Finite Element Analysis with ANSYS Workbench

Van-Hoi Nguyen

Multiphysics Systems Design Laboratory

Department of Mechanical Systems Engineering

Jeonbuk National University

Nov 03, 2025

Main contents

Finite Element Analysis with ANSYS Workbench

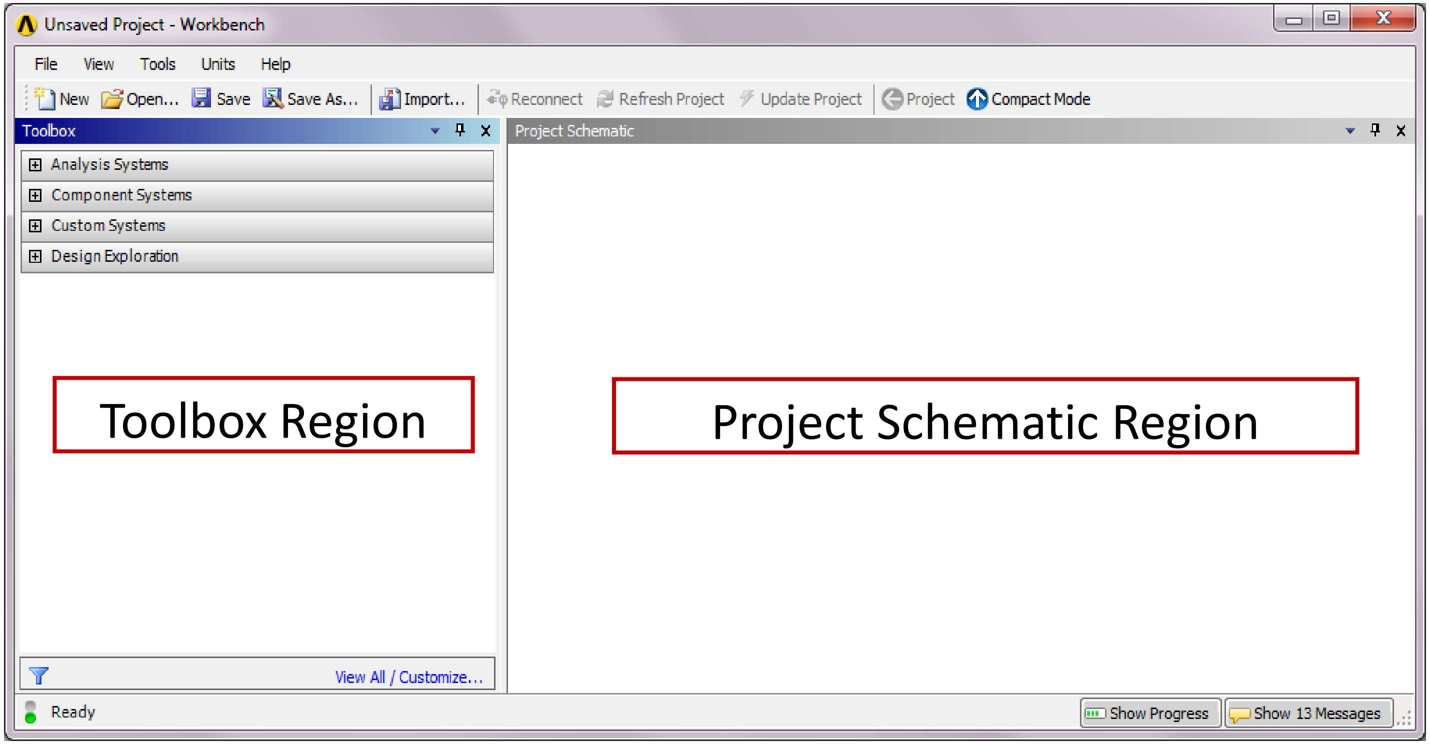
Introduction of ANSYS Workbench

Introduction of ANSYS Workbench

The User Interface

I

- The Workbench interface is composed primarily of a Toolbox region and a Project Schematic region.



Introduction of ANSYS Workbench

The User Interface

I

- The toolbox contains four groups of systems.

<u>Analysis Systems</u>: Pre-defined analysis templates used to build your project, including static structural, steady-state thermal, transient thermal, fluid flow, modal, shape optimization, linear buckling and many others.

<u>Component Systems</u>: Component applications that can be used to build or expand an analysis system, including geometry import, engineering data, mesh, post processing and others.

<u>Custom Systems</u>: Coupled-field analysis systems such as fluid solid interaction, pre-stress modal, thermal-stress and others.

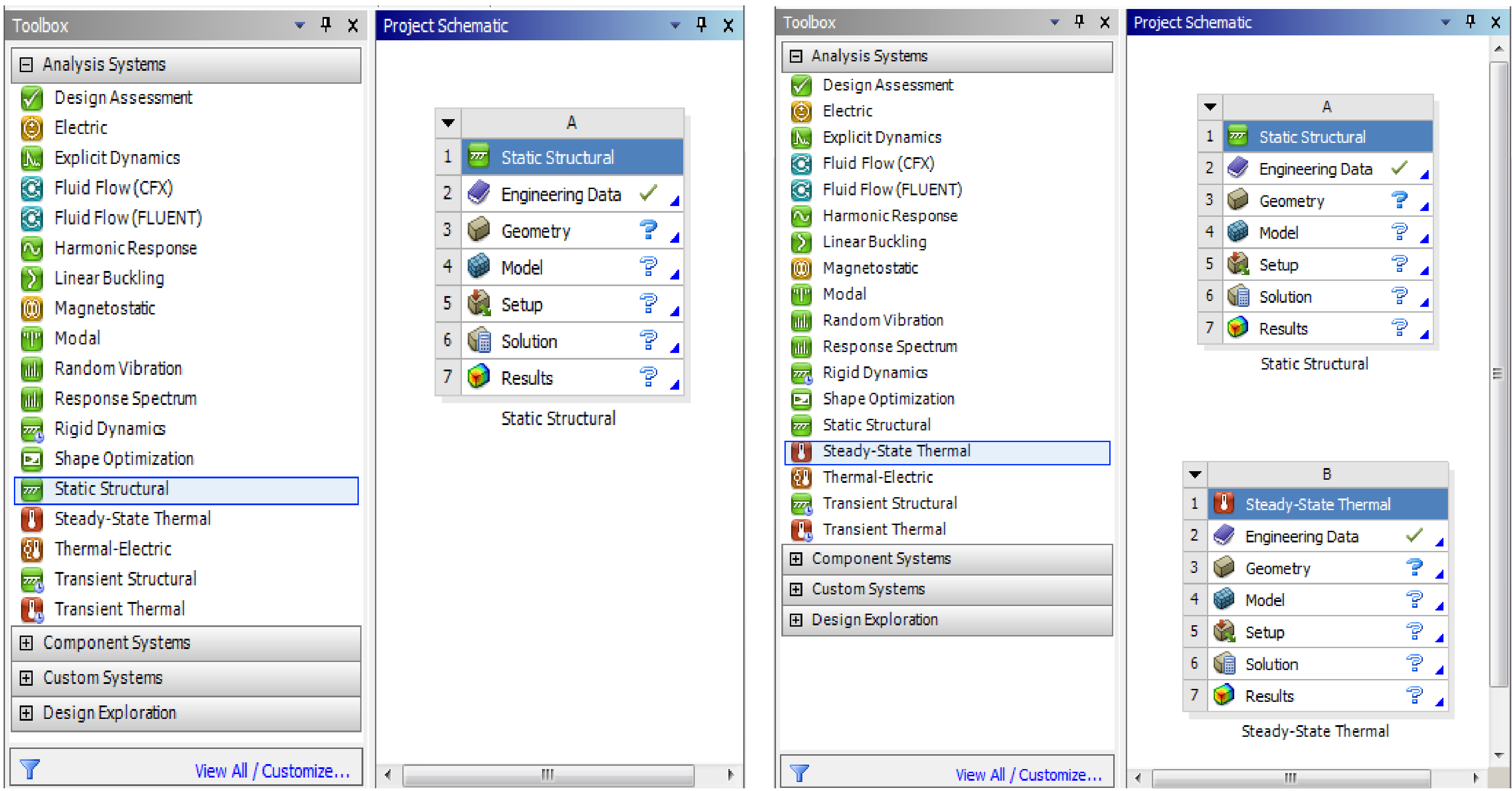
<u>Design Exploration</u>: Parametric optimization studies such as goal-driven optimization, parameters correlation, six sigma analysis and others.

Introduction of ANSYS Workbench

The User Interface

I

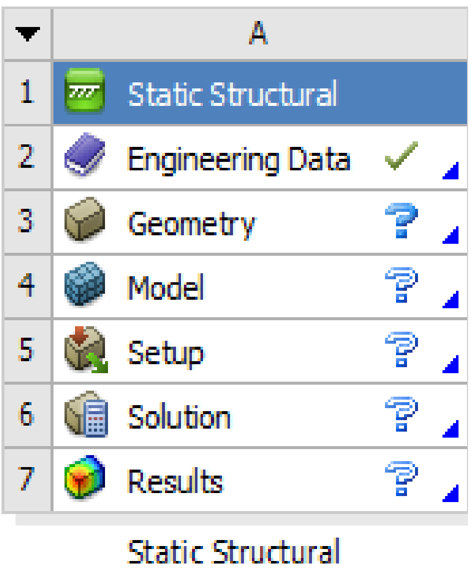
- ❑ The project schematic is a graphical representation of the workflow process.



Introduction of ANSYS Workbench

The User Interface

- Cells are components that make up an analysis system.



Engineering Data: Define or edit material models.

Geometry: Create, import or edit the geometry model.

Model/Mesh: Assign materials, define coordinate system & generate mesh.

Setup: Apply loads, boundary conditions and configure analysis settings.

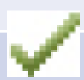
Solution: Access the model solution or share solution data with other downstream systems.

Results: Indicate the results availability and status (i.e., post-processing).

Introduction of ANSYS Workbench

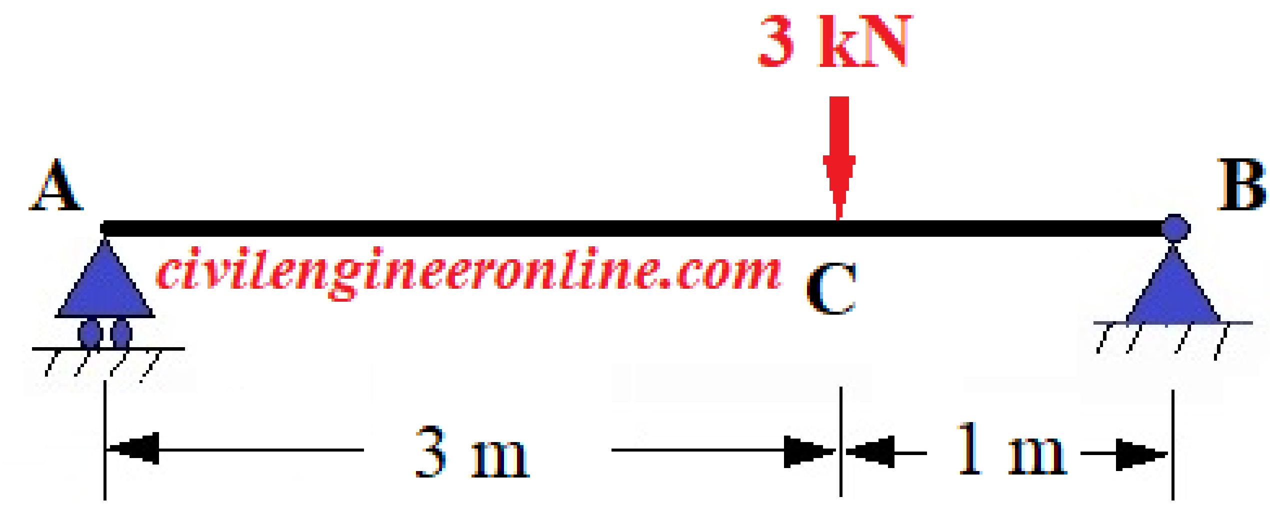
The User Interface

- Indicator icons and descriptions of the various cell states

Cell State	Indicator	Description
Unfulfilled		Need upstream data to proceed.
Refresh Required		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 on upstream data.
Up to Date		Data is up to date and no attention is required.
Input Changes Pending		An update or refresh action is needed to recalculate based on changes made to upstream cells.
Interrupted		Solution has been interrupted. A resume or update action will make the solver continue from the interrupted point.
Pending		Solution is in progress.

Structural Static Analysis

<Problem Description> The beam cross-section is rectangular with a width of 5cm and depth of 10 cm. The modulus of elasticity of beam material is 200 GPa. Compute the deflection.

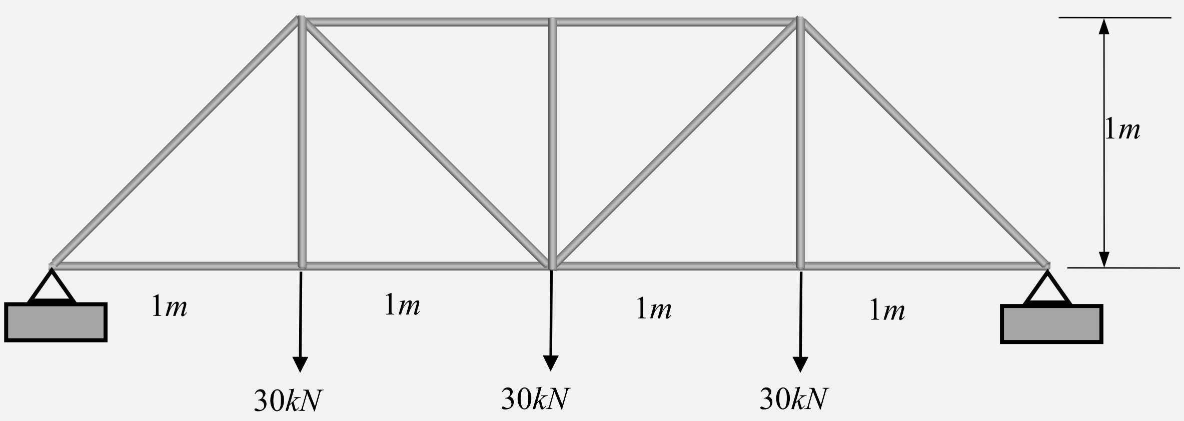


Structural Static Analysis

Truss Bridge

II

<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.



Material: Douglas Fir

$E = 13.1\ GPa$

$\nu = 0.29$

Member cross section:

height = 6 cm

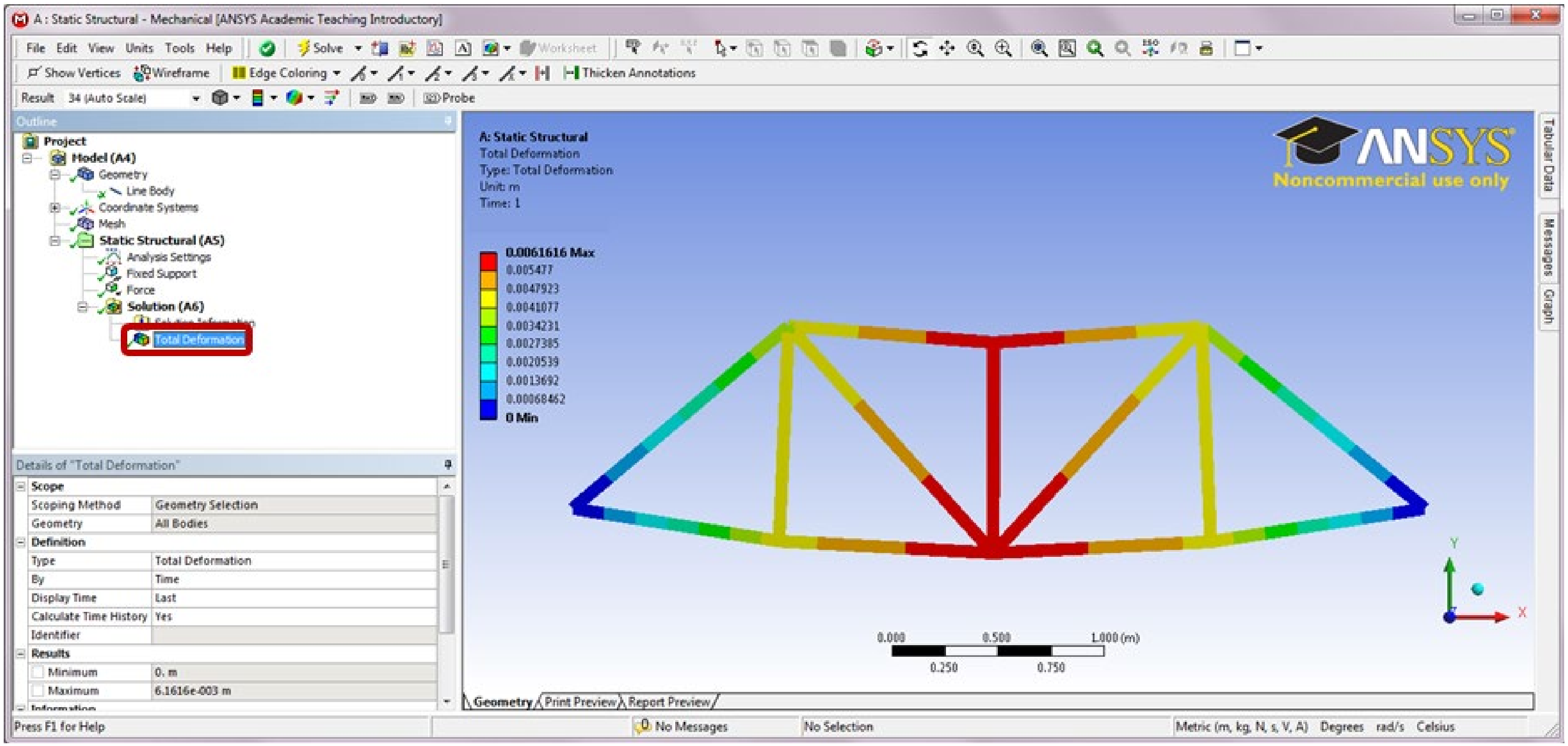
width = 6 cm

Structural Static Analysis

Truss Bridge

II

Run a ***Static Structural Analysis*** to review the truss deformation results.

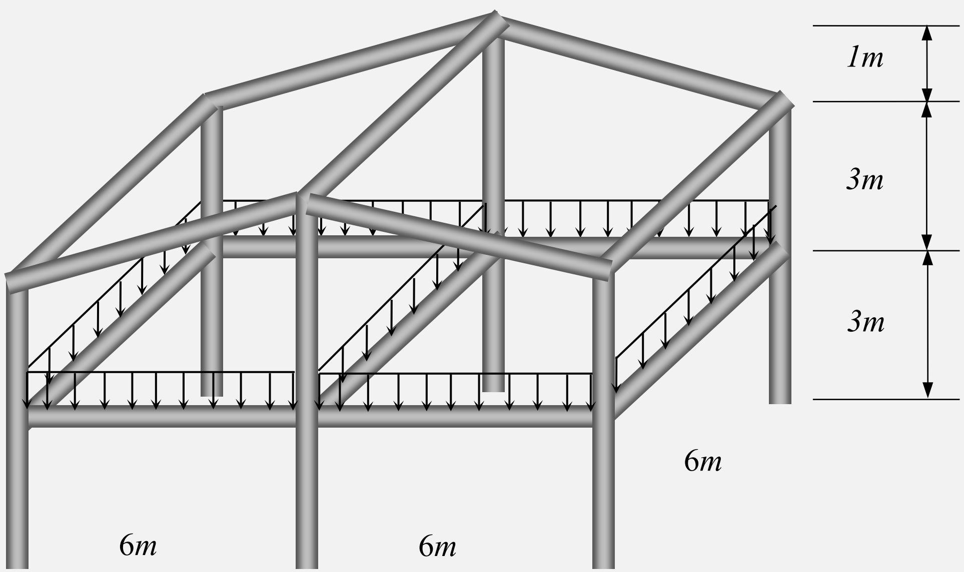


Structural Static Analysis

Beam Building

II

<Problem Description> Steel framing systems provides cost-effective solutions for low-rise buildings. They have high strength-to-weight ratios, and can be pre-fabricated and custom-designed. Consider the following two-story 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*

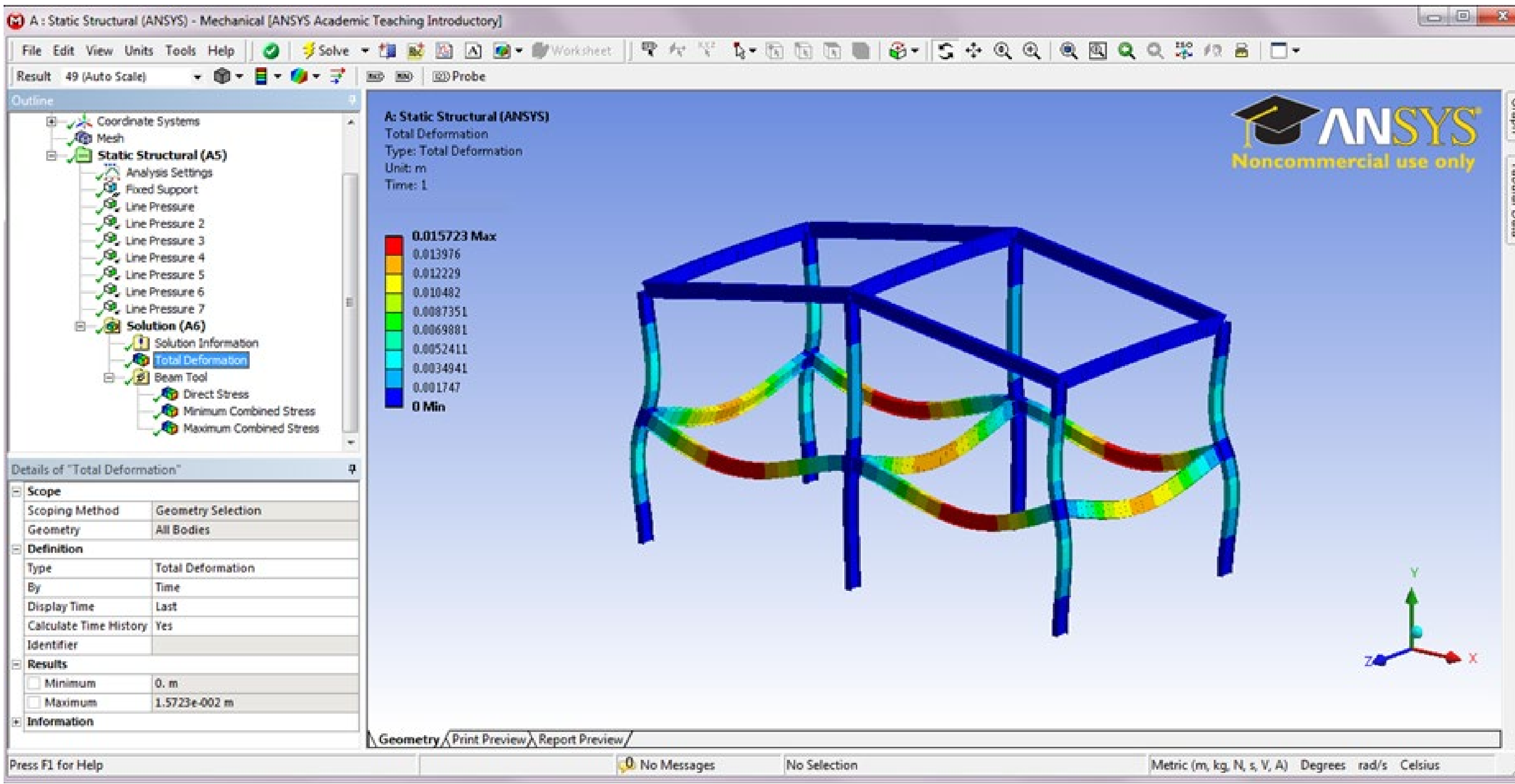
web thickness = 11.5 *mm*

flange thickness = 7.3 *mm*

Structural Static Analysis

Beam Building

Run a ***Static Structural Analysis*** to review the frame deformation results.

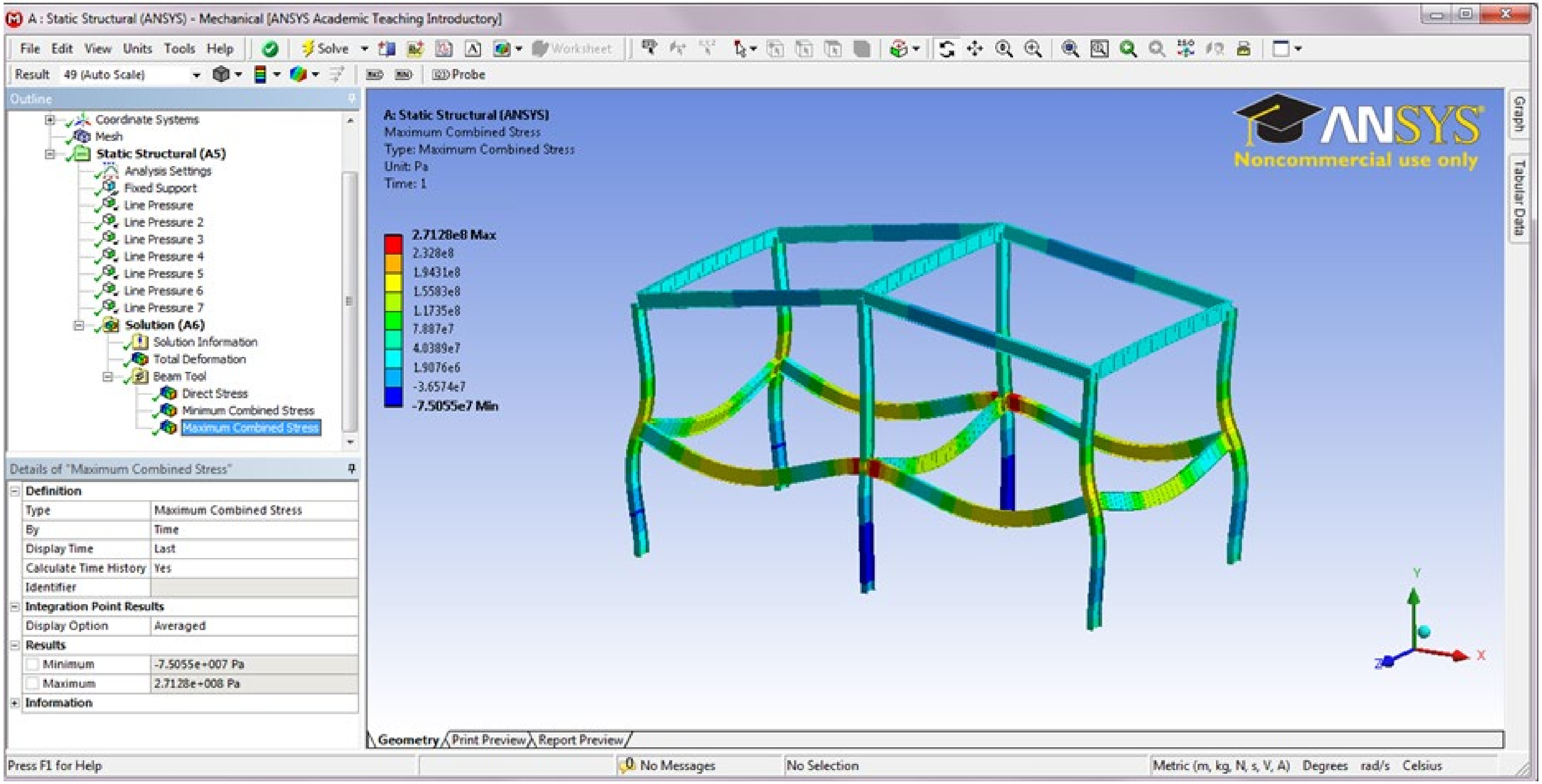


Structural Static Analysis

Beam Building

II

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

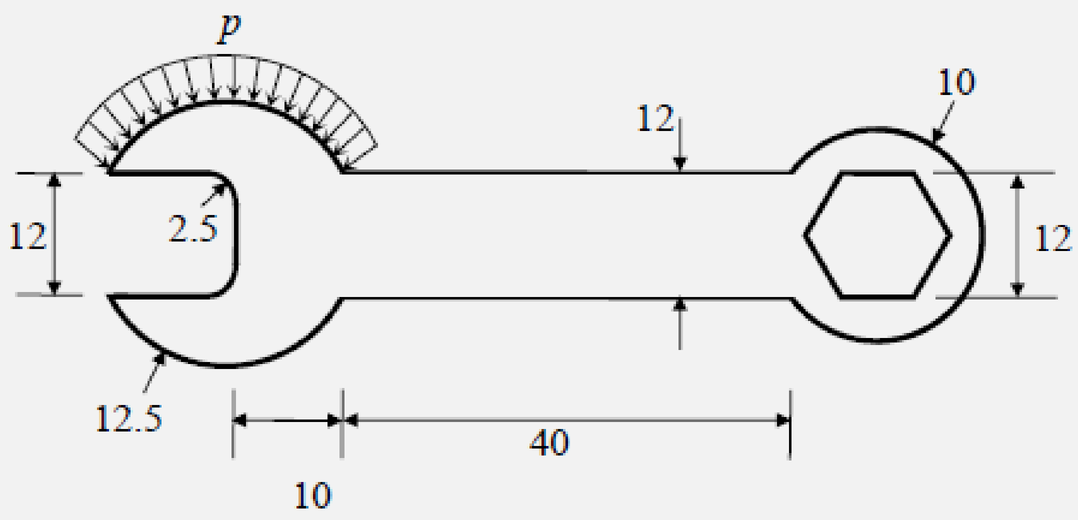


Structural Static Analysis

2D Wrench

II

<Problem Description> A combination wrench is a convenient tool that is used to apply torque to loosen or tighten a fastener. The wrench shown below is made of stainless steel and has a thickness of 3*mm*. Determine the maximum deformation and the distribution of von Mises stresses under the given distributed load and boundary conditions.



Material: Stainless Steel

$E = 193$ GPa, $\nu = 0.27$

Boundary Conditions:

The hexagon on the right is fixed on all sides.

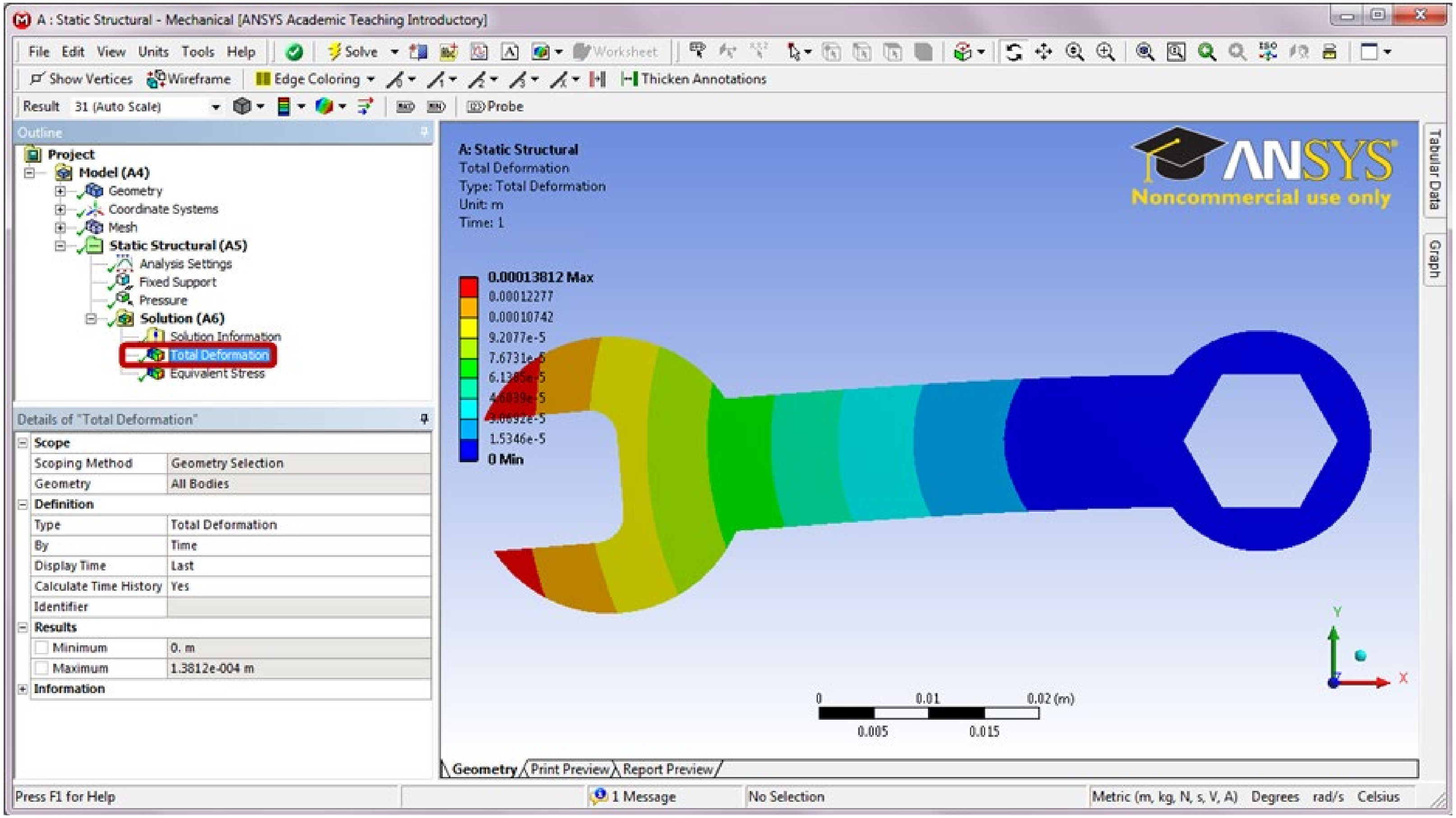
Pressure $p = 2$ MPa

(All units are in millimeters.)

Structural Static Analysis

2D Wrench

Run a ***Static Structural Analysis*** to review the wrench deformation results.

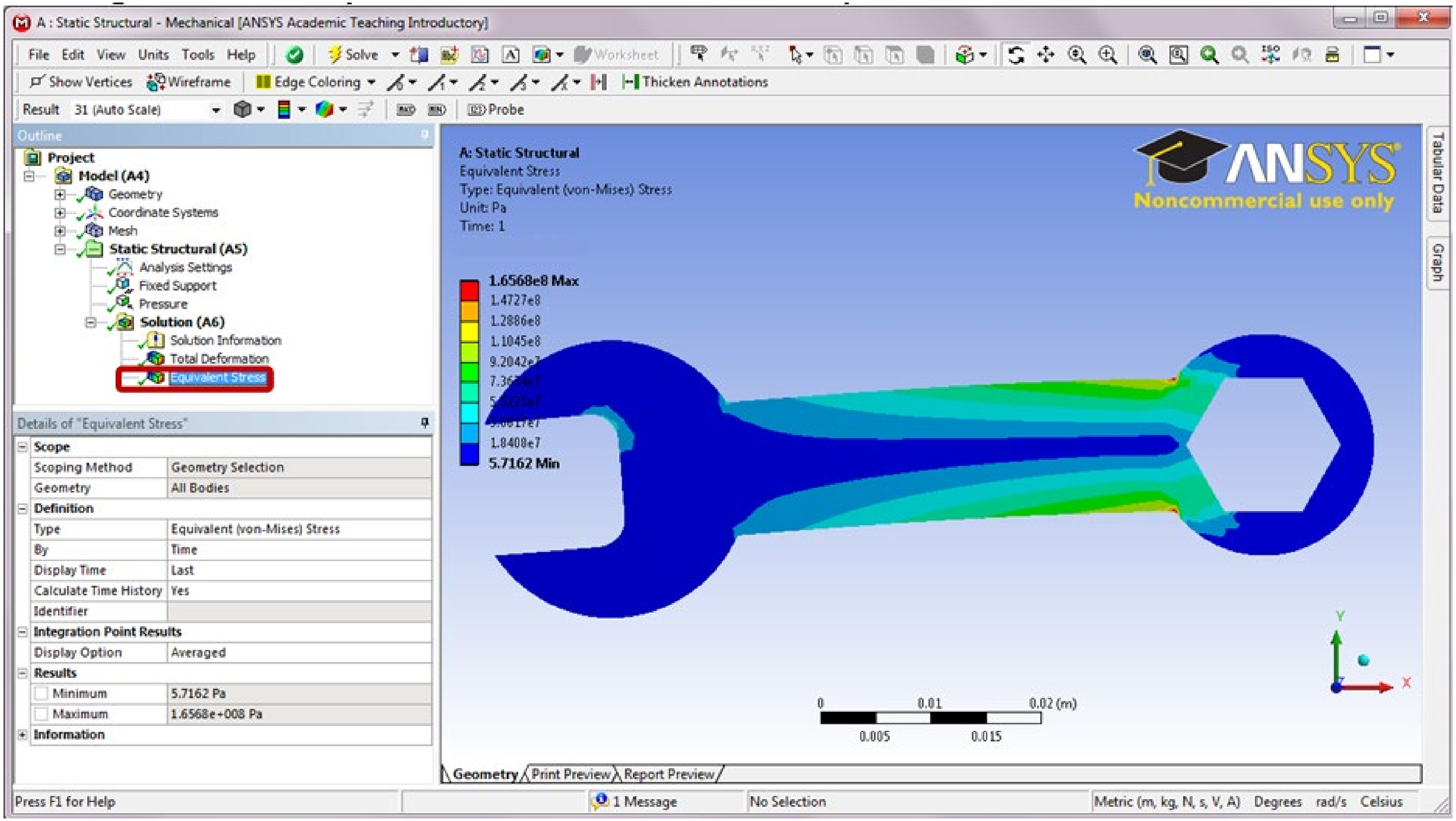


Structural Static Analysis

II

2D Wrench

Equivalent Stress in the ***Outline*** to review the von-Mises stress distribution.

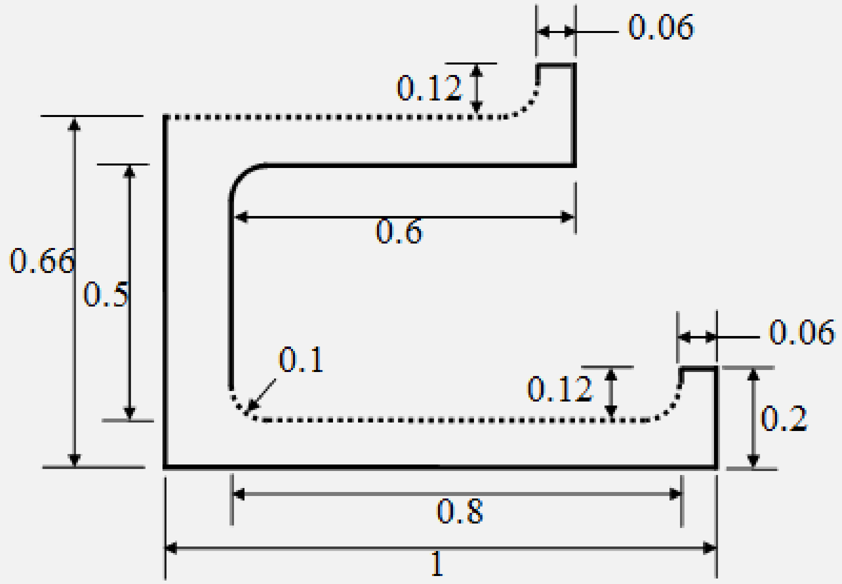


Structural Static Analysis

Garden Fountain

II

<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 dimensions of the planar cross section 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

$\nu = 0.15$

Boundary Conditions:

Bottom edge: fixed.

Left edge: axis of symmetry.

Hydro pressure on dotted edges.

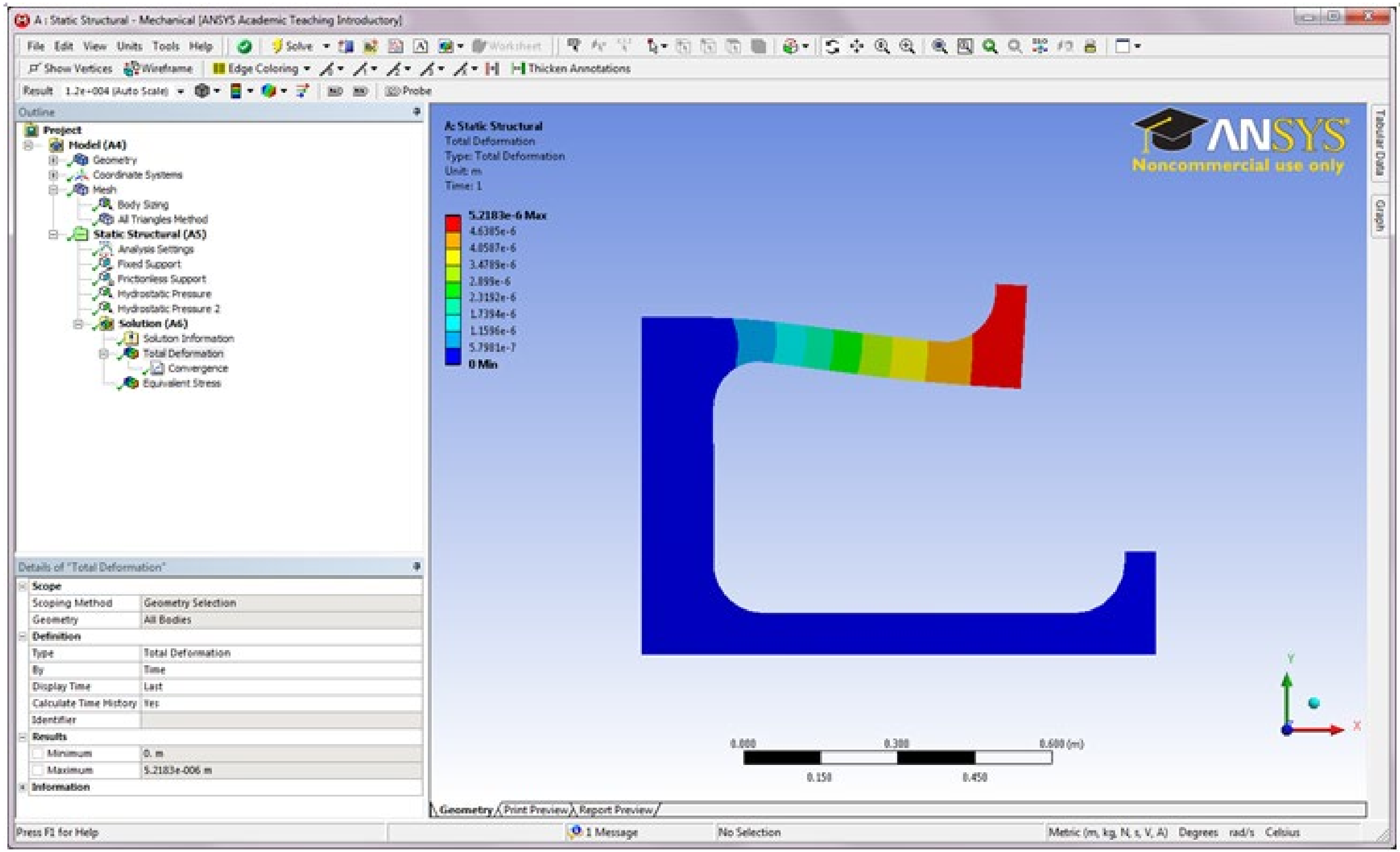
(All units are in meter)

Structural Static Analysis

Garden Fountain

II

Run a ***Static Structural Analysis*** to review the axisymmetric deformation results.

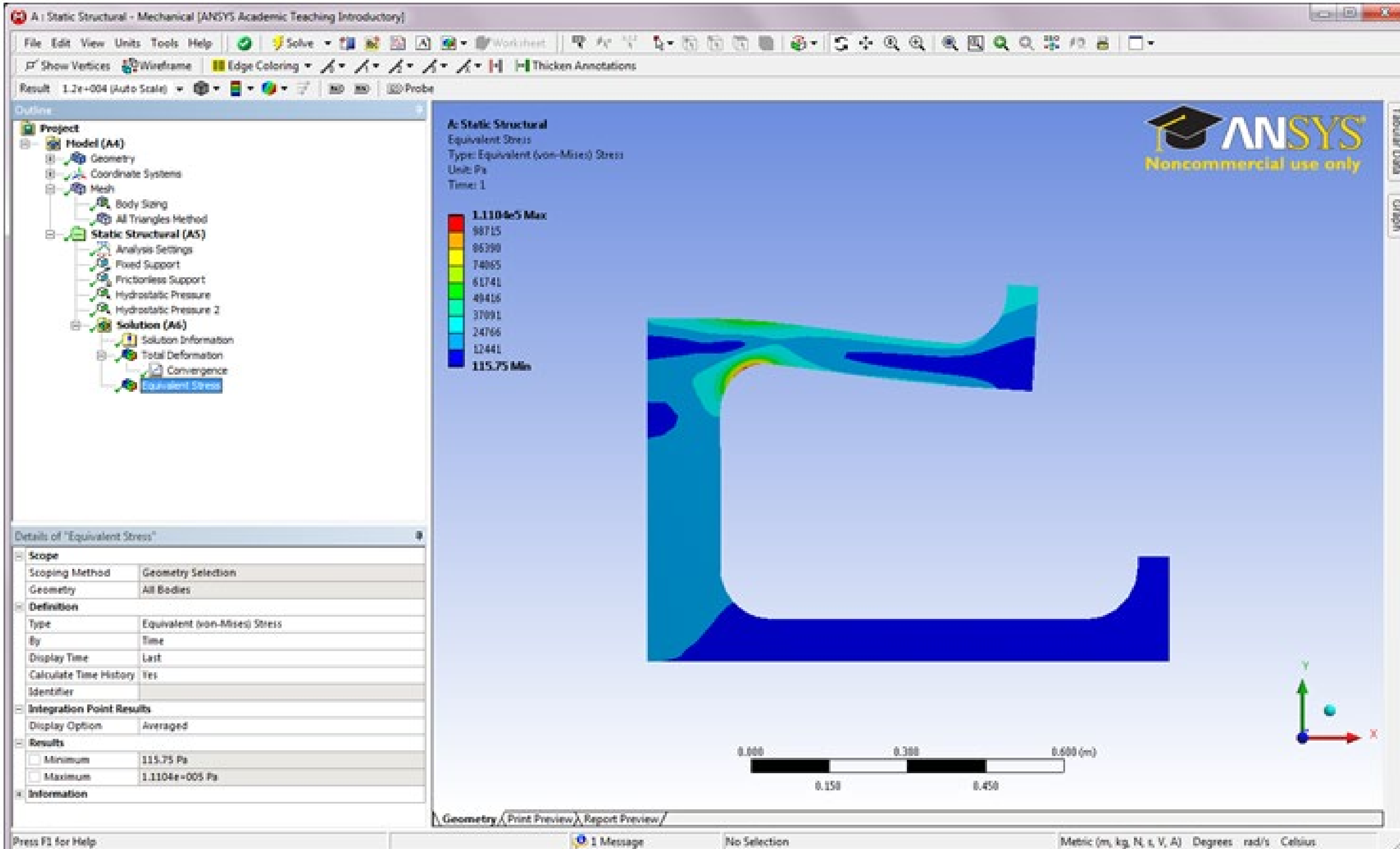


Structural Static Analysis

Garden Fountain

II

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

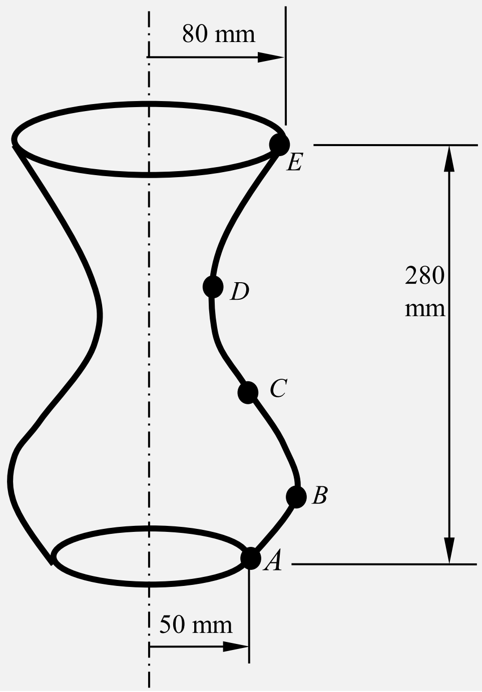


Structural Static Analysis

Shell Vases

II

<Problem Description> Vases are decorative pieces that can be of any artistic shapes. The figure below gives the dimensions of a flower vase made of glass. Assume that the vase has a uniform thickness of 4 mm. The water level reaches 100 mm below the opening of the vase. Determine the maximum deformation and von Mises stress under the hydrostatic pressure.

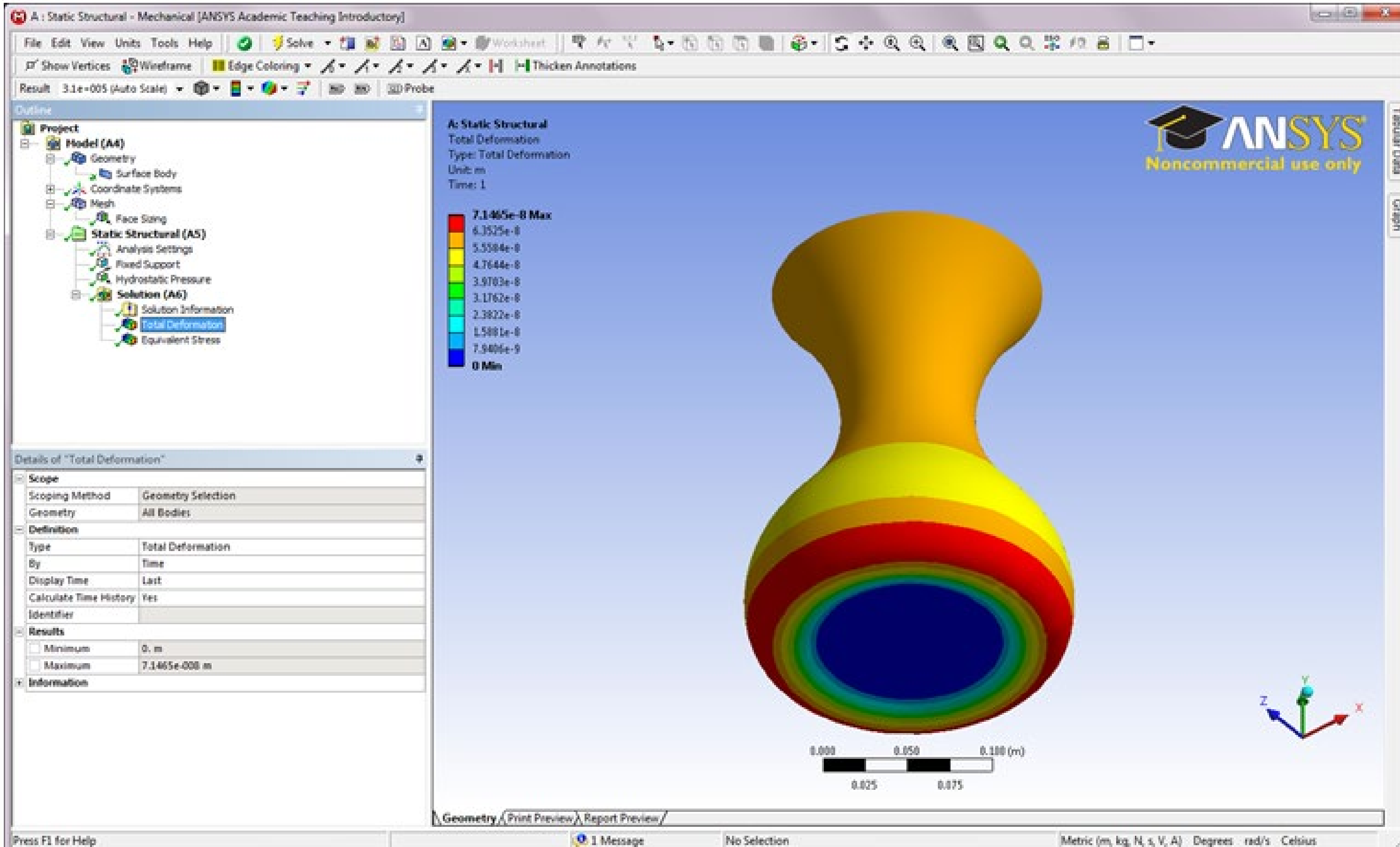


Material: Glass
$E = 70\ GPa$
$\nu = 0.17$
Boundary Conditions:
Bottom surface: fixed.
Coordinates of Construction Points:
A: (50 mm, 0 mm, 0mm)
B: (90 mm, 40 mm, 0mm)
C: (60 mm, 120 mm, 0 mm)
D: (40 mm, 180 mm, 0 mm)
E: (80 mm, 280mm, 0mm)

Structural Static Analysis II

Shell Vases

Run a ***Static Structural Analysis*** to review the deformation results of the vase.

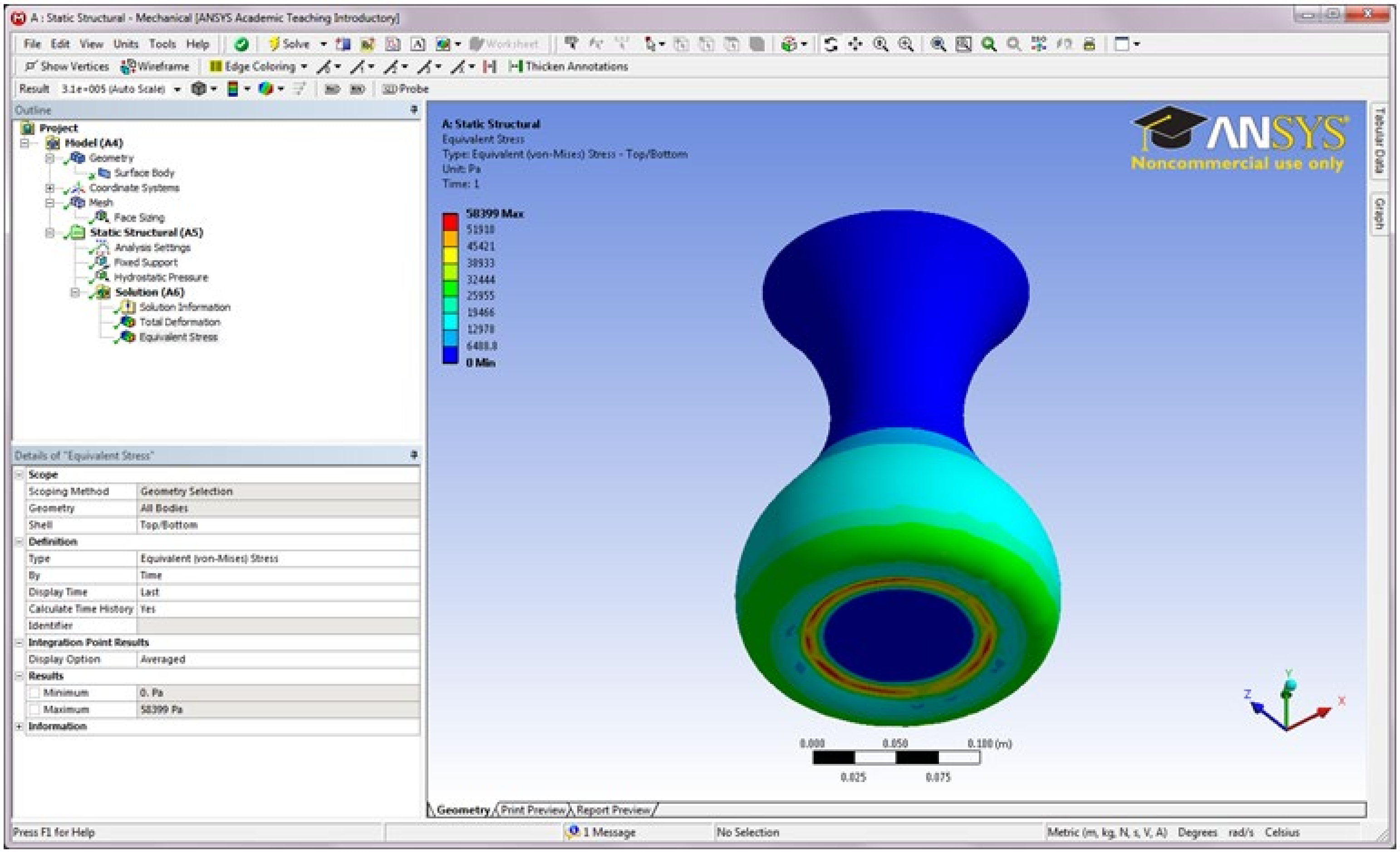


Structural Static Analysis

Shell Vases

II

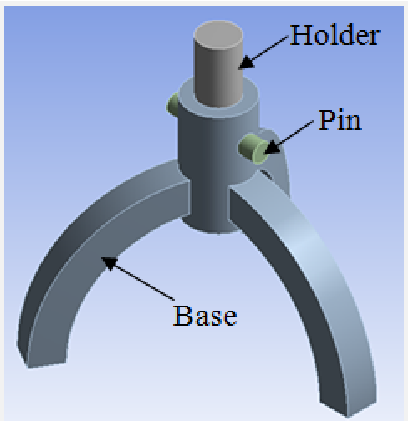
Click on ***Equivalent Stress*** in the ***Outline*** to review the stress results.



Structural Static Analysis

3D Base

<Problem Description> A base stand assembly includes a base, a holder and a pin, as shown in the following figure. The stand assembly is made of structural steel. Assume a no-separation condition for all contact regions. Determine the deformation and von Mises stress distributions of the assembly under the given load and boundary conditions.



Material: Structural Steel (E = 200 GPa, ν = 0.3)

Boundary Conditions:

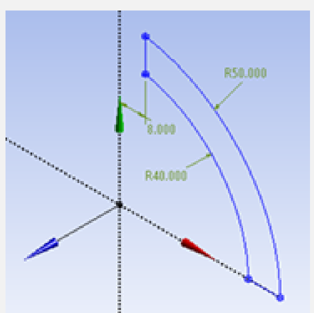
The bottom faces of the leg-base are fixed.

A downward force of 1kN is applied to the holder's top face.

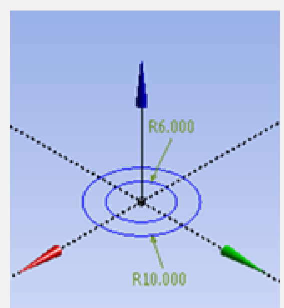
Geometry Construction:

The bottom of the hub-base is 35 mm above the ground level.

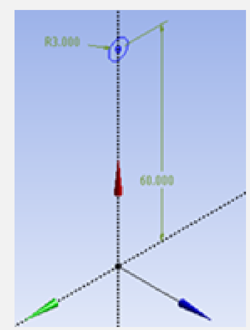
The holder is 36 mm tall, 18 mm of which is in contact with the hub-base.



Sketch of the leg-base (extrude 5mm on both sides)



Sketch of the hub-base (extrude 35mm on one side)

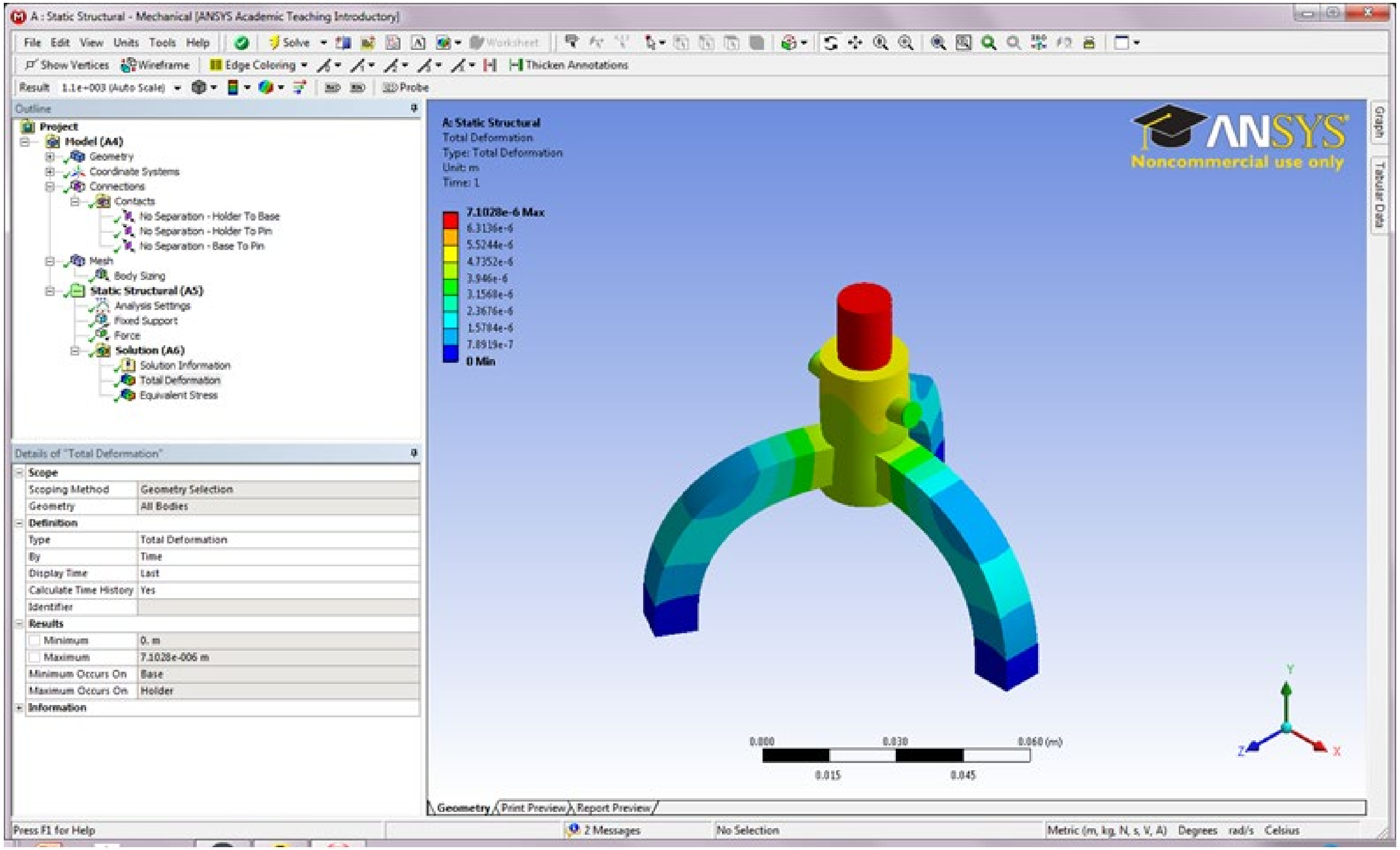


Sketch of the pin (extrude 15mm on both sides)

Structural Static Analysis

3D Base

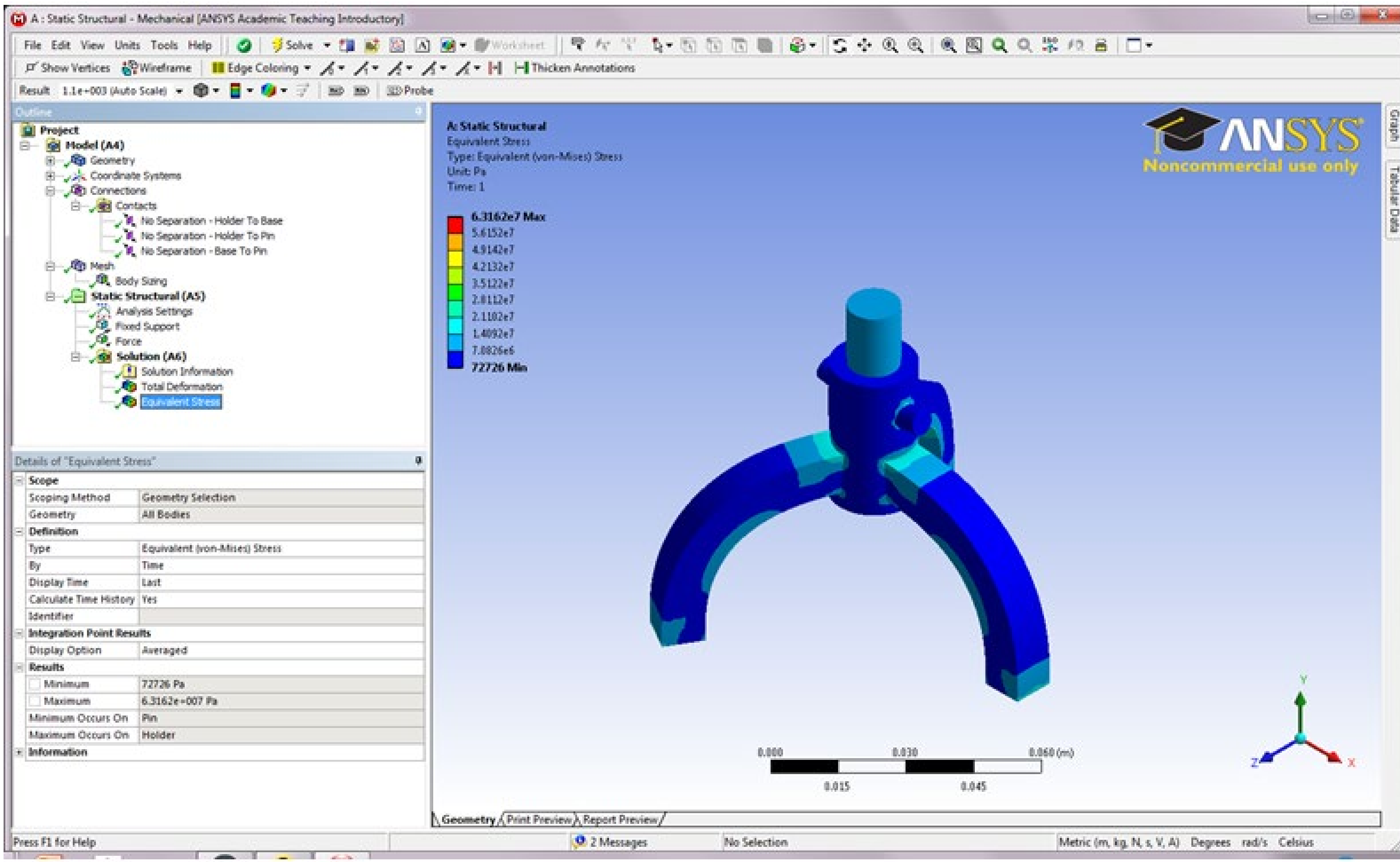
Run a ***Static Structural Analysis*** to review the assembly deformation results.



Structural Static Analysis

3D Base

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

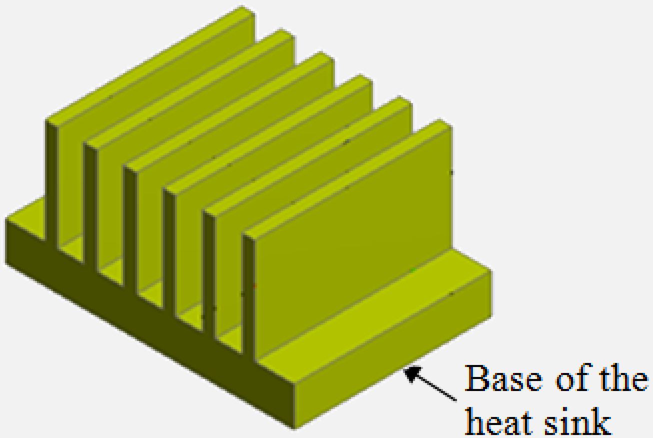


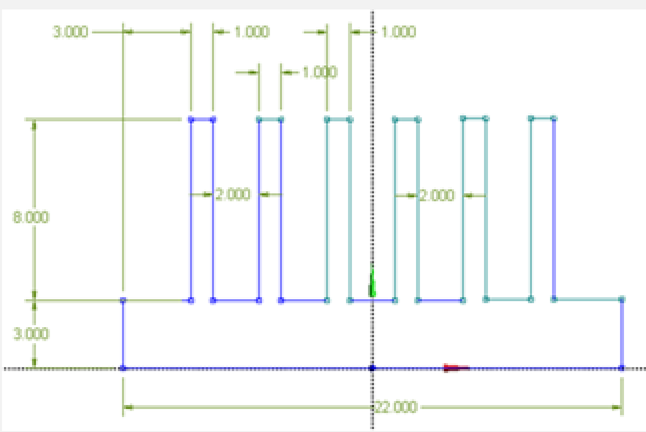
Structural Dynamic Analysis

Structural Dynamic Analysis

Heat Sink

II





All dimensions are in millimeters.

Material: Aluminum

k= 170 W/(m·K)

ρ= 2800 kg/m^3; c = 870 J/(kg·K)

E = 70 GPa; ν = 0.3

α = 22x10^{-6} 1/°C

Boundary Conditions:

Air temperature of 28°C; h = 30W/(m^2·°C).

Steady-state: q' = 1000W/m^2 on the base.

Transient: Square wave heat flux on the base.

Initial Conditions:

Steady-state: Uniform temperature of 28°C.

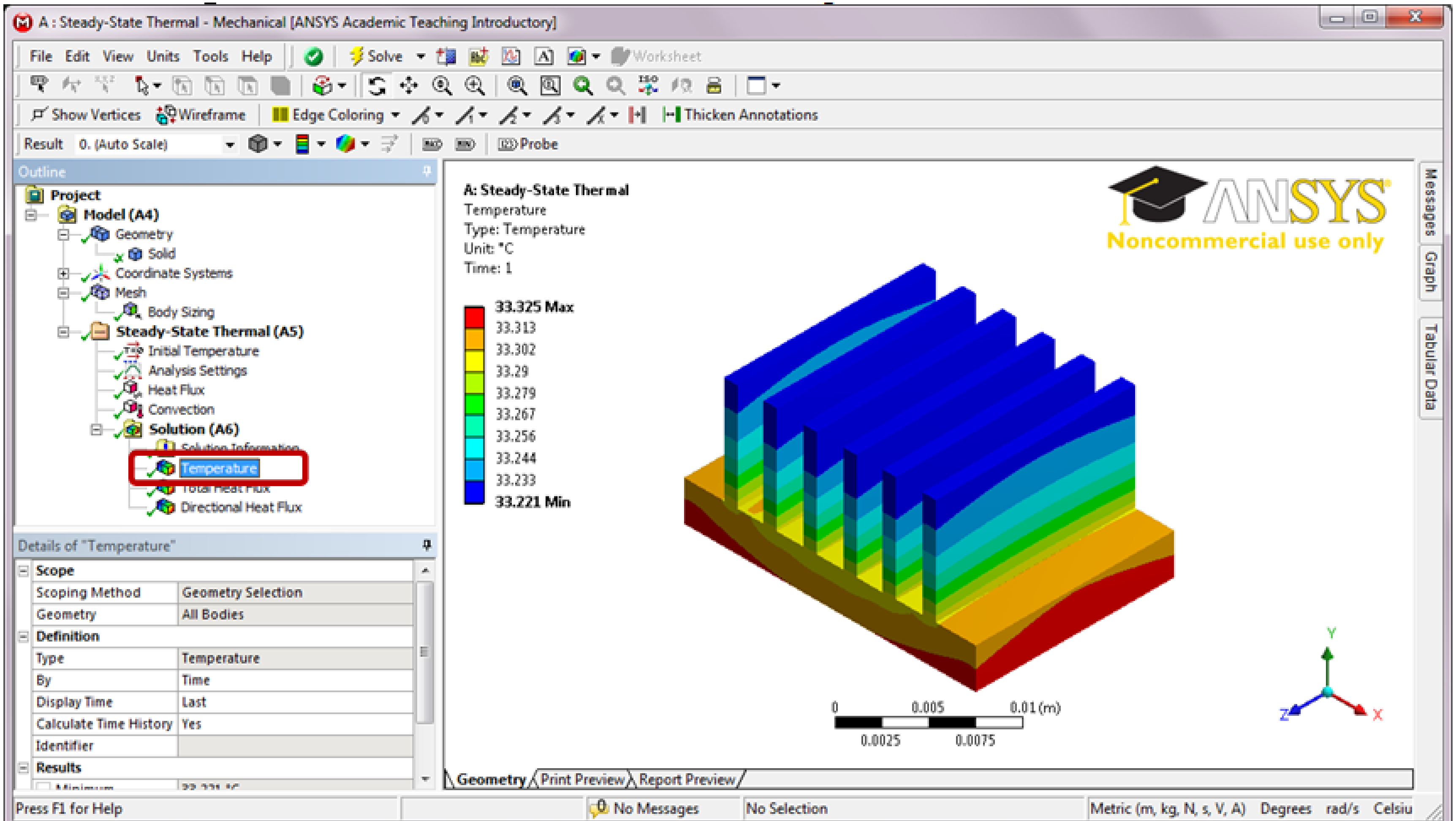
Transient: Steady-state temperature results.

Structural Dynamic Analysis

II

Heat Sink

Run a ***Steady-State Thermal Analysis*** to view the temperature results.

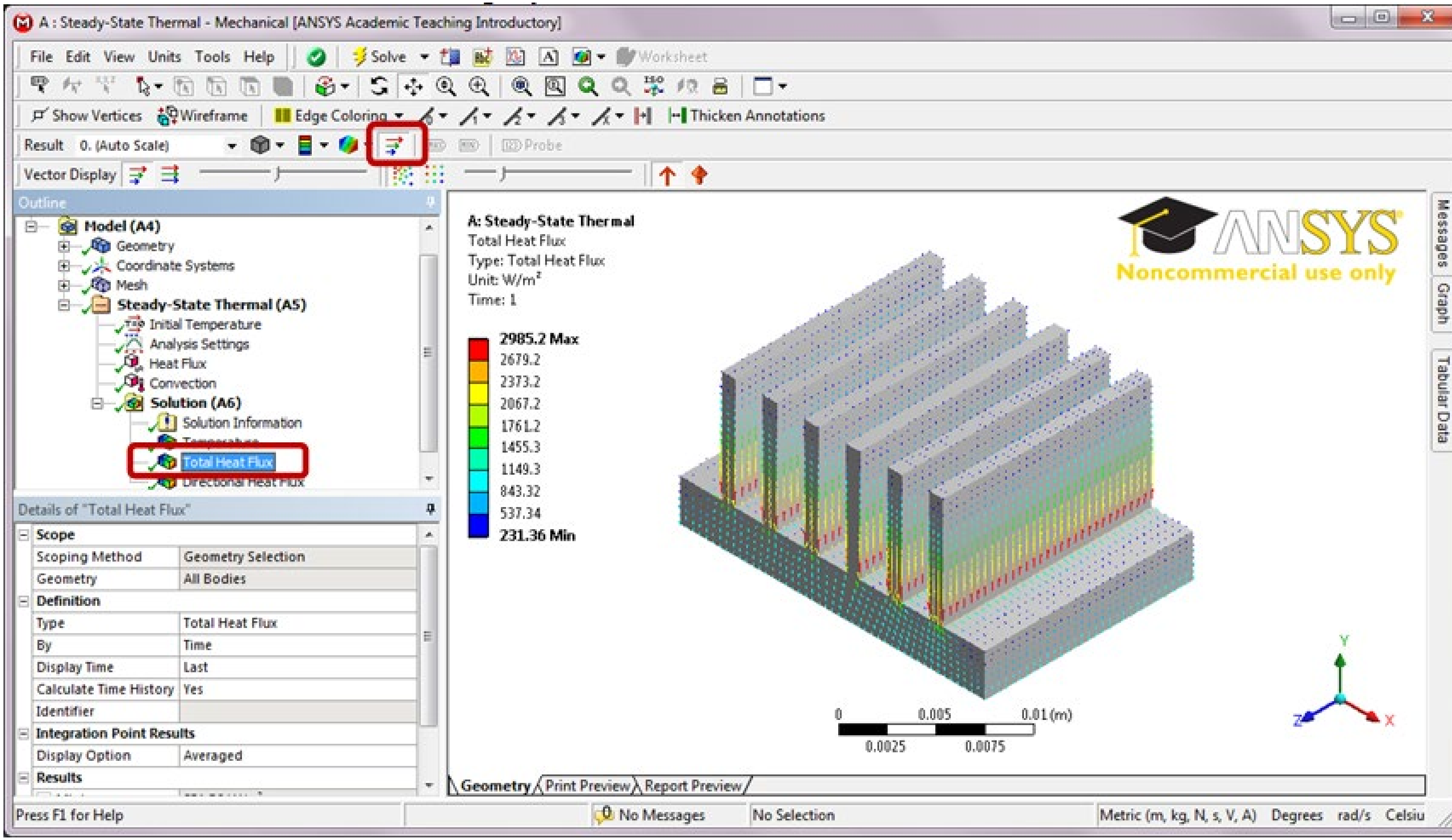


Structural Dynamic Analysis

II

Heat Sink

Select ***Total Heat Flux*** to display the heat flux with directional arrows.

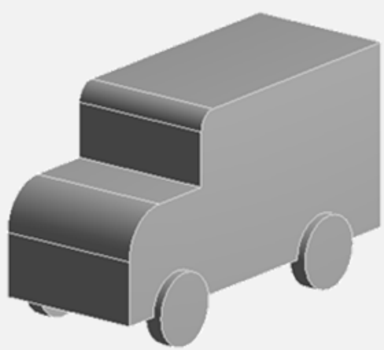


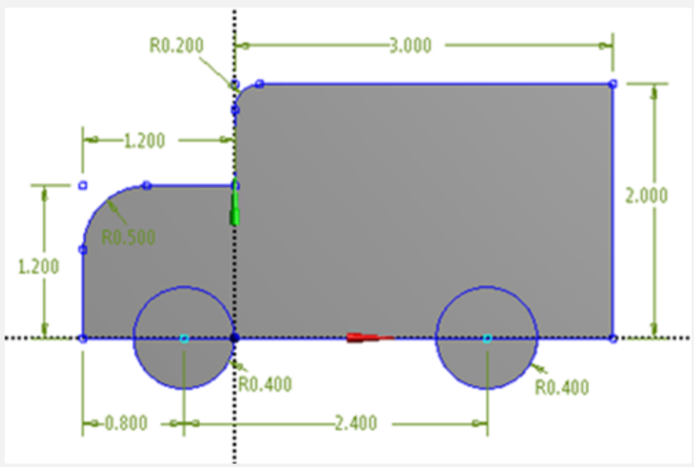
Structural Dynamic Analysis

Structural Dynamic Analysis

Aerodynamic Vehicles

<Problem Description> The aerodynamic performance of vehicles can be improved by utilizing computational fluid dynamics simulation. Conduct fluid analysis of the air flow passing through a truck. Assume air at room temperature of 25°C for the flow field with an air velocity of 40 km/hr blowing from left to right. Use non-slip boundary conditions along the walls of the truck and the ground surface. Find the airflow pattern as well as the pressure and velocity distributions of the flow field around the truck.





All dimensions are in meters.

Flow Field: Incompressible air at 25°C.

Boundary Conditions: Air is blowing from left to right at a velocity of 40 km/hr.

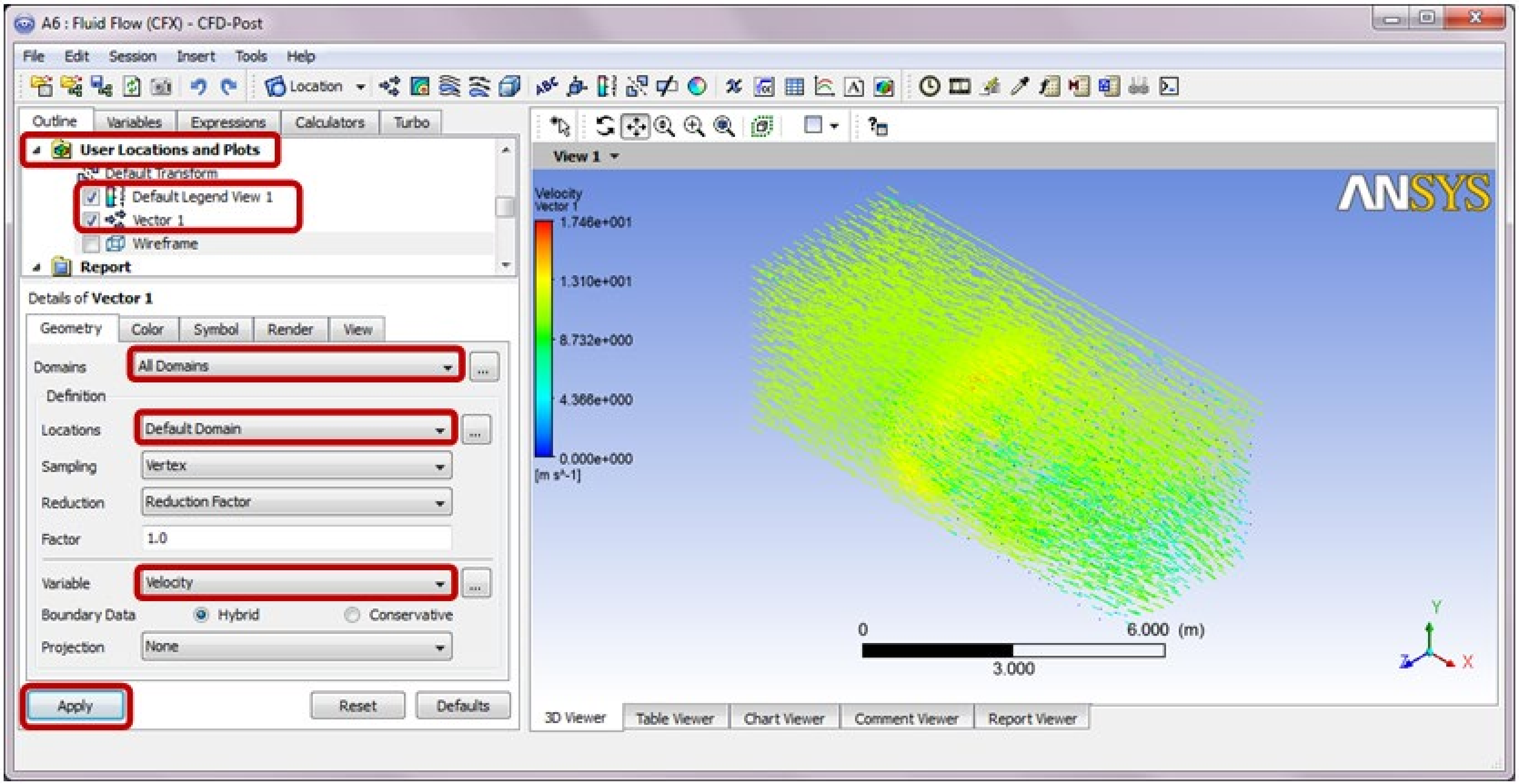
Non-slip boundaries along the walls of the truck and the ground surface.

Structural Dynamic Analysis

II

Heat Sink

Plot Velocity Vector – Right-click ***User Locations and Plots*** in the ***Outline***, and select ***Insert*** and then ***Vector***.

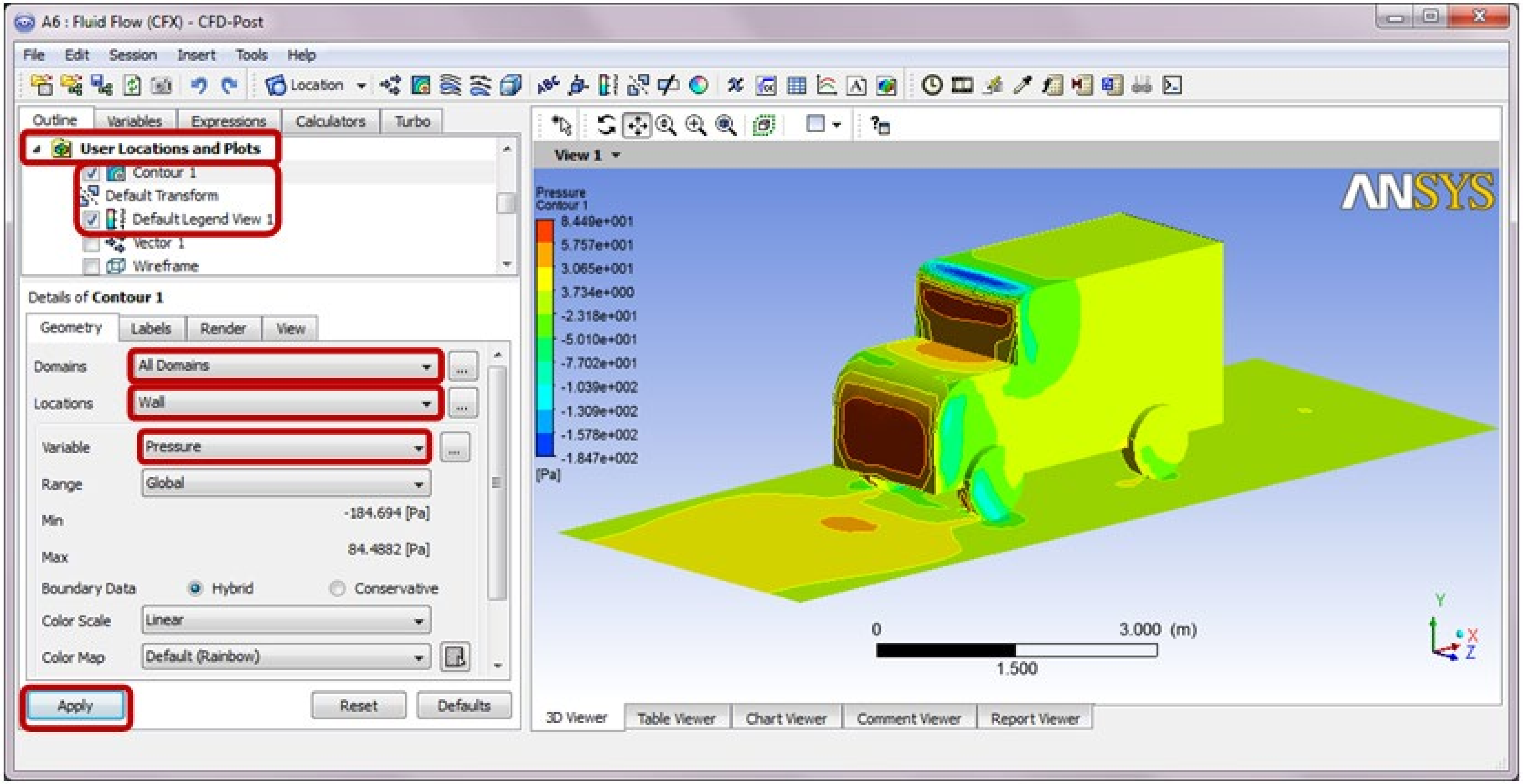


Structural Dynamic Analysis

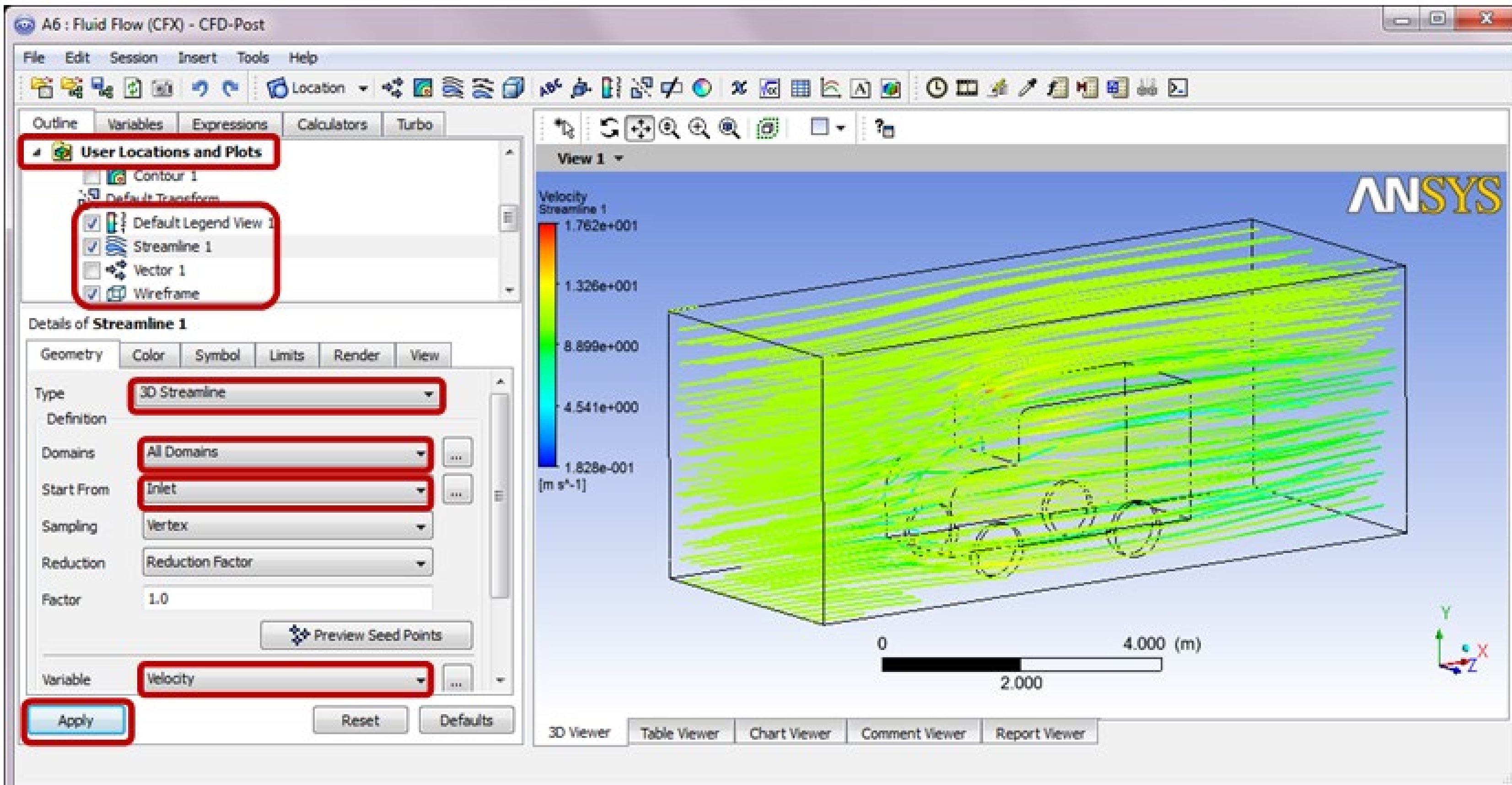
Heat Sink

II

Plot Wall Pressure Contour – Right-click ***User Locations and Plots*** in the ***Outline***, and select ***Insert*** and then ***Contour***.



Plot Streamline – Right-click ***User Locations and Plots*** in the ***Outline***, and select ***Insert*** and then ***Streamline***.



Thank You for Your Attention

Q&A