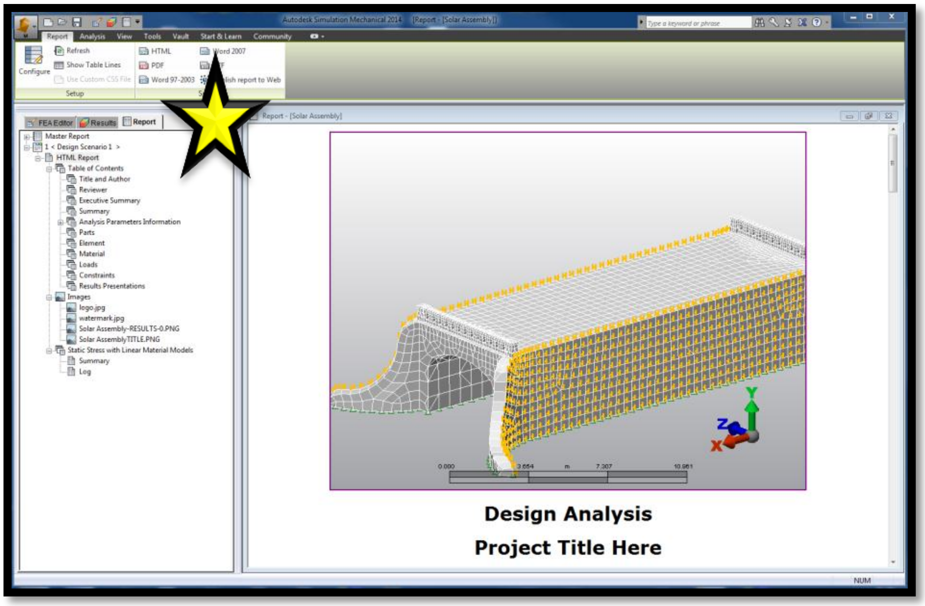
Toggle between the load cases to understand the relationship between the thermal load and the hydrostatic load induced by the ground. Notice the large displacements and stresses near the ends of the retaining walls as you toggle between Displacement and von Misses:

Now consider this image of the bridge where the concrete has cracked and separated in the same area:

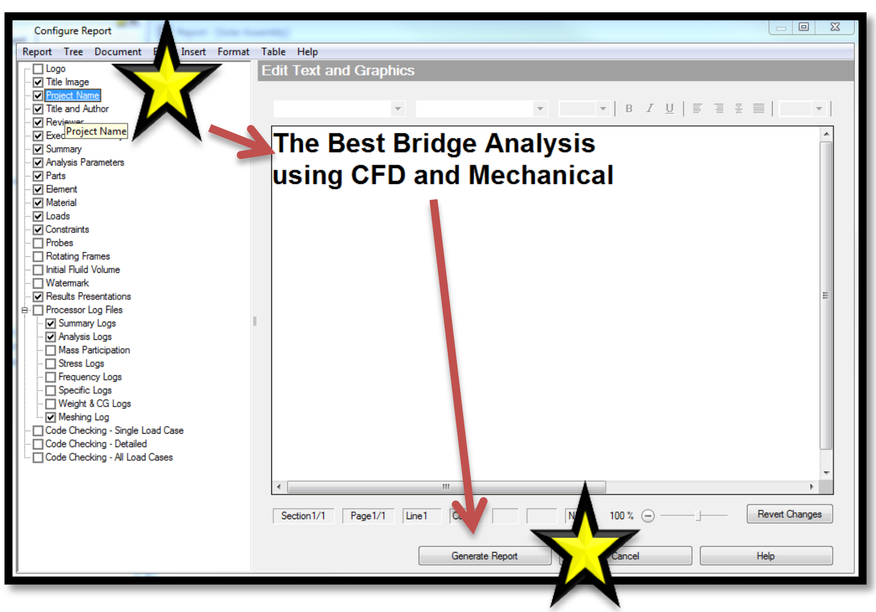


Reports

When you are ready, shift to the Report tab to record your analysis results:



Right select on HTML Report in the Report Tree and select to configure to customize the information you would like to record. For example, click on Project Name and in the Edit Text and Graphics window, type in – "The Best Bridge Analysis using CFD and Mechanical":



Have fun configuring your report and use the refresh to update.

Don't forget to save your report!