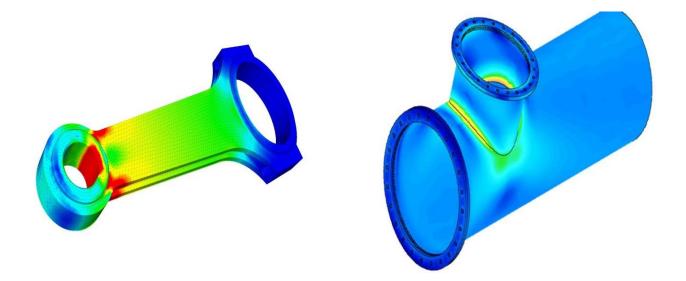


MODELLING AND ANALYSIS LAB MANUAL (I SEM, M.E MANUFACTURING ENGINEERING)



DEPARTMENT OF MECHANICAL ENGINEERING AARUPADAI VEEDU INSTITUTE OF TECHNOLOGY Vinayaka Nagar, Rajiv Gandhi Salai, (Old Mahabalipuram Road) Paiyanoor - 603104.Kanchipuram (Dt)

HOD/MECH

INDEX

S.NO	DATE	DESCRPTION	PAGE NO
1.		INTRODUCTION TO ANSYS	01
2.		CANTILEVER BEAM WITH POINT LOAD AT FREE END	08
3.		DISTRIBUTED LOADING OF A 1D CANTILEVER BEAM	30
4.		APPLICATION OF DISTRIBUTED LOADS	47
5.		BUCKLING FAILURE	66
6.		STRESS ANALYSIS OF AXI-SYMMETRY STRUCTURE	76
7.		ANALYSIS OF 2D TRUSS	87
8.		THERMAL ANALYSIS	99
9.		MODAL ANALYSIS OF A CANTILEVER BEAM	118
10.		MODAL ANALYSIS OF STEPPED SHAFT	134
11.		HARMONIC ANALYSIS OF GUITAR	139
12.		RADIATION EXCHANGE BETWEEN SURFACES	158

INTRODUCTION

Some Basic Concepts

The finite element method (FEM), or finite element analysis (FEA), is based on the idea of building a complicated object with simple blocks, or, dividing a complicated object into smaller and manageable pieces.

Why FEA?

Computers have revolutionized the practice of engineering. Design of a product that used to be done by tedious hand drawings has been replaced by computer-aided design (CAD) using computer graphics. Analysis of a design used to be done by hand calculations and many of the testing have been replaced by computer simulations using computer-aided engineering (CAE) software. Together, CAD, CAE, and computer-aided manufacturing (CAM) have dramatically changed the landscape of engineering.

Among all the computational tools for CAE, the FEM is the most widely applied method or one of the most powerful modern "calculators" available for engineering students and professionals. FEA provides a way of virtually testing a product design. It helps users understand their designs and implement appropriate design changes early in the product development process. The adoption of FEA in the design cycle is driven by market pressure since it brings many benefits that will help companies make better products with reduced development costs and time-to-market.

Finite Element Applications in Engineering

The FEM can be applied in solving the mathematical models of many engineering problems, from stress analysis of truss and frame structures or complicated machines, to dynamic responses of automobiles, trains, or airplanes under different mechanical, thermal, or electromagnetic loading. There are numerous finite element applications in industries, ranging from automotive, aerospace, defense, consumer products, and industrial equipment to energy, transportation and construction, as shown by some examples in Table. The applications of the FEA have also been extended to materials science, biomedical engineering, geophysics, and many other emerging fields in recent years.

FEA with ANSYS Workbench

Over the last few decades, many commercial programs have become available for conducting the FEA. Among a comprehensive range of finite element simulation solutions provided by leading CAE companies, *ANSYS® Workbench* is a user-friendly platform designed to seamlessly integrate *ANSYS, Inc.*'s suite of advanced engineering simulation technology. It offers bidirectional connection to major CAD systems. The *Workbench* environment is geared toward improving productivity and ease of use among engineering teams. It has evolved as an indispensible tool for product development at a growing number of companies, finding applications in many diverse engineering fields

A General Procedure for FEA

To conduct an FEA, the following procedure is required in general:

- Divide the CAD/geometric model into pieces to create a "mesh" (a collection of elements with nodes)
- Describe the behavior of the physical quantities on each element.
- Connect (assemble) the elements at the nodes to form an approximate system of equations for the entire model.
- Apply loads and boundary conditions (e.g., to prevent the model from moving).
- Solve the system of equations involving unknown quantities at the nodes (e.g., the displacements).
- Calculate the desired quantities (e.g., strains and stresses) at elements or nodes.

In commercial FEA software, this procedure is typically rearranged into the following phases:

- Preprocessing (build FEM models, define element properties, and apply loads and constraints)
- ▶ FEA solver (assemble and solve the FEM system of equations, calculate element results)
- Postprocessing (sort and display the results)

Overview of ANSYS Workbench

ANSYS Workbench is a simulation platform that enables users to model and solve a wide range of engineering problems using the FEA. It provides access to the ANSYS family of design and analysis modules in an integrated simulation environment. This section gives a brief overview of the different elements in the ANSYS Workbench simulation environment or the graphical-user interface (GUI). Readers are referred to ANSYS Workbench user's guide for more detailed information.

The User Interface

The *Workbench* interface is composed primarily of a *Toolbox* region and a *Project Schematic* region. The main use of the two regions is described next.

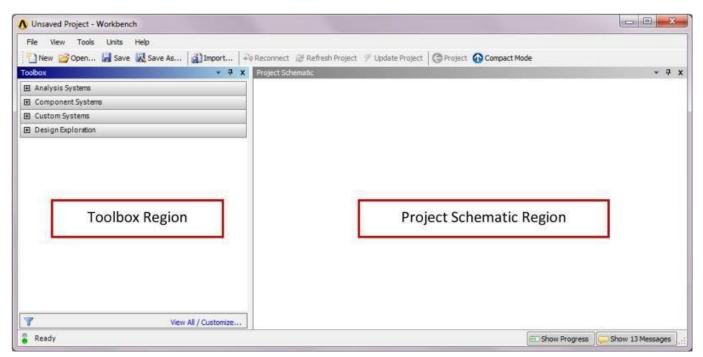


Figure: ANSYS Workbench user interface.

The Toolbox

The *Toolbox* contains the following four groups of systems:

Analysis Systems: Predefined analysis templates to be used to build your project, including static structural, steady-state thermal, transient thermal, fluid flow, modal, shape optimization, linear buckling, and many others.

Component Systems: Component applications that can be used to build or expand an analysis system, including geometry import, engineering data, mesh, postprocessing, and others.

Custom Systems: Coupled-field analysis systems such as fluid solid interaction, prestress modal, thermal-stress, and others.

Design Exploration: Parametric optimization studies such as response surface optimization, parameters correlation, six sigma analysis, and others.

The Project Schematic

A project schematic, that is, a graphical representation of the workflow, can be built by dragging predefined analysis templates or other components from the *Toolbox* and dropping them into the *Project Schematic* window. "Drag" here means to move the mouse while holding down the left mouse button, and "drop" means to release the mouse button.

To build a project for static structural analysis, for instance, drag the *Static Structural* template from the *Toolbox* and drop it into the rectangular box that appears in the *Project Schematic* window. A standalone analysis system that contains the components needed for static structural analysis is added to the project schematic as shown in Fig. The system consists of seven individual components called cells.

Alternatively, a standalone analysis can be created by double-clicking. For example, doubleclick the *Steady-State Thermal* template from the *Toolbox*, and an independent

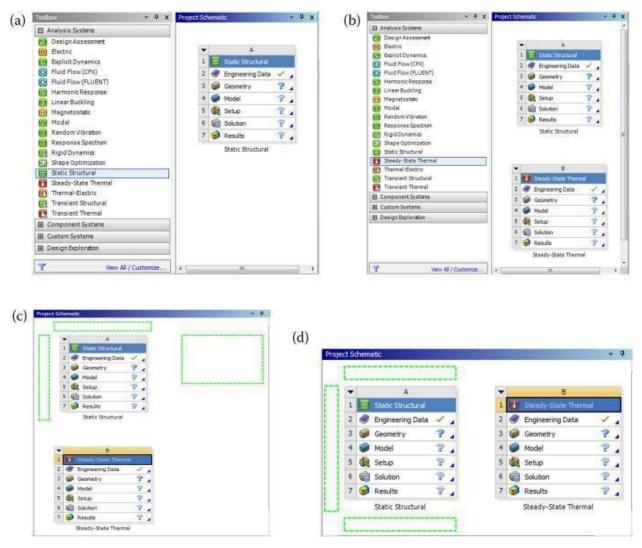


Figure: Defining standalone analysis systems in the project schematic: (a) a standalone system; (b) two independent standalone systems; (c) moving a system in a top-bottom configuration; and (d) moving a system in a side-byside configuration.

Steady-State Thermal system will be placed in the default location below the existing *Static Structural* system.

A system can be moved around another system in the project schematic. To move a system, click on the header cell (i.e., the cell titled *Steady-State Thermal* for the thermal system) and drag it to a new place. Once you drag the header cell, dashed rectangles appear for the possible new locations to drop the system. This is illustrated in Fig c and d for two systems with initial top–bottom and side-by-side configurations, respectively.

To delete a system, click on the down arrow button at the upper left corner of the system from the *Project Schematic* window, and then choose *Delete* from the drop-down context menu.

In somecases, a project may contain two or more analysis systems that share data. For example, a downstream modal analysis may use the same material, geometry, and model data from the preceding structural analysis. To build such a project, create a standalone system for *Static Structural* analysis. Then, drag the *Modal* analysis template from the *Toolbox* and drop it onto the *Model* cell of the *Static Structural* system. Immediately before the subsequent system is dropped, bounding boxes will appear on the *Engineering Data, Geometry,* and *Model* cells of the first system, as shown in Fig a. After the system is released, a project including two linked systems is created, as shown in Fig b, where the linked cells indicate data sharing at the *Model* and above levels.

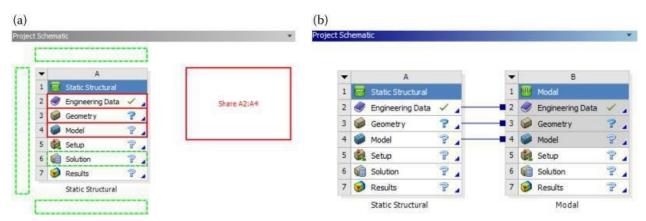


Figure: Defining linked analysis systems in the project schematic: (a) dropping the second (subsequent) system onto the Model cell of the first system to share data at the model and above levels; (b) two systems that are linked.

Working with Cells

Cells are components that make up an analysis system. You may launch an application by double-clicking a cell. To initiate an action other than the default action, right-click on a cell to view its context menu options. The following list comprises the types of cells available in *ANSYS Workbench* and their intended functions:

Engineering Data: Define or edit material models to be used in an analysis.

Geometry: Create, import, or edit the geometry model used for analysis.

Model/Mesh: Assign material, define coordinate system, and generate mesh for the model.*Setup:* Apply loads, boundary conditions, and configure the analysis settings.*Solution:* Access the model solution or share solution data with other downstream systems.*Results:* Indicate the results availability and status (also referred to as postprocessing).

As the data flows through a system, a cell's state can quickly change. *ANSYS Workbench* provides a state indicator icon placed on the right side of the cell. Table describes the indicator icons and the various cell states available in *ANSYS Workbench*. For more information, please refer to *ANSYS Workbench* user's guide.

The Menu Bar

The menu bar is the horizontal bar anchored at the top of the *Workbench* user interface. It provides access to the following functions:

File Menu: Create a new project, open an existing project, save the current project, and so on. *View Menu:* Control the window/workspace layout, customize the toolbox, and so on. *Tools Menu:* Update the project and set the license preferences and other user options. *Units Menu:* Select the unit system and specify unit display options. *Help Menu:* Get help for *ANSYS Workbench*.

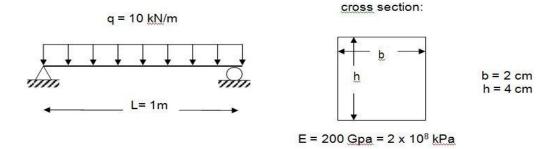
Cell State	Indicator	Description		
Unfulfilled	P	Need upstream data to proceed		
Refresh required	ē.	A refresh action is needed as a result of changes made on upstream data		
Attention required		User interaction with the cell is needed to proceed		
Update required		An update action is needed as a result of changes made o upstream data		
Up to date		Data are up to date and no attention is required		
Input changes pending		An update or refresh action is needed to recalculate base on changes made to upstream cells		
Interrupted		Solution has been interrupted. A resume or update acti will make the solver continue from the interrupted po		
Pending	17-	Solution is in progress		

Ex. No :

Date :

SIMPLY SUPPORTED BEAM WITH UNIFORMLY DISTRIBUTED LOAD

Problem Description



This is a simple, single load step, structural analysis of a simply supported beam. The beam is supported at both ends while there is a distributed load of 10kN/m. The objective of this problem is to demonstrate a simple ANSYS Workbench problem with a textbook solution: finding Von Mises' stresses and total deflection throughout the beam.

1. Preprocessing

Choose the preprocessor from the ANSYS main menu

Define the element type:

a) element type >> add/edit/delete >> add >>

The box on the left lists the general element categories, and the subheadings are lists of the element type. The category gives the element type a unique prefix.

a) Choose Beam

A list of element types for beams should appear in the right box

b) Choose 2D elastic 3 and click OK.

The three is the unique suffix for this element type. You have now specified that you will be using the ANSYS element Beam3, and it will be referred to as element type 1 in your model.

c) Click Close

- 2. Define material properties:
 - a) material properties >> Material Models...

A two part box will open. The left part indicates you are defining Material Model 1. In the right part double click:

b) Structural >> Linear Elastic >> Isotropic."

In the first box labeled EX input 2e11 and in the second boxlabeled PRXY input 0.35, click OK. Close the window.

This creates an isotropic material and assigns it the number 1.

For this simple analysis, the only elastic constants that need to be defined are Young's modulus and Poisson's ratio. You're responsible for keeping track of units since ANSYS is "unitless" Young's modulus is in N/m^2 or Pa.

- 3. Create a set of real constants for the cross-section
 - a) real constants >> add >>

Make sure the correct element type is shown

b) OK

Type the following in the correct boxes

- c) area=8e-4, Izz=1.067e-7, height=.04 and click OK.
- d) Click Close
- 4. Create nodes
 - a) Modeling create >> nodes >> in active cs>>

Enter the following node numbers and coordinates

Selecting apply will execute the command and query theuser for more input

- b) Node 1 is at 0,0
- c) Apply
- d) Node 2 is at .5,0
- e) Apply

Selecting OK will exit the node query and apply the currententry

- f) Node 3 is at 1,0
- 5. Create elements
 - a) Create >> elements >> auto numbered >> thru nodes >>
 - b) Pick node 1 then node 2
 - c) Apply
 - d) Pick 2 then 3

Plot the elements if they disappear.

a) Utility Menu: plot >> elements

Also under plot controls, you can opt to see the boundaryconditions and loads that you are creating.

a) Utility Menu: plot controls >> symbols >> All Applied BCs>>OK

Now apply a set of boundary conditions

The Loads menu is in both the Preprocessor menu and the Solutions menu. They are the same. From the Preprocessormenu, first you need to pick the analysis type:

- a) Loads >> new analysis >> analysis type >> static
- b) loads >> define loads >> apply >> structural >> displacement >> on nodes >>
- c) pick node 1
- d) apply

A window will pop up asking you for the displacement

- e) choose ux and uy, enter the value 0
- f) apply
- g) Pick node 3 this time
- h) apply

- i) Select only uy and enter the value 0
- j) OK
- k) Boundary conditions "tie down" the structure at points. It is very important to tie the structure down, otherwise you'll get zero stress everywhere, large deflections and an invalid solution in general. The only exception is in a modal analysis, where you can run the analysis "free-free", that is, unconstrained.

Apply the surface load

- a) loads >> define loads >> apply >> Structural >> pressure >> on beams >> pick all
- b) enter the value 10000 in the box for node I (ANSYS will automatically set the load for node J assuming a uniform pressure load)
 - The load is in N/m, thus we are using a fully consistent set of units N, m, and Pa. ANSYS does not keep track of theunits for you so beware!
 - You should see the pressure loading graphically since you turned on the boundary conditions earlier.
- c) Click OK

Select all of the nodes (go to the Utility Menu)

- a) Select >> everything
- b) OK

SAVE_DB (from the ANSYS Toolbar) Finish (ANSYS Main Menu, at the bottom)

Solution phase

Select Solution from the ANSYS main menu.

- a) Solve ->> current LS
- b) OK

Post-processing

Select general post-processor from the ANSYS main menu.

Read in the results

a) Read results - first set

Plot the results

a) Utility menu: Plot>>results>>deformed shape>>OK

this plots the deformed shape.

Looking at the deformed shape always gives a first checkto see if the loads, boundary conditions, etc. were applied correctly. If the shape does not make physical sense, examining this plot first will save time and paper.

b) Select PlotCtrls>>Hard copy>> To file and giving the *filename* then hit OK

This basically dumps the graphics window image into apostscript file.

General Postprocessor

- a) List results>>nodal solution>> Displacement vector sum>>OK
- b) Print it out
- c) Close

Then obtain a list of element stresses

a) List results>>element solution>> Line Element Results >>Element Results>> OK

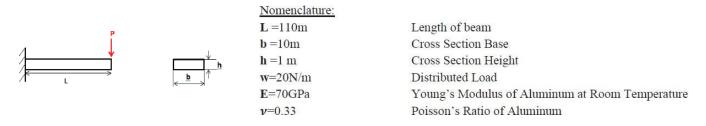
List the reaction forces

List Results>>Reaction Solu...>>All items>>OK

Ex. No : Date :

CANTILEVER BEAM WITH POINT LOAD AT FREE END

Problem Description



This is a simple, single load step, structural analysis of a cantilever beam. The left side of the cantilever beam is fixed while there is a distributed load of 20N/m. The objective of this problem is to demonstrate a simple ANSYS Workbench problem with a textbook solution: finding Von Mises' stresses and total deflection throughout the beam. The beam theory for this analysis is shown below:

Theory

Von Mises Stress

Assuming plane stress, the Von Mises Equivalent Stress can be expressed as:

 $\sigma' = (\sigma_x^2 - \sigma_x \sigma_y + \sigma_y^2 + 3\tau_{xy}^2)^{\frac{1}{2}}$ Additionally, since the nodes of choice are located at the top surface of the beam, the shear stress at this location is zero.

$$(\tau_{xy} = 0, \ \sigma_y = 0).$$

Using these simplifications, the Von Mises Equivalent Stress from equation 1 reduces to:

$$\sigma' = \sigma_x$$

Bending Stress is given by:

 $\sigma_x = \frac{P(x-L)c}{I}$ Where $I = \frac{1}{12}bh^3$ and $c = \frac{h}{2}$. From statics, we can derive:

$$\sigma_{\rm x} = \frac{6P({\rm x}-{\rm L})}{b{\rm h}^2} = 66k{\rm Pa}$$

Beam Deflection

As in module 1.1, the beam equation to be solved is:

$$\frac{d^2y}{dx^2} = \frac{M(x)}{EI}$$

Using Shigley's Mechanical Engineering Design, the beam deflection is:

$$\delta(x) = \frac{Px^2(x-3L)}{6EL}$$

With Maximum Deflection at:

$$\delta = \frac{PL^3}{3EI} = 7.61 \mathrm{mm}$$

Workbench Analysis System

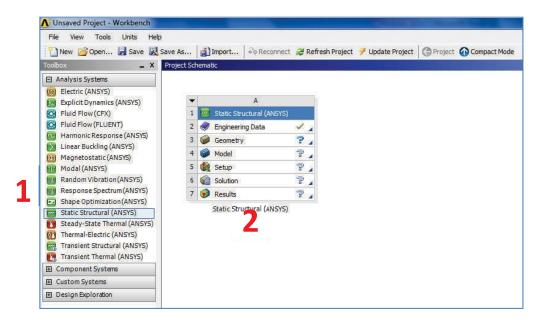
Opening Workbench

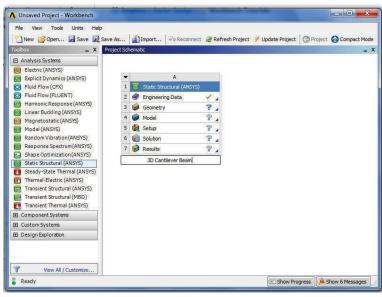
- 1. On your Windows Desktop click the Start button.
- 2. Under Search Programs and Files type "ANSYS"
- 3. Click on ANSYS Workbench to start workbench. This step may take time.



Static Structural Analysis

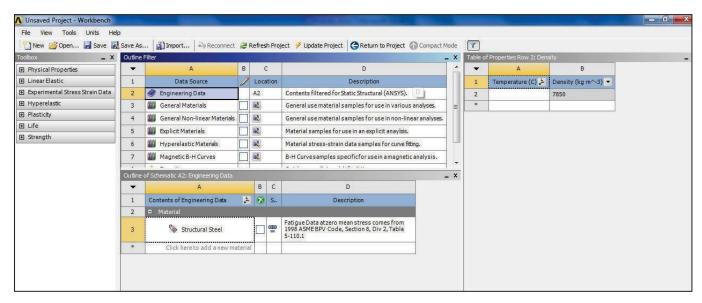
- 1. As you open ANSYS you can see the entire array of problems on the left had side this software can help you solve. The problem at hand is a *Static Structural* problem. Double click **Static Structural (ANSYS)** to open the task manager for your problem set in the Project Schematic area.
- 2. ANSYS allows you to build on each problem, so it is smart to name each project. At the bottom of the task manager you will see **Static Structural (ANSYS)**, double click this to change the name. For this problem choose "*3D Cantilever beam*."





Engineering Data

To begin setup for your cantilever beam, double click or right click on *Engineering Data* and click *edit*. This will bring up another screen.



This new window will allow you to alter the material properties of your cantilever beam. Under **Outline of Schematic A2: Engineering Data**, it shows *click here to add a new material*, this menu allows you to input the material of your cantilever beam, double click and type *Aluminum*.



WARNING Do not delete or change the Structural Steel, just another material.

Now expand *Linear Elastic* by double clicking on E Linear Elastic or on the plus

- 🗆 Linear Elastic
 - Isotropic Elasticity
- 🚰 Orthotropic Elasticity

symbol shown. 🎦 Anisotropic Elasticity

Double click on Isotropic Elasticity to give the material the same properties across the beam. This action brought up a new table on the right; this allows us to add necessary properties. As show on the top right of the screen in *Table of Properties Row 2: Isotropic Elasticity*:

- 1. Click in Temperature and type 25
- 2. Click in Young's Modulus and type 70E9 or 7E10
- 3. Click in Poisson's Ratio and type 0.33

•	A	В	С
1	Temperature (C) 📮	Young's Modulus (Pa) 🔻	Poisson's Ratio
2	25	7E+10	0.33
*			

WARNING Make sure to DELETE the Temperature entry after property input before continuing! Failure to do so will lead to errors later

	Table of	Properties Row 2: Isot	ropic Elasticity	asticity _ X		
	-	А	В	с		
Delete!	1	Temperature (C) 🏓	Young's Modulus (Pa) 🔻	Poisson's Ratio	Bulk Mod	
Derete	2	.	7E+10	0.33	6.8627E+	
	*					

After filling in the properties, this concludes the Engineering Data, to return to the project schematic area, click on seen on the upper tab.

Return to Project

Geometry

Base Geometry

1. Go to **Workbench -> Project Schematic -> Geometry** and double click. This will open a new window for *ANSYS Design Modeler* where the Geometry will be created.

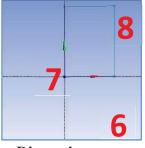
Select desired length u	init:
Meter	C Foot
C Centimeter	C Inch
C Millimeter	
C Micrometer	
Always use proje	
Enable large mode	

Note: Select meters and hit ok

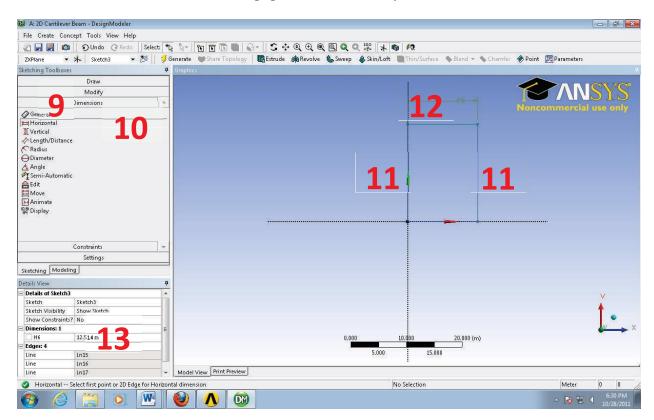
- 2. In the new window, click the **Display Plane** icon to toggle the coordinate system.
- Go to Design Modeler -> Tree Outline -> right click on YZPlane. Click Look At to view the YZ plane.



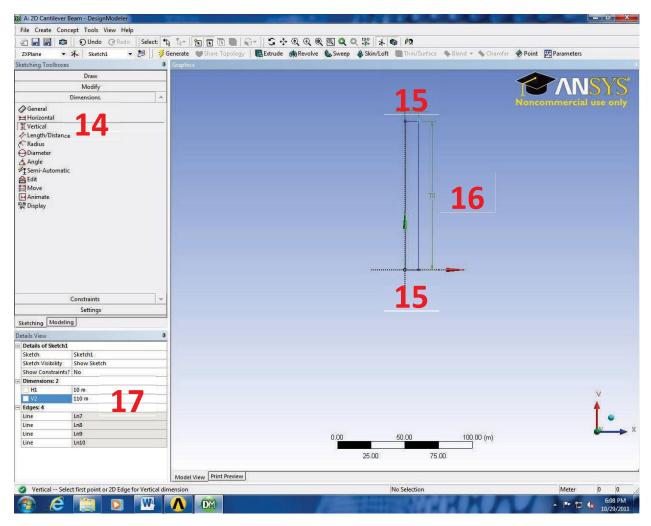
- 4. Go to **Design Modeler -> Tree Outline -> Sketching**
- 5. Click on **Rectangle** and Click off **Auto-Fillet**:
- 6. Bring your cursor into the workspace at point 0,0, over the origin until 'P' appears directly above the origin.
- 7. Click on the origin to place the lower left corner of our rectangle on the origin.
- 8. Click on a point in the first quadrant to define the top right corner of our rectangle. The point is arbitrary as we will be fixing dimensions momentarily.



- 9. Go to Sketching Toolboxes -> Dimensions
- 10. Click Horizontal to specify a horizontal dimension.
- 11. Click the left and right faces of the rectangle in the sketch to specify that we will be dimensioning this horizontal length. A green line with a symbol should appear.
- 12. Drag the green line above the sketch and click to set its location.
- 13. Go to **Detail View -> Dimension 1.** In the first subcategory, replace the current dimension with 10. The units should populate automatically.



- 14. Go to **Sketching Toolboxes -> Dimensions -> Vertical** to specify the vertical dimension.
- 15. Click the bottom and top faces of the sketch to specify the vertical dimension. A green line should appear.
- 16. Drag the green line to the right of the sketch and click.
- 17. Go to **Detail View -> Dimension 2**. Replace the value with 10. The units should populate automatically (meters).



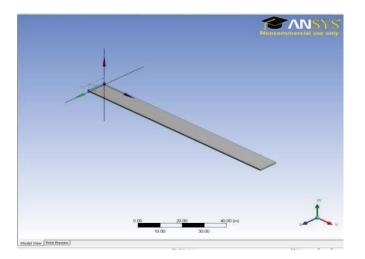
Now that we have modeled the base geometry, we will extrude it to create a 3D volume.

Extrude Sketch

- 1. Go to Main Toolbar -> and select Extrude
- 2. Go to Modeling -> FD1, Depth (>0) -> enter in 110
- **3.** Go to **Design Modeler** -> *i* **Generate**.
- **4.** To verify our geometry, look at the isometric view. Click the *blue dot* in the **triad** in the lower right corner of the screen to look at the isometric view.

Details of Extrude2		
Extrude	Extrude2	
Base Object	Sketch1	
Operation	Add Material	
Direction Vector	None (Normal)	
Direction	Normal	
Extent Type	Fixed	
FD1, Depth (>0)	110 m	
As Thin/Surface?	No	
Merge Topology?	Yes	

Your 3D turface should look like this:

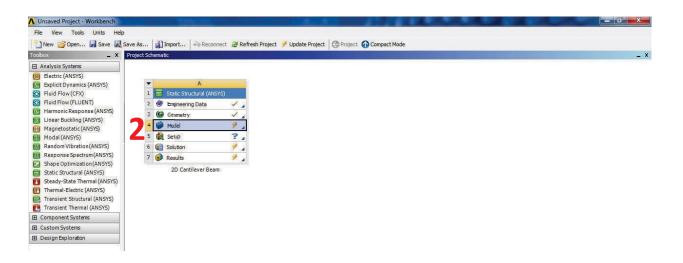


Now that we have the geometry, we will mesh the beam using 3D Elements.

Model

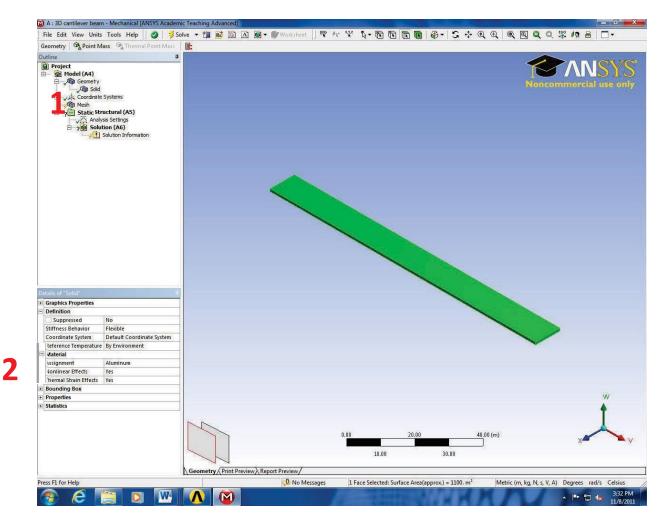
Open ANSYS Mechanical

- 1. **_____** out of Design Modeler. Don't worry, your work will be saved.
- 2. Go to Workbench -> Project Schematic -> Model This will open ANSYS Mechanical



Material Assignment

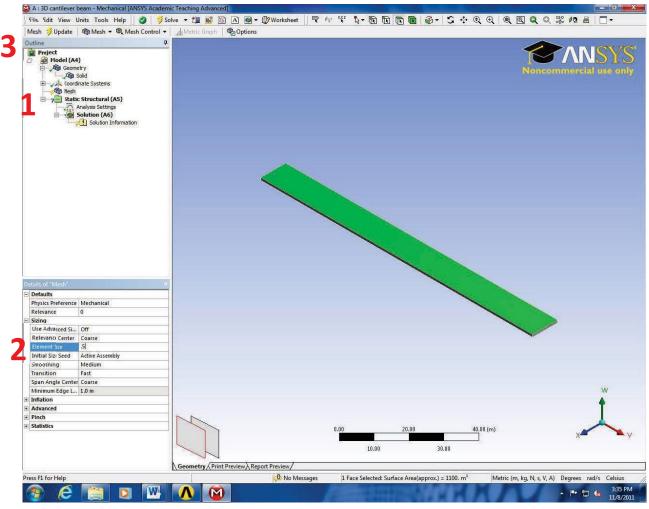
- 1. Go to Mechanical -> Outline -> Project -> Model -> Geometry -> Surface Body
- 2. Under Mechanical -> Details of "Surface Body" -> Material -> Assignment, change *Structural Steel* to *Aluminum*.



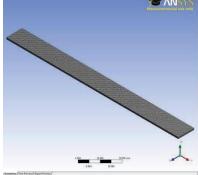
22

Mesh

- 1. Go to Mechanical -> Outline -> Project -> Model -> Mesh
- 2. Go to Mechanical -> Details of 'Mesh' -> Sizing -> Element Size and change the value from *Default* to .5 m. This will give us 2 elements through the thickness of the beam.



3. Click Mechanical -> Update. This may take some time. Your mesh should look as shown below:

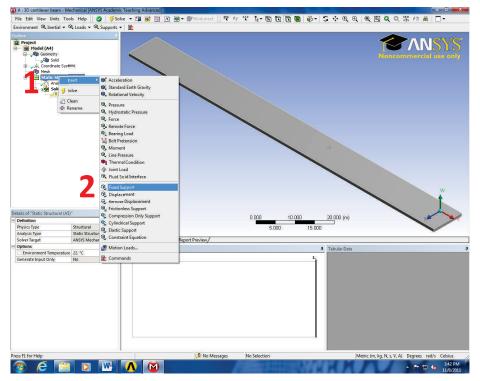


<u>Setup</u>

You can perform the rest of your analysis for this problem in the *ANSYS Mechanical* window. The other options in the *Workbench* window will link you back to the same screen (i.e. Setup, Solution, Results)

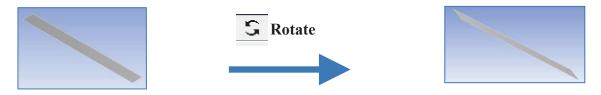
Fixed Support

- 1. Go to Mechanical -> Outline -> right click Static Structural (A5)
- 2. Go to Insert -> Fixed Support

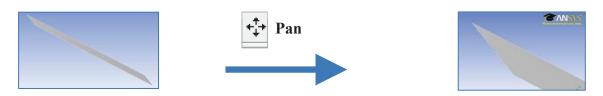


We are going to fix the elements at the left end of the beam. In order to do this, we will use the **Edge** tool to select the left edge. However, from the current orientation of the beam, it is difficult to select this surface.

3. Using the **S** Rotate tool click on the graphic area and move the mouse to the right. This will cause the left end of the beam to be oriented in a manner that can be clicked



4. Using the **Pan** tool, click the graphic area and drag the left face to the center of the graphic window. Use the *mouse scroll* to zoom in on the left face



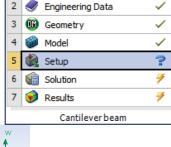
- 5. Click the **Edge** tool.
- 6. Go to Mechanical -> Outline -> Static Structural (A5) -> Fixed Support
- 7. Run the cursor across the left end face. When it becomes red, click it to select it.
- 8. Go to Mechanical -> Details of "Fixed Support" -> Geometry and select Apply

<u>Setup</u>

While in the Project Schematic double click Setup This will _ en a new window similar to Model Space

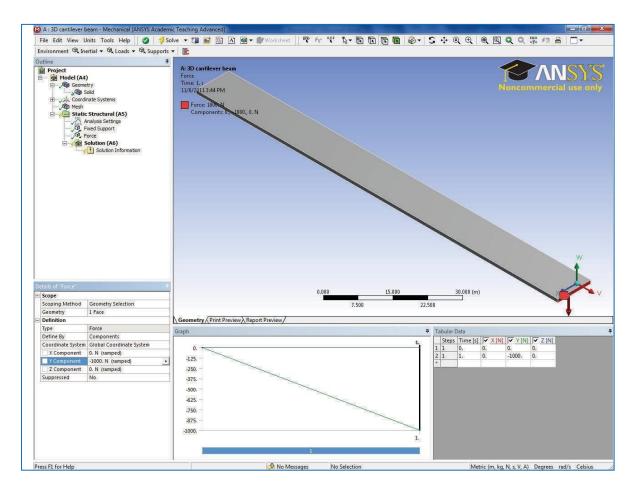
Loads

- 1. Click the x-axis icon to get a side view of the cantilever beam
- 2. Click Fixed end On the tool bar, make sure vertex option is selected.
- 3. Go to Mechanical -> Outline -> Static Structural (A5) -> Fixed Support
- 4. Run the cursor across the left end face. When it becomes red, click it to select it.
- 5. Click the left side of the geometry; this will add a green box to select the point.
- 6. Right click 2 Static Structural (A5)
- 7. Click insert, and 🚱 Fixed Support
- 8. This will add a fixed end to your cantilever beam in the work space.
- 9. Point Load On the tool bar, change selection option to Edge: vertex.
- 10. Click on the geometry, this will highlight
- 11. Right click ; Static Structural (A5)
- 12. click insert, and A table will appear "Details of Line Pressure" 🔍 Line Pressure
- 13. Under "Definition" you will see "Defined by"→Change this to "Components"
- 14. As shown, Y Component force is zero. \rightarrow Change this to value to -20
- 15. This will show your cantilever beam with a load applied as shown. Leave the Setup screen open this time.



edge instead of

A Static Structural (ANSYS

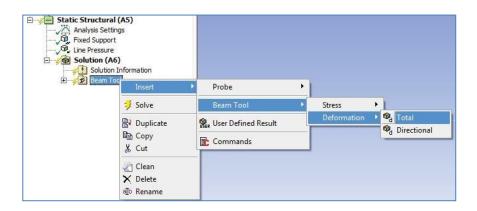


Solution

Go to Mechanical -> Outline -> Project -> Model(A4) -> Static Structural (A5) -> Right Click Solution (A6) -> Insert -> Beam Tool

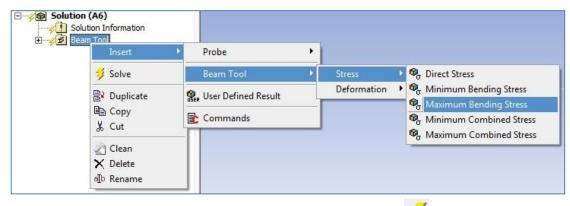
Deformation

```
Go to Mechanical -> Outline -> Project -> Model(A4) -> Static Structural (A5) -> Solution (A6) -> Beam Tool -> Insert -> Beam Tool -> Deformation -> Total
```



Stress

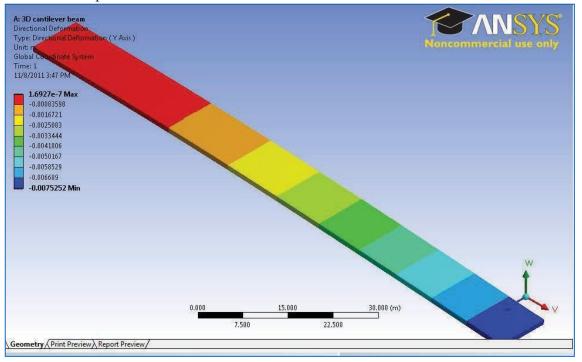
Go to Mechanical -> Outline -> Project -> Model(A4) -> Static Structural (A5) -> Solution (A6) -> Beam Tool -> Insert -> Beam Tool -> Stress -> Maximum Bending Stress



Now that our solvers have been defined, go to **Mechanical** -> **Solve**. The calculations in Workbench may take up to a minute to solve.

Go to Mechanical -> Outline -> Project -> Model (A4) -> Solution (A6) -> Maximum Bending Stress

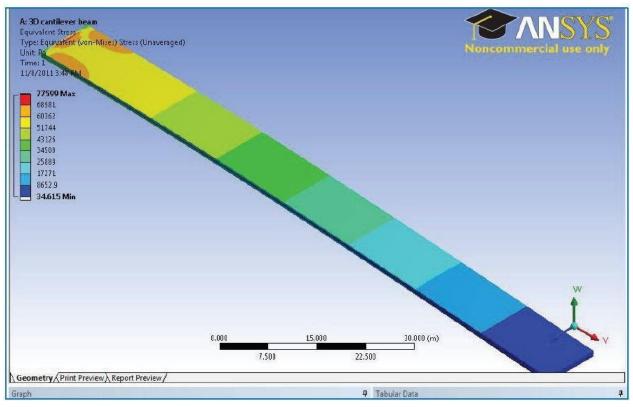
Go to **Details of "Maximum Bending Stress"-> Integration Point Results -> Display Option ->** Change to **Unaveraged**



Your Stress plot should look as shown below:

Go to Mechanical -> Outline -> Project -> Model(A4) -> Solution(A6) -> Total Deformation

Your Von-Mises plot should look as shown below:



Results

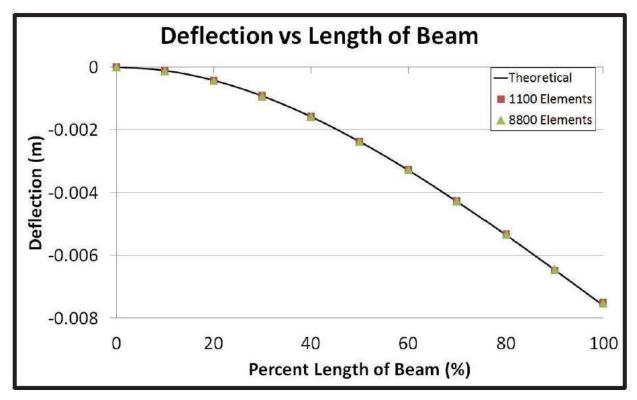
Max Deformation Error According to equation, the theoretical max deflection is 7.16 mm. The percent error (%E) in our model can be defined as:

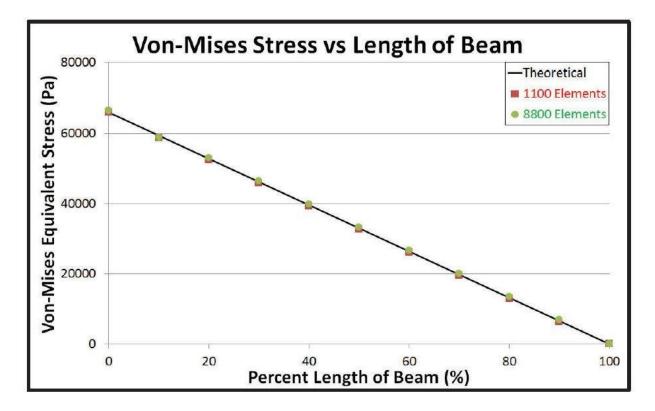
$$\% E = abs\left(\frac{\delta_{theoretical} - \delta_{model}}{\delta_{theoretical}}\right) * 100 = 1.28\%$$

Max Equivalent Stress Error

According to equation, the theoretical max equivalent stress is 66000 Pa. Using the same definition of error as before, we derive that our model has **6.3%** error in the max equivalent stress. The reason for the elevated stress level is singularity resulting from Poisson's effect at the fixed support. In the validation section, it is shown that with increased mesh size, the analytical answers for Max Equivalent stress are closely represented in nodes close to but not at the region where singularity occurs. The effect of singularity is also reduced with the implementation of higher order elements.



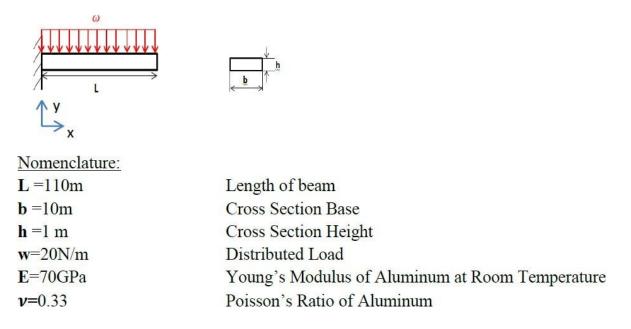




Ex. No : Date :

DISTRIBUTED LOADING OF A 1D CANTILEVER BEAM

Problem description



This is a simple, single load step, structural analysis of a cantilever beam. The left side of the cantilever beam is fixed while there is a distributed load of 20N/m. The objective of this problem is to demonstrate ANSYS Workbench with a textbook problem, finding Von Mises' stresses and total deflection throughout the beam. The beam theory for this analysis is shown below:

Theory

Von Mises Stress

Assuming plane stress, the Von Mises Equivalent Stress can be expressed as:

$$\sigma' = (\sigma_x^2 - \sigma_x \sigma_y + \sigma_y^2 + 3\tau_{xy}^2)^{\frac{1}{2}}$$

Additionally, since the nodes of choice are located at the top surface of the beam, the shear stress at this location is zero.

$$(\tau_{xy} = 0, \ \sigma_y = 0).$$

Using these simplifications, the Von Mises Equivelent Stress from equation 1 reduces to:

$$\sigma' = \sigma_x$$

Bending Stress is given by:

$$\sigma_{\rm x} = -\frac{\omega(L-x)^2 c}{2I}$$

Where I = $\frac{1}{12}$ bh³ and c = $\frac{h}{2}$. From statics, we can derive:
 $\sigma_{\rm x} = \frac{-3\omega(L-x)^2}{bh^2} = 72.6 k Pa$

Beam Deflection

As in module 1.1, the equation to be solved is:

$$\frac{\mathrm{d}^2 \mathrm{y}}{\mathrm{d}\mathrm{x}^2} = \frac{\mathrm{M}(\mathrm{x})}{\mathrm{EI}}$$

Plugging in equation 1.3.5, we get:

$$EI\frac{d^2y}{dx^2} = \frac{w}{2}(2Lx - L^2 - x^2)$$

Integrating once to get angular displacement we get:

 $EI\frac{dy}{dx} = \frac{w}{2}(L\frac{x^2}{2} - xL^2 - \frac{x^3}{3}) + C_1$ At the fixed end (x=0), $\theta(0) = \frac{dy(0)}{dx} = 0$, thus $C_1 = 0$ $EI\frac{dy}{dx} = \frac{w}{2}(L\frac{x^2}{2} - xL^2 - \frac{x^3}{3})$ Integrating again to get deflection: $EIy = \frac{w}{2}(L\frac{x^3}{3} - \frac{x^2}{2}L^2 - \frac{x^4}{12}) + C_2$ At the fixed end.y(0)= 0 thus $C_2 = 0$, so deflection ($\delta = y$) is:

$$\delta = \frac{wx^2}{24EI} (4Lx - 6L^2 - x^2)$$

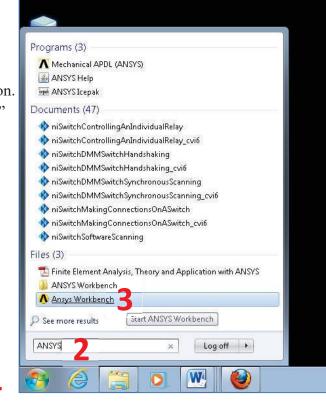
The maximum displacement occurs at the point load(x=L)

$$\delta_{max} = -\frac{\mathrm{wL}^4}{\mathrm{8EI}} = 6.27\mathrm{mm}$$

Workbench Analysis System

Opening Workbench

- 1. On your Windows 7 Desktop click the **Start** button.
- 2. Under Search Programs and Files type "ANSYS"
- 3. Click o ANSYS Workbench to start workbench. This step may take time.



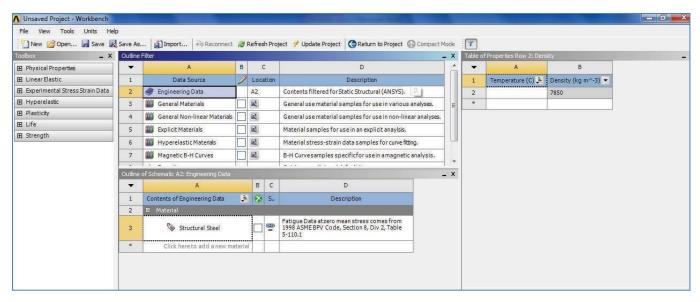
Static Structural Analysis

- 1. As you open ANSYS you can see the entire array of problems on the left had side this software can help you solve. The problem at hand is a *Static Structural* problem. Double click **Static Structural (ANSYS)** to open the task manager for your problem set in the Project Schematic area.
- 2. ANSYS allows you to build on each problem, so it is smart to name each project. At the bottom of the task manager you will see **Static Structural (ANSYS)**, double click this to change the name. For this problem choose "*1D Cantilever beam*."

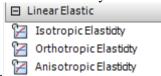


Engineering Data

To begin setup for your cantilever beam, double click or right click on *Engineering Data* and click *edit*. This will bring up another screen.



This new window will allow you to alter the material properties of your cantilever beam. Under **Outline of Schematic A2: Engineering Data**, it shows *click here to add a new material*, this menu allows you to input the material of your cantilever beam, double click and type *Aluminum*. Now expand *Linear Elastic* by double clicking on E Linear Elastic or on the plus



symbol shown.

Double click on Isotropic Elasticity to give the material the same properties across the beam. This action brought up a new table on the right; this allows us to add necessary properties. As show on the top right of the screen in *Table of Properties Row 2: Isotropic Elasticity*:

- 1. Click in Temperature and type 25
- 2. Click in Young's Modulus and type 70E9 or 7E10
- 3. Click in Poisson's Ratio and type 0.33

Table of	Table of Properties Row 2: Isotropic Elasticity						
•	А	В	с				
1	Temperature (C) 📮	Young's Modulus (Pa) 🔻	Poisson's Ratio				
2	25	7E+10	0.33				
*							

WARNING: Make sure to DELETE the Temperature entry after property input before **continuing!** Failure to do so will lead to errors later.

	-	A	В	с	
Delete!	1	Temperature (C) 📮	Young's Modulus (Pa) 🔻	Poisson's Ratio	Bulk Mod
Defete	2	1	7E+10	0.33	6.8627E+
	*				

After filling in the properties, this concludes the Engineering Data, to return to the project schematic area, click on **Return to Project** seen on the upper tab.

Geometry

Right click on the geometry \rightarrow *Properties* to bring up *Properties of Schematic A3: Geometry*. Expand *Basic Geometry Options*, the only change to make is uncheck *Parameters* and check *Line Bodies*. Your Table should be identical to the one provided below:

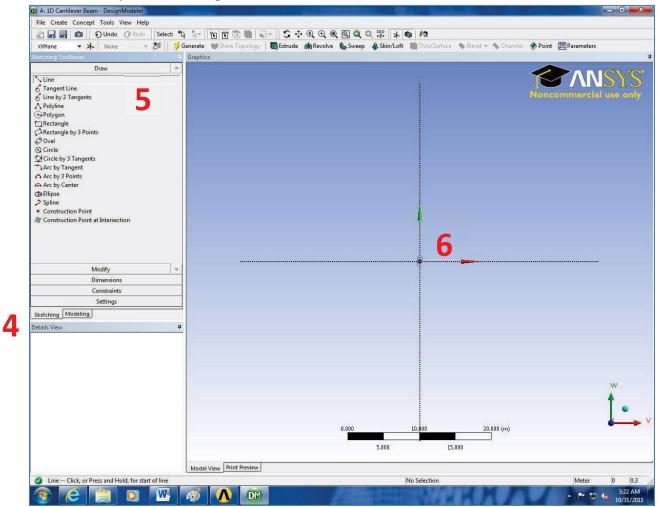
•	A	8		
1	Property	Value		
2	General			
3	Cell ID	Geometry		
4	Geometry Source	a		
5	Geometry File Name			
6	Basic Geometry Options			
7	Solid Bodies			
8	Surface Bodies			
9	Line Bodies			
10	Parameters			
11	Attributes			
12	Named Selections			
13	Material Properties			
14	Advanced Geometry Options			
15	Analysis Type	3D		
16	Use Associativity			
17	Import Coordinate Systems			
18	Import Work Points			
19	Reader Mode Saves Updated File			
20	Import Using Instances			
21	Smart CAD Update			
22	Enclosure and Symmetry Processing			
23	Mixed Import Resolution	None		

Base Geometry

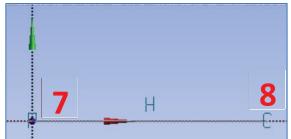
- 1. Go to **Workbench -> Project Schematic -> Geometry** and double click. This will open a new window for *ANSYS Design Modeler* where the Geometry will be created.
- 2. In the new window, click the **Display Plane** icon to toggle the coordinate system.
- 3. Go to **Design Modeler -> Tree Outline ->** right click on XYPlane. Click **Look At** to view the xy plane.

A: 1D Cantilever Beam - DesignModeler	A REAL PROPERTY AND A REAL		
ile Create Concept Tools View Help			
🖞 🔚 🛃 📫 🗍 🐑 Undo 📿 Redo 🗍 Select:	*\$ \$- \$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$	2 🛱 🔺 🖤 🕅	
(YPlane 🔻 🔺 🛛 None 👻 🗍	ジ Generate 🖤 Share Topology 🖪 Extrude 🛭 🚓 Revolve 🛭 🌜 Sweep	🚯 Skin/Loft 📲 Thin/Surface 👒 Blend 🤻	🔨 💊 Chamfer 🛛 🛷 Point 😰 Parameters
	U Graphics		
→ A: 1D Cantilever Beam → XYPlans → XZPlan 12 Look at → X ZXPlan 10 Rename			

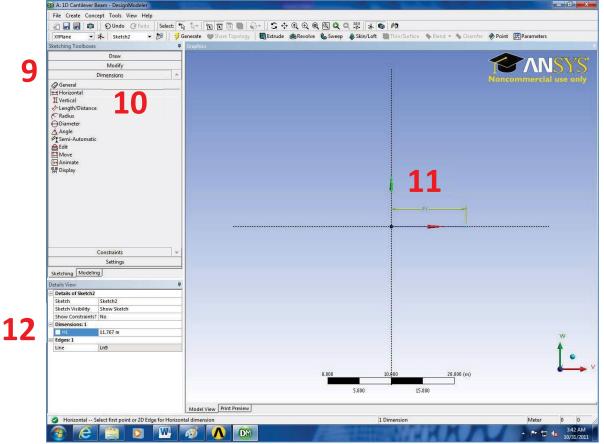
- 4. Go to Design Modeler -> Tree Outline -> Sketching
- 5. Click on Line:
- 6. Bring your cursor into the workspace at point 0,0, over the origin until 'P' appears directly above the origin.



- 7. Click on the origin to start the line and.
- 8. As it follows the x-axis a *C* will appear, click any point along this axis.



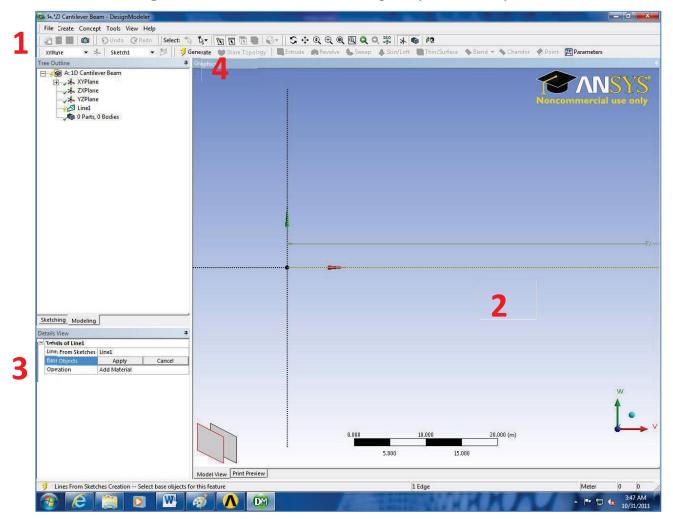
- 9. Go to Sketching Toolboxes -> Dimensions
- 10. Click Horizontal to specify a horizontal dimension.
- 11. In the workspace, click somewhere on the y axis and then the endpoint of your line segment. A green line with a symbol should appear.
- 12. Go to **Detail View -> H1.** In the first subcategory, replace the current dimension with 110. This is the length of your beam.



Now that we have modeled the base geometry, we will model the beam as a 1D surface with an area.

Surface from Sketch

- 1. Go to **Design Modeler -> Concept -> Lines From Sketches** to make this a line body.
- 2. Click your line segment, this will turn it yellow.
- 3. Go to **Detail View -> Base Objects -> Apply**.
- 4. Go to **Design Modeler ->** Click **Generate** to update your line body.



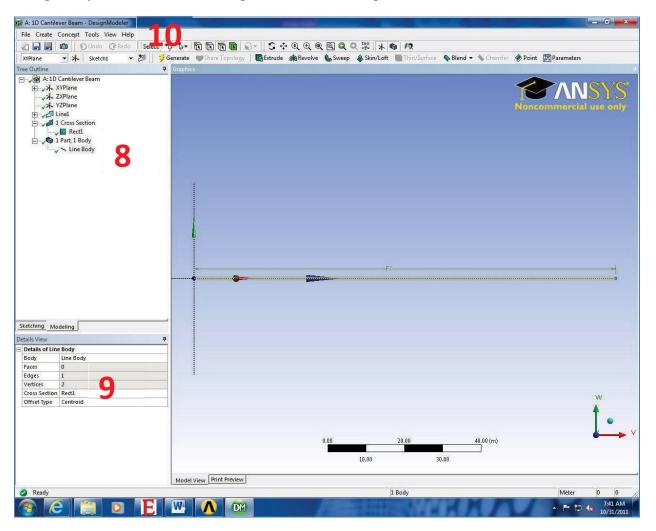
- 5. Go to Design Modeler -> Concept -> Cross Section -> Rectangular
- 6. Go to **Detail View -> Dimensions 2 -> B**. In the first subcategory, replace the current dimension with 1.
- 7. Go to **Detail View -> Dimensions 2 -> H**. In the first subcategory, replace the current dimension with 01.

	Concept Tools View Help									
	🖾 🛛 🔊 Undo @Redo 🗍	Select: *>		8-115 + 0	+ Q Q Q	Q 150 🗼 🕲	10			
								lend 🔻 🌭 Chamfer	Point 🛐 Parameter	·c
Tree Outline	· Ma Sketchi	Graphic		Carterance 040	nevolve Sourcep	Skill/Loit	Transformere -	icita de la citalitat	Witchite [B3] and increase	
	C (1) B	Graphic	S .		_					
	ZXPlane YZPlane								Noncommerci	NSY al úse o
Sketching Mo	adalian									
Sketching Mo	odeling									
Details View		-	.							
Details View	: 1		-							
Details View Details of Rect Sketch	ti Recti									
Details View Details of Rect Sketch Show Constra	t1 Rect1 aints? No									
Details View Details of Rect Sketch Show Constra Dimensions: 2	t1 Rect1 aints? No 2									
Details View Details of Rect Sketch Show Constra Dimensions: 2 B	t1 Rect1 aints? No 2 10 m									
Details View Details of Rect Sketch Show Constra Dimensions: 2 B H	t1 Rect1 aints? No 2									
Details View Details of Rect Sketch Show Constra Dimensions: 2 B H Edges: 4	t1 Rect1 aints? No 2 10 m									w
Details View Details of Rect Sketch Show Constra Dimensions: 2 B H Edges: 4 Line	t1 Rect1 aints? No 2 10 m 1 m									
Details View Details of Rect Sketch Show Constra Dimensions: 2 B H Edges: 4 Line Line	tti Recti asints? No 2 10 m 1 m Ln10 Ln11									
Details View Details of Rect Sketch Show Constra Dimensions: 2 B H Edges: 4 Line Line Line	tti Recti aaints? No 2 10 m 11 m Linio									
Details View Details of Rect Sketch Show Constra Dimensions: 2 B H Edges: 4 Line Line Line Line Line Line	tti Recti aanta 2000 2000 2000 2000 2000 2000 2000 20				0.00	51	90.00	1000.000 (m)		
Details View Details of Rect Sketch Show Constra Dimensions: 2 B H Edges: 4 Line Line Line	tti Recti aanta 2000 2000 2000 2000 2000 2000 2000 20				0.00			1000.00 (m)		¥.
Details View Details of Rect Sketch Show Constra Dimensions: 2 B H Edges: 4 Line Line Line Line Physical Prope	Recti aints? No 2 10 m 1 m im Ln10 Ln11 Ln12 Ln12 Ln13 im				0.00	51	00.00 750.00			ţ.
Details View Details of Rect Sketch Show Constra Dimensions: 2 B H Edges: 4 Line Line Line Line Line Physical Prope (A)	t1 Rect1 ints? No 2 10 m i m int. Int. Ln10 Ln11 Ln12 Ln13 nerties: 10 10 m ²	E			0.00			1000.00 (m)		ě.
Details View Details of Reci Sketch Show Constra Dimensions: 2 B H Edges: 4 Line Line Line Line Line Line Line Line	tti Recti anno 2000 Recti anno	E	View Print Preview		0.00			1000.00 (m)		¥.

- 8. Go to Design Modeler -> Tree Outline -> 1Part, 1 Body -> Line Body
- 9. Go to Detail View -> Cross Section -> select Rect1

10. Go to Design Modeler -> View -> check Cross Section Solids

This concludes Geometry, exit out of the window and back to the Project Schematic. Before doing this, you should have an image similar to the one provided below.



Model

While in the Project Schematic screen double click **Model** This will open a new screen.

▼	А	
1	😇 Static Structural (ANSYS)	
2	Engineering Data	
3	Geometry	× .
4	🞯 Model	1
5	🍓 Setup	? .
6	Solution	7 🖌
7	😥 Results	1
	Cantilever beam	

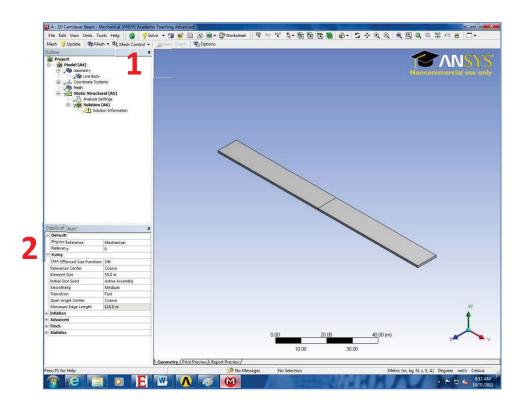
Material

- 1. Go to Design Modeler -> Outline -> Geometry -> Line Body
- 2. Go to Detail of "Line Body" -> Assignment -> Aluminum

+ Graphics Properties	
- Definition	
Suppressed	No
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Offset Mode	Refresh on Update
Offset Type	Centroid
- Material	
Assignment	Structural Steel
Nonlinear Effects	Yes Aluminu
Thermal Strain Effects	Yes
+ Bounding Box	
+ Properties	
+ Statistics	

Mesh

- 1. Go to Design Modeler -> Outline -> Mesh
- 2. Go to Detail of "Mesh"-> Sizing -> Element Size. Replace the current dimension with 55. This sets one element every 55m along the cantilever beam.
- 3. Go to Design Modeler -> Click Update. This will update your geometry will the designated elements.



WARNING: The element size will lead to incorrect results as we will explore in the 'Results' section (page 16)

Exit out of the Model screen to the Project Schematic.

<u>Setup</u>

While in the Project Schematic double click Setup

This will open a new window similar to Model Space

Loads

Click the x-axis icon to get a side view of the cantilever beam

1) Fixed end

On the tool bar, make sure vertex option is selected.

Click the left side of the geometry, this will add a

green box to select the point.

Right click ? Static Structural (A5),

Click insert, and 🔍 Fixed Support

This will add a fixed end to your cantilever beam

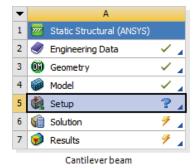
in the work space.

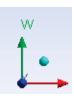
2) Distributed Load

On the tool bar, change selection option to

edge instead of vertex.

Click on the geometry, this will highlight

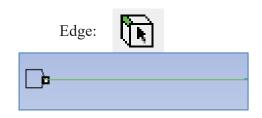












the cantilever beam.

Right click? Static Structural (A5), click insert, and Line Pressure A table will appear "Details of Line Pressure" Under "Definition" you will see "Definied by" Change this to "Components"

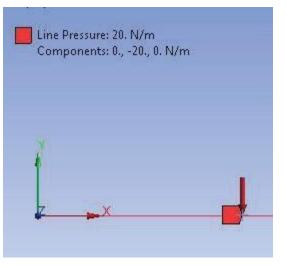
As shown, Y Component force is zero.

Change this to value to -20

This will show your cantilever beam with a load applied as shown.

Leave the Setup screen open this time.

De	Details of "Line Pressure"							
Ξ	Scope							
	Scoping Method	Geometry Selection						
	Geometry	1 Edge						
Ξ	Definition							
	Туре	Line Pressure						
	Define By	Components						
	Coordinate System	Global Coordinate System						
	X Component	0. N/m (ramped)						
	Y Component	0. N/m (ramped)						
	Z Component	0. N/m (ramped)						
	Suppressed	No						
	X Component	0. N/m (ramped)						
	📃 Y Component	-20. N/m (ramped)						
	Z Component	0. N/m (ramped)						



Solution

Go to Mechanical -> Outline -> Project -> Model(A4) -> Static Structural (A5) -> Right Click Solution (A6) -> Insert -> Beam Tool

Deformation

Go to Mechanical -> Outline -> Project -> Model(A4) -> Static Structural (A5) -> Solution (A6) -> Beam Tool -> Insert -> Beam Tool -> Deformation -> Total

Static Structural Analysis Settir Sixed Support Suite Pressure Solution (Ad	ngs 5) Information		11		
🗄 👾 🏂 Beam To	Insert	Probe	•		
		Beam Tool	•	Stress	•
	B Duplicate	🕵 User Defined Result		Deformation	0
	■ Copy 从 Cut	Commands	_		[●] d Directional
	 ∠ Clean X Delete a] Rename 				

Stress

Go to Mechanical -> Outline -> Project -> Model(A4) -> Static Structural (A5) -> Solution (A6) -> Beam Tool -> Insert -> Beam Tool -> Stress -> Maximum Bending Stress

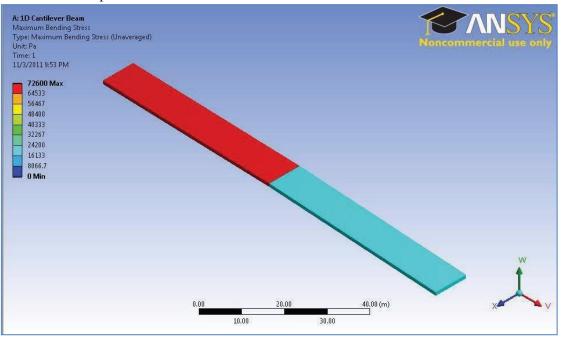
am Tool Insert	Probe	•			
芺 Solve	Beam Tool	•	Stress	•	💁 Direct Stress
Puplicate	👷 User Defined Result		Deformation	+	💁 Minimum Bending Stress
B Copy					💁 Maximum Bending Stress
K Cut	Commands				💁 Minimum Combined Stres
do Cui		_			💁 Maximum Combined Stres
Clean					
🗙 Delete					
allo Rename					

Now that our solvers have been defined, go to **Mechanical** -> **Solve**. The calculations in Workbench may take up to a minute to solve.

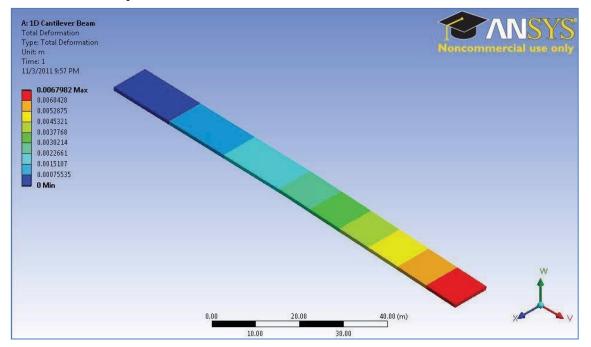
Go to Mechanical -> Outline -> Project -> Model (A4) -> Solution (A6) -> Maximum Bending Stress

Go to **Details of "Maximum Bending Stress"-> Integration Point Results -> Display Option ->** Change to **Unaveraged**

Your Stress plot should look as shown below:



Go to Mechanical -> Outline -> Project -> Model(A4) -> Solution(A6) -> Total Deformation



Your deflection plot should look as shown below:

Results

Max Deformation Error

According to equation 1.3W.9, the theoretical max deflection is 6.27 mm. The percent error (%E) in our model can be defined as:

$$\% E = EEE \left(\underbrace{\overset{\delta the Ereti EEE - \delta EEE eE}{\delta_{the Ereti EEE}}} \right) * 100 = 8.42\%$$
(1.3W.13)

This error is due to that fact, the first mesh was coarse. The 1D elements used interpolate between the nodes in the elements to estimate the total deflection. Since there are two elements, a node at the fixed end, middle, and end point, there is an expected degree of truncation error. In the validation section, our model will converge to the expected solution with a finer mesh, proving mesh independence.

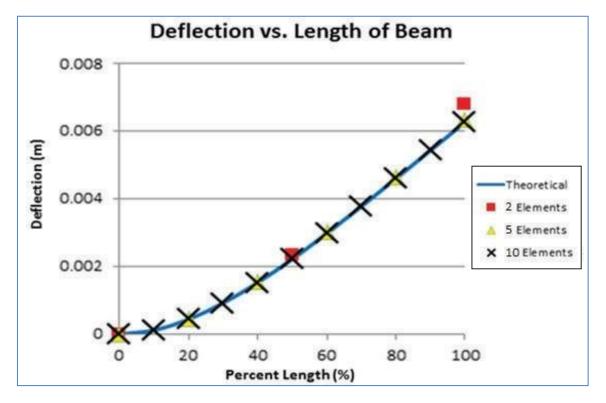
Max Bending Stress Error

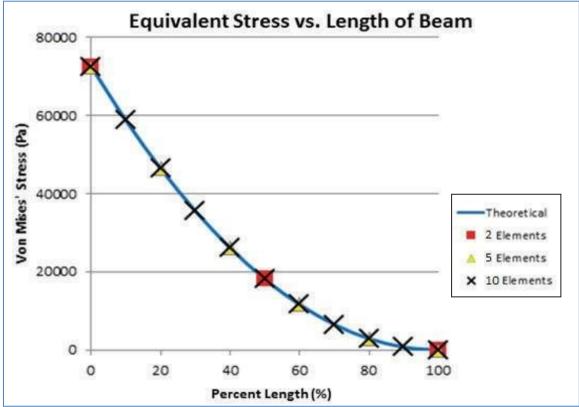
According to equation 1.3W.6, the theoretical max stress is 72.6kPa. The percent error (%E) in our model can be defined as:

$$\% E = EEE \left(\frac{\delta the Ereti EEE - \delta EEE eE}{\delta_{the Ereti EEE}}\right) * 100 = 0\%$$
(1.3W.14)

According to equation 1.3W.6, the theoretical max equivalent stress is 72600 Pa. Using the same definition of error as before, we derive that our model has **0%** error. Even with an extremely coarse mesh like 2 elements, there is no error in the beam.

Validation

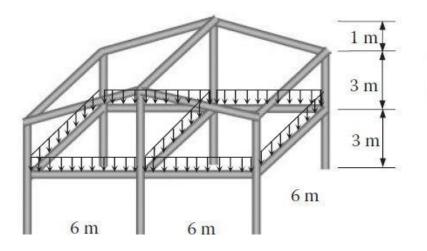




Ex. No : Date :

APPLICATION OF DISTRIBUTED LOADS

Problem Description: Steel framing systems provide cost-effective solutions for low-rise buildings. They have high strength-to-weight ratios, and can be prefabricated and custom designed. Consider the following two-storey building constructed with structural steel I-beams. Determine the deformations and the stresses in the frame when a uniform load of 50 kN/m is applied on the second floor as shown below.



Material: Structural steel *Line pressure:* 50 kN/m *I-beam size:* W356 × 171 Beam depth = 355.6 mm Flange width = 171.5 mm Web thickness = 11.5 mm Flange thickness = 7.3 mm

Solution

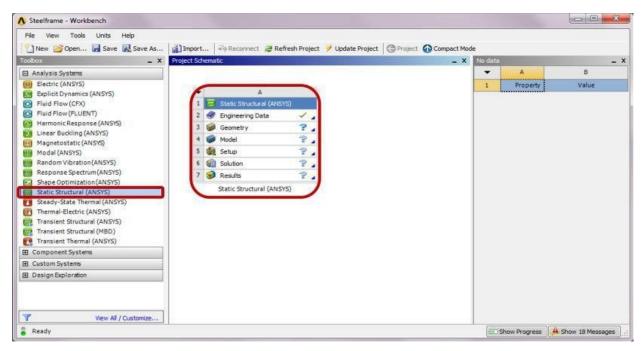
To solve the problem with ANSYS® Workbench, we employ the following steps:

Step 1: Start an ANSYS Workbench Project

Launch ANSYS Workbench and save the blank project as "Steelframe.wbpj."

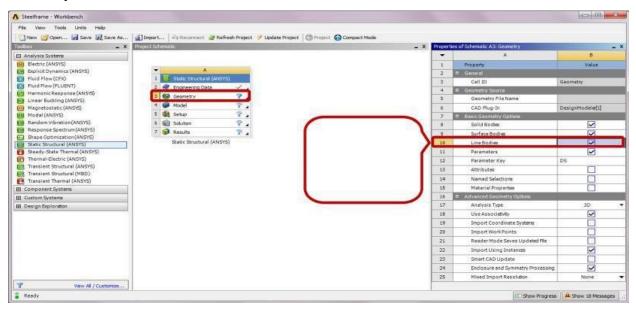
Step 2: Create a Static Structural (ANSYS) Analysis System

Drag the Static Structural (ANSYS) icon from the Analysis Systems Toolbox window and drop it inside the highlighted green rectangle in the Project Schematic window to create a standalone static structural analysis system.



Step 3: Launch the DesignModeler Program

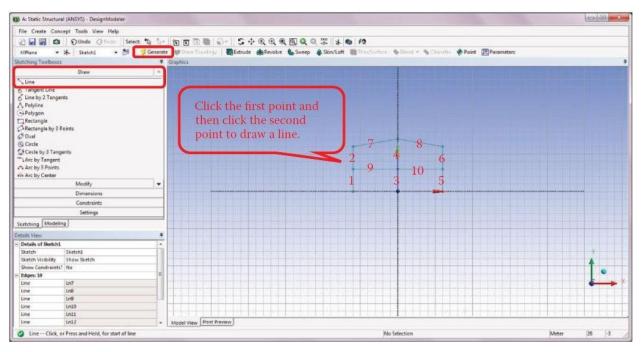
Double-click the Geometry cell to launch DesignModeler, and select "Meter" as length unit in the Units pop-up window. Ensure Line Bodies is selected in the Properties of Schematic A3: Geometry window.



Step 4: Create Line Sketch

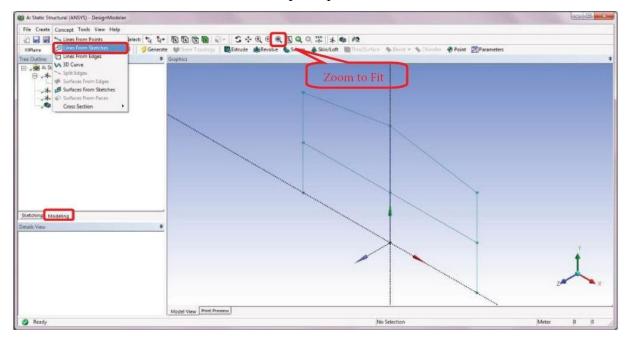
Click the Sketching tab and select Settings. Turn on Show in 2D and Snap under Grid options. Use the default value of "5 m" for Major Grid Spacing and "5" for Minor-Steps per Major. Click

a start point and then an end point in the Graphics window to draw a line. Draw 10 lines as shown in the sketch below. After completion, click Generate to create a line sketch.

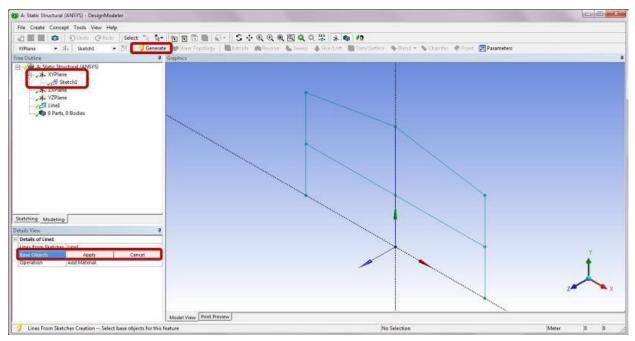


Step 5: Create Line Body from Sketch

Check off the Grid options under Settings of Sketching Toolboxes. Switch to the Modeling tab. Note that a new item named Sketch1 now appears underneath XYPlane in the Tree Outline. Select Lines from Sketches from the Concept drop-down menu. Click Zoom to Fit.

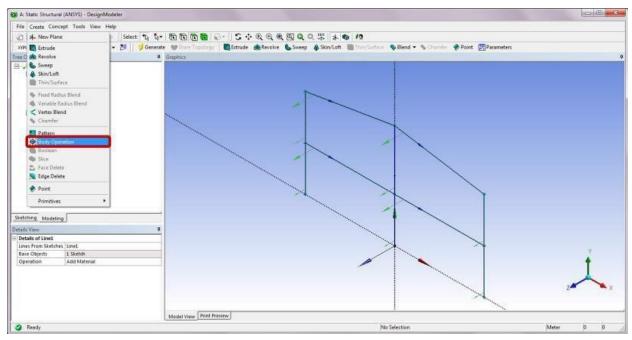


Select Sketch1 from the Tree Outline and click Apply to confirm on the Base Objects selection in the Details of Line1. Click Generate to complete the line body creation.

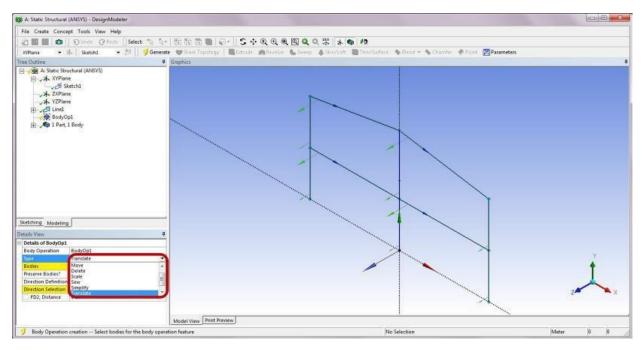


Step 6: Create Line Body through Translation

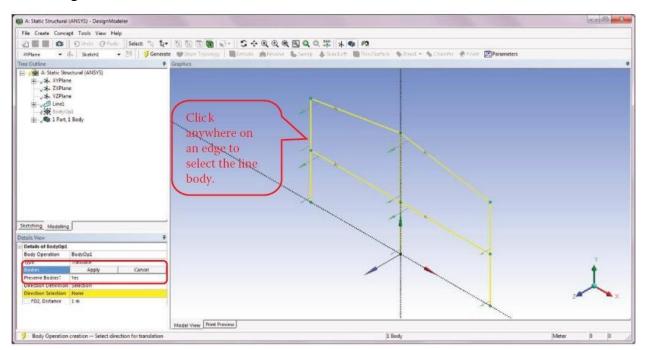
Select Body Operation from the Create drop-down menu. A new item named BodyOp1 is now added to the Tree Outline.



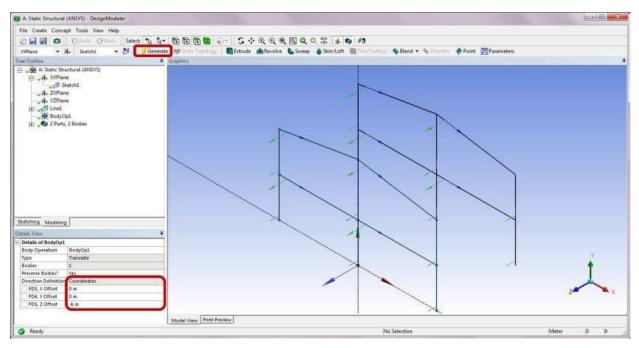
In the Details of BodyOp1, click anywhere on the Type cell and select Translate from the dropdown menu.



Select the line body from the Graphics window and then click Apply to confirm on the Bodies selection in the Details of BodyOp1. After completion, change the Preserve Bodies? selection to Yes. This will help preserve a copy of the selected line body at the current location while translating it to a new location.

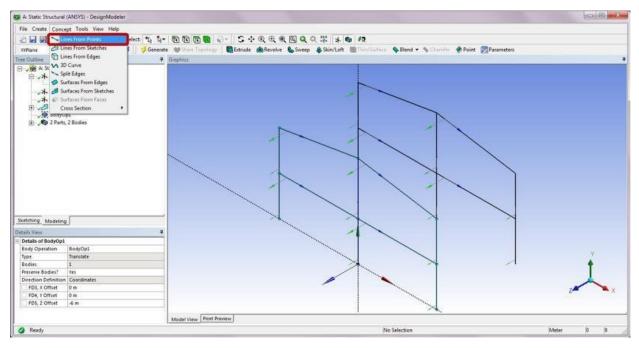


In the Details of BodyOp1, change the Direction Definition to Coordinates, and enter "-6" for the Z Offset. Click Generate. After completion, the line body will be copied backward by 6 m.

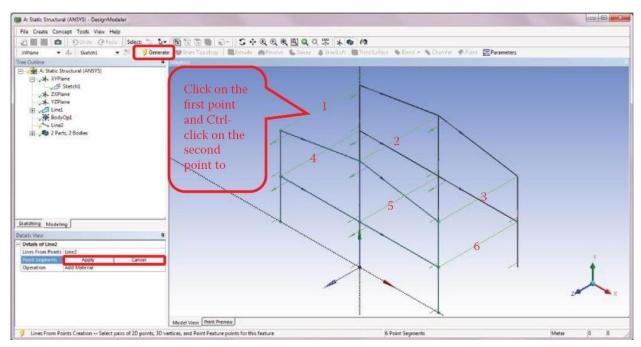


Step 7: Create Line Body from Points

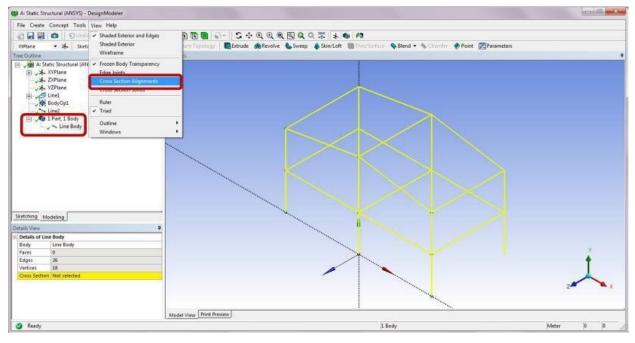
Select Lines From Points from the Concept drop-down menu. After completion, a new item named Line2 is added to the Tree Outline.



In the Graphics window, select a start point and Ctrl-select an end point to draw a line. Draw six new lines connecting the two planar frames as shown below. Click Apply to confirm on the Point Segments selection in the Details of Line2. Click Generate to complete the line creation.

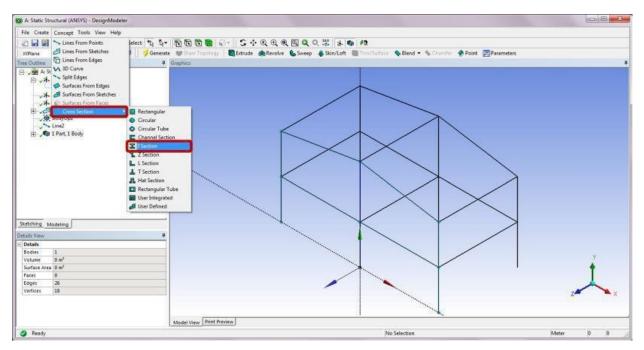


Note that the line bodies created in all previous steps now merge into a single line body. Check off the Cross Section Alignments from the View drop-down menu to switch-off the display of local coordinate systems.

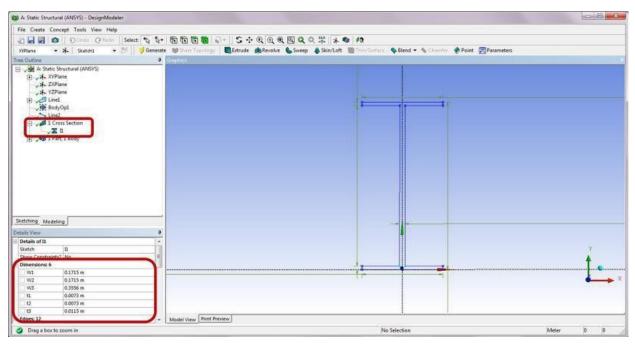


Step 8: Create a Cross Section

Select a Cross Section of I Section from the Concept drop-down menu. A new item named I1 is now added underneath the Cross Section in the Tree Outline.

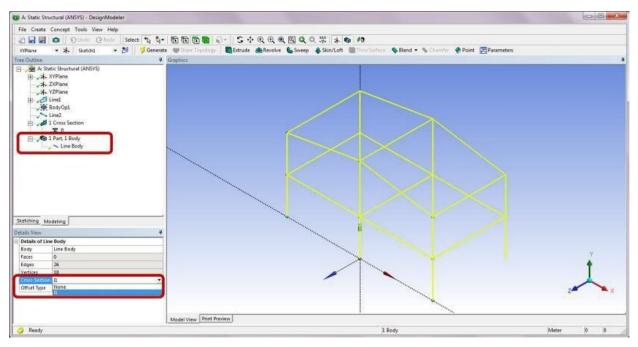


In the Details of I1 under Dimensions, enter "0.1715" for W1 and W2, "0.3556" for W3, "0.0073" for t1 and t2, and "0.0115" for t3.

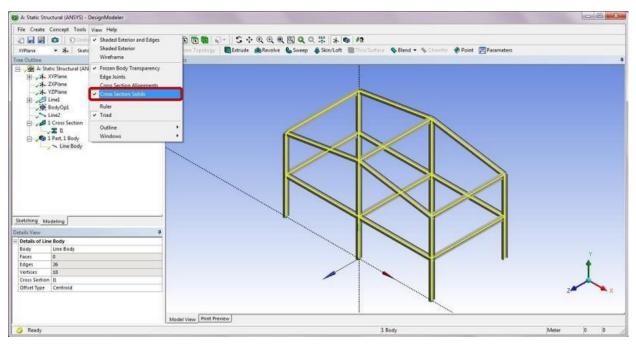


Step 9: Assign Cross Section to Line Body

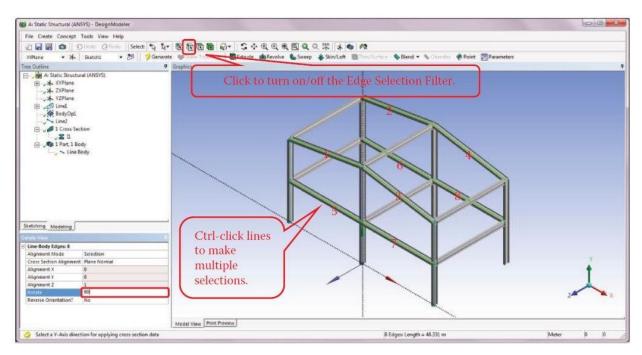
Select the Line Body underneath 1Part, 1 Body in the Tree Outline. In the Details of Line Body, assign I1 to the Cross Section selection.



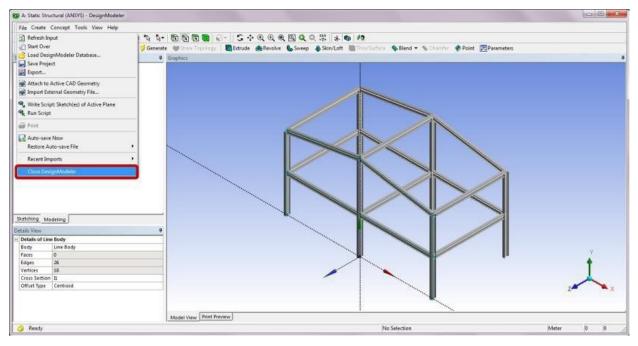
Turn on the Cross Section Solids from the View drop-down menu to view the frame as a solid structure.



Note that some I-beams in the above structure are used as H-beams. To fix the misaligned cross sections, turn on the Edge Selection Filter and select the eight line edges shown below from the Graphics window. In the Details of Line-Body Edges, enter "90" for Rotate to turn the beams 90° about their neutral axes.

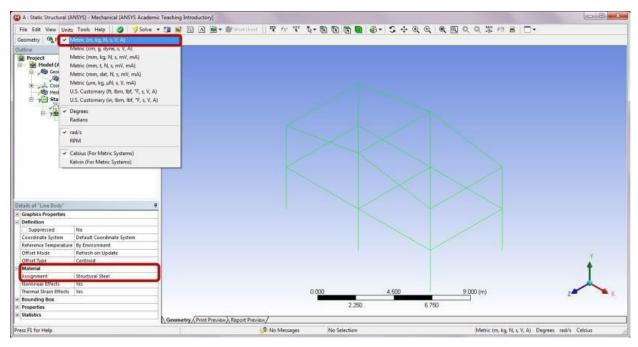


The adjusted frame shown below now has all the I-beams oriented in the strong axis configuration. This completes the geometry creation of a frame structure. Click Close Design Modeler to exit the program.



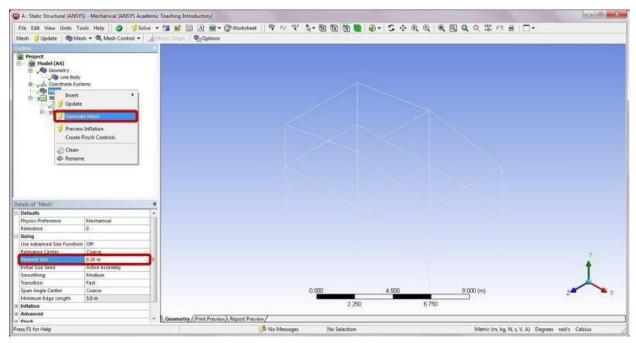
Step 10: Launch the Static Structural (ANSYS) Program

Double-click the Model cell to launch the Static Structural (ANSYS) program. From its Units drop-down menu, select Metric (m, Kg, N, s, V, A). Note that in the Details of "Line Body" the material is assigned to Structural Steel by default.



Step 11: Generate Mesh

In the Details of "Mesh," enter "0.2 m" for the Element Size. In the Outline of Project, rightclick on Mesh and select Generate Mesh.

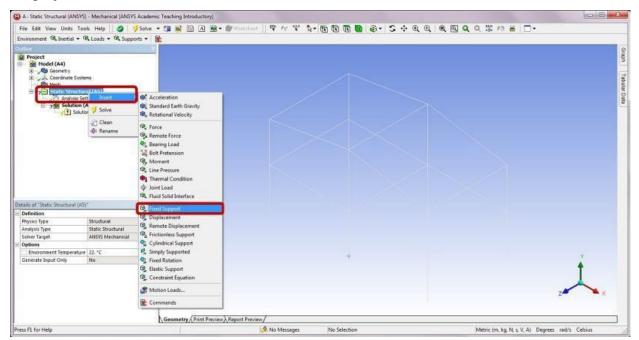


After completion, the meshed structure appears in the Graphics window. You may deselect the Ruler from the View drop-down menu to turn off the ruler display in the Graphics window.

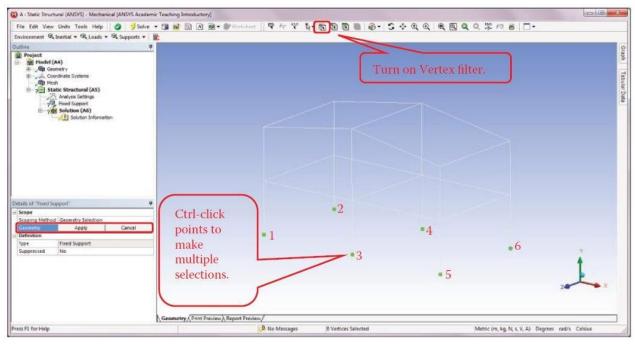
		lemic Teaching Introductory]	
File Edit View Units T	Tools Help 🛛 🥥 💖	ne + 🗊 😹 🔟 🛆 🞯 + 🕼 Worksheet 🛛 🕾 fr 🚏 💱 🏷 🛅 🛅 🗑 🗑 🗑 🗳 S 💠 Q Q Q Q Q	第 /2 8 0.
Mesh 🔰 🗸 🛩 Shaded Ext Suttine Shaded Ext Project Wireframe	terior	ad Metchic Graph ♥2 Options	
□ 🙆 M → A → Thick Shell	is and Beams		
Wisual Expa	insion (Beta)	A MARKEN AND A MARKEN	
Annotation			
🗄 💦 Custom An	nnotations		
Rulet			
✓ Legend			
🗸 Triad			
Eroded No.	des	Provide a state of the state of	
✓ Annotation	n Labels		
	22		
Outline	•		
Toolbars	•	Better Broke Made	All and a second
Windows			AL
Details of "Mesh"			
- Defaults	The state of the state		28
Physics Preference	Mechanical		RULE E
Relevance	0		
- Sizing			E .
Use Advanced Size Functio	on Off		10
Relevance Center	Coarse		
Element Size	0.20 m		
Initial Size Seed	Active Assembly		+
Smoothing	Medium		
Transition	Fast		A
Span Angle Center	Coarse		7 1 1
Minimum Edge Length	3.0 m		
+ Inflation	States and the		
+ Advanced			
+ Dinch		* \Geometry \Print Preview \Report Preview /	
Press F1 for Help		9 No Messages No Selection Met	tric (m, kg, N, s, V, A) Degrees rad/s Celsius

Step 12: Apply Boundary Conditions

In the Outline of Project, right-click on Static Structural (A5) and select Insert and then Fixed Support. After completion, a Fixed Support item is added underneath Static Structural (A5) in the project outline tree.

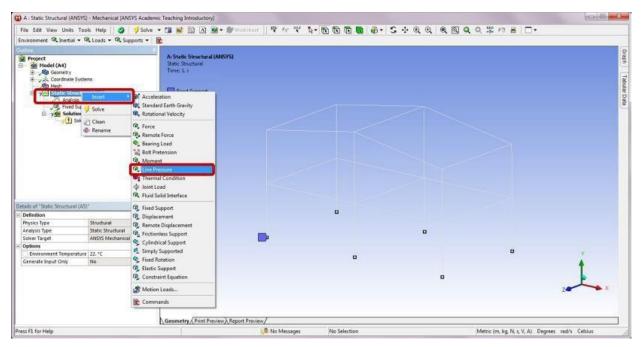


Select the six points as shown below in the Graphics window. In the Details of "Fixed Support," click Apply to confirm on the Geometry selection. After completion, a Fixed Support boundary condition will be added to the selected six points.



Step 13: Apply Loads

In the Outline of Project, right-click on Static Structural (A5) and select Insert and then Line Pressure.



Select the line as shown below in the Graphics window. In the Details of "Line Pressure," click Apply to confirm on the Geometry selection.

🚳 A : Static Structural (ANSVS) - Mechanical (ANSVS Academic T	aching Introductory]	
File Edit View Units Tools Help 🛛 🍏 Solve 💌	🕻 🖄 🗷 🖉 + 🕼 Watsat 🛛 🗣 🖓 💱 🗞 🕲 🔞 🕲 🚳 - S 수 Q Q 🔍 🔍 🕲	9.9.2 /2 8 0-
Environment R Inertial + R Loads + R Supports +		
Cardination of Learning of Lea		
Details of "Line Pressure" # Scope Scope Scoping Method Geometry Selection Cancel Orderation Apply Orderation Orderation Type Line Pressure Outries by Vector Maghtude Direction Clink to Detine Suppressed No	Click a line to make a selection.	, zotex
	eametry / Print Dreview / Report Dreview /	
Press F1 for Help	0. No Messages 1 Edge Selected: Length = 6. m	Metric (m, kg, N, s, V, A) Degrees rad/'s Celsius

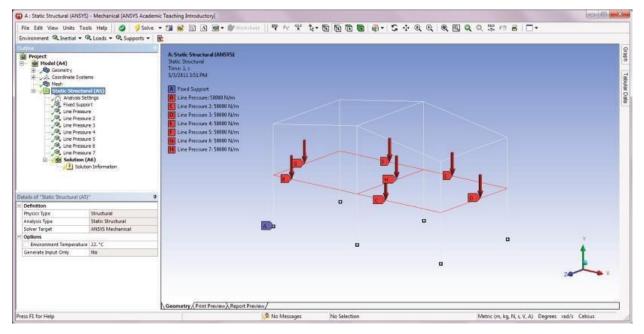
In the Details of "Line Pressure," change the Define By selection to Components and enter "-50000" for the Y Component. A downward red arrow will appear on the selected line in the Graphics window.

Environet & betal + Q Lods + Q Logs + Q Lods + Q Logs + Q Lods + Q Logs + Q	File Edit View Units Tools Help 🛛 🥥 🥩 Solve 🕶	· 御 😹 🔟 Δ 🕮 • #Weathart 聖 ケン 聖 str 簡 簡 簡 🔞 🕹 • ち 수 Q Q 奥 図 Q Q 黒 ノ2 岳 〓 +	
Project Image: Comparison of the system Image: Comparent () Image			
Implementation Solution (Ar) Solution (Ar) <t< th=""><th>Duffine 0</th><th></th><th></th></t<>	Duffine 0		
Image: Model (A) B - Goodrade System Station (A) Component: B - Goodrade System	DI Project		
Advance Market State Statemark Advance Statemark Advance Statemark Advance Statemark States (Advance) States (Advance) States (Advance) States (E Model (A4)		
Methem Methem Advises Standbard (AS) Advises Standbard (AS) Advises Standbard (AS) Solution (AS) Solution (AS) Solution (AS) Solution (AS) Solution (Infernation Sope Solution (AS) It large Infernation Perfultion Resultion (Signed Space) Sopen Muthod Geometry Stateding It large Infernation Perfultion Resultion (Signed Space) Sopen States (Sobol Not Infernation) It large Perfultion Resultion (Signed Space) Sopen States (Sobol Not Infernation) Resultion (Signed Space) Component 0. Not Impredient Sobol Not Impredient Sopen States (Sobol Not Impredient Space) Sobol Not Impredient Space) Component 0. Not Impredient (Sobol Not Impredient Space) Sobol Not Impredient Space) Sopensition 0. Not Impredient (Sobol Not Impredient Space) Sobol Not Impredient (Sobol Not Impredient Space) Sopensition 0. Not Impredient (Sobol Not I	E Geometry	Time: 1.3	
Analysis Statings Find Support Solution (A) Solution Information Scope Sco	H Coordinate Systems		
Analysis Settings Find Support Solution (A6) Solution (A6) Solution (A6) Socie Second Method Geometry Setection Geometry 1 Edge Definition The Mine Second System X Component Consider System X Component Consider System Component Consider System Consider System X Component Consider System X Consider System X Consider System X Consider System X Consider System X Cons	(a) Static Structural (AS)	Line Pressure: S1010 N/m	
Solution (Ab) Solution Information Stable of Lippe Directoure Scope Scope Scopenty I Edge Definition Term Edge Constructor Stopport Component Other System Component Other System Component Other System Scope Scopent Component Other System Scope Stoppent Other System Scope Stoppent Other System Scope Stoppent Component No	Analysis Settings	Components: 0, -50100, 0, N/m	
Solution (A6) Composition (A6) Solution Information Participation Componention Componention Componention Componention Solution Information Solution Information Solution Information Componention Componention Componention Solution Information Solution Information Solution Information <td< td=""><td>- Fixed Support</td><td></td><td></td></td<>	- Fixed Support		
Interference 0 Scope Scope Scope	, C. Line Pressure		
Stope 0 Scope 0 Scope 0 Scope 0 Scope 0	Solution (A6)		
Scope	Solution Information		
Scope Scoping Mithod Geometry Selection Georetry 1 Loge Definition Type Definition Type Components Coordinate System X Component O Nim (samped) X Component O Nim (samped) X Component Scope No Nim (samped) X Component Scope Nim (samped) X Component Scope Nim (samped) X Component Scope Nim (samped) X Component Scope Nim (samped) X Component Scope Nim (samped) X Component Scope Nim (samped) Scope Nim (samped) Scope Scope Scope Scope Nim (samped) Scope Sco			
Scope Scoping Mithod Geometry Steledion Geometry 1 Loge Definition Type Definition Type Components Coordinate System X Components Constante System X Components O Nim (samped) X Components Scope Steledion A Nim (samped) X Components Scope Steledion A Nim (samped) X Components Scope Steledion Scope Steledion A Nim (samped) X Components Scope Steledion Scope Steledion Scope Steledion Steledi			
Scope Scoping Mithod Geometry Steledion Geometry 1 Loge Definition Type Definition Type Components Coordinate System X Components Constante System X Components O Nim (samped) X Components Scope Steledion A Nim (samped) X Components Scope Steledion A Nim (samped) X Components Scope Steledion Scope Steledion A Nim (samped) X Components Scope Steledion Scope Steledion Scope Steledion Steledi			
Scope Scope defined Geometry Stedion Gennetry I I dge Definition Type Define By Components Controlinet System X Component 0.NWn (amped) X Component			
Scope Scope defined Geometry Stedion Gennetry I I dge Definition Type Define By Components Controlinet System X Component 0.NWn (amped) X Component			
Scope Scope defined Geometry Stedion Gennetry I I dge Definition Type Define By Components Controlinet System X Component 0.NWn (amped) X Component			
Scope			
Scope + Scoping Method Geometry Selection Greenetry 1 Edge Definition - Type Line Pressure Define By Components Constrainte System O-Nim (Instrand) X Component			
Scoping Method Generity Stedion Generity I Edge Definition Par Une Pressure Continuers System X Components 0. N/m (samped) Component 0. N/m (samped) Component 0. N/m (samped) Component 0. N/m (samped) Suppressed No			
Geometry I I dage Definition Type Une Pressure Define by Component Coordinate System (Global Coordinate System) X Component O. N/m (smped) X Component O. N/m (smped) X Component Suppressed No		+	
Definition Type Line Pressure Component Control to Line Views Control to L			
Type Line Pressure Define By Components Coordinate System Components X Component 0.10/m (samped) X Component 0.10/m (samped) X Component 0.10/m (samped) Suppressed No		T T	
Define By Components Coordinate System X Component 0. Nrm (namped) X Component 0. Num (namped) Coordinate System X Component 0. Num (namped) Coordinate System X Component 0. Num (namped) Suppressed No			
Coordinate System X. Component: 0. RVm (sumped) X. Component: 0. RVm (sumped) Z. Component: 0. RVm (sumped) Suppressed No		Y Y	
X Composent 0. N/m (samped) X Composent - 4000 N/m (samped) C Composent 0. N/m (samped) Suppressed No			
Z Component -5000 N/m (insped) Z Component 0. H/m insped) Suppressed No		1	
Z Component IO. N/m. Itempedi Suppressed No			li in
Suppressed No			
			2.0
\Geometry_Print Preview_Report Preview/	Suppressed No		
\Geometry_{Print Preview_Report Preview/			
I comment/V kum sussiew/vielbur sussiew/		Generates / Denie Denies / Benerat Denies /	
ess F1 for Help No Messages No Selection Metric (m, kg, N, s, V, A) Degrees rad/s Celsius	ESTER COMPANY AND AND ADDRESS OF ADDRESS OF ADDRESS ADDRE		

Repeat the steps of adding a line pressure, and insert the second Line Pressure item underneath Static Structural (A5) in the Project Outline tree. Apply the same exact load to the selected line highlighted in the following figure.

Re Edit Viene Unda Tools Help Enversment Reventive Control Control <th>A : Static Structural (ANSYS) - Mechanical (ANSYS Academ)</th> <th>ic Teaching Introductory </th> <th>0.0.0</th>	A : Static Structural (ANSYS) - Mechanical (ANSYS Academ)	ic Teaching Introductory	0.0.0
Enveneers & benels + @ Loods + @ bageots + @	File Edit View Units Tools Help 🛛 🥥 🥳 Solve	- 加減部回用・2010年には、マケマない間面面にお・ちゃのの、ののののスパミロ・	
Outline Image: Imag			
Imposed Imposed (A)			
Contain of Line Presents Scappe Scap	Podel (A4) Coordinate Systems Coordinate Systems Coordinate Systems Static Structural (A5)	Line Pressure 2 Time: L s SY3/05113-849 PM	
Scape Scape Scape Scape Scape Scape Scape Scape Weight Stape Definition Type Une Pressure Scape	Solution (A6)		-
Scaping Method Geometry Selection Geometry 1 Edge Click on the line and apply it to the geometry Scaping Annotation System Scaping Method Constraints System Consident System Global Constraints System Scaping Annotation System Scaping Method Constraints	Cartain of "Low Pressons 7"		
Geometry 1 1699 Definition Proce Definition Considence System Consolute System Click on the line and apply it to the geometry selection of line Composed Suppressed No Click on the line and apply it to the geometry selection of line Composed Composed Composed Click on the line and apply it to the geometry selection of line Composed Composed Composed Click on the line and apply it to the geometry Selection of line Composed Click on the line and apply it to the geometry Selection of line			
se Verninon Type Unine Pressure Contrainte System Contrainte System			
Type Une Pressure Generative System Components Consponents Gibbal Coolinste System R Consponents Generative System Scomponent Generative System		Click on the line	
Consolence System Global Condust System See See Condust See See Condust System See See Condust System See Condust System See See Condust System See Condust See See Condust See See Condust See Condust See See Condust See Condust See See Condust See			
Coordinate System Global Coordinate System X Composed G. New Iranged Composed G. New Iranged Composed G. New Iranged Suppressed Na		and apply it to	¥.
X Composed 0. Nem (ramped) Scamposed Selection of line Suppressed No		the geometry	
Suppressed No			
Suppressed (Na Geometry / Finit Preview / Report Preview /		selection of line	•
Suppressed No Geometry (Print Preview) Report Preview /		selection of mic	ZA X
Geometry (Frint Preview) Report Preview/			1251 0100
	Notzingen - NV2	Generate (Print Provide) Broad Brasins /	
	Press F1 for Help	Cocometry Annu Annuel Applie Hereitery (() No Messages No Selection Metric (m, kg, N, s, V, A) Degrees	andle Cables

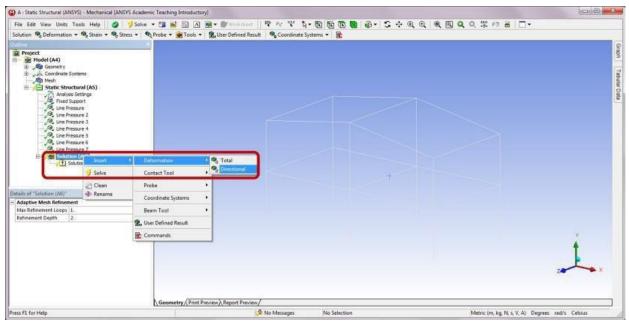
Repeat the steps until the same load is applied to all seven edges highlighted in the figure below.



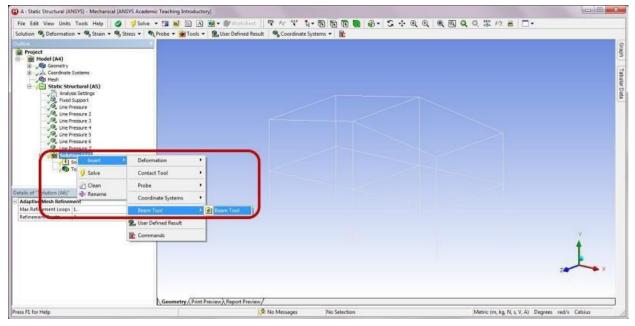
Alternative Procedure: Note that the Line Pressure item in the Outline can be copied and pasted under Static Structural (A5) for repeated use. To make a copy, right-click on Line Pressure and select Copy from the menu. To paste, right-click on the Static Structural (A5) and select Paste. Remember to apply each newly pasted Line Pressure to a different line edge on the Geometry selection in the Details of "Line Pressure" until the same load is applied to all seven edges.

Step 14: Retrieve Solution

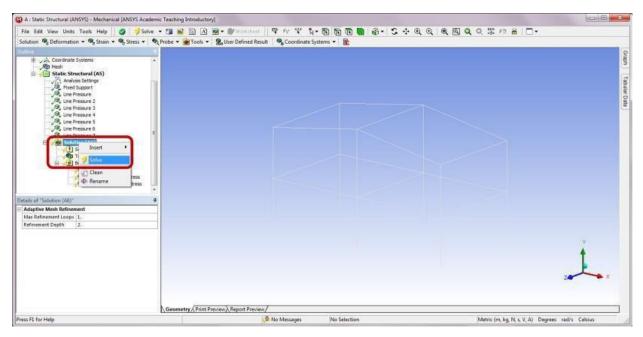
Insert a Total Deformation item by right-clicking on Solution (A6) in the Outline tree. Insert a



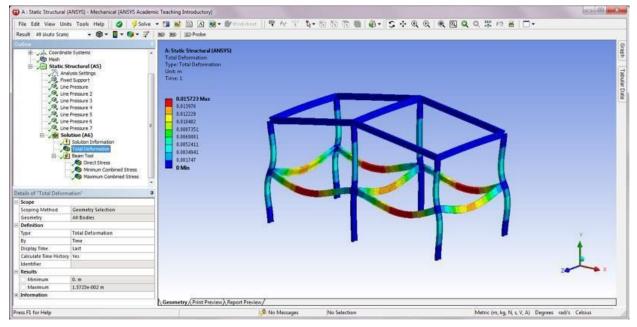
Insert a Beam Tool item by right-clicking on Solution (A6) in the Outline tree.



Right-click on Solution (A6) in the Outline tree and select Solve. The program will start to solve the model.



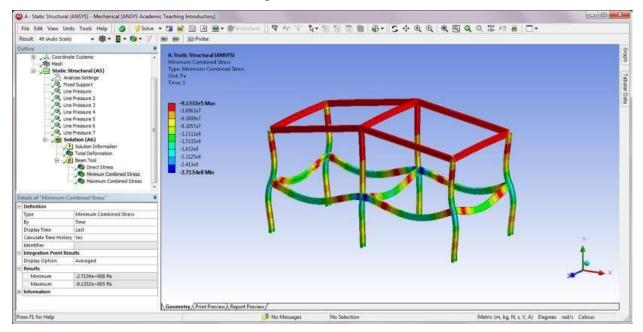
After completion, click Total Deformation in the Outline to review the total deformation results.



Click Direct Stress under Beam Tool in the Outline to review the axial stress results in beams.

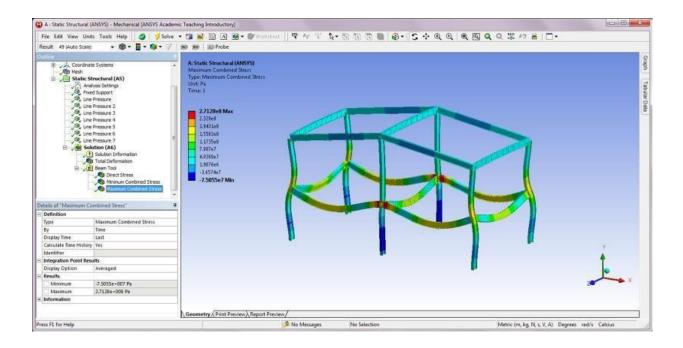
A : Static Structuri	al (ANSYS) - Mechanical (ANSYS Academi	raching Introductory]	
File Edit View U	Inits Tools Help 🛛 🥥 😏 Solve	🖀 🖬 🖄 📾 • 🗰 Workillert 🗣 🎶 🙄 🗞 🖄 🖄 👘 🚳 • 💽 💠	00000000000000000000000000000000000000
Result 49 (Auto Sca		DProbe	
Mesone as padro sta		and an and a second sec	
Outline		A: Static Structural (ANSYS)	
Coords	nate Systems +	Direct Stress	3
E G Stati	: Structural (A5)	Type: Direct Stress	
	knalysis Settings	Unit: Pa	ā
Q 1	wed Support	Time: 1	
-,9,1	ine Pressure		
10, 0	ine Pressure 2 ine Pressure 3	- 8.8319e5 Max	
	ine Pressure 3	-7.545166	
19	ine Pressure 5	-1.6775e7	
	ine Pressure 6	-2.5605e7	
Q. U	ine Pressure 7	-3.4434#7	
	Solution (A6)	-4.3263eJ	
Sides.	Solution Information	-5.2093e7	
	Total Deformation	-6.8922e7	
	Beam Tool	-6.9751e7	
	Mininum Combined Stress	-7.858e7 Min	
	Naximum Combined Stress		
	<u> </u>		
Details of "Direct Stree	st* • •		
E Definition			
Type	Direct Stress		
By	Time		
Display Time	Last		
Calculate Time Histo	ory Yes		
Identifier	Statistics and statistics		
E Integration Point R	lesults		
Display Option	Averaged		
- Results	The second se		·
Minimum	-7.858e+007 Pa		Z A X
Maximum	8.8319e+005 Pa		
+ Information			
		Seometry / Print Preview / Report Preview /	
Press F1 for Help		0 No Messages No Selection	Metric (m, kg, N, s, V, A) Degrees rad/s Celsius

Click Minimum Combined Stress under Beam Tool to retrieve the linear combination of the



Direct Stress and the Minimum Bending Stress results in beams.

Click Maximum Combined Stress under Beam Tool to retrieve the linear combination of the Direct Stress and the Maximum Bending Stress results in beams.



Close the Static Structural (ANSYS) program. Save project and exit Workbench.

Ex. No : Date :

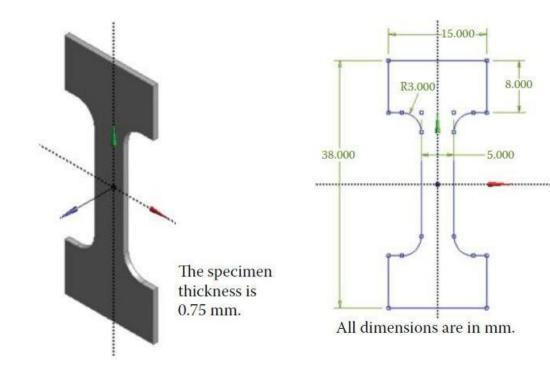
BUCKLING FAILURE

Problem Description: A dog-bone shaped specimen is examined for static, fatigue, and buckling failures. The specimen is made of structural steel with geometric dimensions shown below. The bottom face of the specimen is fixed, and the top face of the specimen is applied a static pressure load of 50 MPa.

(a) Determine whether or not the specimen undergoes plastic deformation under the given static pressure load.

(b) If the static pressure load is changed into a fully reversed cyclic load with a magnitude of 50 MPa, find the life of the specimen, and also determine whether or not fatigue failure occurs in the specimen assuming a design life of 10^6 cycles.

(c) Determine whether or not the specimen buckles under the given static pressure load, and obtain the first three buckling mode shapes.



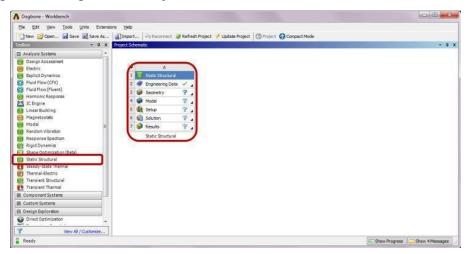
Solution steps for portion (A and B):

Step 1: Start an ANSYS Workbench Project

Launch ANSYS Workbench and save the blank project as "Dogbone.wbpj."

Step 2: Create a Static Structural Analysis System

Drag the Static Structural icon from the Analysis Systems Toolbox window and drop it inside the highlighted green rectangle in the Project Schematic window.

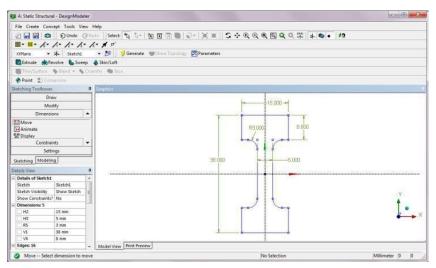


Step 3: Launch the Design Modeler Program

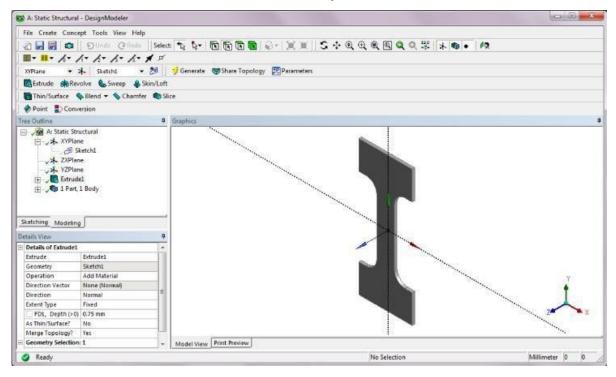
Double-click the Geometry cell to launch Design Modeler, and select "Millimeter" in the Units pop-up window.

Step 4: Create the Geometry

Click on the Sketching tab. Draw a sketch of the dog bone shape on the XY Plane, as shown below. An entity named Sketch1 will be shown underneath XY Plane of the model's Tree Outline.

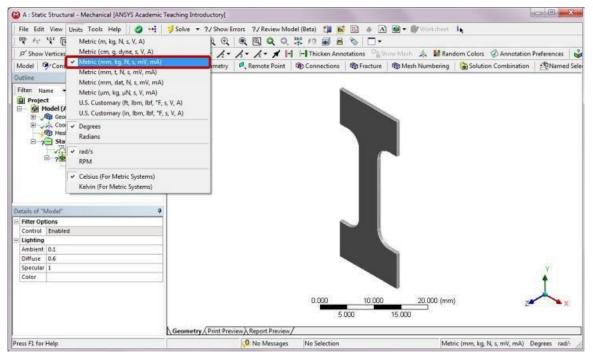


Extrude Sketch1 to create a 0.75 mm thick solid body, as shown below.



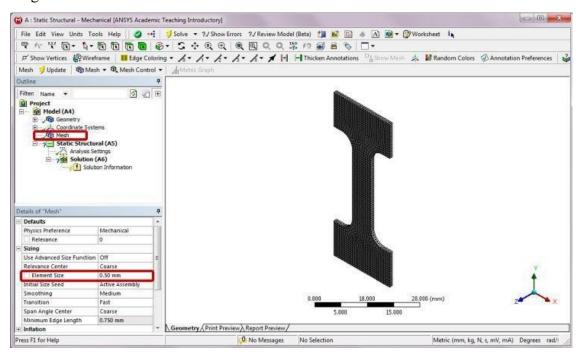
Step 5: Launch the Static Structural Program

Double-click on the Model cell to launch the Static Structural program. Change the Units to Metric (mm, kg, N, s, mV, mA).



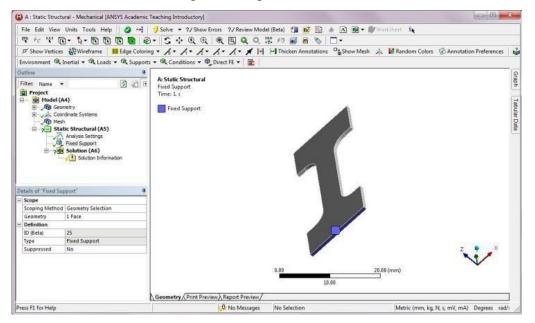
Step 6: Generate Mesh

Click on Mesh in the Outline tree. In the Details of "Mesh," enter "0.5 mm" for the Element Size. Right-click on Mesh and select Generate Mesh.



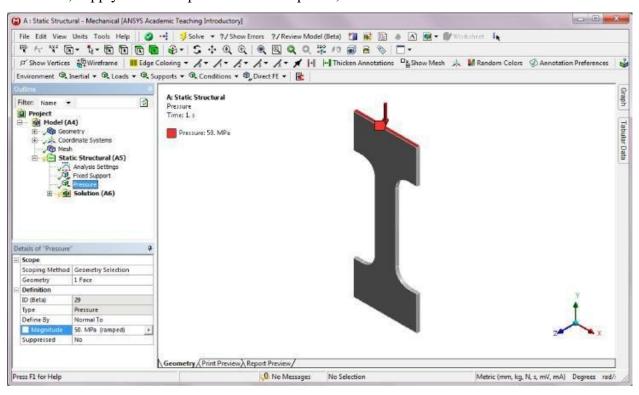
Step 7: Apply Boundary Conditions

Right-click on Static Structural (A5). Choose Insert and then Fixed Support from the context menu. Click on the bottom face, and apply it to the Geometry selection in the Details of "Fixed Support." The bottom face of the dog bone shape is now fixed as shown below.



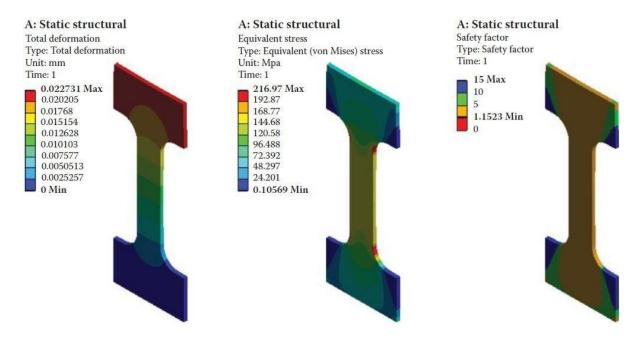
Step 8: Apply Loads

Right-click on Static Structural (A5). Choose Insert and then Pressure. In the Details of "Pressure," apply a 50 MPa pressure to the top face, as shown below.



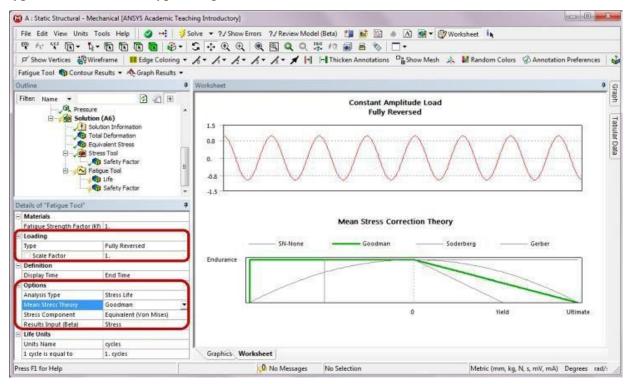
Step 9: Retrieve Static Analysis Results

- First, insert a Total Deformation item by right-clicking on Solution (A6) in the project Outline.
- Then, insert an Equivalent Stress item by right-clicking on Solution (A6) in the project Outline.
- Next, right-click on Solution (A6) in the project Outline, and select Insert -> Stress Tool Max Equivalent Stress. The initial yielding in the test sample may be predicted by comparing the maximum von Mises stress in the specimen with the tensile yield strength of the specimen material. The Stress Tool is used here to show the safety factor results.
- Right-click on Solution (A6) and select Solve. The computed total deformation, von Mises stress and safety factor distributions are shown below. From the static analysis results, it is apparent that the neck portion of the specimen will not yield if loaded statically.



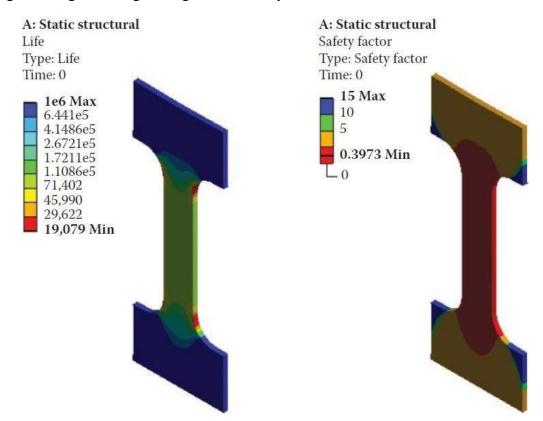
Step 10: Retrieve Fatigue Analysis Solution

Right-click on Solution (A6) in the Outline, and select Insert -> Fatigue -> Fatigue Tool. In the Details of "Fatigue Tool," set the Mean Stress Theory to Goodman. Note that the default loading type is Fully Reversed constant amplitude load, and that the default analysis type is the Stress Life type using the von Mises stress calculations.



Right-click on the Fatigue Tool in the Outline, and select Insert -> Life. Next, right-click on the Fatigue Tool and select Insert -> Safety Factor. In the Details of "Safety Factor," change the Design life from the default value of 10^9 cycles to 10^6 cycles.

Finally, right click on the Fatigue Tool and select Evaluate All Results. From the fatigue analysis results, the shortest life is at the undercut fillets (19,079 cycles) followed by the neck portion of the specimen. The neck portion of the specimen has a fairly small safety factor with a minimum value of 0.3973. The results show that the specimen will not survive the fatigue testing assuming a design life of 10^6 cycles.



Solution steps for portion (C):

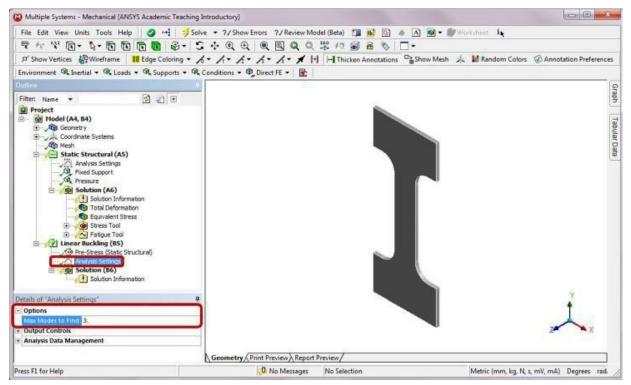
Step 1: Create a Linear Buckling Analysis System

In the Project Schematic window, right-click on the Solution cell of the Static Structural analysis system and select Transfer Data to New -> Linear Buckling. A linear buckling analysis system will be added, with the static structural results being used as initial conditions. The engineering data, geometry, and model will be shared by both analyses.

File Edit View Tools Units Exten	isions Help	
New Copen 🛃 Save 😹 Save As	👔 Import 🖓 Reconnect 😹 Refresh Project 💛 Update Project 🕼 Project 🞧 Compact Mode	
- • • ×		* 4
Analysis Systems Cooling Analysis Systems Cooling C	and a second s	
Orrect Optimization		
View All / Customize		

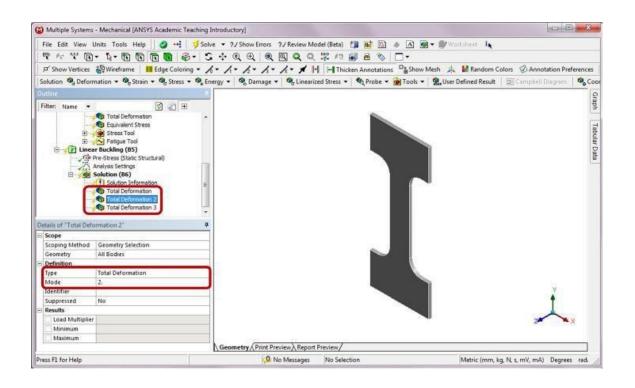
Step 2: Launch the Multiple Systems-Mechanical Program

Double-click the Setup cell of the Linear Buckling system to launch the Multiple Systems– Mechanical program. Click on the Analysis Settings under Linear Buckling (B5) in the Outline. In the Details of "Analysis Settings," set the Max Modes to Find to 3.



Step 3: Retrieve Linear Buckling Analysis Results

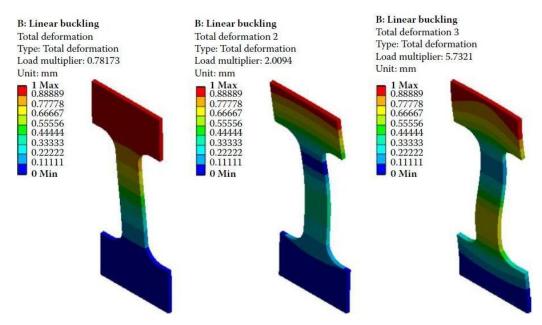
- Insert three Total Deformation items by right-clicking on Solution (B6) in the Outline.
- In the Details of "Total Deformation," set Mode to 1. In the Details of "Total Deformation 2," set Mode to 2. In the Details of "Total Deformation 3," set Mode to 3.



- Right-click on Solution (B6) and select Solve to view the buckling modes. To use the default window layout as shown below, select View -> Windows -> Reset Layout from the top menu bar.
- Note that the load modifier for the first buckling mode is found to be 0.78173. To find the load required to buckle the structure, multiply the applied load by the load multiplier.
- For example, the first buckling load will be 39.0865 MPa (0.78173 × 50 MPa), thus the applied pressure of 50 MPa will cause the specimen to buckle. In the Graph window, you can play the buckling animation.

Multiple Systems - Mechanical [ANSYS	cademic Teaching Introductory]	
File Edit View Units Tools Help	🥥 💀 🔰 Solve 🔹 ?/ Show Errors ?/ Review Model (Beta) 🎁 🙀 🔕 📣 🖪 👼 🖝 💕 Worksher	i i
	■ @• 5 * Q Q Q Q Q Q X /2 @ 8 % □•	
F Show Vertices 🖓 Wireframe	dge Coloring 🔹 🍾 🖌 * 🍾 * 🖈 🖈 💾 🖂 Thicken Annotations 🕒 Show Mesh 🎄 🕍 R	andom Colors 🛛 🛞 Annotation Preferences
Result 2. (Auto Scale) - 🝘 -	▼ 🐠 😴 😥 📧 🗇 Probe	
Dutfine		
Filter: Name +	B: Linear Buckling Total Deformation	
Solution (A6) Solution Informatic Solution Informatic Solution Informatic Solution Informatic Solution Informatic Solution (B6) Solution (B6) Solution Information Total Deformation Total Deformation Total Deformation	Type: Total Deformation Load Multiplier. 0.78173 Unit: mm 1. Max 0.80698 0.77778 0.66667 0.33333 0.2222 0.11111 0. Min	× ×
letails of "Total Deformation"		
Scope Scoping Method Geometry Selection	Geometry (Print Preview) Report Preview/	
Geometry All Bodies	Graph Ø Tabular Data	
- Definition		oad Multiplier
Type Total Deformation	Animation	
Mode 1.	# 3. 2.2.2.2000	
Identifier	5.7321 3 3. 5.732	
Suppressed No	3	
Results		
Load Multiplier 0.78173	1 2 3	
Minimum 0, mm		
ress F1 for Help	Q No Messages No Selection Metr	ic (mm, kg, N, s, mV, mA) Degrees rad,

The following figures show the first three buckling mode shapes. The corresponding load multipliers for the first, second, and third mode shapes are 0.78173, 2.0094, and 5.7321, respectively.



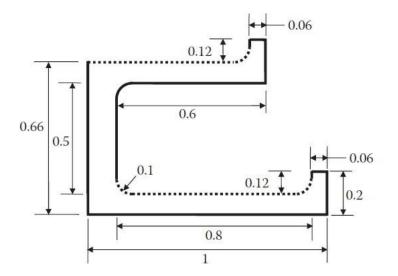
Note that the max value in the total deformation plots is scaled to 1 when displaying the buckling mode shapes. Here, the deformation plot is used for mode shape visualization, with the actual values of deformation carrying no physical meaning.

Ex. No :

Date :

STRESS ANALYSIS OF AXI-SYMMETRY STRUCTURE

Problem Description: Garden fountains are popular amenities that are often found at theme parks and hotels. As a fountain structure is usually an axisymmetric geometry with axisymmetric loads and support, only a 2-D model, sliced through the 3-D geometry, is needed to correctly predict the deformation of or stress in the structure. The figure below gives the cross section of an axisymmetric model of a two-tier garden fountain made of concrete. Determine the maximum deformation and von Mises stress under the given hydrostatic pressure. Use adaptive meshing to improve solution convergence.



Material: Concrete E = 29 GPa v = 0.15 Boundary conditions: Bottom edge: fixed. Left edge: axis of symmetry. Hydro pressure on dotted edges.

(All units are in meter)

Solution

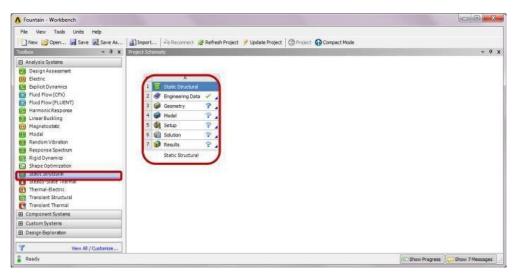
To solve the problem with ANSYS® Workbench, we employ the following steps:

Step 1: Start an ANSYS Workbench Project

Launch ANSYS Workbench and save the blank project as "Fountain.wbpj"

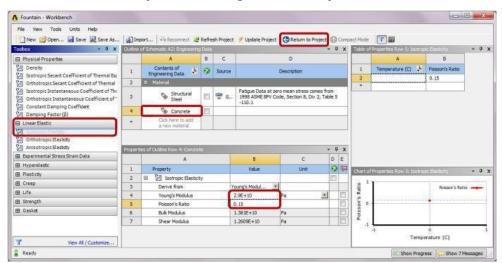
Step 2: Create a Static Structural (ANSYS) Analysis System

Drag the Static Structural (ANSYS) icon from the Analysis Systems Toolbox window and drop it inside the highlighted green rectangle in the Project Schematic window to create a standalone static structural analysis system.



Step 3: Add a New Material

Double-click on the Engineering Data cell to add a new material. In the following Engineering Data interface which replaces the Project Schematic, type "Concrete" as the name for the new material, and double-click Isotropic Elasticity under Linear Elastic in the leftmost Toolbox window. Enter "29E9" for Young's Modulus and "0.15" for Poisson's Ratio in the Properties window. Click the Return to Project button to go back to Project Schematic.



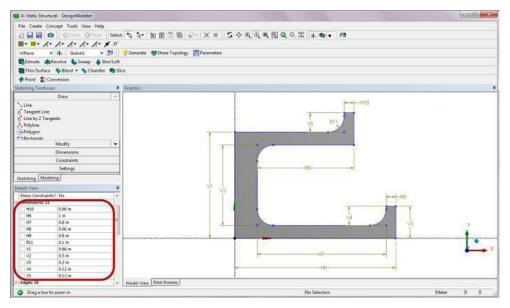
Step 4: Launch the Design Modeler Program

Ensure Surface Bodies is checked in the Properties of Schematic A3: Geometry window (select Properties from the View drop-down menu to enable display of this window). Select 2D for Analysis Type in this Properties window. Double-click the Geometry cell to launch DesignModeler, and select "Meter" in the Units popup window.

Pile View Tools Units Help	👔 Import 🏻 🖗 Reconnect 🛛 Ø Refresh Project 🏓 Ups	late Project @ Project @ Compact N	fode	
cobas - 9 X F	Project Schematic	* 9 X Pro	perties of Schematic A3: Geometry	- 9
Analysis Systems			A	8
🕜 Design Assessment		1	Property	Value
B) Electric	• A	2	E General	
Bxplicit Dynamics	1 🔤 Static Structural	3	Component ID	Geometry
Fluid Flow (CFX)	2 🦪 Engineering Data 🗸	4	Directory Name	SYS
Fluid Flow (FLUENT)	3 🥪 Geometry 📪	5	Geometry Source	
Harmonic Response	4 Model	6	Geometry File Name	
Magnetostatic	6 40 cm	7	F Basic Geometry Options	
Modal		8	Solid Bodies	
Random Vibration		9	Surface Bodies	
Response Spectrum	7 🥩 Results 👕 🛓	3	0 Line Bodies	1
Rigid Dynamics	Static Structural	1	1 Parameters	2
Shape Optimization		1	2 Parameter Key	05
Static Structural		1	3 Attributes	10
Steady-State Thermal		1	4 Named Selections	100 E
1) Thermal-Electric		1	5 Material Properties	121
Transient Structural		2	6 Advanced Geometry Options	
Transient Thermal		T	7 Analysis Type	20 .
Component Systems		1 U	8 Use Associativity	2
a Custom Systems		2	9 Import Coordinate Systems	1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
B Design Exploration		2		(F)
View All / Customize		2	Dander Made Cause	E

Step 5: Create Surface Sketch

Click on the Sketching tab. Select the Draw toolbox and then Line. Draw a closed loop line profile as dimensioned below. Make sure a horizontal constraint (H) and a vertical constraint (V) appear when drawing a horizontal and a vertical line, respectively. Use the Fillet tool in the Draw toolbox to create line fillets with a radius of 0.1 m as shown below.



Step 6: Create Surface Body

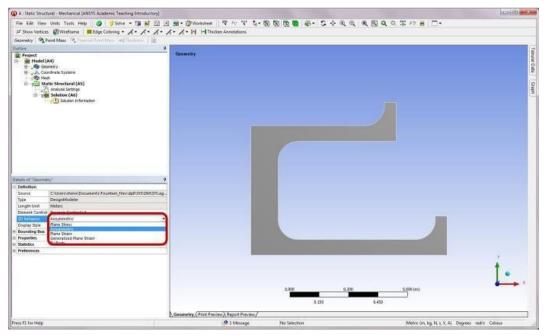
Switch to the Modeling tab and choose Surfaces from Sketches from the Concept menu. Select Sketch1 from the Tree Outline shown below, and apply it to the Base Objects selection in the Details of SurfaceSK1. Then click Generate.

🙀 A: Static Structural - DesignModeler			
File Create Concept Tools View Help	and the second second second second		
21 1 Colorida Colorida Selecti Ta	5- mm mm	·····	
h. h. h. h. h. h. # =			
XTPlane + H- Sketchi + 21 10	marte better Territoria		
Strong Country Strong Strong		The second se	
		are II A sum "Accordings.	
Tree Outline	a Graphics		
A Static Structural A Static			
Skettining Modeling Tensik Vise Dealin of Stanbackit Dealin of Stanbackit Surface From Stetches Surface Stat Surface Stat			n
Orient With Plane Normal? Yes			
Thickness (>=0) 0 m		1	B
		10000	
		0.000 0.300 0.600 (m) 0.150 0.450	Ŀ
	Model View Print Preview	1.)	
	Tuener Anim Trune Leaners 1		Meter 0 0

A Surface Body will be created from the surface sketch. Exit the Design Modeler.

Step 7: Launch the Static Structural Program

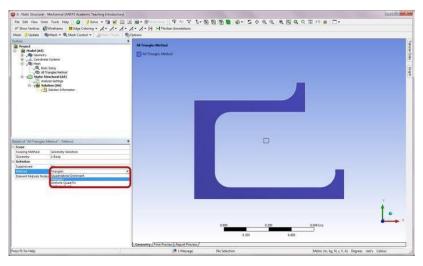
Double-click on the Model cell to launch the Static Structural program. Click on Geometry in the Outline. In the Details of "Geometry," choose axisymmetric for 2D Behavior.



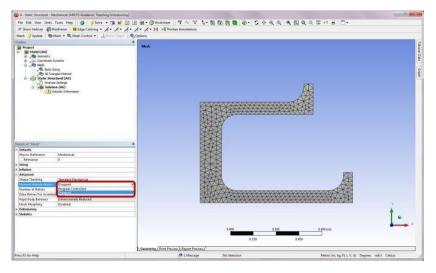
In the Details of "Surface Body," click to the right of the Material Assignment field and select Concrete from the drop-down context menu.

Step 8: Generate Mesh

- Right click on Mesh in the Outline. Select Insert and then Sizing from the context menu. In the Details of "Body Sizing," enter "0.05 m" for Element Size. Click on the surface body in the Graphics window and apply it to the Geometry selection.
- Right click on Mesh.
- Select Insert and then Method. In the Details of "Automatic Method," click on the surface body, and apply it to the Geometry. Select Triangles for Method. This will make use of triangular elements for the mesh generation.



In the Details of "Mesh," choose Dropped for the Element Midside Nodes field. This will specify the use of linear elements in the mesh. Note that linear triangular elements are employed here to show the convergence of linear FEA approximate solutions; they are in general not recommended for stress analysis. Right click on Mesh and select Generate Mesh.

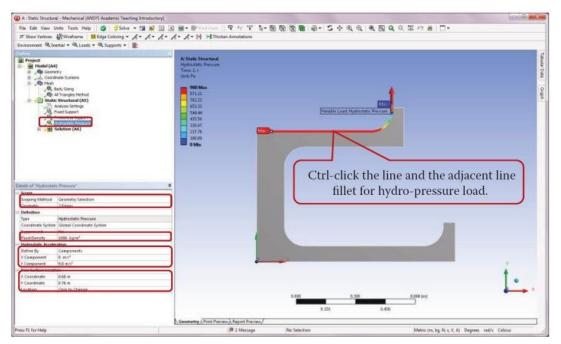


Step 9: Apply Boundary Conditions

- Right-click on Static Structural (A5). Choose Insert and then Fixed Support from the context menu. Apply this support to the horizontal line at the bottom.
- Right-click on Static Structural (A5). Choose Insert and then Frictionless Support from the context menu. Apply this support to the leftmost vertical line (center line of the fountain). The frictionless support prevents the line from moving or deforming in the normal direction, and thus is equivalent to a symmetry condition.

Step 10: Apply Loads

- In the Project Outline, right-click on Static Structural (A5), Choose Insert and then Hydrostatic Pressure. The hydrostatic load simulates pressure due to fluid weight.
- In the Details of "Hydrostatic Pressure," ctrl-click the horizontal line and the adjacent line fillet shown below, and apply the two edges to the Geometry selection.
- Enter 1000 kg/m³ for Fluid Density. Change the Define By selection to Components, and enter 9.8 m/s² for the Y Component of Hydrostatic Acceleration.
- Enter 0.68 m for the X Coordinate and 0.76 m for the Y Coordinate for the Free Surface
 Location. The location corresponds to the upper endpoint of the line fillet.

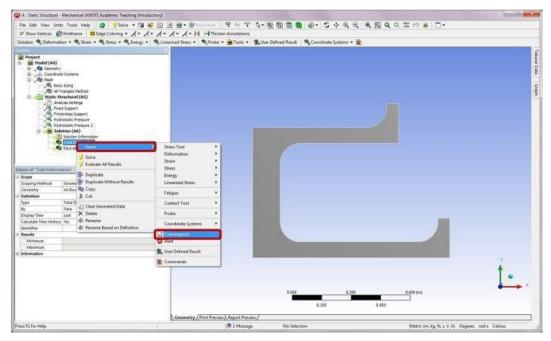


Insert another Hydrostatic Pressure load, and apply the pressure to the line and its two adjacent line fillets as shown below.

A : Static Structural - Mechanical (ANSYS Academic Teaching Int Contemport Contemport (A) - Static Structural - Mechanical (ANSYS Academic Teaching Int (A) - Static Structural - Mechanical (ANSYS Academic Teaching Int (A) - Static Structural - Mechanical (ANSYS Academic Teaching Int (A) - Static Structural - Mechanical (A) - Static Structu	aductory]	California (B) and Ca
File Edit View Units Tools Help 👩 🦻 Solve • 🗇	※回回週・#******* マルマ \$* ●回回● ●* 5 수 Q Q 天田Q Q 早か # □*	
9" Show Vertices		
Environment @ Inental + @ Loads + @ Supports + 🔡		
Dattern		
© Propert S → Concerty → Co	A Math Shared Hard Shared Hard Price 1990 Anno 1990	
Denits of "Nychotatic Prostane 2" "Scane Scane (Genetity Selection Scane Scane Sca	Ctrl-click the line the adjustment two fillets for hydro-pressure load.	line
Coordinate Syntex Glassi Coordinate Syntem Provide Syntem Pr		
Component 6.m.1"		
Pren landace Several ani Il Costantination Odd-m Il Coordination Odd-m Il Coordination Odd m Facebook District Star Principa	530 <u>64</u> 06(m)	1.
From F1 for Help	1.55 3.60 \Geometry/DrittProvies/ExportProvies/ @ 1 Message No Selection Meric (m, ig, N, c	V.A) Degrees rad/s Cebius

Step 11: Retrieve Solution

Insert a Total Deformation item by right-clicking on Solution (A6) in the Outline tree. Rightclick on the Total Deformation in the Outline tree, and choose Insert then Convergence.



- > In the Details of "Convergence," enter 1% for the Allowable Change field.
- In the Details of "Solution (A6)," set Max Refinement Loops as 10, and Refinement Depth as 1. The refinement depth controls the aggressiveness of the mesh refinement; it has a range from 0 to 3 with a larger number indicating more aggressive refinement.

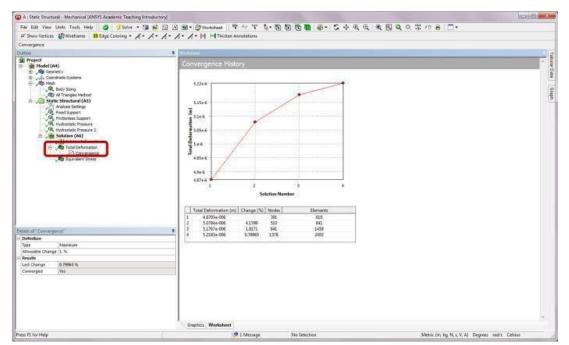
-	Definition	
	Туре	Maximum
	Allowable Change	1.%
-	Results	
	Last Change	0.%
	Converged	No

Ξ	Adaptive Mesh Refinen	nent
	Max Refinement Loops	10.
	Refinement Depth	1.
=	Information	
	Status	Done

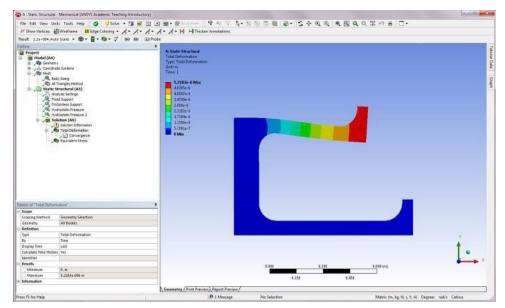
Press F1 for Help

Press F1 for Help

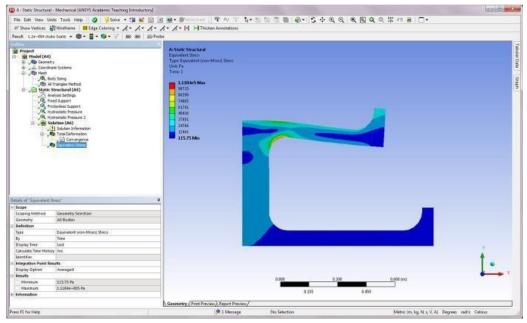
- ♦ Insert an Equivalent Stress item by right-clicking on Solution (A6) in the Outline tree.
- Right-click on Solution (A6) in the Outline tree and choose Solve. The program will start to iterate the solution until the difference between two consecutive iterations is less than 1% or the maximum number of mesh refinement loops reaches 10.
- After completion, click on Convergence in the Outline to review the convergence curve.
 The resulting maximum deformations at different mesh iterations are also recorded in the table below the curve.



Click on Total Deformation in the Outline to review the converged deformation results.



Click on Equivalent Stress in the Outline to review the stress results.



Modeling tips: A model may be subjected to body forces such as gravitational or radial centrifugal/inertia forces, in addition to the hydrostatic pressure load. To consider such forces, the density of the structure's material needs to be given as an input, and the forces are typically calculated as follows:

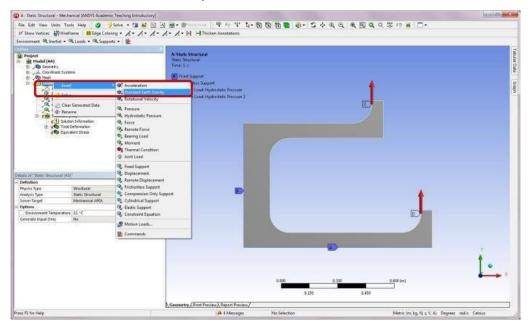
$$f_r = \rho r \omega^2$$
, Equivalent radial centrifugal/inertial force $f_z = -\rho g$, Gravitational force

where ρ is the mass density and g the gravitational acceleration (=9.8 m/s²).

Take the following steps to add body forces to the fountain model. First, double-click on Density in Physical Properties Toolbox. Enter 2.38e3 for Density in the Properties of Outline Row 3: Concrete. Click on the Return to Project button.

File Edit View Tools Units Help	d) in	port PeReconnect 🥥	Refre	esh Projec	t 🥖 Update Project	GReturn to Pr	oject	00	Comp	act Mode				
rachan 🔹 🔍 🗙	Outline	of Schematic A2: Engineering	Data				*	ņ	×	Table of	Properties Row 210	Seculty		- 4
Physical Properties		A	8	с		D					A		8	
2 Isotropic Secant Coefficient of Thermal Do	i.	Contents of Engineering Data	ø	Source		Description				1	Temperature (C)	J.	Density (kg m ^-3)	4
Orthotropic Secant Coefficient of Thermal	2	 Material 								1.000				-13
🔁 Isotropic Instantaneous Coefficient of The	3	2 Concrete		ም c										
Orthotropic Instantaneous Coefficient of ⁺ Constant Damping Coefficient Damping Factor (B)		Structural Steel		🔮 G.	Fatigue Data at zero 1998 ASME BPV Cox -110.1									
Linear Bastic		Click here to add a new material												
업 에 Orthotropic Elasticity Anisotropic Elasticity	Property	tes of Outline Row 3: Concret					- 0	0	×					
Experimental Stress Strain Data		Δ.	1		R	с	-	1	E					
Hyperelastic		Property	-	_	Value	Unit		2		-		-		_
Plasticity	2	가 Density				ka m^-3		-	104	Chart of	Properties Row 2: 0	Data N	2	, ą
E Creep	3	E A Isotropic Elasticit		-	2,3963	40 m - + 5	-							
🗉 Life	4	Derive from	8	-	Young's Modul		-	-	-					
Strength	5	Young's Modulus				Pa			17					
🗉 Gasket	6	Poisson's Ratio	_		0.15				F					
19-10-10-10-10-10-10-10-10-10-10-10-10-10-	7	Bulk Modulus		-	(57.55)	Pa	-		ři.					
	8	Shear Modulus				Pa			E					
	-				Allocation and the	1.0.0		-	-					

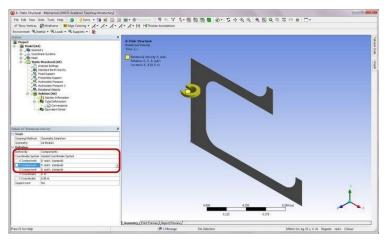
- Next, refresh the Model cell of Project Schematic after the above change is made on the Engineering Data cell. Double-click on the Model cell to launch the Static Structural Program.
- In the Project Outline shown below, right-click on Static Structural (A5). Choose Insert and then Standard Earth Gravity.



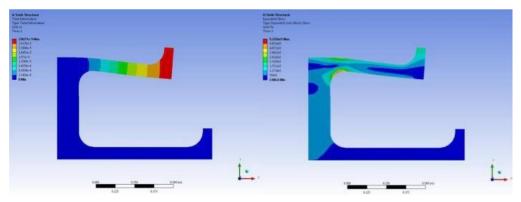
Right-click on Static Structural (A5) and choose Insert then Rotational Velocity.

Project Proje	bens	Al Stable Structu Date Discurre Time: 8 /			0.02	
, G, Hydrastal , G, Hydrastal , G, Solution , J) Solution , J) Solution , G, Topo , G	ter f - Const ter f - Const of StatC Clear Generated Data is PriC Rename (A6) (A6)	Al Acceleration Constrained Vectory Constrained Constrained Constrained Constrained Constrained Constrained Constrained	ntatic Perman Intel Perman J Par 3.8865 m/1 ²			
ntain pf "Dato Unictual (# Definition Physics Type Analysis Type Solver Target Options	Structural Static Structural Mechanical APOL	Field Support Field Support Displacement Frectorkers Support Connections Support Connections Support Euclid Support Euclid Support Euclid Support Constant Equation	Ð		¶	
Convolutent Temperatur Generate Input Only	No.	🔮 Motion Loads				
				1.15	1.350 a.500/m) 0.315	Ŀ,
Press FL for Help		Geometry (First	Preview), Report Preview/	No Selection	Materia in he bi	s V. Al Degrees rad/s Celsius

In the Details of "Rotational Velocity," change Define By to Components. Enter 5 rad/s for Y Component.



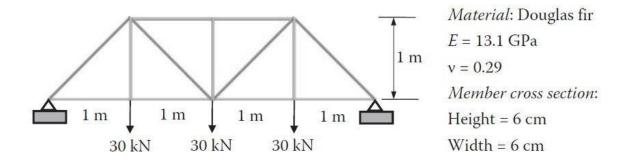
- Right-click on Solution (A6) in the Outline tree, and select Solve to update the model results. The new deformation and stress results are shown below.
- Both the maximum deformation and the maximum von Mises stress values are shown to be slightly increased, as compared to the results considering only the hydrostatic pressure load.



Ex. No : Date :

ANALYSIS OF 2D TRUSS

Problem Description: Truss bridges can span long distances and support heavy weights without intermediate supports. They are economical to construct and are available in a wide variety of styles. Consider the following planar truss, constructed of wooden timbers, which can be used in parallel to form bridges. Determine the deflections at each joint of the truss under the given loading conditions.



SOLUTION

To solve the problem with ANSYS® Workbench, we employ the following steps:

Step 1: Start an ANSYS Workbench Project Launch ANSYS Workbench and save the blank project as "Woodtruss.wbpj."

Step 2: Create a Static Structural (ANSYS) Analysis System Drag the Static Structural (ANSYS) icon from the Analysis Systems Toolbox window and drop it inside the highlighted green rectangle in the Project Schematic window to create a standalone static structural analysis system.

Step 3: Add a New Material

Double-click (or right-click and choose Edit) on the Engineering Data cell in the above Project Schematic to edit or add a material. In the following Engineering Data interface which replaces the Project Schematic, click the empty box highlighted below and type a name, for example, "Douglas Fir," for the new material.

Woodtruss - Workbench													00	×
File View Tools Units Help								_						
🗋 New 💕 Open 📕 Save 🔣 Sa	ave As	Jimport	Po Reco	onnect	🗃 Refre	esh Project 🍠 Updat	e Project	Prote		act Mode				
	Des Latition Vite	Project Schemati	10100				- 1							,
Analysis Systems														8
	_													
Design Assessment B) Electric		-		A										
Explicit Dynamics		1 20	Static St	tructur	ral									
Fluid Flow (CFX)		100	Enginee	Sector and	- 11									
Fluid Flow (FLUENT)			Geomet		?									
Marmonic Response			Model	e.y.	2									
Linear Buckling					7	4								
Magnetostatic Modal			Setup			4								
Modal Random Vibration		and the second second	Solution		?.	4								
Response Spectrum		7 🥩	Results		?.	•								
Rigid Dynamics			Static St	tructur	ral									
Shape Optimization														
Static Structural														
Steady-State Thermal														
Thermal-Electric														
Transient Structural Transient Thermal														
Transient Thermal Component Systems														
] Custom Systems														
B Design Exploration														
Ready Woodtruss - Workbench											Show Pro		now 16 Mess	
Ready Woodtruss - Workbench le Edit View Tools Units	Help	Import	Po Reco	mect	2 Refre	sh Project 🝠 Updat	e Project	Retur	n to Project					
Ready Woodtruss - Workbench le Edit View Tools Units] New 🎯 Open 🛃 Save 🔀 Sa	Help				🥔 Refre	sh Project 🍠 Updat	e Project (Retur		Comp	ect Mode		-	
Ready Woodtruss - Workbench le Edit View Tools Units] New @Open 글 Save 및 Sa Soox · 우 유 X 인	Help	hematic A2: Engi		Data		sh Project 🍠 Updat		Retur	n to Project • 4 X	Comp	ect Mode		-	, 4
Ready Woodtruss - Workbench le Edit View Tools Units New @Open 글 Save 값 Sa Soo	Help Ive As	hematic A2: Engir A	neering (Data B	с		D	Retur		Comp Table of	act Mode Properties	Row 2: Denst		р р В
Ready Woodtruss - Workbench le Edit View Tools Units New 20 Open Save & Save toox • P × 0 Physical Properties Linear Elastic	Help we As	hematic A2: Engi	neering (Data				Retur		Comp Table of	act Mode Properties		Density (р р В
Ready Woodtruss - Workbench le Edit View Tools Units New Copen Save S Sa Sox 7 X C Physical Properties Linear Elastic	Help we As utline of Sci 1	hematic A2: Engl A Contents of	neering (Data B	с		D		* 4 X	Comp Table of	act Mode Proper Les Temper	Row 2: Denst		р р В
Ready Woodtruss - Workbench le Edit View Tools Units New Open Save & Sa Sox • F X Physical Properties Linear Elastic Sotropic Elastidy	Help we As utline of Sci 1	A Contents of Engineering Data Material	heering (B	с	Fatigue Data at zero	D Description	a comes f	• 9 ×	Comp Table of 1 2	act Mode Proper Les Temper	Row 2: Denst	Density (р р В
Ready Woodtruss - Workbench le Edit View Tools Units New Open Save & Save hysical Properties Linear Elastic Sorthotropic Elastidty Orthotropic Elastidty	Help we As utline of Sci 1 2 = 3	A Contents of Engineering Data Material Structur Steel	neering (B	C Source		D Description	a comes f	• 9 ×	Comp Table of 1 2	act Mode Proper Les Temper	Row 2: Denst	Density (р р В
Ready Woodtruss - Workbench le Edit View Tools Units New Copen Save S Sa Cox • 4 × • Physical Properties Linear Elastic Stortopic Elastiday Orthotropic Elastiday Anisotropic Elastiday Experimental Stress Strain Data	Help we As utine of Sci 1 e	A Contents of Engineering Data Material	neering (B	C Source	Fatigue Data at zero	D Description	a comes f	• 9 ×	Comp Table of 1 2	act Mode Proper Les Temper	Row 2: Denst	Density (р р В
Ready Woodtruss - Workbench le Edit View Tools Units New 20 Open Save 20 Sa Cox • 4 × 1 Physical Properties Linear Elastic 1 Stortopic Elastiday Orthotropic Elastiday Anisotropic Elastiday Experimental Stress Strain Data Hyperelastic	Help we As utline of Sci 1 2 = 3	A Contents of Engineering Data Material Structu Steel Click here to a	neering (B	C Source	Fatigue Data at zero	D Description	a comes f	• 9 ×	Comp Table of 1 2	act Mode Proper Les Temper	Row 2: Denst	Density (р р В
Ready Woodtruss - Workbench le Edit View Tools Units New 20 Open Save S Sa tox • 4 × • Physical Properties Linear Elastic 1 Stotropic Elastidty Orthotropic Elastidty Anisotropic Elastidty Experimental Stress Strain Data Hyperelastic Plasticty	Help we As utline of Sci 1 2 = 3	A Contents of Engineering Data Material Structu Steel Click here to a	neering (B	C Source	Fatigue Data at zero	D Description	a comes f	• 9 ×	Comp Table of 1 2	act Mode Proper Les Temper	Row 2: Denst	Density (р р В
Ready Woodtruss - Workbench le Edit View Tools Units New Open Save S Save toox • P × O Physical Properties Linear Elastic Orthotropic Elastidy Anisotropic Elastidy Experimental Stress Strain Data Hyperelastic Plasticty Creep	Help we As utline of Sci 1 2 = 3	A Contents of Engineering Data Material Structu Steel Click here to a	neering (B	C Source	Fatigue Data at zero	D Description	a comes f	• 9 ×	Comp Table of 1 2	act Mode Proper Les Temper	Row 2: Denst	Density (р р В
Ready Woodtruss - Workbench le Edit View Tools Units New Open Save S Save New Open Save S Save toox Physical Propertes Linear Elastic Sortopic Elastidy Anisotropic Elastidy Experimental Stress Strain Data Hyperelastic Plasticty Creep Life	Help we As utline of Sci 1 2 = 3	A Contents of Engineering Data Material Structu Steel Click here to a	neering (B	C Source	Fatigue Data at zero	D Description	a comes f	• 9 ×	Comp Table of 1 2	act Mode Proper Les Temper	Row 2: Denst	Density (х ф В
Ready Woodtruss - Workbench le Edit View Tools Units New Open Save S Sa toox Physical Properties Linear Elastic Dorthotropic Elastidy Anisotropic Elastidy Anisotropic Elastidy Experimental Stress Strain Data Hyperelastc Plasticty Creep Life Strength	Help we As utline of Sci 1 2 = 3	A Contents of Engineering Data Material Structu Steel Click here to a	neering (B	C Source	Fatigue Data at zero	D Description	a comes f	• 9 ×	Comp Table of 1 2	act Mode Proper Les Temper	Row 2: Denst	Density (х ф В
Ready Woodtruss - Workbench le Edit View Tools Units New Open Save S Sa Los Physical Properties Linear Elastic Dorthotropic Elastidy Anisotropic Elastidy Anisotropic Elastidy Creep Life Strength	Help we As utline of Sci 1 2 = 3	A Contents of Engineering Data Material Structu Steel Click here to a	neering (B	C Source	Fatigue Data at zero	D Description	a comes f	• 9 ×	Comp Table of 1 2 *	act Mode Properties	Eov 21 Dens t	Density (, a B kg m
Ready Woodtruss - Workbench le Edit View Tools Units New Open Save S Se Sox Physical Properties Linear Elastic Storropic Elastidy Orthotropic Elastidy Anisotropic Elastidy Anisotropic Elastidy Creep Life Strength Gasket	Help we As utine of Sc 1 2 = 3 2 *	Hematic A2: Engi A Contents of Engineering Data Material Structu Click here to new material	ral	B B	C Source	Fatigue Data at zero	D Description	a comes f	• 4 x	Comp Table of 1 2 *	act Mode Properties	Rov 2: Denst	Density (, a B kg m
Ready Woodtruss - Workbench le Edit View Tools Units New Copen Save S So Soo Physical Properties Linear Elastic Storropic Elastidy Orthotropic Elastidy Anisotropic Elastidy Creep Life Strength Gasket	Help we As utine of Sc 1 2 = 3 2 *	A Contents of Engineering Data Material Structu Steel Click here to a	ral add a	B B	C Source	Faligue Data at zero ASME BPV Code, Sec	D Description	a comes f	• 4 x itom 1998 -110.1	Comp Table of 1 2 *	Act Mode Properties	Eov 21 Dens t	Density (, a B kg m
Ready Woodtruss - Workbench le Edit View Tools Units New Copen Save S So Soo Physical Properties Linear Elastic Storropic Elastidy Orthotropic Elastidy Anisotropic Elastidy Creep Life Strength Gasket	Help we As utine of Sc 1 2 = 3 2 *	Hematic A2: Engi A Contents of Engineering Data Material Structu Click here to new material	ral	B B	C Source	Fatigue Data at zero	D Description	a comes f	• 4 X from 1998 -110.1	Comp Table of 1 2 *	ect Mode Proper lies	Eov 21 Dens t	Density (7850	, a B kg m
Ready Woodtruss - Workbench le Edit View Tools Units New Copen Save S So Soo Physical Properties Linear Elastic Storropic Elastidy Orthotropic Elastidy Anisotropic Elastidy Creep Life Strength Gasket	Help we As utine of Sc 1 2 = 3 2 *	Hematic A2: Engl A Contents of Engineering Data Material Structu Click here to new material	ral add a	B B	C Source	Faligue Data at zero ASME BPV Code, Sec	D Description	s comes f	• 4 x itom 1998 -110.1	Comp Table of 1 2 *	Properties	Eov 21 Dens t	Density (7850	, a B kg m
Ready Woodtruss - Workbench le Edit View Tools Units New Copen Save S So Soo Physical Properties Linear Elastic Storropic Elastidy Orthotropic Elastidy Anisotropic Elastidy Anisotropic Elastidy Experimental Stress Strain Data Hyperelastic Plasticby Creep Life Strength Gasket	Help we As 1 2 C 3 2 *	Hematic A2: Engi A Contents of Engineering Data Material Steel Click here to a new material	ral add a	B B	C Source	Fatigue Data at zero ASME BPV Code, Sec	D Description mean stress tion 8, Div 2	s comes f	• 4 X from 1998 -110.1	Comp Table of 1 2 * Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart Chart of Chart of Char	Proper lies	Eov 21 Dens t	Density (7850	, a B kg m
Ready Woodtruss - Workbench le Edit View Tools Units New Copen Save S So Soo Physical Properties Linear Elastic Storropic Elastidy Orthotropic Elastidy Anisotropic Elastidy Anisotropic Elastidy Experimental Stress Strain Data Hyperelastic Plasticby Creep Life Strength Gasket	Help we As 1 g 2 = 3 * *	Hematic A2: Engineering Oata A Contents of Engineering Oata Material Click here to- new material Click here to- new material Click here to- new material Click here to- new material Click here to- steel Click here to- new material Click here to- new to- click here to- new to- click here to- new to- click here to- new to-	eering (add a add a Structure A	Data B Coeffici	C Source	Patigue Data at zero ASME BPV Code, Sec B B Value	D Description mean stress tion 8, Div 2	; comes f , Table 5	▼	Comp Table of 1 2 * Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart Chart of Chart of Char	Proper lies	Eov 21 Dens t	Density (7850	, a B kg m
Ready Woodtruss - Workbench le Edit View Tools Units New Open Save S Sa toox	Help we As Utine of Sc 2 = 3 = *	Contents of Engineering Data Material Steel Click here to . new material Click here to . new material	reering to rail add a A Structure A	Data B Coeffici	C Source	Patigue Data at zero ASME BPV Code, Sec B B Value	D Description mean stress tion 8, Div 2	; comes f , Table 5	v 4 X irom 1998 -110.1 D E D E D E D E	Comp Table of 1 2 * Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart of Chart Chart of Chart of Char	Properties	Eov 21 Dens t	Density (7850	- 4 8 kg m ²
Ready Woodtruss - Workbench le Edit View Tools Units New Copen Save S Sa box Physical Properties Linear Elastic Orthotropic Elastidy Anisotropic Elastidy Experimental Stress Strain Data Hyperelastic Plasticty Creep Life Strength Gasket P	Help we As 1 g 2 = 3 * *	Hematic A2: Engineering Oata A Contents of Engineering Oata Material Click here to- new material Click here to- new material Click here to- new material Click here to- new material Click here to- steel Click here to- new material Click here to- new to- click here to- new to- click here to- new to- click here to- new to-	reering to rail add a A Structure A	Data B Coeffici	C Source	Patigue Data at zero ASME BPV Code, Sec B B Value	D Description mean stress ction 8, Div 2	; comes f , Table 5	▼	Comp Table of 1 2 * Chart of Chart of Chart of Chart of Chart of Chart of Chart Chart of Chart of Chart of Chart of Chart Chart of Chart of Chart of Chart of Chart of Chart Chart of Chart of C	Properties	rov 2: Densit	Density (7850	ж ф В

Select "Douglas Fir" from the Outline window, and double-click Isotropic Elasticity under Linear Elastic in the leftmost Toolbox window. Enter "1.31E10" for Young's Modulus and "0.29" for Poisson's Ratio in the bottom center Properties window.

Click the Return to Project button to go back to the Project Schematic.

File Edit View Tools Units	Help					-		_	1.1				
🎦 New 💕 Open 🛃 Save 🔣	Save As	👔 Import 🗟 Reco	nnect	Refr	esh Project 🍠 Upd	ate Project 🔇	Return to	Project	O Con	ipact Mod	e 🝸 🛍		
Toolbox 👻 🕈 🗙	Outline o	f Schematic A2: Engineering	Data		-	-		4 X	Table	of Proper	ties Row 6: Iso	bropk Ela	4 >
Physical Properties		A	в	С		D					A		в
Linear Elastic Isotropic Elasticity	1	Contents of Engineering Data	ø	Source		Description			1	Tem	perature (C)		
	2	Material							2			1.0397E	+10
Anisotropic Elastidty	3	Structural Steel	阙	😅 G	Fatigue Data at ze								
🖽 Experimental Stress Strain Data		and the second sec	100	-	ASME BPV Code, S	ection 8, Div 2, 12	spie 5-110	.1					
🗄 Hyperelastic	4	🖗 Douglas Fir	-		1								
Plasticity	*	Click here to add a											
Plasticity		new material											
		new material											
🗈 Creep		new material											
E Creep		new material											
E Creep E Life E Strength		new material			1]		771		
E Creep E Life E Strength		new material							C.C. Martin	of Proper		atropic Ela	4
⊞ Creep ⊞ Life ⊞ Strength									Chart e	222		otropic Ela	D
 B Creep B Life B Strength 	Propertie	es of Outline Row 4: Douglas	Fe					₽ X	Chart dl	of Proper	ties Row 6: Iso	htropic Ela 🔹	.
E Creep E Life E Strength	Propertie		Fr		8	c	D	E	Chart A	222	ties Row 6: Iso	Louis and	.
E Creep E Life E Strength	1	es of Outline Row 4: Douglas A Property			B Value	C	D		Chart dl (m01")	222	ties Row 6: Iso	Louis and	
 B Creep B Life B Strength 	1 2	es of Outline Row 4: Douglas A Property S 15otropic Elasticity			Value		D	E	Chart dl (m01")	1.5	ties Row 6: Iso	Louis and	
 B Creep B Life B Strength 	1 2 3	er of Outline Row 4: Douglas A Property Isotropic Elasticity Derive from			Value ung's Modul 💽	Unit	D	E 1	Chart dl (m01")	1.5	ties Row 6: Iso	Louis and	-
 B Creep B Life B Strength 	1 2 3 4	A Property Isotropic Elasticity Derive from Young's Modulus		1	Value ung's Modul 💌 .31E+10		D	E 1	Chart dl	1.5	bes Row 6: Iso	Louis and	
	1 2 3	er of Outline Row 4: Douglas A Property Isotropic Elasticity Derive from		1	Value ung's Modul • .31E+10 .29	Unit	D	E 1	Chart Bulk Modulus (*10**)	1	Bes Row 6: Iso	tulk Modulus –	a :

Step 4: Launch the DesignModeler Program

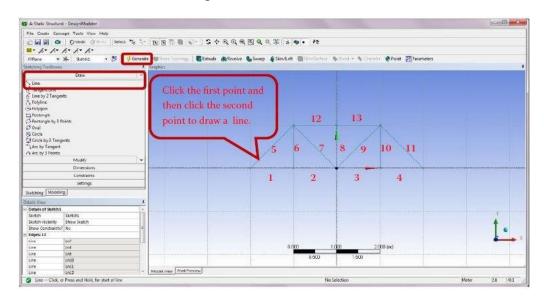
Ensure Line Bodies is checked in the Properties of Schematic A3: Geometry window. Doubleclick the Geometry cell to launch DesignModeler, and select "Meter" as length unit in the Units pop-up window.

File View Tools Units Help						
🎦 New 📸 Open 🛃 Save 🔣 Save As	. Import 🖓 Reconnect 😹 Refresh Project	🍠 Update Project 🛛 🕝	Project	Compact Mode		
oolbox 👻 🕈 🗙	Project Schematic	- 4 x	Propert	ies of Schematic A3: Geometry	*	ņ.
E Analysis Systems				A	В	
Design Assessment			1	Property	Value	
e Electric	▼A		2	= General	J.	
Explicit Dynamics	1 💹 Static Structural		3	Component ID	Geometry	1.
Fluid Flow (CFX)	2 🥏 Engineering Data 🗸 🖌		4	Directory Name	SYS	
Fluid Flow (FLUENT)	2 Geometry 2		5	Geometry Source		
Harmonic Response	4 Model		6	Geometry File Name		
Linear Buckling			7	Basic Geometry Options	57	
Magnetostatic			8	Solid Bodies	V	
Modal Random Vibration	6 🍘 Solution 👕 🧣	1	9	Surface Bodies		
Response Spectrum	7 🥪 Results 👕 🧣		10	Line Bodies		_
Rigid Dynamics	Static Structural	-	11	Parameters		
Shape Optimization			12	Parameter Key	DS	
Static Structural			13	Attributes	1	
Steady-State Thermal			14	Named Selections	(E)	_
Thermal-Electric			15	Material Properties	1	_
Transient Structural			16	Advanced Geometry Options		
💘 Transient Thermal 👻 👻		1	17	Analysis Type	3D	14
View All / Customize			18	Use Associativity		-

Step 5: Create Line Sketch

Click the Sketching tab and select Settings. Turn on Show in 2D and Snap under Grid options. Use the default value of "5 m" for Major Grid Spacing and "5" for Minor-Steps per Major.

Click a start point and then an end point in the Graphics window to draw a line. Draw 13 lines as shown in the sketch below. After completion, click Generate to create a line sketch.



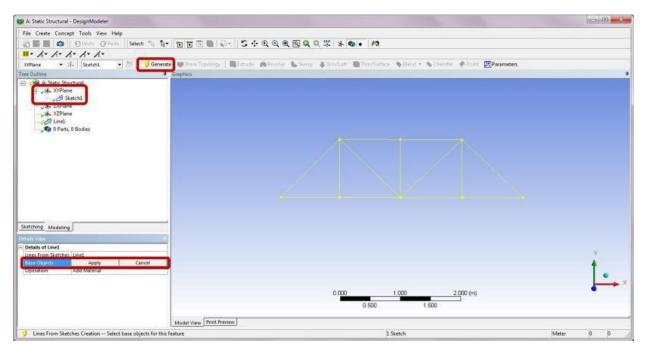
Step 6: Create Line Body from Sketch

Check off the Grid options under Settings of Sketching Toolboxes. Switch to the Modeling tab. Note that a new item named Sketch1 now appears underneath XYPlane in the Tree Outline.

Select Lines from Sketches from the Concept drop-down menu.

🚥 A: Static Structure	al - DesignModeler			
File Create Con	cept Tools View Help			
	Lines From Points	lelect to De	DDDD000-5+QQQQX+0+10	
11- 1- 2	Lines From Sketches			
	Lines From Edges			A
XiPlane	3D Curve		1 Divie Topology Bistrude Brevolve & Sweep & Skin/Loft Tran/Surface Allend	
Tree Outline	Split Edges	9	rephics	9
E St A St	opni, eoges Surfaces From Edges Surfaces From Edges			
100	Surfaces From Faces			
*	Cross Section			
	rts, 0 Bodies			
Sketching Modelin	ng			
Details View		Ģ		
 Details of Sketch1 				
Sketch	Sketch1			1
Sketch Visibility	Show Sketch			4
Show Constraints	7 No	12		
E Edges: 13	1.1.W			X
Line	Ln7		0.000 1.000	2.000 (m)
Line	Ln8 Ln9			
Line	Ln9 Ln10		0.500	1.500
Line	Inll			
Line	Ln12		Model View Print Preview	
A Ready	(MMA)		1 Sketch	Meter 10 0
C Ready			1 Sketon	Meter 0 0

Select Sketch1 from the Tree Outline and click Apply to confirm on the Base Objects selection in the Details of Line1. Click Generate to complete the line body creation.



Step 7: Create a Cross Section

Select a Cross Section of Rectangular from the Concept drop-down menu. A new item named Rect1 is now added underneath the Cross Section in the Tree Outline.

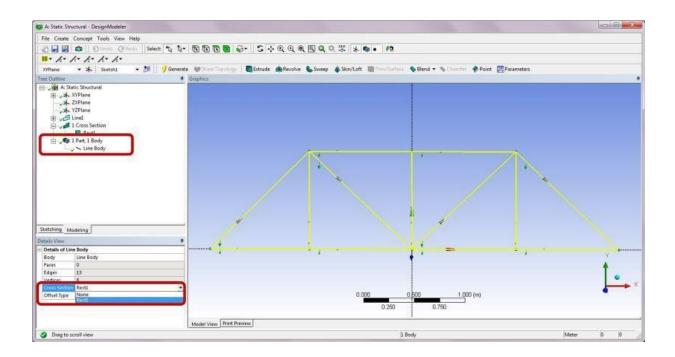
📖 A: Static Structural - DesignModeler			
File Create Concept Tools View Help	p		
Construction C	8 🥑 Generate - 🖤 Shire Topology 4 Graphics	장· S 상 및 및 및 및 및 및 및 및 사 이후 約 國Extrude 最Revolve & Sweep & Skin/Loft 圖Thin/Suffice	
Grading Cress Section Grading Cress Section Grad Secti	Restangeder Circular Tube Circular Tube Circular Tube Circular Section Section Section Section Het Section Retangular Tube User Defined		
Sketching Modeling			
Detaily View	4		
Details of Line Body Body Line Body Faces 0 Edges 13 Vetices 8 Oross Section: Not selected		8.403 L010 2.608 (m) 0.510 1.500	¥ • ×
Ø Ready	Model View Print Pr	niew 1 Body	Meter 0 0

3 A: Static Structural - DesignModeler		
File Create Concept Tools View Help		
2	· S · Q Q Q Q Q X * • • 12	
1 · h · h · h · h · h ·		
	🛛 🐻 Extrude 🚓 Revolve 🐁 Sweep 🐐 Skin/Loft 🔚 Thin/Surface 💊 Blend 🕶 🗞 Chamfer 🚸 Point 🕎 Pe	arameters
ree Outline # Etablica		
A Static Structural A Staticon A Static Structuran A Static Structuran A Static S		
Sketching Modeling		
Netals View		
Details of Rect1 A		
Sketch Rect1		2
Show Constraints? No		1
Dimensions: 2 8 0.06 m		•
H 0.06 m		
Edges: 4	0.000 0.050 0.100 (m)	
Line Ln22	0.025 0.075	
Line Ln23	0.075 0.075	
Line Ln24		
Line Ln25 - Model View Print Pre		
Orag to scroll view	1 Sketch	Meter 0 0

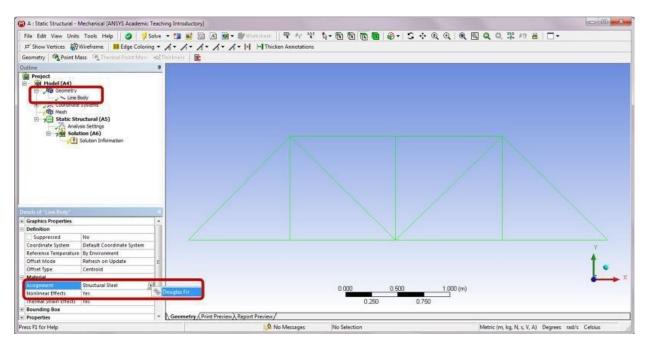
In the Details of Rect1 under Dimensions, enter "0.06 m" for both B and H.

Step 8: Assign Cross Section to Line Body

Select the Line Body underneath 1Part, 1 Body in the Tree Outline. In the Details of Line Body, assign Rect1 to the Cross Section selection. Click Close Design Modeler to exit the program.

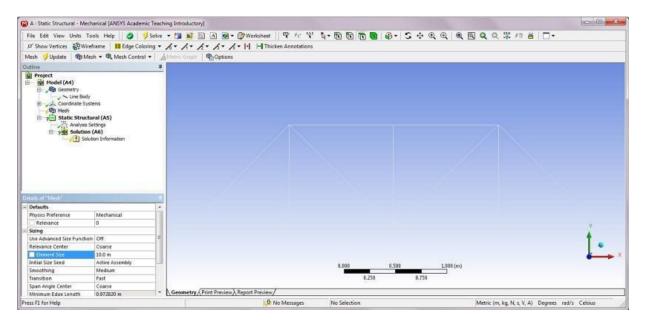


Step 9: Launch the Static Structural (ANSYS) Program Double-click the Model cell to launch the Static Structural (ANSYS) program. Note that in the Details of "Line Body" the material is assigned to Structural Steel by default. Click to the right of the Assignment field and select Douglas Fir from the drop-down context menu.



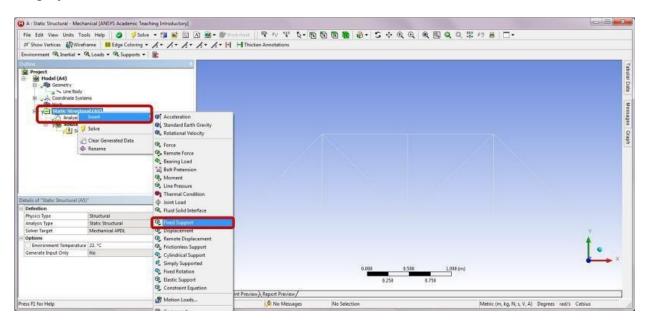
Step 10: Generate Mesh

In the Details of "Mesh," enter a fairly large number, say, "10 m," for the Element Size, to ensure each member is meshed with only one element. In the Outline of Project, right-click on Mesh and select Generate Mesh.



Step 11: Apply Boundary Conditions

In the Outline of Project, right-click on Static Structural (A5) and select Insert and then Fixed Support. After completion, a Fixed Support item is added underneath Static Structural (A5) in the project outline tree.

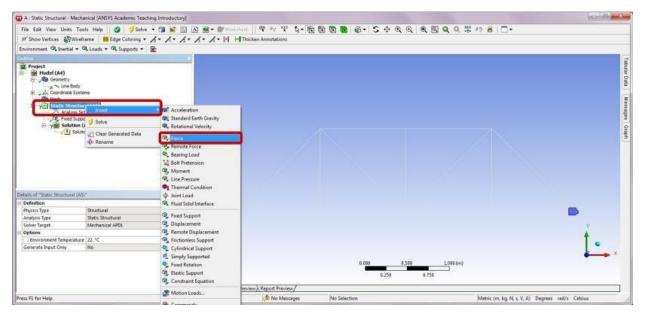


Select the two points as shown below in the Graphics window. In the Details of "Fixed Support," click Apply to confirm on the Geometry selection. After completion, a Fixed Support boundary condition will be added to the selected two points.

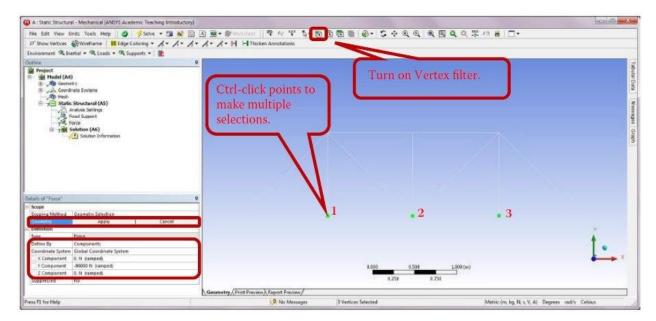
A : Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Teaching Int Content of the Static Structural - Mechanical (ANSYS Academic Structural - Mechanical (ANSYS Academic Structural - Mechanical (ANSYS Academic Structural - Mechanical - Mechani			(6 X
File Edit View Units Tools Help 👩 🦻 Solve 🕶 🗍	🛚 😢 🔯 🖉 🖉 Worksheet 🕿 Ar 😤 🔖 🔞	100 B+ 5 + Q Q Q Q Q	0、第 /2 音 □•
P' Show Vertices 🖓 Wireframe 🛛 🖬 Edge Coloring 🔹 🔏 🔹	/ * / * / * / * H H Thicken Annotations		
Emironment 🥵 Inertial + 🔍 Loads + 🔍 Supports + 👔			
Outline		Turn on Vertex f	
-20 Analyse satisfys -70 Finds Support -70 Finds Support -70 Finds Solution (Ab) -70 Solution Information	Ctrl-click points to make multiple selections.]	
Details of "Frand Support"			
Scope	V.		
Consist Multiple Consults Estadion	 1		• 2
Geogetry Apply Cance			· ·
Type Pixed Support			
Suppressed No			
			x
		9.800 0.598 L00	1 (m)
		8.250 0.758	
	Geometry (Print Preview) Report Preview/	21095-1 82205-1	
Press F1 for Help	6. No Messages	2 Vertices Selected: Distance = 4.0139 m	Metric (m, kg, N, s, V, A) Degrees rad/s Celulus

Step 12: Apply Loads

In the Outline of Project, right-click on Static Structural (A5) and select Insert and then Force.



Select the three points as shown below in the Graphics window. In the Details of "Force," click Apply to confirm on the Geometry selection. Also underneath the Details, change the Define By selection to Components and enter "-90000N" for the Y Component. A downward red arrow will appear on the selected three points in the Graphics window.



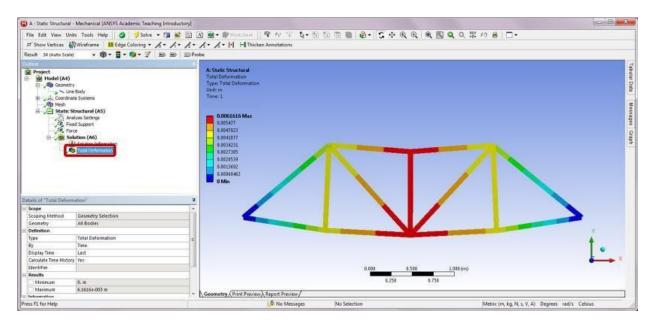
Alternatively, the load can be applied to each of the three points individually by inserting Force three times under Static Structural (A5). In this case, enter "-30000N" for the Y Component of each individual Force item.

Step 13: Retrieve Solution

Insert a Total Deformation item by right-clicking on Solution (A6) in the Outline tree.

A : Static Structural - Me	echanical (ANSYS Academic Teac	hing Introductory]				
File Edit View Units	Tools Help 🛛 🥑 🔰 Solve	• 📁 🖬 🗇 🖄 🖷 • 🕼 🗰	orksheet 🛛 🕾 🕂 🧐 🖓 - 🛐	10 10 10 S · S · Q	Q Q Q Q X 10 8	
		6. 1. 1. 1. h. h. H.				
Solution @ Deformation	+ @ Strain + @ Stress + @	Energy • 🍕 Probe • 🙀 Tools •	2. User Defined Result @ Coord	śinate Systems 👻 👰 Beam Results 👻 📗	6	
Continue Project Continue Continue Continue Static Strue Continue Static Strue Static Strue Continue Static Strue Continue Static Strue Continue Static Strue Continue Static Strue Continue Static Strue Continue Static Strue Static Static Strue Static Strue Static Strue Static Strue Static Strue Static Strue Static Strue Static Static Strue Static Static Strue Static Strue Static Strue Static Strue Static Strue Static Static Strue Static Static Strue Static Static Strue Static Static Strue Static Static Strue Static Static Static Strue Static Static Strue Static Static Static Strue Static Static Static Static Static Strue Static Static Static Strue Static Static Stati	stems ctural (AS) s Settings upport sty http:///internationality.com/ http://internationality.com/ http	1	Sectional			Tobusi Data Messapei Graph
	Solve Evaluate All Results		-s onecoonar			S
		Probe •	/			
	Clear Generated Data	Coordinate Systems >				
		Beam Results •				
Details of "Solution (A6)"		Beam Tool +				
Adaptive Mesh Refineme		2. User Defined Result				
Max Refinement Loops 1 Refinement Depth						
E Information	61	Commands]			Υ.
Status 3	loke Required			0.000 0	1500 L.008 (m) 0.750	<u>t-</u> .
		\ Geometry √P	rint Preview), Report Preview/			
Press F1 for Help			0 No Messages	No Selection	Metric (m, kg, N	s, V, A) Degrees rad/s Celsius

Right-click on Solution (A6) in the Outline tree and select Solve. The program will start to solve the model. After completion, click Total Deformation in the Outline to review the total deformation results.



Modeling tips: To get the reaction force, a Force Reaction probe can be inserted by right-clicking on Solution (A6) in the Outline tree as shown below.

A : Static Structural - Mechan	nical (ANSYS Academic Teaching	Introductory]				
File Edit View Units Tool	s Help 🛛 🥥 🔰 Solve 🔹	🗊 😼 🖾 🛆 🏽 🕷 🕫	Warkshart 🛛 🗣 fr 🦞 🗞 - 🕞	i 🕼 🐚 💼 🚳 + 🖸 🔅	QQ QQ Q Q 0 8 17 8 [1+
	ime 🛛 Edge Coloring 🔹 🔏					
Solution 🤏 Deformation 👻	🗣 Strain 🔹 🍕 Stress 🔹 🎭 Ene	rgy + 🍳 Probe + 🎃 Tools	- 😤 User Defined Result 🧠 Coo	ordinate Systems 👻 💁 Beam Results	• 🗈	
Project Model(A4) Model(A4)	d (AS) Inga It	Deformation				
- Orecto	🚽 Solve	Contact Tool	•			
	② Clear Generated Data 과 Rename	Probe Coordinate Systems Deam Results	Contention Strain Strain Strain Stress Op Position			ĺ.
etails of "Solution (A6)"			• • Position	0.001	8.500 L 808 (m)	
Adaptive Mesh Refinement		Beam Tool	Force Reaction		250 0.750	
Max Refinement Loops 1.		B. How Defined Read	Moment Keachon			
Refinement Depth 2. Information		Commands	Noint P		0 Tabular Data	
Status Solve	Required	6.	Image: Spring Image: Spring		3	
		Mensage	is Graph			le not con la Barro vice de contra de la contr
ess F1 for Help			0 No Messages	No Selection	Metric (m, k	I, N, s, V, A) Degrees rad/s Celsius

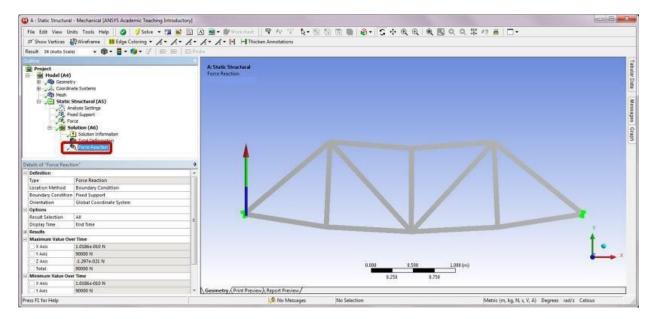
In the Details of "Force Reaction," select the Fixed Support as the Boundary Condition.

A : Static Structural - Mechanical (ANSYS Academic Te.)	eaching Introductory]	
File Edit View Units Tools Help 👩 🚽 Soli	alen • 🗊 💰 🖾 🖄 👾 🖤 Watchard 🗌 🗣 fr 🦞 🗞 🔞 🔞 🔞 🚳 🗳 • 💲 🕂 🍳	0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0
	* / * / * / * / * / * * + Thicken Annotations	
	🗣 Energy + 🔍 Probe + 🍘 Tools + 😭 User Defined Result 🔍 Coordinate Systems + 🔍 Beam Results + 👔	•
Outline		
Project Model (A4) Model (A4) Model (A4) Model Address Systems Model Address Model		
The Direction Details 2000		
	0 0.001	8.500 L101 (m)
ietalis of "Force Reaction". Definition	0 0.000 0.250	8.500 LE00 (m)
Intelling of "Force Reaction" Optimition Type Force Reaction	. 9,250	
Datalis of "Forus Reaction" Ordination Type Force Reaction Learning Methods - Englisher	1,230 AGeometry / Print Preview / Report Preview /	8,750
Intelais of "Force Reaction" Definition Type Force Reaction Instantion Attion Boundary Condition Strandor Science Force Strandor Science Force Science Force	. 9,250	
Intellis of "Fours Reaction" (Definition Type Force Reaction Coeffice Method Reurofan Condition	1,230 AGeometry / Print Preview / Report Preview /	8,750
India of "Force Reaction" Definition Typ: Force Reaction Foundary Condition Foundary Condition Found	1,230 AGeometry / Print Preview / Report Preview /	8,750
Istails of "Force Reaction" Cefinition Type Force Reaction Isoundary Constant Commentation Resultance Condition Commentation Result Result Selection All Display Time End Time	1,230 AGeometry / Print Preview / Report Preview /	8,750
table of "Force Reaction" Definition Type Force Reaction Contract of the second sec	1,230 AGeometry / Print Preview / Report Preview /	8,750
Intellik of "Force Reaction" Definition Type Force Reaction Institution Attacher Result Selection Result Selection	1,230 AGeometry / Print Preview / Report Preview /	8,750
Interfacts of "Force Reaction" Interfaction Type Force Reaction Type Force Reaction Constraints Force Reaction Constraints Force Reaction Force	Graph	8,750
Details of "Force Reaction" Opfinition Type Force Reaction Leadina Mithain Roundary Condition Contraction Weak Springs Revul Selection Dipplay Time Dipplay Time Buildary Mithain Maximum Value Over Time	1,230 AGeometry / Print Preview / Report Preview /	8,750

Right-click on Solution (A6) in the Outline tree and select Evaluate All Results.

A - Static Structural - Mechanical JANSYS Academic Teaching Introd	uctory!				
File Edit View Units Tools Help 👩 🥩 Solve 🕶 🛄 🖠	🕼 🕼 🗑 • 🕐 Worksheet 🛛 🕾 🎌 💱 🖒 • 🕅	nn n 8. 5 + Q 0	1	8 0.	
P Show Vertices					
Solution @ Deformation - @ Strain - @ Stress - @ Energy -		dinate Systems - 🤨 Beam Results - 👔			
Outine					-
Project B B Hodel (A4) B √ B cometry					Tabusar Data
B Coordinate Systems					8
Analysis Settings					Messages Graph
Analysis Settings					aŭe
Solution and					
Sol Insert					deni
Tab 🥩 Evaluate AX Results					2
Clear Generated Data					
alb Rename					
Details of "Solution (A6)"	¢.				
Adaptive Mesh Refinement					
Max Refinement Loops 1.					
Refinement Depth 2					
E Information					Y.
Status Post-processing Required					+
					•
					X X
		0.000 0.50	0 L000(m)		
		0.250	0.750		
	Geometry (Print Preview) Report Preview/				1
Press F1 for Help	Q No Messages	No Selection	6.4	etric (m, kg, N, s, V, A) Degrees rad/s	Celdus
	The Interseduce			and the other of all of a posterior rates	

After completion, click Force Reaction in the Outline to review results.



Note here that the reaction force is found to be 90,000 N in the positive Y-direction. This is because a boundary condition has been applied earlier to the two fixed ends in one step (see Step 11). To avoid summing of the force reaction, two fixed conditions can be inserted instead in Step 11, one for each end. The reaction forces at an individual support can then be displayed by selecting the support of interest from the drop-down menu of Boundary Condition in the Details of "Force Reaction."

Ex. No :

Date :

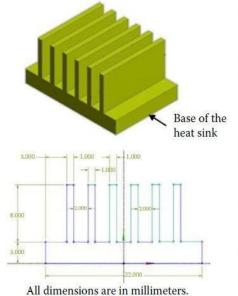
THERMAL ANALYSIS

Problem Description: Heat sinks are commonly used to enhance heat dissipation from electronic devices. In the case study, we conduct thermal analysis of a heat sink made of aluminum with thermal conductivity k = 170 W/(m K), density $\rho = 2800$ kg/m³, specific heat c = 870 J/(kg K), Young's modulus E = 70 GPa, Poisson's ratio v = 0.3, and thermal expansion coefficient $\alpha = 22 \times 10^{-6/\circ}$ C. A fan forces air over all surfaces of the heat sink except for the base, where a heat flux q' is prescribed. The surrounding air is 28°C with a heat transfer coefficient of h = 30 W/(m²°C).

Part A: Study the steady-state thermal response of the heat sink with an initial temperature of 28°C and a constant heat flux input of $q' = 1000 \text{ W/m}^2$.

Part B: Suppose the heat flux is a square wave function with period of 90 s and magnitudes transitioning between 0 and 1000 W/m². Study the transient thermal response of the heat sink in 180 s by using the steady-state solution as the initial condition.

Part C: Suppose the base of the heat sink is fixed. Study the thermal stress response of the heat sink by using the steady-state solution as the temperature load.



Material: Aluminum $k = 170 \text{ W}/(\text{m} \cdot \text{K})$ $\rho = 2800 \text{ kg/m}^3; c = 870 \text{ J}/(\text{kg} \cdot \text{K})$ $E = 70 \text{ GPa}; \nu = 0.3$ $\alpha = 22 \times 10^{-6}/^{\circ}\text{C}$

Boundary conditions: Air temperature of 28°C; $h = 30 \text{ W}/(\text{m}^2 \cdot ^{\circ}\text{C})$.

Steady state: $q' = 1000 \text{ W/m}^2$ on the base. *Transient*: Square wave heat flux on the base.

Initial conditions:

Steady state: Uniform temperature of 28°C. Transient: Steady-state temperature results.

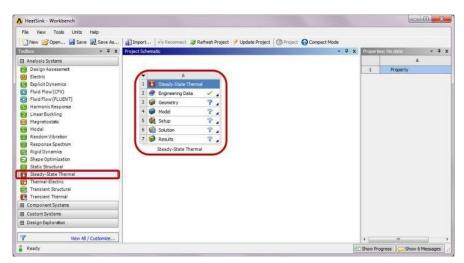
Part A: Steady-State Thermal Analysis

Step 1: Start an ANSYS Workbench Project

Launch ANSYS Workbench and save the blank project as HeatSink.wbpj.

Step 2: Create a Steady-State Thermal Analysis System

Drag the Steady-State Thermal icon from the Analysis Systems Toolbox window and drop it inside the highlighted green rectangle in the Project Schematic window.



Step 3: Add a New Material

Double-click on the Engineering Data cell to add a new material. In the following Engineering Data interface which replaces the Project Schematic, type "Aluminum" as the name for the new material, and double-click Isotropic Thermal Conductivity under Thermal in the Toolbox window. Change the Unit to "Wm⁻¹K⁻¹" and enter "170" for Isotropic Thermal Conductivity in the Properties window. Click the Return to Project button to go back to Project Schematic.

File Edit View Tools Units Help									_			
New 🧭 Open 🛃 Save 😹 Save As		port	R	efresh Pr	piect 🥖 Update	Project GRetu	m to R	Proie	ct	Compa	ct Mode 🛛 🏹 🛍	
- -		of Schematic A2: Engineer				-		4	-		Properties Row 2! Isotr	rocThemalC + 4 3
3 Thermal		A	в	c	1	D					A	8
Conductivity	1	Contents of Engineering Data	-	Source	8	Description				1	Temperature (C) 🌙	
	2	Material								2		170
	-3	Structural Steel	直	🚍 G.	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1			0.25		17		
	4	Aluminum	123	1								
	۲	Click here to add a new matenal										
	Propert	Properties of Outline Row 4; Aliminum 🔹 🔻 🗴										
		A			8	c	D	-	E	-		
	1	Property			Value	Unit		×		1000	Properties Row 2: Isob	vectores c · · · · ·
	2	Isotropic Ther	mal		170	Wm^-1K^-1		-		É 25	0.4	
		1.0								Thermal Conductivity [W	0	mal Conductivity
View All / Customize										hermal	-1 Temp	o i erature [C]

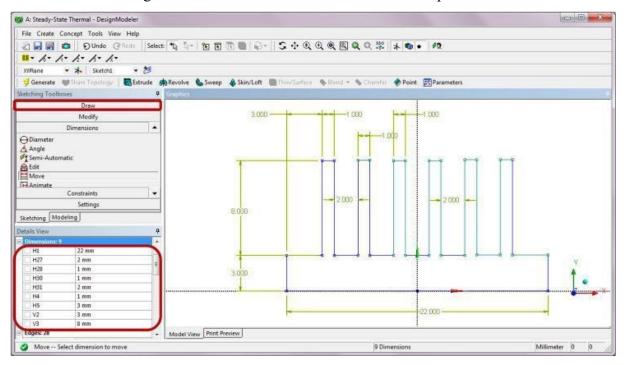
Step 4: Launch the Design Modeler Program

Double-click the Geometry cell to launch Design Modeler, and select "Millimeter" in the Units pop-up window.

New 📴 Open 🛃 Save 🔀 Save As	import	🖓 Reconnect 🥔 Refresh Project 🗡 Update Project 🌘 Project 🌘	Compact I	fode		
roolbox 👻 🗣 🗙	Project 5ch	ematic 👻 🖣	× Prop	erbes of Schematic A3: Geometry		ů
Analysis Systems				A	в	
Design Assessment			1	Property	Value	
Electric	-	A	2	= General		
Explicit Dynamics	1	🛄 Steady State Thermal	3	Component ID	Geometr	y .
S Fluid Flow (CFX)	2	🔗 Engineering Data 🛛 🗸	4	Directory Name	SYS	
Fluid Flow (FLUENT)	3	Geometry ?	5	= Geometry Source		
Marmonic Response	4	Model ?	6	Geometry File Name		_
Linear Buckling	e.	de auso	7	 Basic Geometry Options 		
Magnetostatic	2		8	Solid Bodies		
Modal Random Vibration	6	Solution 😨 🖌	9	Surface Bodies		
Response Spectrum	7	😥 Results 🛛 🍸 🖌	10	Line Bodies	间	_
Rigid Dynamics		Steady-State Thermal	11	Parameters		_
Shape Optimization			12	Parameter Key	DS	-
Static Structural			13	La construction of the second s	171	_
Steady-State Thermal			14	Named Selections	111	_
Thermal-Electric			15	Material Properties	191	
Transient Structural			16	and the second se	in the second se	
🔁 Transient Thermal			17	English Kondutarin color (holo constantic), dhat at an at an directed	30	
Component Systems			18	and the second se	1	
E Custom Systems			19		10	-
Design Exploration			20		111	_
				Dearder Mode Sauer	-	_
View Al / Customize	1		21	Updated File	10	

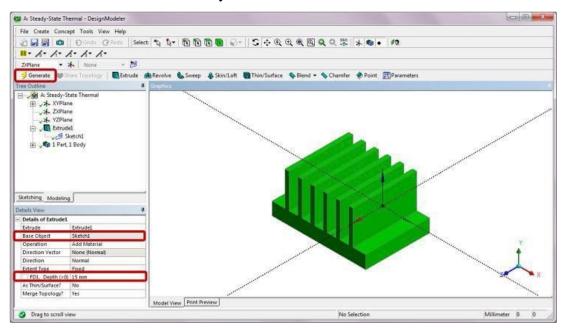
Step 5: Create a Profile Sketch

Click on the Sketching tab. Select the Draw toolbox and draw a 2D profile as shown below.



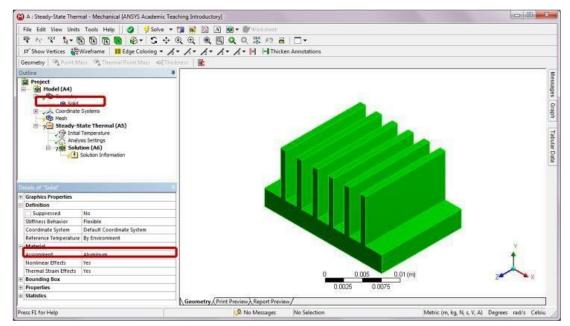
Step 6: Create an Extruded Body

Switch to the Modeling tab and click on the Extrude feature. The default Base Object is set as Sketch1 in the Details of Extrude1. Change the extrusion depth to 15 mm in the field of FD1, Depth and click Generate. A solid body is created as shown below.



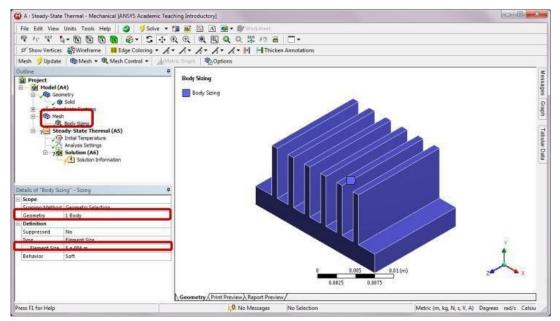
Step 7: Launch the Steady-State Thermal Program

Double-click on the Model cell to launch the Steady-State Thermal program. Click on the Solid under Geometry in the Outline tree. In the Details of "Solid," click to the right of the Material Assignment field and select Aluminum from the drop-down menu.

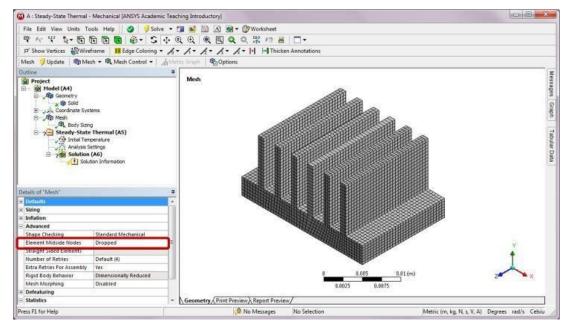


Step 8: Generate Mesh

Right click on Mesh in the Project Outline. Select Insert and then Sizing from the context menu. In the Details of "Face Sizing," enter "5e-4 m" for the Element Size. Click on the top, bottom surfaces, and the side walls of the guitar in the Graphics window and apply the five faces to the Geometry selection.

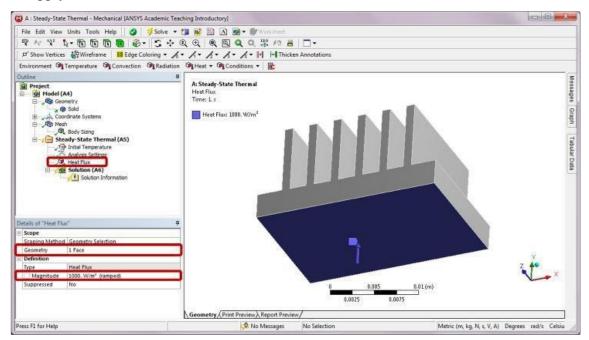


In the Details of "Mesh," select Dropped for the Element Midside Nodes under the advanced option. This helps reduce the total number of nodes to an acceptable level not exceeding the requested resources of educational licenses. Right-click on Mesh and select Generate Mesh.

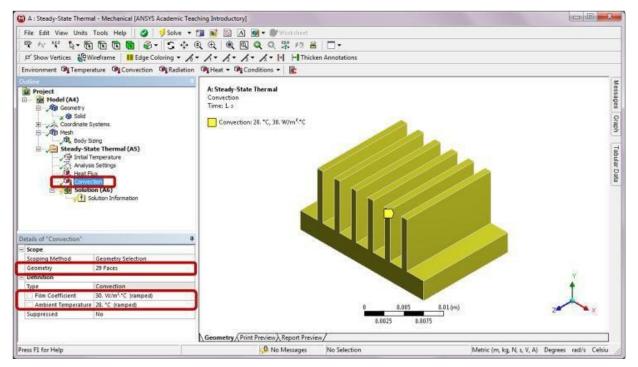


Step 9: Apply Boundary Conditions

Right-click on Steady-State Thermal (A5). Choose Insert and then Heat Flux from the context menu. Apply a heat flux of 1000 W/m^{-2} to the base of the heat sink.

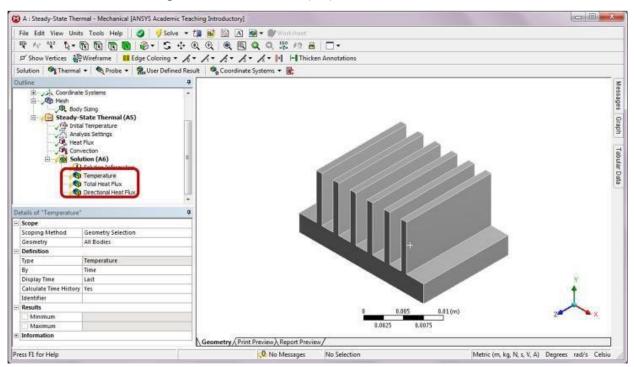


Right-click on Steady-State Thermal (A5). Choose Insert and then Convection from the context menu. In the Details of "Convection," enter 30 W/($m^{2\circ}C$) for Film Coefficient and 28°C for Ambient Temperature to all surfaces (a total of 29 faces) except for the base of the heat sink.

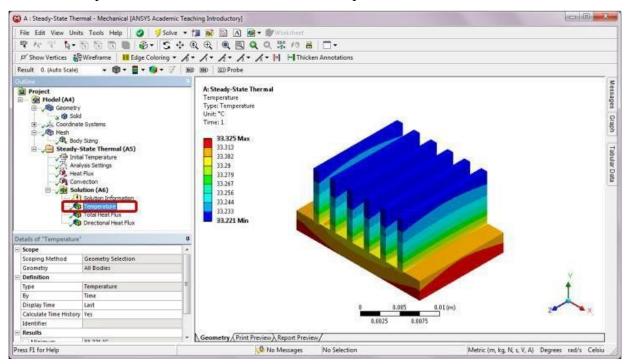


Step 10: Solve and Retrieve Results

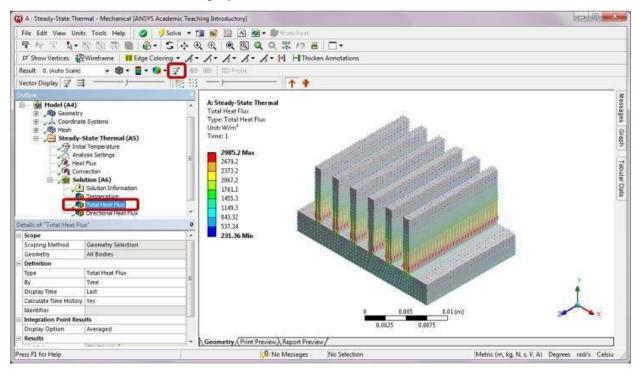
Right-click on Solution (A6) in the Outline, and insert Temperature, Total Heat Flux, and Directional Heat Flux to the solution outline. In the Details of "Directional Heat Flux," set the Orientation to Y-axis. Right-click on Solution (A6) and click Solve.



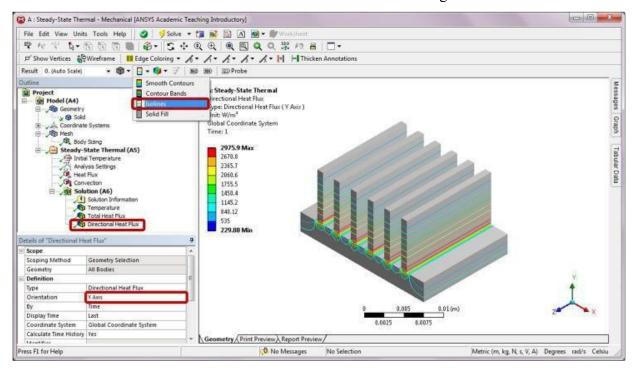
Click on Temperature in the Outline to review the temperature distribution.



Click on Total Heat Flux to display the heat flux with directional arrows.



Click on Directional Heat Flux to review the heat flux isolines along Y-axis.



Part B: Transient Thermal Analysis

Step 1: Add a Transient Thermal Analysis System

Drag the Transient Thermal icon from the Analysis Systems Toolbox window and drop it onto the Solution cell of the highlighted Steady-State system in the Project Schematic.

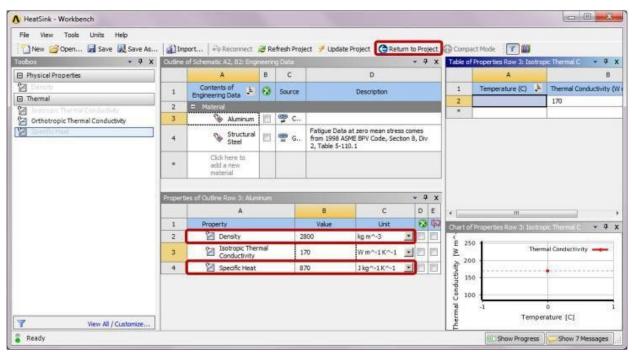
File View Tools Units Help	. 🎼 Import 🗟 Reconnect 🖉 Refresh Project 🦻	Update Project 🛛 🕲 Project 🕢 Co	mpact Mod		
Tooloox - 🗣 🗙	Project Schematic	* 7 X	Property	es: No data	* 7 >
Analysis Systems	in a second s			A	8
Design Assessment Electric Explicit Dynamics Enplicit Dynamics Fluid Flow (CFX) Fluid Flow (FLUENT) Sel Assessment Magnetostatic Modai Magnetostatic Modai Regionse Spectrum Regionse Spectrum Static Structural Static Structural Transient Structural Transient Thermal B Component Systems B Design Exploration	Image: Construction of the second	Share A21A4 Transfer A6	1	Property	Value
Yew All / Customize					

This creates a Transient Thermal system that shares data with Steady-State Thermal system. The temperature distribution from the Steady-State Thermal analysis is now the initial temperature for the Transient Thermal analysis. If the initial temperature is uniform for the Transient Thermal analysis, then this data sharing is not needed.

A HeatSink - Workbench									
File Wew Tools Units Help	4			1.1					
🎦 New 📴 Open 🛃 Save 🔣 Save As	Import	a Reconnect 🔒	Refresh Proje	ct 🍠 Update Proje	ct OProje	ect OCo	mpact Mod	e	
Toolbox 👻 🕈 🗙	Project Schen	natic				• 7 X	Propert	es: No data	* 7 >
E Analysis Systems	a bear the same							A	8
👩 Design Assessment	1 mm					-	1	Property	Value
(e) Electric	-	A		-	В				
Explicit Dynamics	1 23	Steady-State Therm	£	1 💽 Tran		li i	1		
G Fluid Flow (CFX)	2 🦪	Engineering Data	1	- 2 🦪 Eng	neering Data	1 1			
G Fluid Flow (FLUENT)	3 600	Geometry	1	- 3 (G) Geo	netry	1			
1 Harmonic Response	4	Model	1	- 4 📦 Mod					
Einear Buckling	1000		1	Contraction of the local division of the		4.			
Magnetostatic	intelligible second	Setup		• 5 🗶 Sen		<i>a</i> .			
Modai	- Linear -	Solution	1.	6 🔬 Solu		2.			
Random Vibration Response Spectrum	7 🎯	Results	× .	7 🕑 Res	lits	8.	1		
		Steady-State Therma	4	Tran	sient Therma	1			
Rigid Dynamics				-		_			
Static Structural									
Steady-State Thermal									
Thermal-Electric									
Transient Structural									
Transient Thermal									
E Component Systems									
Custom Systems									
Design Exploration									
Wew All / Customize			m		_	,			
🖡 Ready								Show Progress	Show 7 Messages

Step 2: Add Material Properties

Double-click on Engineering Data of Steady-State Thermal. Add a Density of 2800 kg/m³ and Specific Heat of 870 J/(kg K) to the Properties of Aluminum. Click Return to Project.

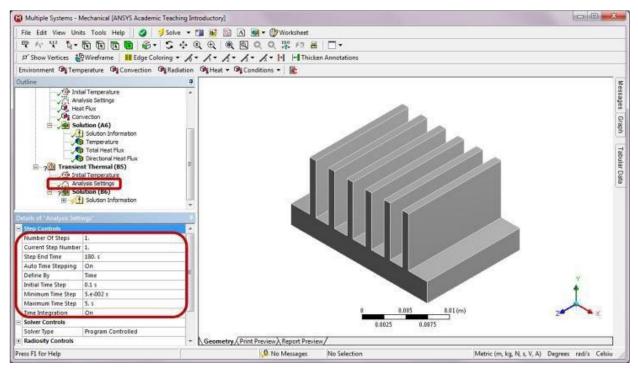


Step 3: Set Up Transient Thermal Analysis

Double-click on the Setup cell of the Transient Thermal system to launch the Multiple Systems– Mechanical program. Click Yes on the pop-up window to read the modified upstream data.

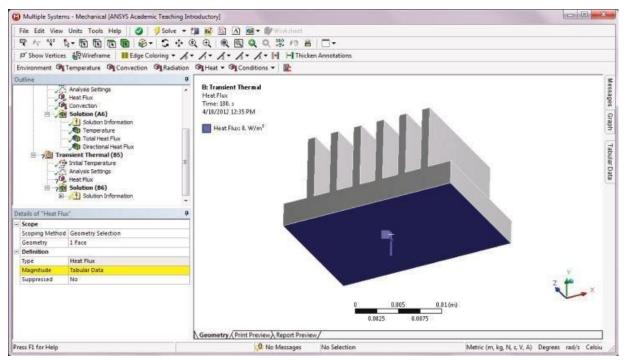
Select Analysis Settings from the Outline tree. In the Details of "Analysis Settings", under Step Controls, set the Step End Time to 180s. Change the Auto Time Stepping to On from Program Controlled. Change Defined By to Time.

The default values for the initial and the maximum time steps are small for this model. Set the Initial Time Step to 0.1. Set the Minimum Time Step to 0.05. Set the Maximum Time Step to 5. A small time step will help increase the accuracy of the model and also produce enough result steps so the animation will have smooth transition between solution steps.



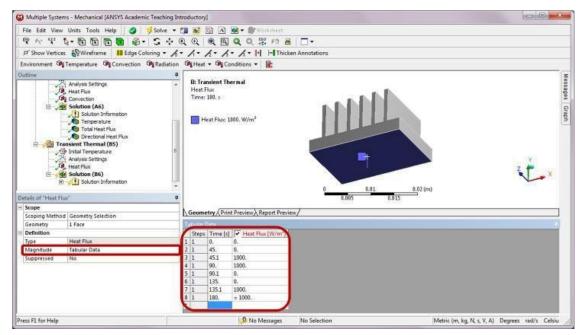
Step 4: Apply Cycling Heat Flux

Right-click on Transient Thermal (B5). Choose Insert and then Heat Flux from the context menu. In the Details of "Heat Flux," change Magnitude to Tabular Data, and apply the heat flux to the bottom surface of the heat sink as shown below.



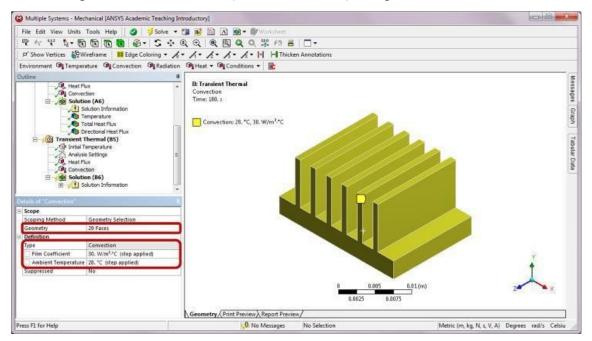
Click on Tabular Data on the right edge of the Graphics window, and then click on the push pin labeled AutoHide to display the Tabular Data window as shown below.

Enter the following values in the Tabular Data table.



Step 5: Apply Convective Boundary Condition

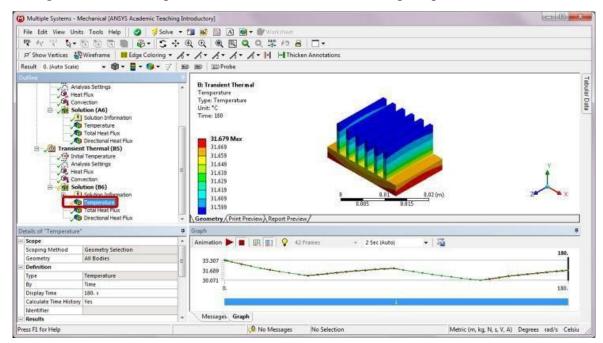
Right-click on Transient Thermal (B5). Choose Insert and then Convection from the context menu. In the Details of "Convection," enter 30 W/(m2 °C) for Film Coefficient and 28°C for Ambient Temperature to all surfaces (a total of 29 faces) except for the base of the heat sink.



Step 6: Solve and Retrieve Results

Right-click on Solution (B6) in the Outline, and insert Temperature, Total Heat Flux, and Directional Heat Flux. In the Details of "Directional Heat Flux," set the Orientation to Y-axis. Right-click on Solution (B6) and click Solve.

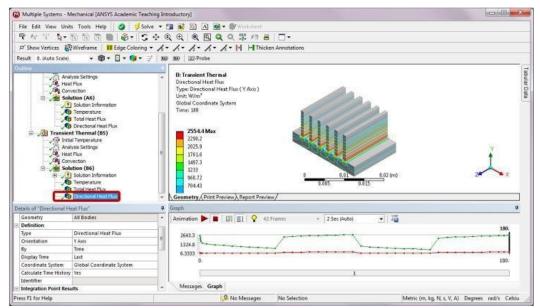
Click on Temperature in the Outline to review the distribution. To show Graph, click on Graph on the right side of the Graphics window, and then click on the push pin labeled AutoHide.



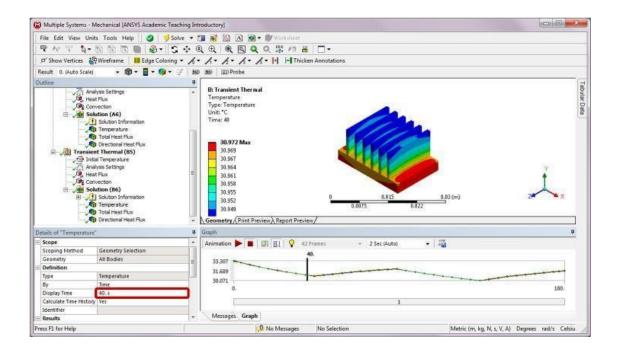
Click on Total Heat Flux to display the heat flux with directional arrows.

😮 Multiple Systems - Me	echanical [ANSYS Academic To	aching Introductory]	
File Edit View Units	Tools Help	olve - 🔝 👪 🔯 \Lambda 🗃 - 🕼 Wartsbeet	
		· · · · · · · · · · · · · · · · · · ·	
	and the second sec	g · / · / · / · / · IThicken Annotations	
Result 0. (Auto Scale)	· @ ·		
Vector Display 😴 📑			
Outline			
Conv Conv Conv Conv Conv Conv Conv Conv	ection tion (A6) Soluton Information Tengerature Total Heat Flux Directional Heat Flux E Thermal (B5) (Tengerature yss Settings Flux ection	B: Transient Thermal Total Heat Flux Type: Total Heat Flux Unit: W/m ³ 2295.2 2093.3 1215.4 1500.5 1235.6 970.68 970.86 0.0075 0.0075 0.03 (m) 0.03 (m)	Tabular Otata
Details of "Total Heat Flux	ić.	Graph	9
E Results	VI 81	🔺 Animation 🕨 🔳 🛄 🕎 42 Frames 🔷 2 Sec (Auto) 🔹 🥁	
	175.99 W/m ²		180.
	2560.1 W/m ²	2649.6	
Minimum Value Over T			
	9.6906 W/m ² 188.54 W/m ²	9,6906	
Maximum Maximum Value Over 1			180.
	542.18 W/m ²	1	
	2649.6 W/m ²		
maximum	20-12-20 (K) III	+ Messages Graph	
Press F1 for Help		9 No Messages No Selection Metric (m, kg, N, s, V, A) Degrees rad/s Celsiu

Click on Directional Heat Flux to review the heat flux isolines along Y-axis.



Modeling tips: In the case study, an end time of 180 s is specified in the transient setup. By default, the last set of results (solution at 180 s) from the transient analysis is used for graphics window display. To display results at a different time point, for example, temperature distribution at 40 s, you may change the Display Time from Last to 40s in the Details of "Temperature." Then right click on Solution (B6) and select Evaluate All Results. A result at the specified time will be displayed at the Graphics window as shown below.



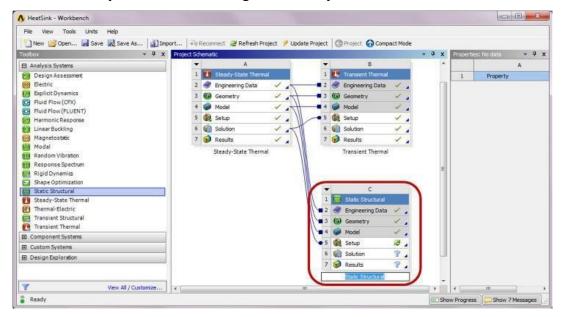
Part C: Thermal Stress Analysis

Step 1: Add a Static Structural Analysis System

Drag the Static Structural icon from the Analysis Systems Toolbox window and drop it onto the Solution cell of the highlighted Steady-State system in the Project Schematic

A HeatSink - Workbench File Wew Tools Units Help	A CONTRACT OF A	
	t 🕫 Reconnect 😹 Refresh Project 🏏 Update Project ③ Project 🚯 Compact Mode	(
Toolbox * 7 X	Project Schematic	👻 🖗 🗶 - Properbes: No dota 🛛 👻 🐺 🗶
Colocx • • • • • • Analysis Systems ① Design Assessment ② Design Assessment ② Electric >> Epplicit Dynamics ○ >> Fluid Flow (FLUENT) ©> Hamonic Response >> Linear Buckling >>> Magnetostatic >>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>	Project Governance A 1 Steady-State Thermal 2 Benjmeering Data 4 Model 5 Setup 6 Solution 7 Results Steady-State Thermal 5 State Athermal 5 State Athermal 5 State Athermal 5 State Athermal 5 State Athermal 5 State Athermal	A 1 Property
View All / Customize	<	, c , , ,

This creates a Static Structural system that shares data with the Steady-State Thermal system as shown below. The temperature distribution from the Steady-State Thermal analysis is now the load input for the Static Structural analysis. If a uniform temperature is specified as a load for the Static Structural analysis, then data sharing of the steady-state thermal solution is not needed.



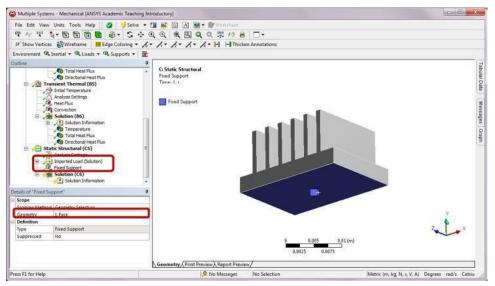
Step 2: Add Material Properties

Double-click on Engineering Data of Steady-State Thermal. Add Young's Modulus of 70 GPa, Poisson's ratio of 0.3, and Isotropic Instantaneous Coefficient of Thermal Expansion of 2.2E-5 1/°C to the Properties of Aluminum as shown below. Click Return to Project.

File Edit View Tools Units Help		Lin Reconnect		afrach D	niect 🧭 Lindate	Project CAR	harn to	Pro	int.	Comme	+Mode [7144				
Toolbox • 9 X	201000000000000000000000000000000000000	of Schematic A2, B2, C2:				(Thojact		-	×	100 COL 10 COL 1	Properties R			Elasticity	- 4	>
Physical Properties A		A	B	с		D			-		,	Ą			в	
Isotropic Secant Coefficient of Thermal Orthotropic Secant Coefficient of Therm	1	Contents of Engineering Data	•	Source		Description				1 2	Temperat		1	Young's Mo 7E+10	odulus (Pa)	
	3	Aluminum	ញ	🔮 c.					1							
Orthotropic Instantaneous Coefficient Constant Damping Coefficient Damping Factor(8)	4	Structural Steel	8	👕 G.	Fatigue Data from 1998 ASI 2, Table 5-110	et zero mean stres ME BPV Code, Sec 1.1										
Linear Elastic	Pressoon P	es of Outline Row 3: Akr		-	1		- 22	.0	×							
🔀 isetropic Flantite	ALC: SOL			D	E											
2 Orthotropic Elastidty	1	Property			Value	Unit			Ga							
Anisotropic Elastidty	2	27 Density			2800	kg m^-3	-	ñ	=							
Experimental Stress Strain Data Hyperelastic		Isotropic Ins									1000					
	3	Coefficient o	fThe	rmal	2.2E-05	C^-1	-			•			_		-	l
Plasticity Plasticity	4	B 1/3 Isotropic Ela	-				_	-		Chart of	Properties R	ow 🕂 Is	otopo	Basticity	* 4	2
	5	Derive from	routy		Young's Mod			-		Î I	1			ung's Modul		
🕀 Life	6	Young's Modulu		-	7E+10	Pa	-		m	1 0 (n01*)	-	-	TO	ing's Modul	lui	
Strength	7	Poisson's Ratio	-		0.3	10	-	-	一		-		-			
🗄 Gasket	8	Bulk Modulus			5.8333E+10	Pa	-	-	-	0.8 0.7 0.6 0.6			•		******	t
Thermal	9	Shear Modulus		-	2.6923E+10	Pa		-		£ 0.0	1					
Contraction of the second and the se	10	Isotropic The Conductivity	rmal		170	Wm^-1K^-1		1		s, 6uno,	L	-0.5	ó	0	5	1
View All / Customize	11	Specific Heat			870	3 kg^-1 K^-1			1	100		Ten	nperat	ure [C]		

Step 3: Set Up Static Structural Analysis

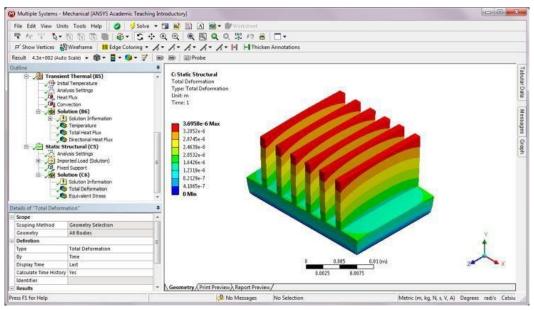
Double-click on the Setup cell of the Static Structural system to launch the Multiple Systems– Mechanical program. Click Yes on the pop-up window to read the modified upstream data. Note an Imported Load item is automatically added to Static Structural (C5) in the Outline tree. Right-click on Static Structural (C5) and insert a Fixed Support to the Outline. Apply the fixed support to the bottom face of the heat sink.



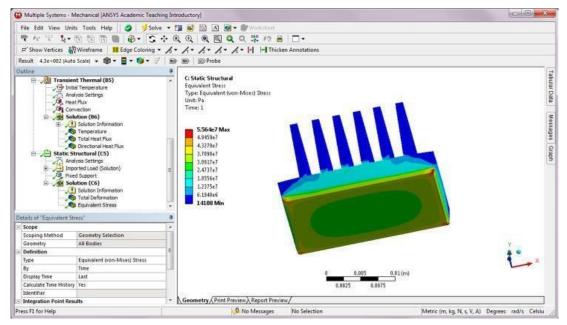
Step 4: Solve and Retrieve Results

Right-click on Solution (C6) in the Outline, and insert Total Deformation and Equivalent Stress to the outline. Then right-click on Solution (C6) and click Solve.

Click on Total Deformation in the Outline to review displacement results.



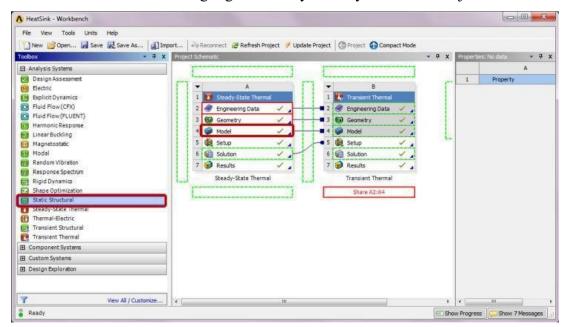
Click on Equivalent Stress in the Outline to review von Mises stress results.



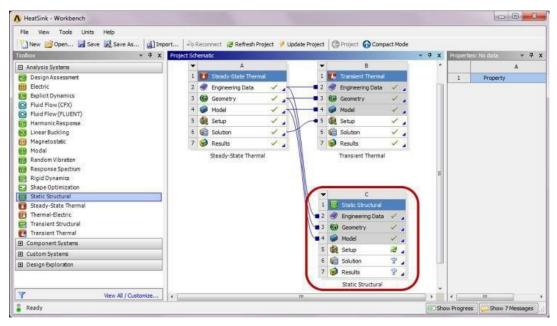
Modeling tips: To apply a uniform temperature load to the heat sink, you may add a Static Structural system that shares data with the Steady-State Thermal system at the model and above

levels. And then insert a constant Thermal Condition load to the heat sink in the Static Structural analysis.

To do this, first drag the Static Structural icon from the Analysis Systems Toolbox window and drop it onto the Model cell of the highlighted Steady-State system in the Project Schematic.

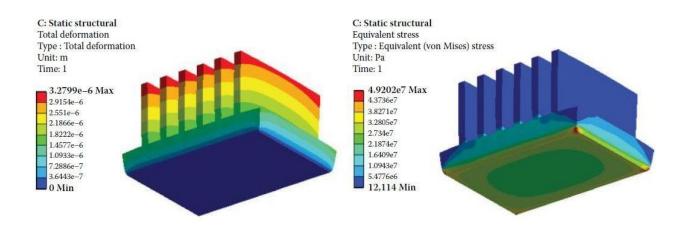


This creates a schematic where Static Structural is sharing data with Steady-State Thermal at the model and above levels as shown below.



Next, add Young's Modulus, Poisson's ratio, and Coefficient of Thermal Expansion data to the Properties of Aluminum in the Steady-State Thermal analysis template following Step 2 of Part C. Set up the Static Structural analysis by following Step 3 of Part C.

Insert a Thermal Condition load to the Static Structural (C5) in the Outline tree and apply a Fixed Support to the base of the heat sink. The total deformation and the equivalent stress distributions of the heat sink with a temperature increase of 10°C can be obtained as follows.



Ex. No : Date :

MODAL ANALYSIS OF A CANTILEVER BEAM

PROBLEM DESCRIPTION:

Consider an aluminum beam that is clamped at one end, with the following dimensions.

Length	Width	Height
4 m	0.346 m	0.346 m

The aluminum used for the beam has the following material properties.

Density	Youngs Modulus	Poisson Ratio
$2,700 \text{ kg/m}^3$	70x10 ⁹ Pa	0.35

Using ANSYS Workbench find the first six natural frequencies of the beam and the mode shapes

PRE-ANALYSIS & START-UP

PRE-ANALYSIS

The following equations give the frequencies of the modes and the mode shapes and are derived from Euler-Bernoulli Beam Theory.

$$\begin{split} & w_n = \alpha_n^2 \sqrt{\frac{EI}{ml^3}} \\ & n = 1, 2, 3, \dots \\ & \alpha_n = 1.875, 4.694, 7.855, \dots \\ & m = \rho V = \rho \cdot l \cdot h \cdot w \\ & I = \frac{w \cdot h^3}{12} \\ & w_1 = 1.875^2 \sqrt{\frac{70 \ E9 \ \frac{kg}{m \cdot s^2} \cdot \frac{0.346m \cdot (0.346m)^3}{12}}{\sqrt{2.7 \ E3 \ \frac{kg}{m^3} \cdot 4m \cdot 0.346m \cdot 0.346m \cdot (4m)^3}} = 111.7 \ \frac{r_{ad}}{s} = 17.8 \ Hz \\ & w_2 = 4.694^2 \sqrt{\frac{70 \ E9 \ \frac{kg}{m \cdot s^2} \cdot \frac{0.346m \cdot (0.346m)^3}{12}}{\sqrt{2.7 \ E3 \ \frac{kg}{m^3} \cdot 4m \cdot 0.346m \cdot 0.346m \cdot (4m)^3}} = 700.4 \ \frac{r_{ad}}{s} = 111.5 \ Hz \\ & w_3 = 7.855^2 \sqrt{\frac{70 \ E9 \ \frac{kg}{m \cdot s^2} \cdot \frac{0.346m \cdot (0.346m)^3}{12}}{\sqrt{2.7 \ E3 \ \frac{kg}{m^3} \cdot 4m \cdot 0.346m \cdot 0.346m \cdot (4m)^3}} = 1961.2 \ \frac{r_{ad}}{s} = 312.1 \ Hz \\ & y_i(x) = \cosh(\frac{\alpha_i x}{L}) - \cos(\frac{\alpha_i x}{L}) - \sigma_i(\sinh(\frac{\alpha_i x}{L}) - \sin(\frac{\alpha_i x}{L})) \\ & \alpha_i = 1.875, 4.694, 7.855, \dots \\ & \sigma_i = 0.73409, \ 1.018647, \ 0.9992245, \ \dots \end{split}$$

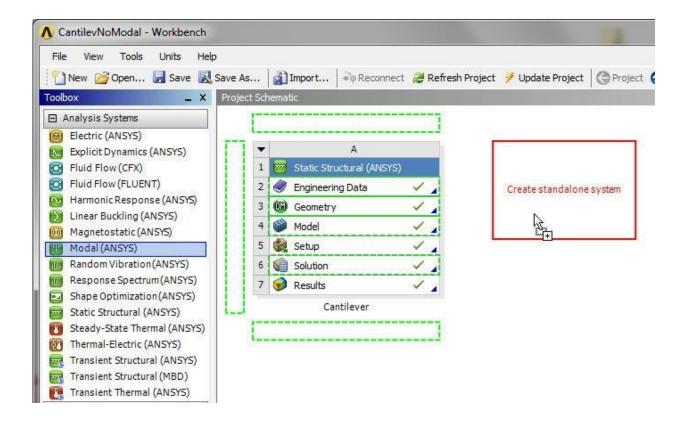
START ANSYS WORKBENCH & LOAD FILES

In this section we will launch ANSYS Workbench and then load the project file, "cantilever.wbpj" that was created in the "Cantilever Beam" tutorial.

Start > All Programs > ANSYS 15.0 > Workbench File > Open

Then choose the "cantilever.wbpj" file that you created in the "Cantilever Beam" tutorial. Modal (ANSYS) Project Selection

Left, click on *Modal ANSYS*, ^{III} ^{Modal (ANSYS)}, and drag it to the right of the "Cantilever" project. You should then see a red box to the right of the "Cantilever" project that says "Create standalone system" as shown below.



Now, release the left mouse button. Your *Project Schematic* window should now look comparable to the image below.

File View Tools Units Help										
🎦 New 🔗 Open 🛃 Save 🔣 Save	e As	1	Import 🏽 🗟 🖗 Reconnect	12 R	Refresh Projec	t 🥠	Up	date Project 🛛 🕻) Proj	ect 🕜
Foolbox _ X Pr	oject Sch	emati	ic							
🖃 Analysis Systems										
electric (ANSYS)	_									
Explicit Dynamics (ANSYS)			A			•		В		
🔇 Fluid Flow (CFX)	1		Static Structural (ANSYS)			1	T	Modal (ANSYS)		
S Fluid Flow (FLUENT)	2		Engineering Data	1		2	0	Engineering Dat	a 🗸	
Marmonic Response (ANSYS)	3	00	Geometry	1		3	0	Geometry	?	
Linear Buckling (ANSYS)	4		Model	~		4		Model	2	
Magnetostatic (ANSYS)	5	-	Setup	1	4		-	Setup	P	
Modal (ANSYS) Random Vibration (ANSYS)	10000		1999-1999-1999-1999-1999-1999-1999-199		<u>.</u>	322 3	-		P	
Random Vibration (ANSYS) Response Spectrum (ANSYS)	6	-	Solution	4	2	-	-	Solution		100 C
Shape Optimization (ANSYS)	7	1	Results	× .	X	7	P	Results	P	1
Static Structural (ANSYS)			Cantilever					Modal (ANSYS)	
Steady-State Thermal (ANSYS)										
Thermal-Electric (ANSYS)										
Transient Structural (ANSYS)										
Transient Structural (MBD)										
Transient Thermal (ANSYS)										

RENAME MODAL (ANSYS)

Double click on Modal (ANSYS) and rename it to "Cantilever Modal".

File View Tools Units Help									
🎦 New 对 Open 🛃 Save 🔣 Sav	e As	🗿 Import	€ø Reconnect	湕 Refre	sh Project	🥖 Up	odate Project	Proje	ect
Foolbox _ X Pr	oject Sch	nematic							
🖃 Analysis Systems									
 Electric (ANSYS) Explicit Dynamics (ANSYS) 	•		A		1	•	В		
Fluid Flow (CFX)	1	😇 Static Stru	ctural (ANSYS)	1		1	Modal (ANSYS)		
G Fluid Flow (FLUENT)	2	Engineerin	g Data	×		2 🦪	Engineering Dat	a 🗸	
Harmonic Response (ANSYS)	3	00 Geometry		× .		3 🈡	Geometry	?	
Dinear Buckling (ANSYS) Magnetostatic (ANSYS)	4	Model		1		4 📦	Model	P	-
Modal (ANSYS)	5	🍓 Setup		× .		5 🚷	Setup	P	
Random Vibration (ANSYS)	6	Solution		1		6	Solution	P	
Response Spectrum (ANSYS)	7	😥 Results		× 🖌		7 😥	Results	?	
Shape Optimization (ANSYS) Static Structural (ANSYS)		Car	itilever				Cantilever Moda	í.	

ENGINEERING DATA

In this section we will input the properties of aluminum (as defined in the the Problem

Specification) in to ANSYS.

First, double click *Engineering Data*, *Section 2018*, in the "Cantilever Modal" Project. Next, click where it says "Click here to add a new material" as shown in the image below.

File Edit View Tools Units	Help									
🎦 New 🚰 Open 🛃 Save 🔣	Save As	🕼 Import 🗟 Reconnect	2	Refre	h Pro	ject 🍠 Update Project	GReturn to Project			
Toolbox 🗕 🗙	Outline	Filter					-			
Physical Properties	•	А	В		C		D			
Linear Elastic	1	Data Source	1	Loc	ation		Description			
IsotropicElastidy	2	🦪 Engineering Data	1	B2		Contents filtered for M	odal (ANSYS).			
Orthotropic Elastidty Anisotropic Elastidty	3	General Materials		R		General use material sa	amples for use in various an			
Anisotropic Elastidy Experimental Stress Strain Data	4	General Non-linear Materials		R		General use material sa	amples for use in non-linear			
Hyperelastic	5	Explicit Materials		R		Material samples for use in an explicit anaylsis.				
Plasticity	6	I Hyperelastic Materials	Ī	R		Material stress-strain	data samples for curve fitting.			
🗉 Life	7	Magnetic B-H Curves	F	R		B-H Curvesamples spe	ecific for use in a magnetic an			
🕀 Strength	8					Quick access list and de	efault items			
	•		-	111		1	- F			
	Outline	of Schematic B2: Engineering Data					_			
	-	A		В	С		D			
	1	Contents of Engineering Data	J.	8	S.,	Desc	ription			
	2	🗖 Material								
	3	📎 Structural Steel			8	Fatigue Data atzero me 1998 ASME BPV Code, 5-110.1				
	*	Click here to add a new ma	ateria	I						
				à						

Next, enter "Aluminum" and press **enter**. You should now have Aluminum listed as one of the materials in table called "Outline of Schematic B2: Engineering Data", as shown below.

Outline	of Schematic B2: Engineering Data			-
-	А	В	С	D
1	Contents of Engineering Data 🗦	8	S.,	Description
2	Material			
3	📎 Structural Steel		@	Fatigue Data atzero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
4	2 📎 Aluminum			
*	Click here to add a new material			

Then, (expand) Linear Elastic, as shown below.

🔥 CantilevNoModal - Workbench
File View Tools Units Help
🎦 New 对 Open 🛃 Save 📓
Toolbox _ X
Physical Properties
Linear Elastic
🔀 Isotropic Elasticity
🔁 Orthotropic Elasticity
🔁 Anisotropic Elasticity
Experimental Stress Strain Data
Hyperelastic ■
❶ Life

Now, (Double Click) Isotropic Elasticity. Then set Young's Modulus to 70e9 Pa and set **Poisson's Ratio** to 0.35, as shown below.

Propertie	Properties of Outline Row 4: Aluminum							
•	А	В	С					
1	Property	Value	Unit					
2	🖃 🎦 Isotropic Elasticity							
3	Derive from	Young's Modulus and Poisson's Ratio 👻						
4	Young's Modulus	7E+10	Pa					
5	Poisson's Ratio	0.35						
6	Bulk Modulus	7.7778E+10	Pa					
7	Shear Modulus	2.5926E+10	Pa					

Next, (expand) Physical Properties, as shown below.

▲ CantilevNoModal - Workbench
File View Tools Units Hel
🌓 🎦 New 💕 Open 层 Save 📓
Toolbox _ X
Physical Properties
🔁 Density
🛛 🔁 Isotropic Secant Coefficient of
🛛 🔁 Orthotropic Secant Coefficient
🛛 🔁 Isotropic Instantaneous Coeffi
🛛 🔁 Orthotropic Instantaneous Coe
🔁 Constant Damping Coefficient
🚰 Damping Factor (β)
Experimental Stress Strain Data ■
Hyperelastic ■
⊞ Life

Now, *(Double Click) Density*. Then, set *Density* to 2,700 kg / m³, as shown below.

Properti	Properties of Outline Row 4: Aluminum								
•	A	В	С						
1	Property	Value	Unit						
2	🔁 Density	2700	kg m^-3 🔻						
3	🖃 🎦 Isotropic Elasticity								
4	Derive from	Young's Modulus and Poisson's Ratio 🔻							
5	Young's Modulus	7E+10	Pa 🔻						
6	Poisson's Ratio	0.35							
7	Bulk Modulus	7.7778E+10	Pa						
8	Shear Modulus	2.5926E+10	Pa						

Now, the material properties for Aluminum have been specified. Lastly, (Click) Return To

Project. GReturn to Project

Save

Save your project now and periodically, as you work. ANSYS does not have an auto-save feature.

GEOMETRY

Attach Geometry from Cantilever to Cantilever Modal

The geometry for the "Cantilever Beam Modal Analysis" tutorial is the same as the geometry for the "Cantilever Beam" tutorial. Instead of recreating the geometry, we will simple attach the geometry from the Static Structural Analysis System (Cantilever) to the Modal Analysis System (Cantilever Modal). In order to attach the geometry, *(left click) Geometry* in the "Cantilever" project and drag it to *Geometry* in the "Cantilever Modal" project, as shown below.

ct Sch	nema	tic							
•		А			•		В		
1		Static Structural (ANSYS)			1	T	Modal (ANSYS)		
2	٢	Engineering Data	\checkmark	4	2	٢	Engineering Data	\checkmark	
3	00	Geometry	\checkmark		3	Ν	Share A3		
+	9	Model	~	4	4	١. T	Model	7	
5		Setup	~	4	5	٢	Setup	7	
6	G	Solution	~	4	6	G	Solution	?	
7	@	Results	~	4	7	۲	Results	7	
		Cantilever					Cantilever Modal		

Then release the left mouse button. You should now see that the geometries are shared as shown in the following image.

Project	Project Schematic										
	•		А				•		В		
	1		Static Structural (ANSYS)				1	T	Modal (ANSYS)		
	2	0	Engineering Data	\checkmark	4		2	٢	Engineering Data	~	4
	3	8	Geometry	\checkmark		-	3	OM	Geometry	~	
	4	6	Model	 Image: A second s	4		4	۲	Model	2	4
	5	٢	Setup	\checkmark	4		5	٢	Setup	?	
	6	1	Solution	\checkmark	4		6	(iii)	Solution	?	4
	7	@	Results	× .	4		7	@	Results	7	4
			Cantilever						Cantilever Modal		

Save

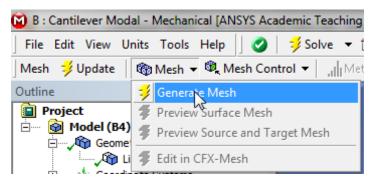
MESH

Launch Mechanical

(double click) Model, ^{Model}, in the "Cantilever Modal" project.

Generate Default Mesh

First, (click) Mesh in the tree outline. Next, (click) Mesh > Generate Mesh as shown below.

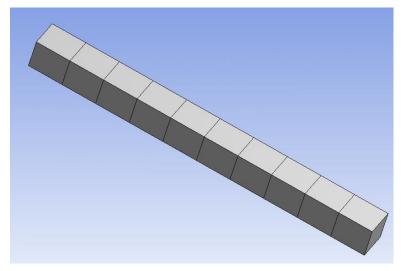


Size Mesh

In this section we will size the mesh, such that it has ten uniform elements. In order to size the mesh, first expand *Sizing* located within the *Details of "Mesh"* table. Next, set *Element Size* to 0.40 m, as shown below.

De	tails of "Mesh"	P
-	Defaults	
	Physics Preference	Mechanical
	Relevance	0
	Sizing	
	Use Advanced Size Function	Off
	Relevance Center	Coarse
	Element Size	0.40 m
	Initial Size Seed	Active Assembly
	Smoothing	Medium
	Transition	Fast
	Span Angle Center	Coarse
	Minimum Edge Length	4.0 m
+	Inflation	~
+	Advanced	
+	Pinch	
+	Statistics	

Now, *(click) Mesh > Generate Mesh* in order to generate the new mesh. You should obtain the mesh, that is shown in the following image.



Note that in this simulation we are working with beam elements, which are simply line segments. As a visualization tool ANSYS displays a beam with width and height. In order to display the actual mesh *(click) View > (deselect) Thick Shells and Beams*. You will then see the mesh displayed in its native form.

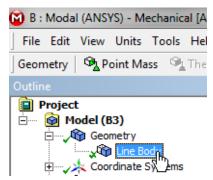


Save

PHYSICS SETUP

Material Assignment

At this point, we will tell ANSYS to assign the Aluminum material properties that we specified earlier to the geometry. First, *(expand) Geometry* then *(click) Line Body*, as shown below.



Then, *(expand) Material* in the "Details of Line Body" table and set *Assignment* to Aluminum, as shown below.

De	tails of "Line Body"	ą		
+	Graphics Properties	1		
Ξ	Definition		1	
	Suppressed	No	1	
	Coordinate System	Default Coordinate		
	Reference Temperature	By Environment	1	
	Offset Mode	Refresh on Update	1	
	Offset Type	Centroid	1	
-	Material		1	
	Assignment	Structural Steel	L	
	Nonlinear Effects	Yes	1	Aluminum
	Thermal Strain Effects	Yes	Π	
+	Bounding Box		1	
+	Properties		1	
+	Statistics		1	

At this point your "Details of Line Body" table, should look comparable to the following image.

De	tails of "Line Body"	4
+	Graphics Properties	
	Definition	
	Suppressed	No
	Coordinate System	Default Coordinate
	Reference Temperature	By Environment
	Offset Mode	Refresh on Update
	Offset Type	Centroid
	Material	
	Assignment	Aluminum
	Nonlinear Effects	Yes
	Thermal Strain Effects	Yes
Ŧ	Bounding Box	
Ŧ	Properties	
Ŧ	Statistics	

Fixed Support

First, (right click) Modal > Insert > Fixed Support, as shown below.



Next, click the *vertex selection filter* button, **(b**). Then, click on the left end of the beam and apply it as the *Geometry* in the "Details of Fixed Support" table.

Constrain Beam to XY Plane

In this section the beam's motion will be constricted to the xy plane.

First, (right click) Modal > Insert > Displacement, as shown below.

File Edit View Units Tools Help Image: Solve → the solution of	😭 C : Cantilever Modal Co	nfined to)	Y Plane -	Mech	nanical [ANSYS	Acade	mic	Teachi
Outline 9 Project Model (C4) Model (C4) Frequency Coordinate Systems 4/19, Mesh Fixed Support Fixed Support Fixed Remarked Support Fixed Remarked Fixed Remarked Support Fixed Rotation Fixed Rotation Fixed Rotation Elastic Support	File Edit View Units	Tools He	lp 📙 🥑	\$	Solve 🤅	- t	Rb¢	N	A
Project C: Ca Model (C4) Frequency Coordinate Systems 4/19, Mesh Fixed Support Mesh Fixed Support Fixed Solution Fixed Support Fixed Solution Clean Project Solution Fixed Support Fixed Support Fixed Solution Fixed Support Fixed Remarked Fixed Remote Displacement Fixed Remarked Fixed Rotation Fixed Rotation Fixed Rotation	Environment 🧐 Inertial	🗸 🔍 Load	ls 👻 🔍 S	uppo	orts 💌	B C			
Commands	Outline Project Project Outline Outl	ystems Insert Solve Clean	•		Therma Fixed St Displac Remote Friction Cylindr Simply Fixed Ro Elastic S Constra	Il Condit upport ement Displac less Sup ical Support otation Support int Equa	emen port port ed		C: Can Moda Freque 4/19/2

Next, click the *edge selection filter* button, **(b)**. Then, click on the geometry and apply it as the *Geometry* in the "Details of Displacement" table. Lastly, set *Z Component* to 0, as shown below.

=	Scope						
	Scoping Method	Geometry Selection					
	Geometry	1 Edge					
Ξ	Definition						
	Туре	Displacement					
	Define By	Components					
	Coordinate System	Global Coordinate System					
	X Component	Free					
	Y Component	Free					
	Z Component	이	5				
	Suppressed	No					

Save

Numerical Solution

Specify Results (Deformation)

Here, we will tell ANSYS to find the deformation for the first six modes. Then, we will be able to see the shapes of the six modes. Additionally, we will be able to watch nice animations of the six modes.

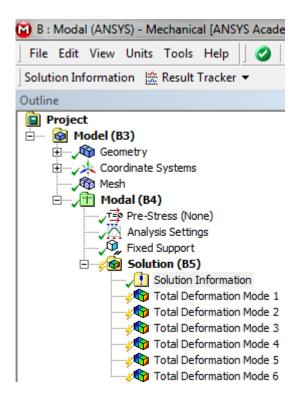
In order to request the deformation results (*right click*) *Solution* > *Insert* > *Deformation* > *Total* as shown below.

🔯 B : Modal (ANSYS) - Mechanical	and the second second				and the second second
File Edit View Units Tools	Help 🔤 🧭 🚺	ジ Solve 🔻 🗄	🔺 🚺 🚺	- 10	Worksheet
Solution 🍕 Deformation 🔻 🎭	Strain 🔻 🍕 Stre	ss 🔻 🔍 🎨 Prol	be 🔻 🙆 Tools 🔻		User Defined Result
Outline		4			
Project Model (B3) Model (B3) Model (B4) Model (B					
Solution I	Insert	Defor	mation	•	📽 Total
	🖉 Clean	Coord	linate Systems	•	💁 Directionଧା
	a <u>l</u> b Rename	🕵 User [Defined Result		
		Comr	mands		

Then, rename "Total Deformation" to "Total Deformation Mode 1". In order to do so *(right click) Total Deformation > Rename*. Next, set *Mode* to 1 as shown in the image below.

Details of "Total Deformation"											
Ξ	Scope										
	Scoping Method Geometry Selection										
	Geometry	All B	od	ies							
	Definition										
	Туре	Tota	I D	eformatior	n						
	Mode	1	T	•			•				
	Identifier										
	Results										
	Minimum										
	Maximum										
Ŧ	Information										

Repeat, this process for the other 5 modes. Make sure that you set *Mode* to the respective mode number. At this point, your *Outline* should look the same as the following image.



Run Calculation

In order to run the simulation and calculate the specified outputs, click the Solve button,

🞺 Solve

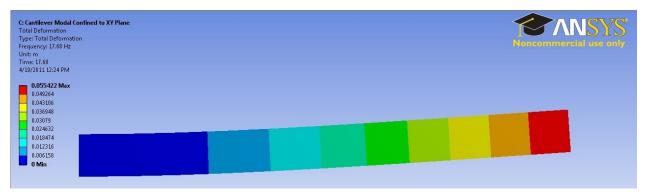
Save

Numerical Results

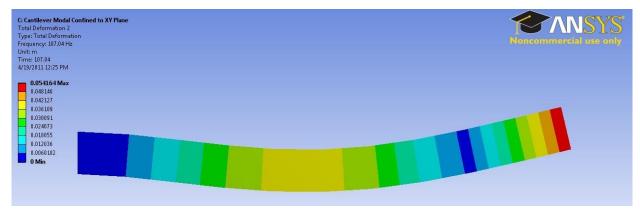
Natural Frequencies

Та	Tabular Data								
	Mode	Frequency [Hz]							
1	1.	17.68							
_	2.	107.04							
3	3.	179.16							
4	4.	285.26							
5	5.	318.23							
6	6.	525.41							

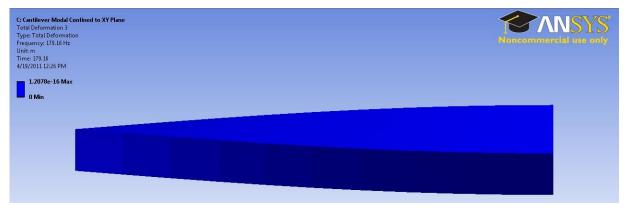
Mode 1



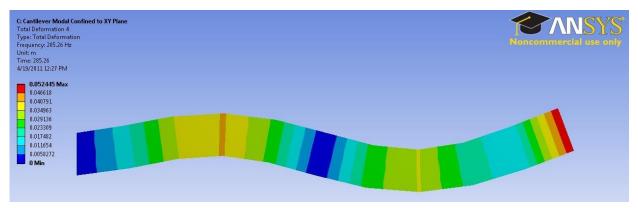
Mode 2



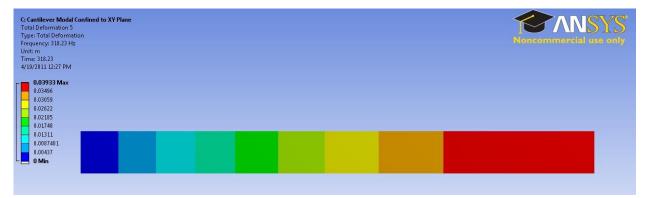
Mode 3



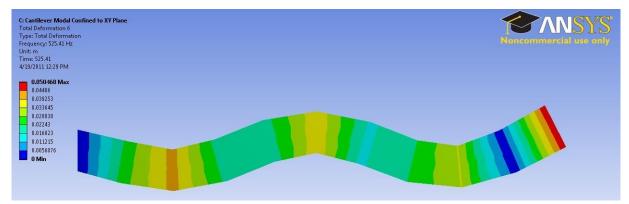
Mode 4



Mode 5



Mode 6



VERIFICATION & VALIDATION

For our verification, we will focus on the first 3 modes. ANSYS uses a different type of beam element to compute the modes and frequencies, which provides more accurate results for relatively short, stubby beams such as the one examined in this tutorial. However, for these beams, the Euler-Bernoulli beam theory breaks down and is no longer valid for higher order modes.

VERIFICATION

Comparison with Euler-Bernoulli Theory

From our Pre-Analysis, based on Euler-Bernoulli beam theory, we calculated frequencies of 17.8, 111.5 and 312.1 Hz for the first three bending modes. The ANSYS frequencies for the first three bending modes are 17.7, 107.0 and 285.2 Hz. Note that in the ANSYS results, the third mode is NOT a bending mode. So the fourth mode reported by ANSYS is the third bending mode. These results give percent differences of 0.6%, 4.2% and 8.7% between ANSYS and theory. Thus the ANSYS results match quite well with Euler-Bernoulli beam theory. Note that the ANSYS beam element formulation used here is based on Timoshenko beam theory which includes shear-deformation effects (this is neglected in the Euler-Bernoulli beam theory).

Comparison with refined mesh

Next, let's check our results with a more refined mesh. We'll run the simulation with 25 elements instead of 10. Following the steps outlined in the Mesh Refinement section of the <u>Cantilever</u> Beam Verification and Validation, refine the mesh.

Meshing the beam with 25 elements yielded the following modal frequencies

Та	Tabular Data								
	Mode	Frequency [Hz]							
1	1.	17.68							
2	2.	107.03							
3	3.	179.16							
4	4.	285.07							
5	5.	318.23							
6	6.	524.21							

These modal frequencies are all very close to those computed with a mesh of 10 elements, meaning that our solution is mesh converged.

Ex. No :

Date :

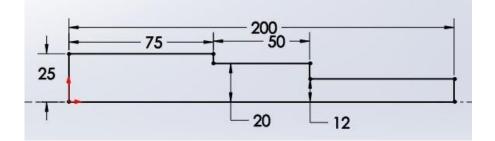
MODAL ANALYSIS OF STEPPED SHAFT

INTRODUCTION

Modal analysis is an important type of structural analysis and play an important role in vibration prediction of engineering structures. Every structure has its number of natural frequencies starting from first, second to infinite. Modal analysis is used to predict these natural frequencies. These natural frequencies are used to avoid resonance as it occurs if applied load frequency matches with one of the natural frequencies of vibrating body.

PROBLEM DESCRIPTION

The geometric dimension of stepped shaft has been shown in fig-1. A 3D model has been shown in fig-2.





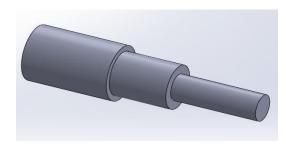


Fig 2

Step-1. Open Ansys workbench and double click on **moda**l analysis module from toolbox, it appears in **Project schematic**, fig-3.

Step-2. In project Schematic right click **Geometry>Import Geometry>Browse** the path where downloaded IGS file of stepped shaft is located, fig-4.

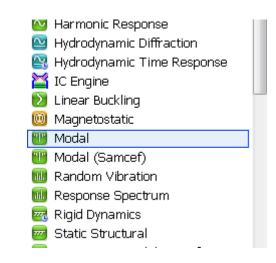


Fig 3

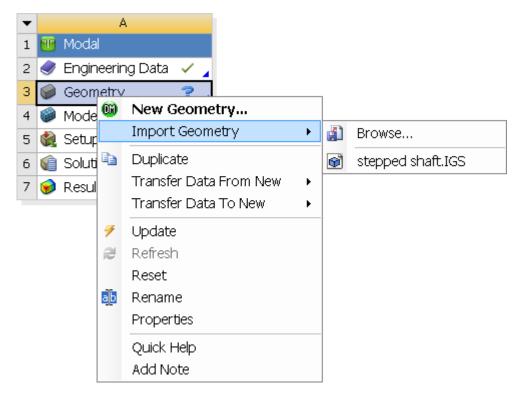


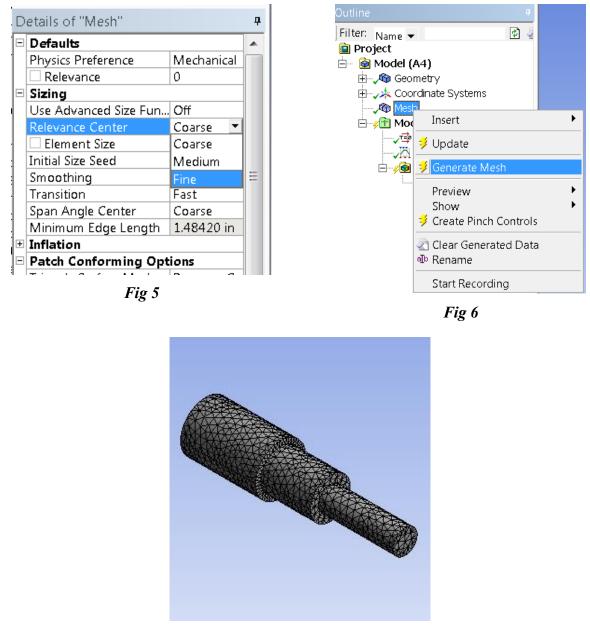
Fig 4

Step-3: Click File>Save, write File Name as "stepped shaft" and press Save.

Step-4: Right click on Model>Edit, mechanical window opens now.

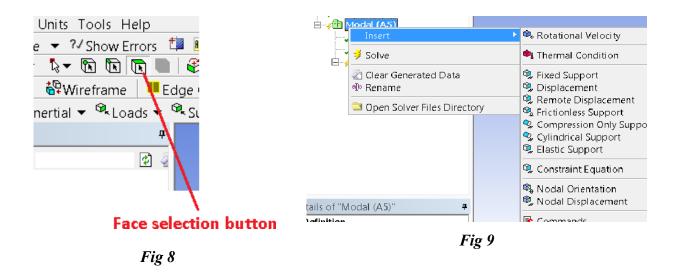
Step-5: Click on **mesh** in **Outline** tree and go to **Details of mesh** and click '+' sign on the left of **Sizing** and change **relevance center** from **Coarse to fine**, fig-5.

Step-6: Right click on **mesh** and click on **Generate mesh**, fig-6. Meshing of shaft will be generated, fig-7.





Step-7: Click on Face selection button to select faces, fig-8. From outline right click **Modal>Insert>Fixed Support**, fig-9. Now select left end face it turns green, fig-10 and "Details view of support" click apply, fig-11.



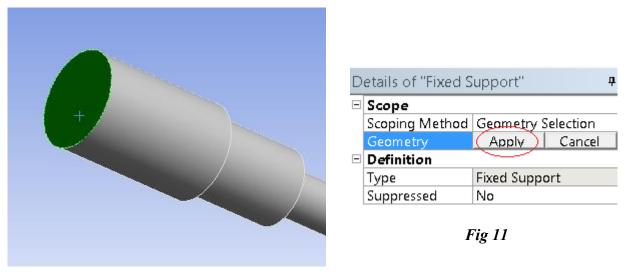


Fig 10

Step-8: Click on analysis settings, that is present under Modal in outline. Go to "Details of analysis settings" and set Max Modes to Find to 6, fig-12. Click File>Save. Right click Solution>Insert>Deformation>Total and set Mode to 1 in "Details of Deformation"., Fig-13.

Details of "Analysis S	ettings" 9	D	etails of "Total D	Deformation"
Options			Scope	
Max Modes to Find	(6)		Scoping Method	Geometry Selection
Limit Search to Rang	e No		Geometry	All Bodies
Solver Controls			Definition	-
Damped	No	L	Туре	Total Deformation
Solver Type	Program Contro		Mode (1.)
Rotordynamics Co			Identifier	$ \sim $
Output Controls			Suppressed	No
			Results	
Analysis Data Mana	agement		🗆 Minimum	

Fig 12

Fig 13

Step-9: Similarly repeat Step-9 again to set mode 2, 3, and so on up to 6.

Step-10: Right click on Solution>Solve and wait until solution status complete.

Step-11: Reduce the view scale from top right corner to 1.7e-2, fig-14.

Step-12: Click on Total Deformation under Solution in order to see contours for any Mode,

fig-16. Frequency Table is also available from Tabular Data, Fig-15.

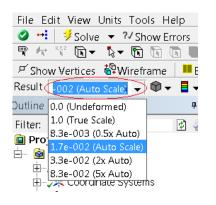


Fig 14

Tabular Data							
	Mode	🗹 Frequency [Hz]					
1	1.	1351.					
2	2.	1351.1					
3	3.	3567.					
4	4.	3567.6					
5	5.	6977.7					
6	6.	8601.8					



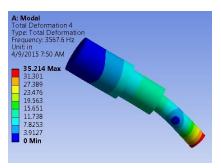


Fig-16: Mode 4 Shape

Result:

So in this modal analysis, we have found natural frequencies using Ansys workbench.

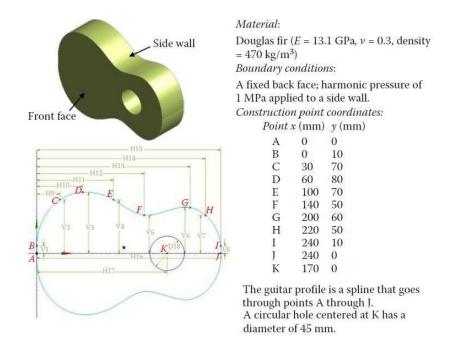
Ex. No :

Date :

HARMONIC ANALYSIS OF GUITAR

Problem Description: Musical instruments such as acoustic guitars create sound by means of vibration and resonance. The body of an acoustic guitar acts as a resonating chamber when the strings are set into oscillation at their natural frequencies. The following figure gives the dimensions of a simplified acoustic guitar model. The guitar has a wall thickness of 3 mm, and is made of Douglas fir wood (E = 13.1 GPa, Poison's ratio v = 0.3, density = 470 kg/m³).

Assuming the back surface of the guitar is fixed, find the first 10 natural frequencies and plot the first five vibration modes of the guitar. Suppose a harmonic pressure loading of magnitude 1 MPa is applied to a side wall of the guitar. Plot the frequency response of the z displacement (along the surface normal direction) of the front surface.



Solution:

Step 1: Start an ANSYS Workbench Project

Launch ANSYS Workbench and save the blank project as "Guitar.wbpj."

Step 2: Create a Modal Analysis System

Drag the Modal icon from the Analysis Systems Toolbox window and drop it inside the highlighted green rectangle in the Project Schematic window to create a standalone modal analysis system.

🐧 Guitar - Workbench		00
Pile View Tools Units Help New [™]	👔 Import Ay Reconnect 🥃 Refresh Project 🦻 Update Project 🕲 Project 🚱 Compact Mode Project Editomatic	- 0 x
Design Assessmet Design Assessmet Design Assessmet Depicit Dynamics Popicit Dynamics Popicit Dynamics Popicit Dynamics Popicit Dynamics Popicit Dynamics Design Bockling Response Spectrum Regid Dynamics Shape Optimization Regid Dynamics Shape Optimization Popicit Dynamics Popi	A 1 Media 2 Pergenering Data 3 Geometry ? 4 Wedal 5 G Solution ? 7 Results ? Modal	
View Al / Customize		Show Progress

Step 3: Add a New Material

- > Double-click on the Engineering Data cell to add a new material.
- In the following Engineering Data interface which replaces the Project Schematic, type "Wood" as the name for the new material, and double-click Isotropic Elasticity under Linear Elastic in the leftmost Toolbox window.
- Enter "13.1E9" for Young's Modulus and "0.3" for Poisson's Ratio in the Properties window. Double-click Density under Physical Properties.
- Enter "470" for Density in the Properties window. Click the Return to Project button to go back to Project Schematic.

File Edit View Tools Units Help						_	_	-						
New 🧭 Open 🛃 Save 🔣 Save As	(i) in	port + Reconnect	2 R	efresh Pro	ject 🥖 Update	Project CRetu	n to Pr	oject	Con	pact I	Hode 🔽 🛍			
Teobox - # X	Outine	of Schematic A2: Engineer	ngli	ete:		200		¥ ¥	Table	of Pro	perties Row 2: D	-	- 1	a i
Physical Properties		A	B	c		D					A		в	
12	1	Contents of Engineering Data	63	Source		Description			1		Temperature (C)	à.	Density (kg m^-3)	•
Isotropic Secant Coefficient of Thermal Exp Orthotropic Secant Coefficient of Thermal	2	 Material 	100	100000			-		2				470	
Isotropic Instantaneous Coefficient of The Orthotropic Instantaneous Coefficient of *	3	Stuctural Steel	23	🔮 G.,	Fatigue Data from 1998 AS 2, Table 5-11	at zero mean stress ME BPV Code, Secti 0.1	comes on 8, D	v						
Constant Damping Coeffident	4	beely @												
E Linear Elastic		Click here to add a new material												
Dirthotropic Elastidy	Propert	bes of Outline Row 4: Waa	d				* 1	J X						
Plasticity		A			в	с	D	E						
	1	Property		1	Value	Unit		5	12100	00720	oerbes Row 2: D	6071		
	2	2 Densty		- C	170	ka m^-3			-	COLUMN	2019 DES 100W 21-0	2020		×10
	3	🖯 🖂 Isotropic Bas	ticity		4b-		12			650			Densty	
	4	Derive from		Ye	ung's Mod				(kg m	550				
	5	Young's Modulus		L	31E+10	Pa.		21		450		1.0		
	6	Poisson's Ratio		0	1.3				Density	350				
	7	Bulk Modulus		3	1.0917E+10	Pa		10	Der 1	250		_		_
	8	Shear Modulus		1	5.0385E+09	Pa				100	1		3	3
													sture [C]	

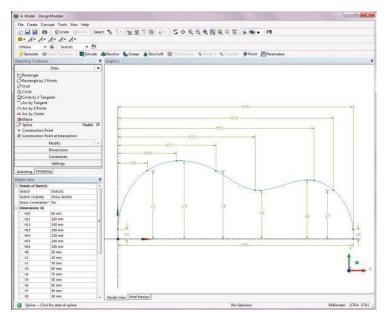
Step 4: Launch the Design Modeler Program

Ensure Surface Bodies is checked in the Properties of Schematic A3: Geometry window (select Properties from the View drop-down menu to enable display of this window). Choose 3D as the Analysis Type in this Properties window. Double click the Geometry cell to launch Design Modeler, and select "Millimeter" in the Units pop-up window.

File View Tools Units Help	Import		sct Mode	
	roject Schematic	and the second se	les of Schematic A3: Geometry	
El Analysis Systems		Return to the Proje	ct Workspace A	8
Design Assessment		1	Property	Value
Bectric	▼	2	 General 	1001-00177
Explicit Dynamics	1 Model	3	Component ID	Geometry
Fluid Flow(CFX)	2 🖉 Engineering Data 🗸	4	Directory Name	SYS
Fluid Flow (FLUENT)	3 Geometry ?	5	 Geometry Source 	And the second s
Harmonic Response		6	Geometry File Name	
UnearBuckling	1 40 m	7	Basic Geometry Options	C. martine and a
3 Magnetostatic		8	Sold Bodies	
Modal	6 🕼 Solution 👕 🍙	9	Surface Bodies	
Random Vibration Resoonse Spectrum	7 😥 Results 🛛 🖓 🔒	10	Line Bodies	171
Rigid Dynamics	Modal	11	Parameters	
Shape Optimization		12	Parameter Key	DS
Static Structural		13	Attributes	E
Steady-State Thermal		14	Named Selections	1
Thermal-Electric		15	Material Properties	199
Transient Structural		15	E Advanced Geometry Ontons	
Transient Thermal		10	And a state of the second s	3D
Component Systems		18	Use Associativity	1
Custom Systems		10	Import Coordinate Systems	
B Design Exploration		and the second se	Import Coordinate Systems	1
		20	Import Work Points Reader Mode Saves	-
View Al / Customize		21	Reader Mode Saves Updated File	173

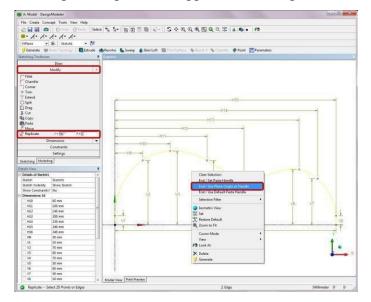
Step 5: Create a Profile Sketch

Click on the Sketching tab. Select the Draw toolbox and then Construction Point. Draw 10 construction points A through J, as shown below. Draw a spline passing through points A through J; right-click at the last construction point and choose Open End from the context menu to finish the spline creation.

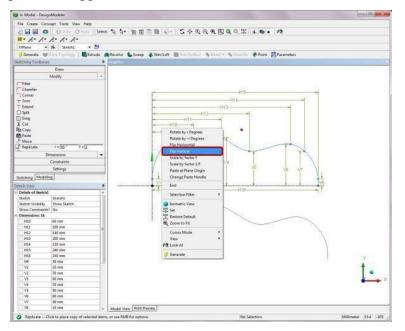


Step 6: Create a Replicate Curve

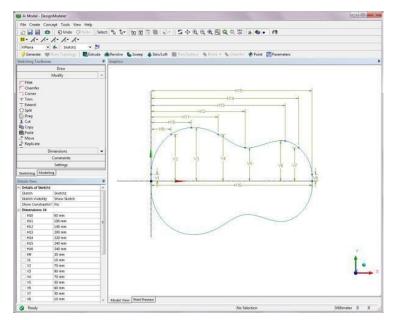
Select the Modify toolbox and then Replicate. Click on the spline from the Graphics window. Right-click anywhere in the Graphics to show the context menu. Select End/Use Plane Origin as Handle as shown below. A replicate spline will appear in the Graphics window.



Next, right-click anywhere in the Graphics, and select Flip Vertical in the context menu. A vertically flipped spline will appear.

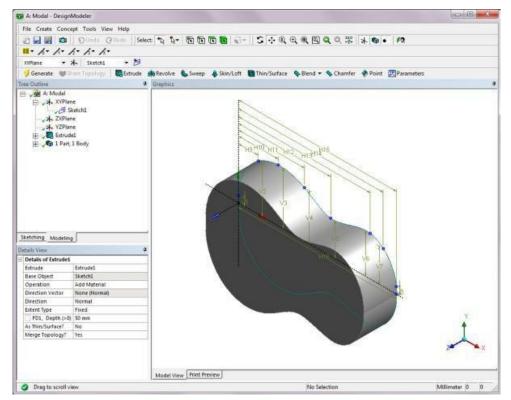


Click on the origin point in the Graphics to place the flipped spline, and press Esc to end the operation. A closed-loop curve is now formed as shown below.



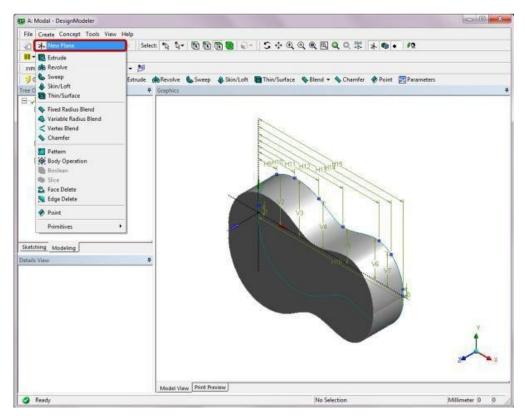
Step 7: Create an Extruded Body

Switch to the Modeling tab and click on the Extrude feature. The default Base Object is set as Sketch1 in the Details of Extrude1. Change the extrusion depth to 50 mm in the field of FD1, Depth and click Generate. A solid body is created as shown below.

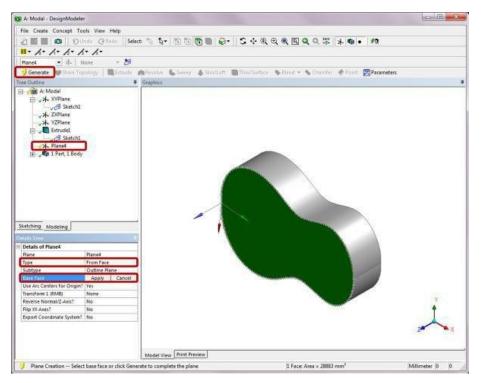


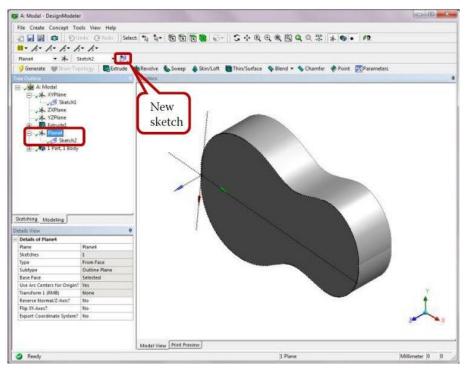
Step 8: Create an Extruded Cut on the Front Face

Create a new plane by selecting New Plane from the Create drop-down menu.



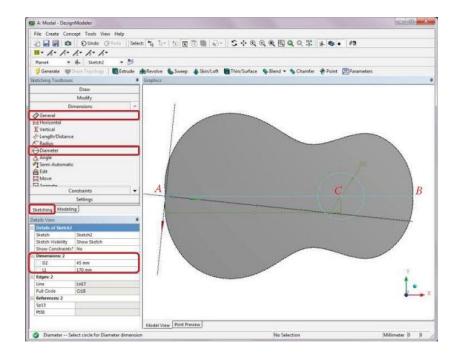
A new plane named Plane4 is now added to the Tree Outline. In the Details of Plane4, set the Type to From Face. Click the front face of the guitar from the Graphics window, and apply it to the Base Face selection in the Details of Plane4. Click Generate.



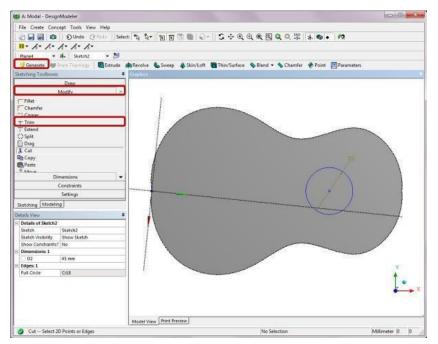


To create a new sketch under Plane4 in the Tree Outline, click on the New Sketch icon.

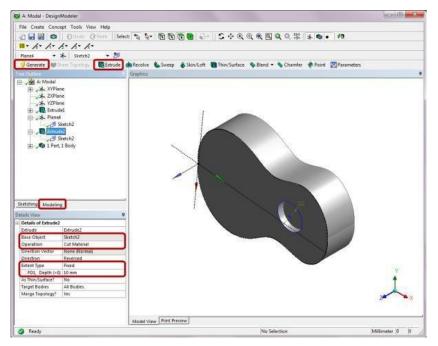
Switch to the Sketching tab for Sketch2. In the sketch, draw a horizontal line by connecting points A and B as shown below. Then draw a circle of diameter 45 mm centered at point C, located 170 mm to the left of point A along line AB.



Next, choose Trim under the Modify tab, and click on line AB in the Graphics window. The sketch line AB will disappear. Click Generate.

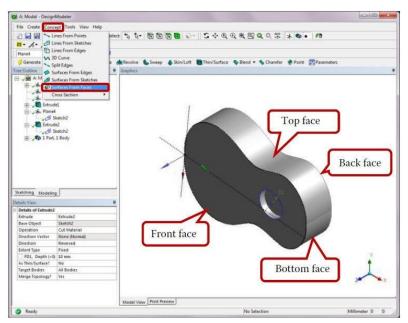


Switch to the Modeling tab, and click on the Extrude feature. The default Base Object is set as Sketch2 in the Details of Extrude2. Set the Operation to Cut Material. Enter an extrusion depth of 10 mm in the field of FD1, Depth and click Generate. An extruded cut feature is now added to the front face as shown below.

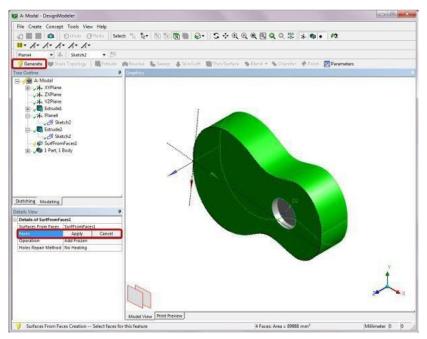


Step 9: Create a Surface Body

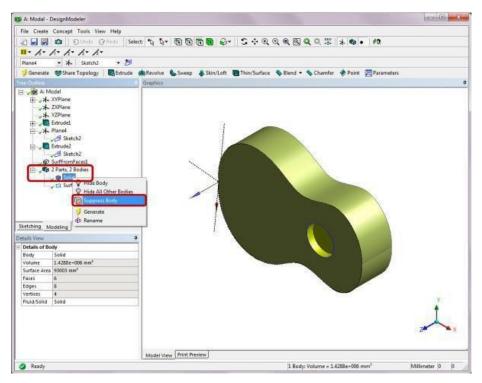
Select Surface from Faces from the Concept drop-down menu. In the Graphics window, Ctrlclick to select four faces, that is, the front, back, top, and bottom faces that enclose the solid body as shown below.



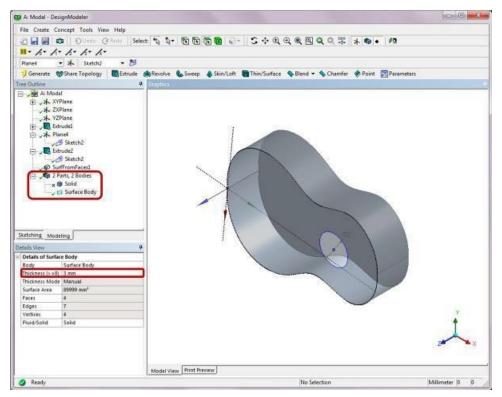
Click Apply next to Faces in the Details of SurfFromFaces1. Then click Generate. A surface body will be generated in the Tree Outline under 2 Parts, 2 Bodies.



Right-click on Solid under 2 Parts, 2 Bodies in the Tree Outline. In the context menu, select Suppress Body.

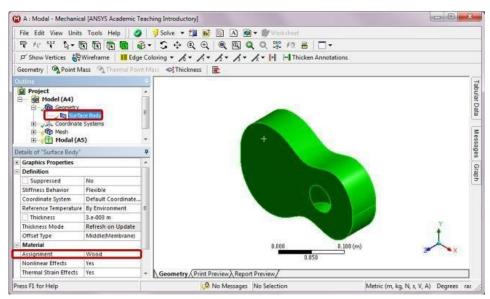


Click on Surface Body under 2 Parts, 2 Bodies in the Tree Outline. Change the Thickness to 3 mm in the Details of Surface Body. Save and exit the Design Modeler.



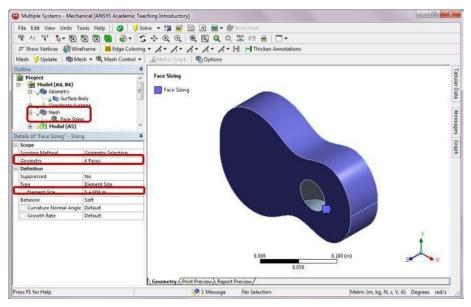
Step 10: Launch the Modal–Mechanical Program

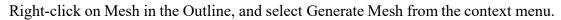
Double-click on the Model cell to launch the Modal–Mechanical program. Click on the Surface Body under Geometry in the Outline tree. In the Details of "Surface Body," click to the right of the Material Assignment field and select Wood from the drop-down menu.

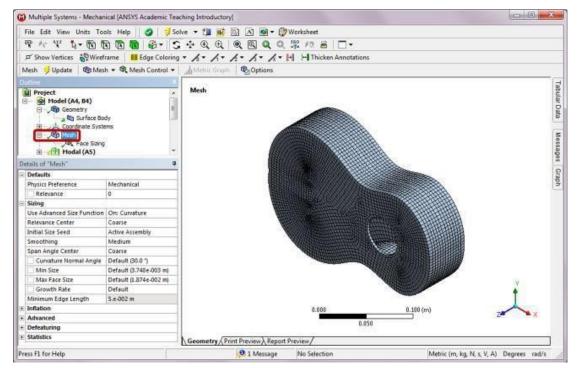


Step 11: Generate Mesh

Right click on Mesh in the Project Outline. Select Insert and then Sizing from the context menu. In the Details of "Face Sizing," enter "5e-4 m" for the Element Size. Click on the front, back, top, and bottom faces of the guitar in the Graphics window and apply the four faces to the Geometry selection.

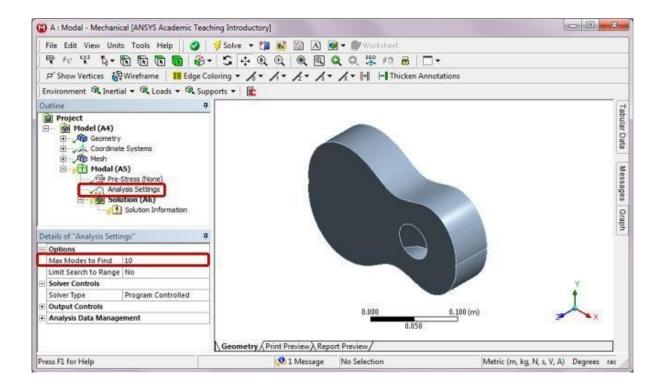




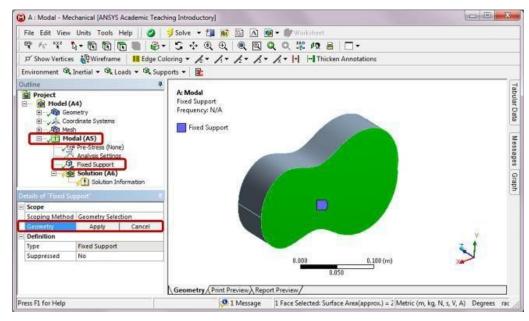


Step 12: Set Up Modal Analysis and Apply Boundary Conditions

Click on Analysis Settings under Modal in the Outline tree. Change the Max Modes to Find to 10 in the Details of "Analysis Settings."

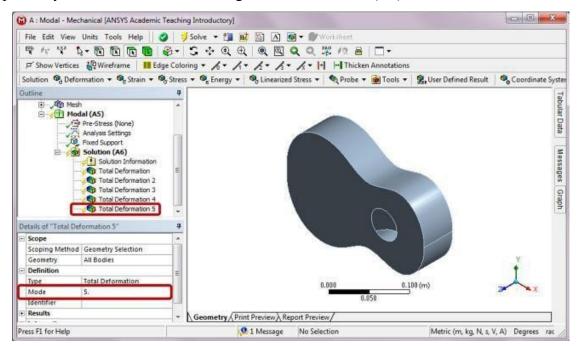


Right-click on Modal(A5). Choose Insert and then Fixed Support from the context menu. Apply the back face to the Geometry selection.

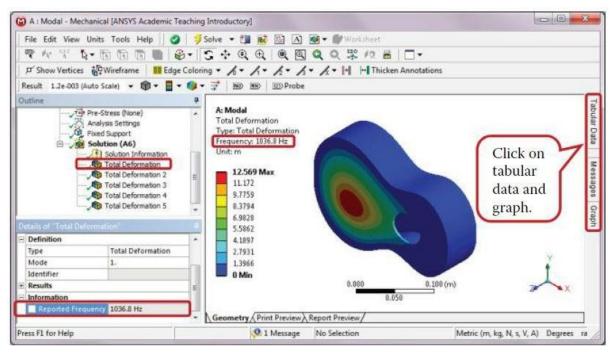


Step 13: Retrieve Results from Modal Analysis

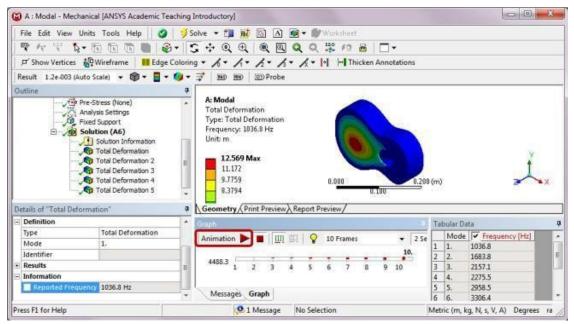
Insert Total Deformation by right-clicking on Solution (A6) in the Outline. In the Details of "Total Deformation," set Mode to 1. Insert another Total Deformation item. In the Details of "Total Deformation 2," set Mode to 2. Repeat this step three more times. Set Mode to 3, 4, and 5, respectively, for each new insertion. Right click on Solution (A6) in the Outline and Solve



Click on Total Deformation in the Outline to review results. The results below show the first natural frequency of 1036.8 Hz and the corresponding mode shape.

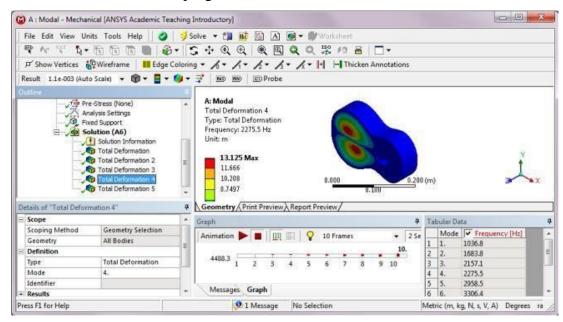


Click on Tabular Data and Graph on the right edge of the Graphics window, and then click on the push pin labeled AutoHide to display the Tabular data and the Graph as shown below.



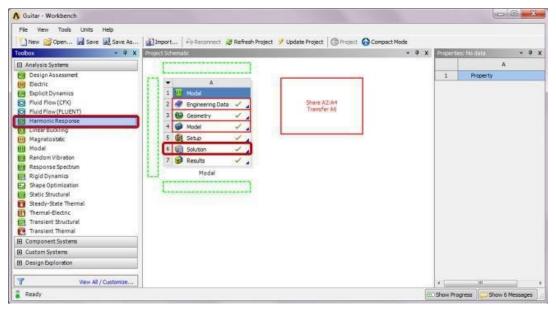
The Tabular data gives the first 10 natural frequencies of the guitar under the fixed bottom boundary condition. The Play/Stop control interface in the Graph

window allows animation of mode shapes. Click on each different Total Deformation item in the Outline to review results, for example, the following figure shows the fourth mode shape, and then exit the Modal–Mechanical program.



Step 14: Create a Harmonic Response Analysis System

Drag the Harmonic Response icon from the Analysis Systems Toolbox window and drop it onto the Solution cell of the highlighted Modal system in the Project Schematic.

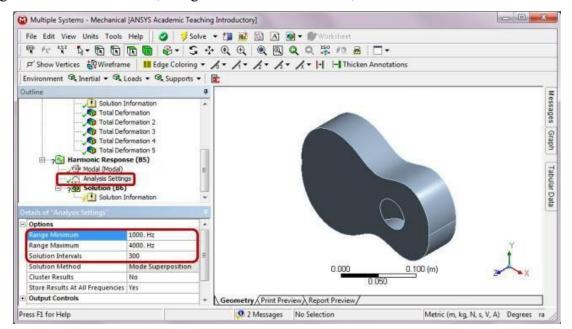


This creates a Harmonic Response system that shares data with Modal system as shown below.

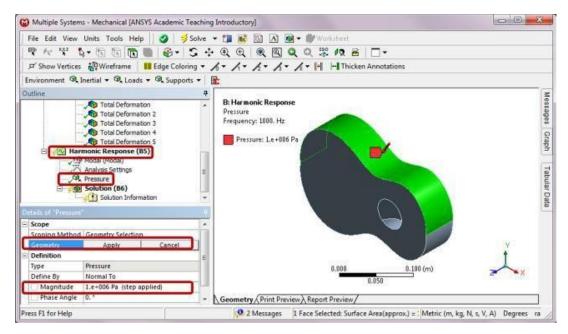
	🛿 Import 🖓 Reconnect 😹 Refresh Project 🥍 Update Project 🕝 Project 🕜 Compact Mo oject Schemabic	A CONTRACTOR OF	operties: No dat		
			and the states	a •	0
	A 1 Medal 2 Connetry Cota 3 Connetry Cota 3 Connetry Cota 4 Model Connetry Cota 5 Connetry Cota 5 Connetry Cota 6 Connetry Cota 7 Results Cota 7 Results Cota 8 Nodel Connetry Cota 9 Nodel Connetry Cota 1 Nodel Cota		I Prop	A serty	
C Transient Thermal Component Systems					
E Custom Systems					
Design Exploration					
View All / Customize					

Step 15: Set Up Harmonic Response Analysis and Assign Loads

Double-click on the Setup cell of the Harmonic Response system to launch the Multiple Systems–Mechanical program. In the program, select Analysis Settings from the Outline. Set the Range Minimum to 1000 Hz, Range Maximum to 4000 Hz, and Solution Intervals to 300.

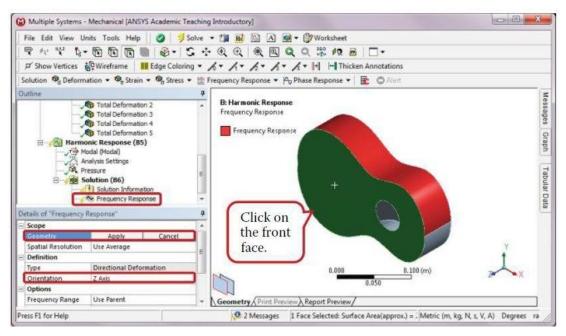


Right-click on Harmonic Response (B5). Choose Insert and then Pressure from the context menu. In the Details of "Pressure," set magnitude as 1 MPa, and apply the top face to the Geometry selection.

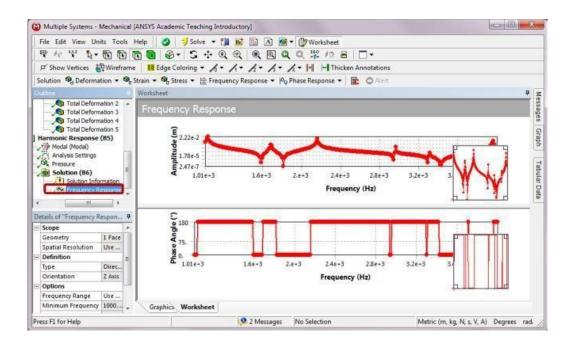


Step 16: Retrieve Results from Harmonic Response

Right-click on Solution (B6). Choose Insert and Frequency Response and then Deformation from the context menu. In the Details of "Frequency Response," set the Orientation of the directional deformation to Z-Axis. Click on the front face and apply it to the Geometry selection.

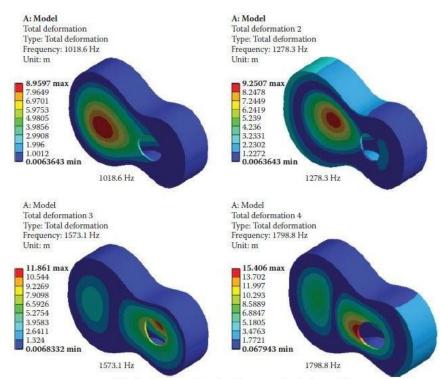


Right-click on Solution (B6) and select Solve. After solution is done, click on Frequency Response in the Outline to review the harmonic response of the guitar.

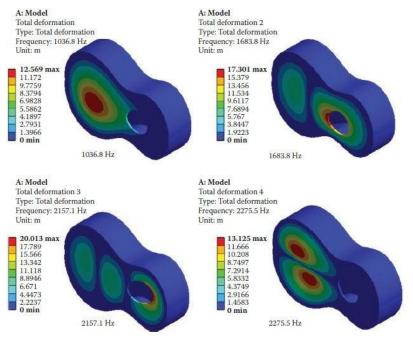


Modeling tips:

- Note that modal analysis can be run as constrained, unconstrained, or partially constrained. Also note that symmetric structure may have asymmetric modes, and thus, it is not recommended to take advantage of symmetry to simplify models for modal analysis.
- In the following, we are going to show a result comparison between an unconstrained model and the fixed model. For the unconstrained model, the first six mode shapes obtained from simulation are rigid body modes that allow the structure to move freely.
- They are not considered as structural modes. The comparison indicates that a free floating guitar has a different set of natural frequencies and mode shapes from that of a fixed guitar.
- In general, constraint conditions have an effect on the vibration characteristics of a structure and should be considered carefully when setting up a model.



The first four structural modes of the unconstrained guitar model.



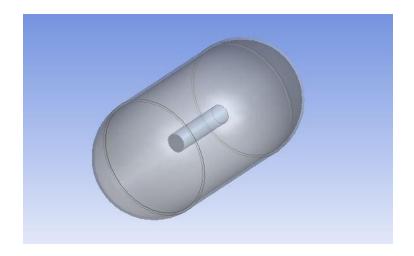
The first four structural modes of the fixed guitar model.

Ex. No : Date :

RADIATION EXCHANGE BETWEEN SURFACES

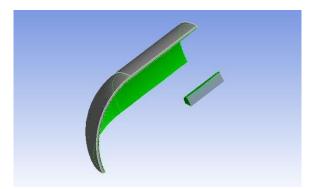
PROBLEM DESCRIPTION:

A very cold specimen is placed in the center of a shell in room temperature. Radiation is exchanged between the surface of the shell and the specimen. Find the emitted radiation, the reflected radiation, the incident radiation, and the net radiation of each surface. Both the shell and the specimen are structural steel. The model is shown below:



PRE-ANALYSIS & START-UP MODEL

We are interested in finding the radiation exchanged between the shell and the specimen surface. We will run a steady state thermal analysis to set the initial conditions of the model. Then we will transfer the initial conditions to transient thermal to complete the radiation analysis. Symmetry boundary conditions are added to the transient thermal model. This is essential to problems involving radiation because it enables the FEA code to compute the view factor between the surfaces in the full model. It is possible to run a full model without symmetry boundary conditions but this example will run faster with 1/8 symmetric model. The following picture shows the 1/8 model and the radiating surfaces in green.



RADIATION

Radiation heat transfer can be derived from the Stefan-Boltzmann Law:

$$Q_R = \sigma \varepsilon FA \left(T_{surface}^4 - T_{ambient}^4 \right)$$

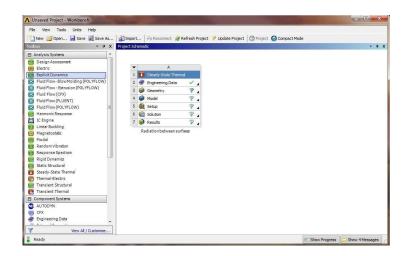
Where:

- $\sigma = \text{Stefan-Boltzman constant}$
- ε = Emissivity
- A = Area of radiating surface
- F = Form factor(1)

The above radiation equation provides correlations for radiation to ambient (form factor assumed to be 1) or surface to surface (view factor calculated).

START-UP

Open ANSYS workbench and drag the *Steady State Thermal* icon from the toolbox to Project Schematic. Name the project Radiation between surfaces.



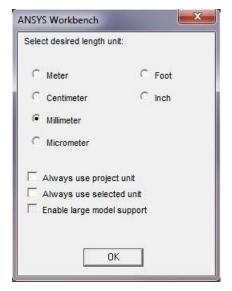
ENGINEERING PROPERTIES

Double click on *Engineering Data* to open the Engineering Data page. Check that *Structural Steel* appears as the default material.

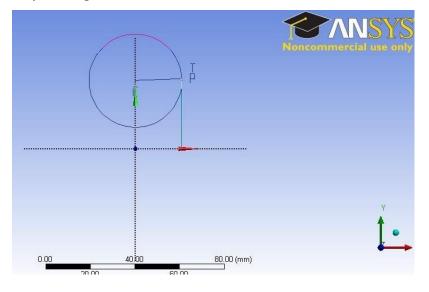
File Edit View Tools Units Help													
🎦 New 对 Open 🛃 Save 🔣 Save As	Imp Imp	oort 🏼 🖗 Reconnect 🔏	Refre	sh Proje	t 🥖 Update Pr	roject	Return	to Projec	t 🕜 Compac	t Mode 🛛 🛐] 🗰		
Foolbox 🔻 🕂 🗙	Outline	of Schematic A2: Engineering	ıg Data	•		φx	Table o	f Properti	es Row 2: Isot	ropic Thermal	Conductivity	т ф	×
Thermal		A	В	С	D				А		В		
 Isotropic Thermal Conductivity Orthotropic Thermal Conductivity 	1	Contents of Engineering Data	8	Source	Description	1	1	Tempe 21	rature (C) 🗦	Thermal Co	nductivity (W	m^-1 C^-1)	
	2	■ Material					2	- 21		00.5			-
	3	Structural Steel		e :	atigue Data at zi iean stress come 998 ASME BPV C ection 8, Div 2, 1 110.1	es from lode,							
	-	Click here to add	-										
	*	a new material											
	*						4			III			•
			ural St	eel		- д х	10000	fProperti	es Row 2: Iso1		Conductivity	× Ţ	, X
		a new material	ural St	ccl	1 1	- д х D Е	Chart o		es Row 2: Isot		Conductivity	* †	×
		a new material tes of Outline Row 3: Struct A Property			с		Chart o) <u> </u>	es Row 2: Isot	ropic Thermal	Condüctivity ermal Conduct		+ ×
	Propert	a new material		В	C E Unit (DE	Chart o) -) -) -	es Row 2: Isot	ropic Thermal			+ ×
	Propert	a new material		B Valu	C E Unit (DE	Chart o		es Row 2: Isot	ropic Thermal			* ×
	Propert	a new material		B Valu	C E Unit (DE	Chart o Chart o Cha) -) -) -) -	es Row 2: Isob	ropic Thermal			* ×

GEOMETRY CREATE THE SHELL SKETCH THE SHELL

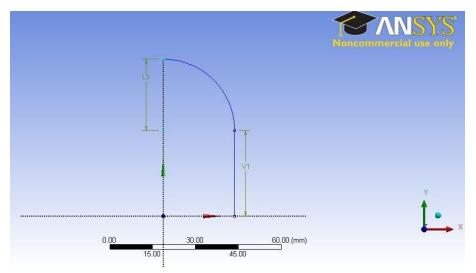
In *Project Schematic*, double click on *Geometry* to open the Design Modeler. When prompted, select *Millimeter* as the unit.



Click on the *XY Plane* and the *z* axis to begin sketching. Use the *Line* sketching tool to create a vertical line starting from the *x* axis. However the cursor around the axis until you see a symbol C to begin your sketch. The symbol C means the line is coincident with the x axis. Next, use the *Arc by Center* to create the dome of the shell. However the cursor near the *y* axis until you see the symbol C. Single click on the y axis and click again on the tip of the line you have just created. You should see a symbol P when you click on the vertex, which means coincident. Finally, click on the *y* axis again to finish the arc.



Use the *General* dimension tool to create dimensions for the line and the radius of the arc. The length of the line is *30 mm* and the radius of the arc is *25 mm*. Your sketch should look like this:



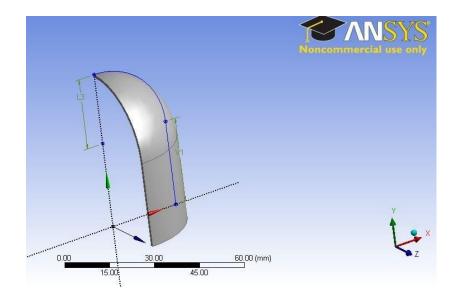
MODEL THE SHELL

Click on Create from the top menu bar and select *Revolve*. The Revolve tool should automatically select your shell sketch for its geometry. If not, highlight the cell next to geometry and select *Sketch1* under the *XYPlane* tree. Select the *Y axis* for *Axis*. This will allow the sketch to revolve around the y axis to create a shell. Change the *Angle* from *360* to *90* degrees. Highlight *As Thin/Surface?* and change the option from *No* to *yes*. Keep the *Inward Thickness* to *1 mm*. Click on *Generate*.

File	Create Concept Tools View	Help
	🔸 New Plane	o Sel
XYPI	💽 Extrude	• 惣
🤣 G	💏 Revolve	arameter
E	Sweep Skin/Loft	Skin/Lo
	Skin/Loft	fer 🐟
	Thin/Surface	ner 🗸
ree O	Circl Dedice Direct	

Details of Revolve1		-
Revolve	Revolve1	
Geometry	Sketch1	
Axis	Selected	
Operation	Add Material	
Direction	Normal	
FD1, Angle (>0)	90 °	
As Thin/Surface?	Yes	
FD2, Inward Thickness (>=0)	1 mm	
FD3, Outward Thickness (> =0)	0 mm	
Merge Topology?	Yes	
Geometry Selection: 1	Arrent (A	-

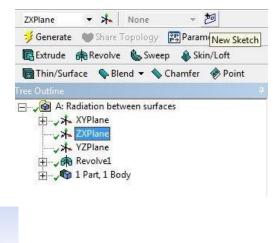
The 1/8 shell model



Create the Specimen

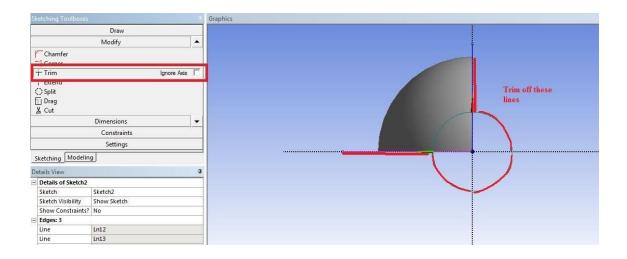
SKETCH THE SPECIMEN

We will create the specimen from the ZX plane. Highlight *ZXPlane* in the Tree Outline and click on *New Sketch* :

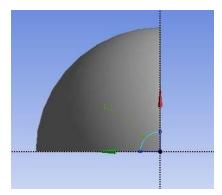


Click on the *Y axis* ² to view the ZX plane.

From the Sketching tab, use the *Circle* tool to draw a circle centered at the origin. Again, Make sure your cursor displays a P near the origin before you begin sketching. Next, use the *Line* tool to draw two lines along the X and Z axis. We only need to create a quarter of the full sketch to create the 1/8 model. Select the *Trim* tool and click on any sketch outside the quarter circle enclosed by the lines and the full circle. Use the *Radius* dimension tool to set the radius of the quarter circle to 4 mm.



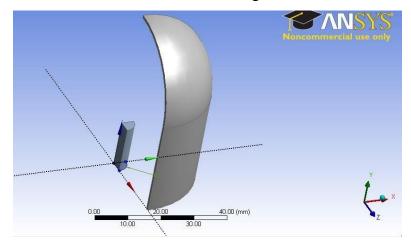
The size of the specimen compared to the shell:



Click on the *Extrude* icon and select the quarter circle for the geometry. In the Details of Extrude1 window, set the *Depth* to *15 mm*.

	Extrude Re	volve 🌭 Sweep 🚯 Skin/Loft
Į	Thin/Surface	💊 Blend 🔻 🥎 Chamfer 🛛 🛷 Poin
De	etails View	đ
-	Details of Extrude1	
	Extrude	Extrude1
	Geometry	Sketch2
	Operation	Add Material
	Direction Vector	None (Normal)
	Direction	Normal
	Extent Type	Fixed
	FD1, Depth (>0	15 mm)
	As Thin/Surface?	No
	Merge Topology?	Yes
÷	Geometry Selection:	:1
	Sketch	Sketch2

Once everything is specified as above, click *Generate*. You should see *2 Parts, 2 Bodies* in the Tree Outline. Your model should look like the following:



You may now close the Design Modeler and move on to the next step.

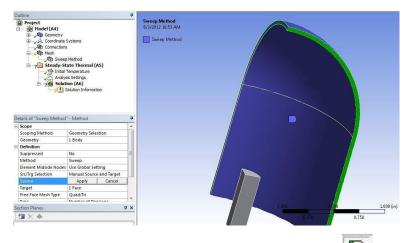
MESH

Double click on *Model* to launch ANSYS Mechanical.

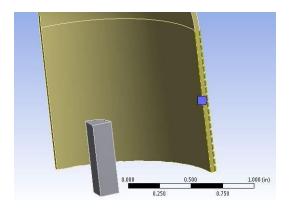
In the Outline window, right click on *Mesh* > *Insert* > *Method*.

	Insert	🕨 🍘 Method
	ジ Update	🔍 Sizing
<u>⊟</u>	誟 Generate Mesh	№ Contact Sizing A Refinement
	Preview Show ジ Create Pinch Controls	Mapped Face Meshing
Details of "Mesh"		
- Defaults	allo Rename	
Physics Prefere	Start Recording	

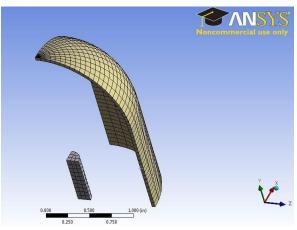
Select the entire shell body for geometry and click on apply. In the *Details of "Automatic Method" -Method* window, change the *Method* from *Automatic* to *Sweep*. Select *Manual Source and Target* for *Src/Trg Selection*. Set the cross sectional face on one side of the shell to source and the other cross sectional face of the shell to target.



Right click on *Mesh* > *Insert* > *Sizing*. Use the edge selection tool to select the outer and inner walls of the shell. Use *Number of Divisions* and set it to *20*.



We will use the default mesh size. Right click on *Mesh* > *Generate Mesh* to create the mesh.



Keep ANSYS Mechanical open and move to Setup.

Physics Setup

Set-up Initial Conditions

Steady-State Thermal

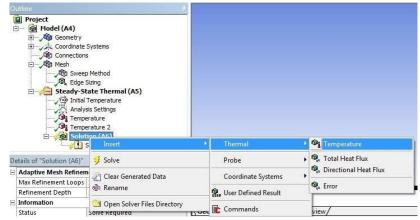
We will need to run the steady state model and use the result as the initial condition for the transient analysis.

Right click on *Steady-State Thermal (A5)* > *Insert* > *Temperature*.

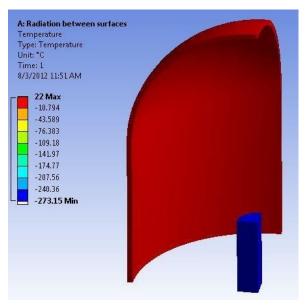


Select the entire *Shell* body and set the temperature to *22 degrees Celsius*. Create another temperature boundary condition but select the *Specimen* instead. Set the temperature of the *Specimen* to *-273.15 degrees Celsius*.

Right click on *Solution (A6)* > *Insert* > *Thermal* > *Temperature*. The default geometry is set to *All Bodies*. Keep it and repeat the step but select only the *Specimen*.



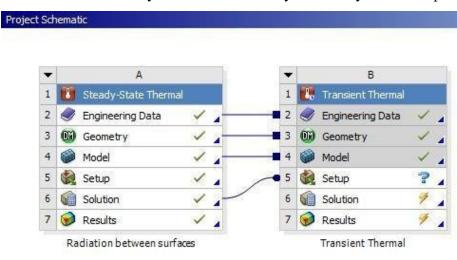
The solution titled *Temperature* will display the temperature distribution of the shell and the specimen and *Temperature 2* will display only the specimen. Notice there isn't any temperature variation because we have done nothing except set the temperature of the two bodies. No heat can be exchanged between the two bodies without specifying additional boundary conditions (convection, radiation, etc).



We are now ready to move on to set up the transient analysis.

Set-up Transient Thermal Analysis

Return to the *Project Schematic* in ANSYS Workbench. Right click on *Solution* > *Transfer Data to New* > *Transient Thermal*. This will export the model, the mesh, and the steady state solution to *Transient Thermal* analysis and the new analysis is ready to be set-up.



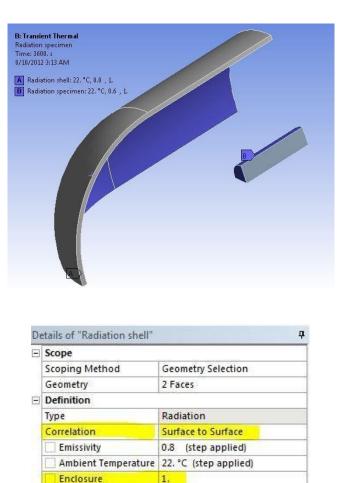
ADDITIONAL MATERIAL PROPERTIES

New material properties have been added in Engineering Data. The new properties are essential to perform transient thermal analysis.

pertie	erties of Outline Row 3: Structural Steel			₹ 4	ιx	
A		В	с	D	E	
1	Property	Value	Unit	8	tp.	
2	🔁 Density	7850	kg 💌			
3	Isotropic Thermal Conductivity	60.5	w 💌			
4	🔀 Specific Heat	434	J 💌			

SURFACE TO SURFACE RADIATION

Surface to surface radiation is applied like a boundary condition. Radiating surfaces are related to one another by their enclosure number. We want to set up the boundary condition to make the shell and specimen surface to "see" one another. This can be done by creating 2 radiation conditions and set their enclosure number to 1. By creating 2 separate conditions, each surface can have different emissivity value.



Once the Convection and Radiation boundary conditions have been set up, you may move on to the next step to set up the solution.

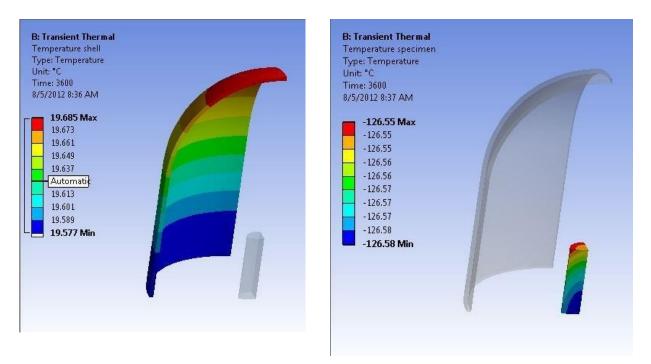
No

Suppressed

NUMERICAL RESULTS

You may receive a warning that says "The initial time increment may be too large for this problem. Check results carefully." Our initial time step is set to 36 seconds, which is rather large for transient analysis. The warning can be eliminated by turning off *Auto Time Stepping* under *Analysis Settings* and manually specify the initial time step.

By the end of end step time, *3600 seconds*, the shell temperature dropped to approximately 19 degrees Celsius and the specimen temperature rose to approximately -126 degrees Celsius.



We will now examine the radiation heat transfer between the surface of the shell and the specimen. Click on *Radiation shell* under the solution tree and expand the *Tabular Data*, located in the lower right corner.

Energy Balance

The net radiation heat flux of a surface can be found by writing the energy balance equation on the surface.

$$q_r = \varepsilon \sigma T^4 + (1 - \varepsilon)q_i - q_i$$

Where

ε is the emissivity

 σ is the Stefan-Boltzmann constant

The three radiation terms on the right hand side of the equation represent different types of radiation associated with a given surface.

The first term is the *emitted radiation*.

The second term is the *reflected radiation*.

The third term is the *incident radiation*.

The sum of these three terms gives the *net radiation heat flux* of a surface.

Tabular data of the shell

	Time [s]	Radiation shell (Net Radiation) [W]	Radiation shell (Emitted Radiation) [W]	Radiation shell (Reflected Radiation) [W]	Radiation shell (Incident Radiation) [W]
1	36.	0.22162	5.5929	1.3433	6.7145
2	72.	0.22104	5.5829	1.3482	6.7101
3	108.	0.22034	5.5739	1.3458	6.6994
4	216.	0.21927	5.5509	1.3401	6.6718
5	540.	0.21743	5.5054	1.3293	6.6173
6	900.	0.21611	5.4732	1.3215	6.5786
7	1260.	0.21518	5.4527	1.3166	6.5542
8	1620.	0.21437	5.4397	1.3136	6.5389
9	1980.	0.21354	5.4316	1.3119	6.53
10	2340.	0.21243	5.4267	1.31	6.5242
11	2700.	0.21064	5.4241	1.307	6.5204
12	3060.	0.20764	5.4232	1.3044	6.5199
13	3420.	0.20375	5.4238	1.3039	6.524
14	3600.	0.2015	5.4245	1.3045	6.5276

Tabular data of the specimen

	Time [s]	Radiation specimen (Net Radiation) [W]	Radiation specimen (Emitted Radiation) [W]	Radiation specimen (Reflected Radiation) [W]	Radiation specimen (Incide
1	36.	-0.21672	1.5787e-010	0.1434	0.36012
2	72.	-0.21615	2.5421e-009	0.14367	0.35982
3	108.	-0.21547	1.2752e-008	0.14373	0.3592
4	216.	-0.21442	2.0044e-007	0.14327	0.35769
5	540.	-0.21262	7.5851e-006	0.14214	0.35477
6	900.	-0.21134	5.7488e-005	0.14129	0.35269
7	1260.	-0.21042	2.1805e-004	0.14073	0.35138
8	1620.	-0.20964	5.8971e-004	0.14032	0.35054
9	1980.	-0.20882	1.3038e-003	0.13991	0.35004
10	2340.	-0.20774	2.5192e-003	0.13943	0.34968
11	2700.	-0.20599	4.403e-003	0.13901	0.3494
12	3060.	-0.20305	7.1669e-003	0.13901	0.34923
13	3420.	-0.19925	1.0961e-002	0.13906	0.34926
14	3600.	-0.19705	1.3313e-002	0.13898	0.34935

The positive sign indicates heat is being transferred to the surrounding through radiation and the negative sign indicates heat is being absorbed from the surrounding. Because the specimen is so cold compared to the shell, some radiation emitted by the shell is absorbed and stored within the specimen. The specimen emits a very small amount of radiation because its initial temperature is near absolute zero but its emitted radiation gradually increases as the specimen gets warmer with time.

The emitted, reflected, and incident radiation over time are also shown in the tabular data.

VERIFICATION & VALIDATION

MESH CONVERGENCE

One way to check the accuracy of the simulation is to refine the mesh and re-run the simulation. The smaller the element in the mesh, the more accurate the simulation will be. The only drawback is longer computation time. To refine the mesh, insert *Body Sizing* on the specimen

and set the element size to 0.001m. Also, enter 0.002m for the element size in the *Details of* "*Mesh*". The original mesh has 620 *Elements* and 4533 *Nodes* and the new mesh has 1600 *Elements* and 11204 *Nodes*.

	Shell Net Radiation at end step time (W)	Specimen Net Radiation at end step time (W)
Original Mesh	0.2015	-0.19705
Refined Mesh	0.20344	-0.19812

The net radiation shows very little change as the number of elements is doubled. No further mesh refinement is need.

VIEW FACTOR

The view factor is calculated for surface to surface radiation. Recall from the radiation equation in pre-analysis, this is an important parameter in computing the radiation between surfaces that are in the same enclosure

$$F_{ij} = \frac{1}{A_i} \int \int \frac{\cos \theta_i \cos \theta_j}{\pi R^2} dA_i dA_j$$

Where

 F_{ij} is the fraction of the radiation leaving surface i that is intercepted by surface j.

 A_i, A_j are the elemental surface area

R is the line that connects the two elemental areas

 θ_i, θ_j are the polar angles formed by the line R with surface normals n_i, n_j

It is difficult to analytically calculate the view factor for this model. Hence, we will use a simplified exercise to show the validity of ANSYS simulation. Proceed to the next step to compare the analytic and ANSYS results.

VIVA VOCE QUESTIONS

- What is meant by finite element? A small unit having definite shape of geometry and nodes is called finite element.
- 2. What is meant by finite element analysis? Finite element method is a numerical method for solving problems of engineering mathematical physics. In the finite element method, instead of solving the problem for the entire body in one operation, we formulate the equations for each finite element and combine them to obtain the solution of the whole body.
- 3. State the methods of engineering analysis. There are three methods of engineering analysis. They are:
 - a. Experimental methods.
 - b. Analytical methods.
 - c. Numerical methods or approximate methods.
- 4. What is meant by node or Joint? Each kind of finite element has a specific structural shape and is interconnected with the adjacent elements by nodal points or nodes. At the nodes, degrees of freedom are located. The forces will act only at nodes and not at any other place in the element.
- 5. What do you mean by discretization? Discretization is the basis of finite element method. The art of subdividing a structure into a convenient number of smaller components is known as discretization.
- 6. What are the three phases of finite element method? The three phases are
 - a. Pre-processing
 - b. Analysis
 - c. Post processing
- 7. What is structural and non-structural problem? Structural problem: In structural problems, displacement at each nodal point is obtained. By using these displacement solutions, stress and strain in each element can be calculated.

Non Structural problem: In non structural problem, temperatures or fluid pressure at each nodal point is obtained. By using these values, Properties such as heat flow, fluid flow, etc for each element can be calculated.

- 8. What are the methods are generally associated with the finite element analysis? The following two methods are generally associated with the finite element analysis. They are
- a. Force method.
- b. .Displacement or stiffness method

9. What is force method and stiffness method?

In force method, internal forces are considered as the unknowns of the problem. In displacement or stiffness method, displacement of the node are considered as the unknowns of the problem. Among them two approaches, displacement method is desirable.

- 10. What is polynomial type of interpolation functions are mostly used in FEM? The polynomial type of interpolation functions are mostly used due to the following reasons:
 - a. It is easy to formulate and computerize the finite element equations.
 - b. It is easy to perform differentiation or integration.
 - c. The accuracy of the results can be improved by increasing the order of the polynomial .
- Name the variational methods.
 a.Ritz method.
 b. Rayleigh Ritz method
- 12. Name the weighted residual methods.
 - a. Point collocation method.
 - b. subdomain collocation method.
 - c. Least square method
 - d. galerkin's method
- 13. What is meant by post processing? Analysis and evaluation of the solution results is referred to as post processing. Post processor computer programs help the user to interpret the results by displaying them in graphical form.
- 14. What is Rayleigh ritz method? Rayleigh ritz method is a integral approach method which is useful for solving complex structural problems, encountered in finite element analysis. This method is possible only if a suitable functional is available.
- 15. What is meant by assemblage FEA? The art of subdividing a structure into a convenient number of smaller components is known as discretization. These smaller components are then put together. The process of uniting the various elements together is called assemblage.
- 16. What is meant by DOF?

When the force or reaction acts at nodal point, node is subjected to deformation. The deformation includes displacement, rotations, and/or strains. These are collectively known as degrees of freedom (DOF).

17. What is aspect ratio?

Aspect ratio is defined as the ratio of the largest dimension of the element to the smallest dimension. In many cases, as the aspect ratio increases, the inaccuracy of the solution increases. The conclusion of many researches is that the aspect ratio should be close to unity as possible.

18. What is truss element?

The truss elements are the part of a truss structure linked together by point joints, which transmit only axial force to the element.

- 19. List the two advantages of post processing?
 - a. Required result can be obtained in graphical form.
 - b. Contour diagrams can be used to understand the solution easily and quickly.
- 20. During discretization, mention the places where it is necessary to place a node

The following places are necessary to place a node during discretization process.

- a. Concentrated load-acting point.
- b. Cross section changing point
- c. Different material interjunction point
- d. Sudden change in load point.
- 21. What is the difference between static and dynamic analysis? Static analysis: The solution of the problem does not vary with time is known as static analysis.

Example: Stress analysis on a beam.

Dynamic analysis: The solution of the problem varies with time is known as dynamic analysis.

Example: vibration analysis problems.

- 22. Name the four FEA softwares?
 - a. ANSYS
 - b. NASTRAN
 - c. COSMOS
 - d. NISA
- 23. Differentiate between global and local axes.

Local axes are established in an element. Since it is in the element level, they change with the change in orientation of the element. The direction differs from element to element.

Global axes are defined for the entire system. They are same in direction for all the elements even though the elements are differently oriented.

- 24. What are the types of loading acting on the structure? There are three types of loading acting on the body. They are:
 - a. Body force (f)
 - b. Traction force (T)
 - c. Point load (P)
- 25. Define body force (f).

A body force is distributed force acting on every elemental volume of the body. Unit: Force per unit volume. Example: Self-weight due to gravity