

Finite Element Analysis Using Pro/Engineer

A.1 INTRODUCTION

Pro/ENGINEER is a three-dimensional product design tool that promotes practices in design while ensuring compliance with industry and company standards. Integrated, parametric, three-dimensional CAD/CAM/CAE solutions allow engineers to design products fast, while improving innovation and quality to ultimately create exceptional products. Although the main function of Pro/ENGINEER is a solid modeler, it also provides various engineering analyses tools such as kinematic analysis, structural and thermal analysis. In this chapter, we will introduce structural finite element modeling and analysis using Pro/ENGINEER Mechanical, which is a CAE (Computer Aided Engineering) tool that enables engineers to simulate the physical behavior of a model and to understand and improve the mechanical performance of the design. It can directly calculate stresses, deflections, frequencies, heat transfer paths, and other factors, showing how the model will behave in a test laboratory or in the real world.

The Pro/ENGINEER Mechanical product line features two modules—Structure and Thermal—each of which solves for a different family of mechanical behaviors. Structure focuses on the structural integrity of the model, while Thermal evaluates heat-transfer characteristics.

Mechanical is available in two basic modes—integrated mode and independent mode. In integrated mode, engineers perform all Mechanical functions within Pro/ENGINEER environment. This version of the product offers the convenience and power of Pro/ENGINEER's parametric feature-creation technology coupled with the full range of Mechanical's solution software. However, some specific capabilities of Mechanical cannot be used in the integrated mode. In independent mode, engineers work in a separate user interface, developing the model from imported geometry or geometry that is created using Mechanical's geometry-creation facilities. In this Chapter we will use the integrated mode in performing finite element analysis.

A.2 GETTING START

To access Pro/Engineer program on Windows systems, choose **Start > All Programs > PTC > Pro Engineer > Pro Engineer**. This command will open Pro/Engineer window as shown in Figure A.1. The Pro/Engineer window is composed of different areas. The drawing area shows part geometry and finite elements as well as results of simulation. Once a part is created, the Navigation window will show part tree. Under the main menu, various toolbars are shown, which collect frequently used commands. Since no part is created, most commands are inactive. Usable commands will be activated depending on the current status of the model. In addition, locations of toolbars can be moved.

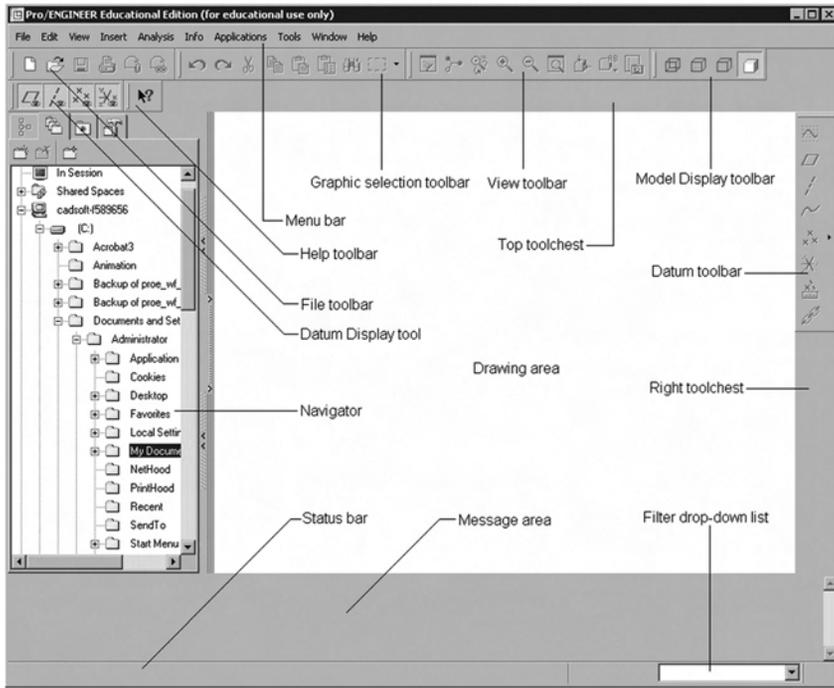


Figure A.1 Layout of Pro/Engineer main window

In order to finish Pro/Engineer, choose **File > Exit** command. Then, **Confirmation** window will show up. Choose **Yes** to finish the program.

Pro/Engineer has its full documents in help menu. In the main menu, choose **Help > Help Center**. Then Pro/Engineer Help windows will show up. In this documentation, the user can find all manuals and tutorials available in Pro/Engineer.

A.3 PLATE WITH A HOLE ANALYSIS

A. Introduction

This example problem illustrates the use of Pro/Engineer for a simple static analysis. You will learn how to build a solid model of a plate with a hole, to create finite elements, to perform the analysis, and to examine the results using Pro/Engineer.

Figure A.2 shows a plate with a hole. The plate is fixed on the left edge and a uniform pressure is applied on the right edge. The magnitude of pressure is chosen such that the total applied force is 300 lb. The purpose of finite element analysis is to estimate the stress concentration along the hole. Even if the model is three-dimensional solid, we will use plane stress elements because stress along the thickness direction is negligible.

B. Creating the Part

In this section, a simple plate with a hole is constructed using Pro/Engineer solid model. This model will be used for finite element analysis.

1. Start Pro/Engineer and change working folder

File > Set Working Directory

Select or create a directory in which a part will be created. Note Pro/Engineer use the term “directory” which is identical to “folder” in windows system.

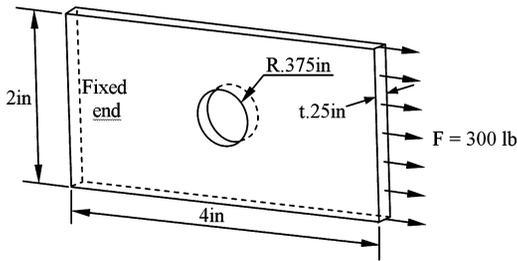
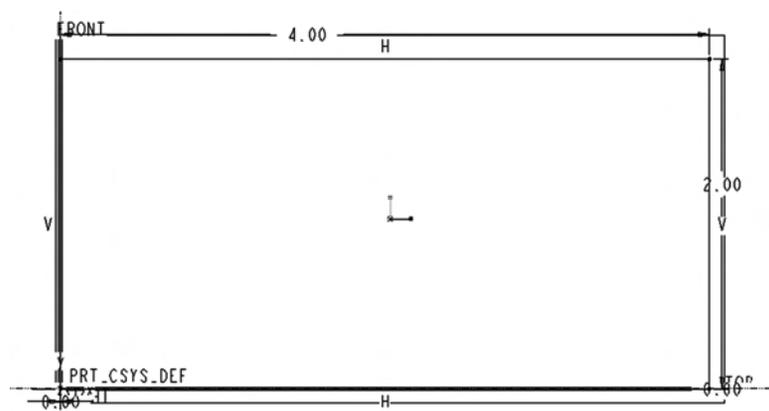


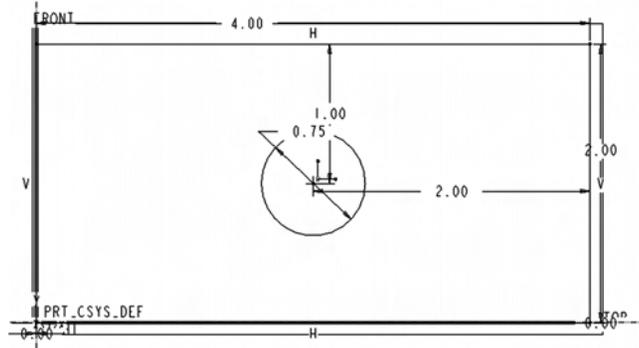
Figure A.2 Plate with a hole model

2. Create a new part
File > New > Part > part name "PlaneStress"
3. Set up a unit to inch (pound f)
Edit > Set up > Unit > Inch Pound Second (IPS) > Set > OK (Changing model unit) > Close (Unit Manager) > Done (Menu manager)
4. Specify part creation method
Insert > Extrude > Placement (in the message area) > Define
 We will sketch the section of the part and extrude it to make a solid. This is the most common way of creating a part. Sketch window will appear
5. Select Sketch Plane
 In the drawing area, three datum planes and a coordinate system are shown. Select the **FRONT** plane and click **Sketch** button in the Sketch window.
 The viewpoint is changed such that the **FRONT** plane is the sketch plane. The sketch menu appears on the right side of the main window.
6. Create 4" × 2" rectangular section
 Select **Sketch > Rectangle** and pick the origin of the coordinate system and pick another point in the drawing area to create a rectangle. The size of the rectangle is not important yet. Click the **middle mouse button** finish creating the rectangle. The same action can be done by selecting rectangle symbol in the toolbar on the right side of the drawing area. Now the size of the rectangle will be shown as dimensions.
 Double click each dimension text and change their values so that the size of rectangle be 4" × 2" as shown in the figure below. You can move the dimension texts by dragging them in the drawing area.



7. Create $\varnothing 0.75$ circle at the center of the rectangle.

Select **circle icon** on the right side toolbar. Select the center location close to the center of the rectangle and click another point to define the circle. Click the **middle mouse button** to finish creating the circle. Change dimensions in the circle so that the location is in the center of the rectangle and its diameter is 0.75 in. Now the section should look like the figure below.

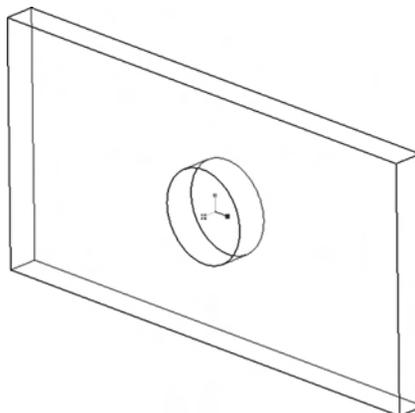
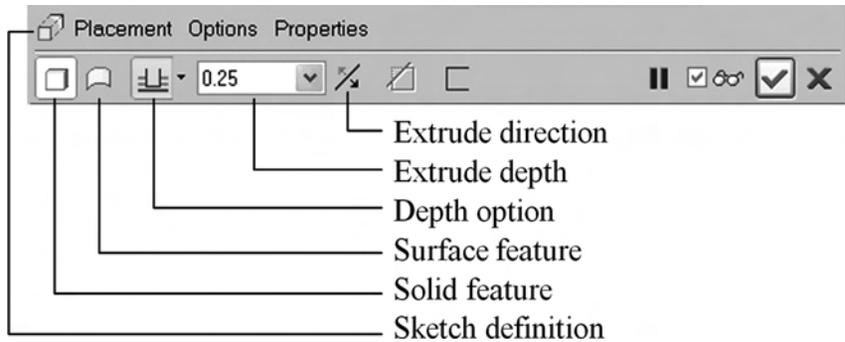


8. Finish creating the section

Click the **checkmark** (continue with the current section) on the right toolbar to finish creating the section. The drawing toolbars on the right will disappear.

9. Extrude the section to create a solid

In the message window (below the **placement** button), type in 0.25 for the depth value; click extrude direction so that the part is extruded in backward; and click **checkmark** to extrude the section by 0.25 in.



10. Save the solid model and exit

File > Save > file name “PLANESTRESS.PRT”

File > Exit > Yes

Check whether file “planestress.prt.1” in your working folder exists.

C. Finite Element Analysis (Integrated Mode)

In the integrated mode, the finite element analysis is performed within Pro/Engineer environment. In the integrate mode, the Mechanics is considered as a sub-menu system of Pro/Engineer.

1. Start Pro/Engineer, change the working directory, and open file “planestress.prt”

File > Set Working Directory

File > Open > planestress.prt

2. Start Mechanics Application

Application > Mechanics > Continue > Structure > OK

3. Idealization

Insert > Midsurface > New > Constant > Select top surface > Hold control-key and select bottom surface > OK > Done/Return

Finite element model does not have to be exactly the same as the solid model, as long as its mathematical behavior is same. Idealization simplifies the solid model to a plate because displacement and stress are constant through the thickness. In order to see the created mid-surface,

Insert > Midsurface > Compress > Shell & Solid > Done > Done/Return > Done/Return

We are creating plane stress element with the thickness the same as the solid

4. Displacement boundary condition

Insert > Displacement constraints

In the Constraint window, type in **displBC** for Name, choose **Edge(s)/Curve(s)** for Reference and select **Edge (1)** in the figure below. Select **OK** to close the Constraint window.

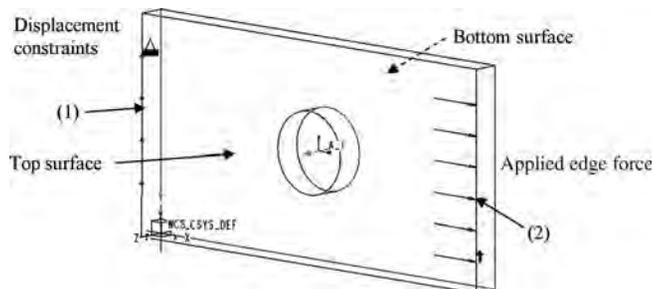
ConstraintSet1 is the name of default set that can include multiple displacement BCs.

5. Applied load

Insert > Force/Moment Load

In the Force/Moment Load window, type in **Traction** for Name, choose **Edge(s)/Curve(s)** for Reference and select **Edge (2)** in the figure below. Type in **X = 300** in force column. Select **OK** to close the Force/Moment Load window.

LoadSet1 is the name of default set that can include multiple loads.



6. Material Data

Properties > Materials

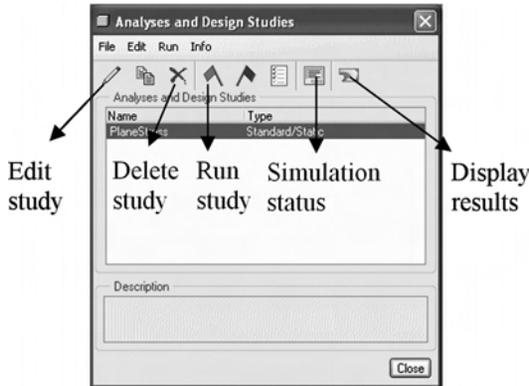
In Material window, choose **STEEL** and click **triple arrow** to add the material to the model. Click **OK** to close the Material window.

Assign the material to the part by clicking **Properties > Material Assignment > OK**. Since PlaneStress is only part and STEEL is only material, we accept defaults.

7. Create Analyses

Analyses > Mechanical Analyses/Studies...

In **Analyses and Design Studies** window, select File > **New Static**



In **Static Analysis Definition** window, type in “**PlaneStress**” for Name, highlight “**ConstraintSet1**” and “**LoadSet1**”. Select Multi-pass adaptive with Polynomial Order, Minimum = 1, Maximum = 9, and Percent convergence = 5% → **OK** → Close

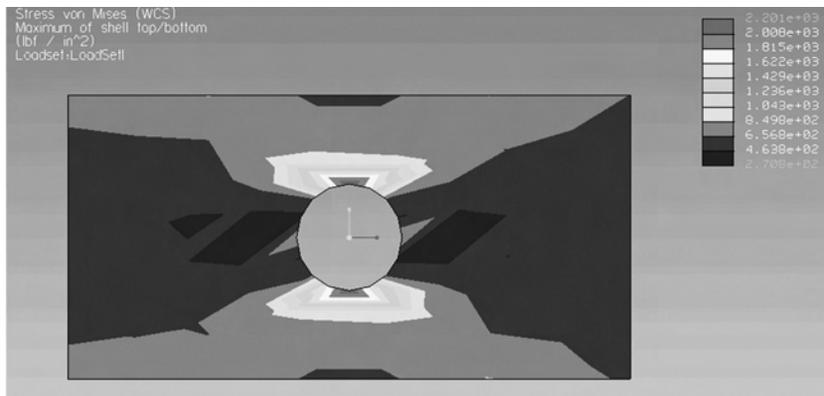
In pro/Mechanica, the order of interpolation is increased progressively until the solution converges.

8. Run the simulation

In **Analyses and Design Studies** window, select “PlaneStress” in the box and click **Run study** icon. The finite element analysis will start in background. The status of analysis can be found by clicking **Simulation status** icon.

9. Post-processing (Stress contour plot)

Once the analysis is finished, the analysis results can be plotted by clicking **Display results** icon. **Result Window Definition** windows will appear. The default values in the window plot the von Mises stress contour. Click **OK and Show** to plot with default values.

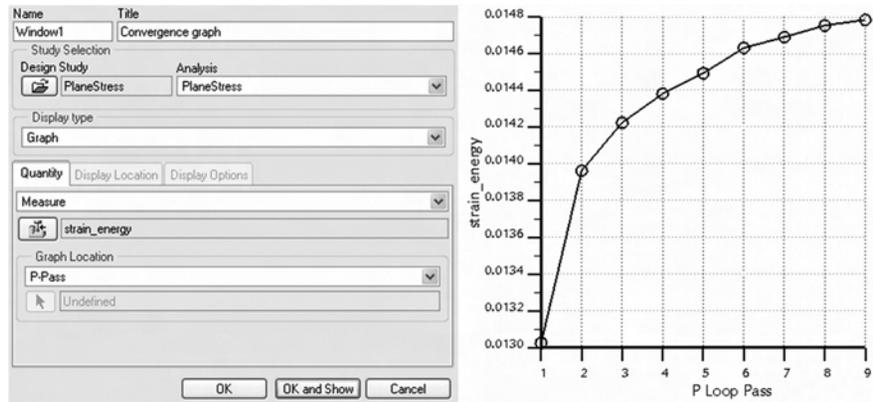


10. Post-processing (Convergence graph)

In addition to the stress contour plot, it is possible to plot a convergence graph of strain energy. Before creating the graph, close the stress contour window by choosing **File > Exit Results > No**

In the **Analysis and Design Studies** window, choose **Display results** icon again. **Result Window Definition** window will appear. Select **Graph** for Display type and **Measure** for Quantity. Click the **Measure** icon below Measure, and select **Strain energy** when the Measure window appears. Select **OK** to close the Measure window and **OK and Show** to display the graph.

The graph shows that the strain energy of the model converges to 0.0148 as higher-order polynomials are used for elements.



Close the convergence graph by selecting **File > Exit Results > No**

A.4 DESIGN SENSITIVITY ANALYSIS/ PARAMETER STUDY

One of the powerful capabilities in finite element analysis is to easily change the design parameters and to perform what-if study without building the actual part. The purpose of this section is to identify how the performance of the model changes according to the design parameters.

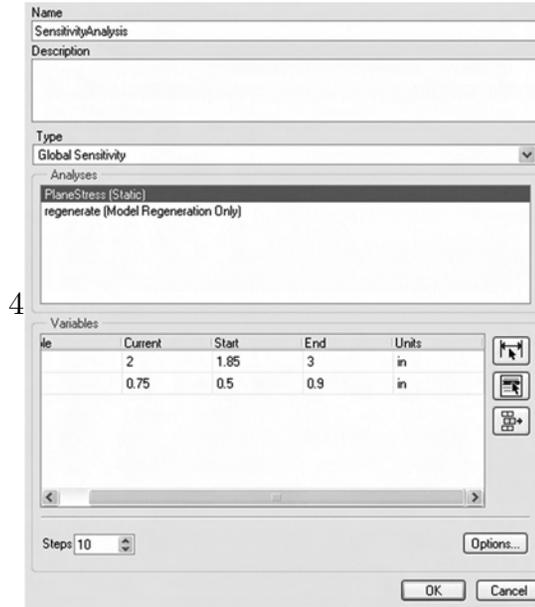
1. Design Sensitivity Study Definition

Select **Analysis > Mechanics Analysis/Studies** from the main menu. In the **Analysis and Design Studies** window, select **File > New Sensitivity Design Study**. Sensitivity Study Definition windows will appear. Type in Sensitivity Analysis for Name and select **PlaneStress** in Analysis section.

2. Design Variables

In the same window, click **Select dimension from model** icon and select the part. Then, dimensions of the part will appear. Select height dimension of the plate. Change the name of the variable to “Height”. Change the start and end value of the height dimension to 1.85 and 3.0, respectively.

Select the diameter dimension using the same way and provide 0.5 for start and 0.9 for end. Change the name of the variable to “Diameter”. Click **OK** to close Sensitivity Study Definition window.



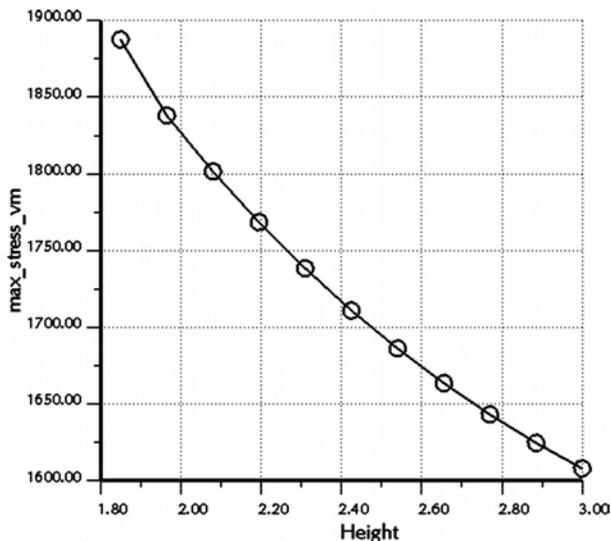
3. Run sensitivity analysis

In Analysis and Design Studies window, click **Start Run** icon to run sensitivity analysis.

4. Review sensitivity results

When analysis is finished, choose **Review Results** icon in the Analysis and Design Study window.

In the Result Window Definition window, type in “Sensitivity w.r.t. Height” in Title. Note that SensitivityAnalysis and PlaneStress are chosen by default in Design Study section. Click Measure icon below Measure section and choose max_stress_vm in the Measure window. Click **OK** to close it. Select Height design variable (or d1) in the Graph Location section and click **OK and Show** button to plot the parameter study results.



Note that the von Mises stress decreases as the height of the plate increases. A similar graph can be plotted for the diameter design variable.

A.5 DESIGN OPTIMIZATION

Design optimization is automatic design process to find what would be the best design given limitations and conditions. First, we have to define what the goal (measure) is and what the constraints (measure) are. Also, the design parameters need to have the lower and upper bounds that the engineer wants to change. All designs that are covered by the lower and upper bounds are called the design domain. Then, the optimization program sequentially changes the design parameters and finds the design that can minimize (or maximize) the goal while satisfying all constraints within the design domain.

1. Optimization Study Definition

Select **Analysis > Mechanics Analysis/Studies** from the main menu. In the Analysis and Design Studies window, select **File > New Optimization Design Study**. Optimization Study Definition windows will appear. Type in “Optimization” for Name.

2. Define optimization problem

Note that minimize total_mass is chosen for the goal by default.

We want to limit the maximum von Mises stress less than 2500 lbf/in². In order to do that, click **Measure** icon on the right side of Design Limits section, and choose max_stress_vm in the Measure window. Type in “2500” in Value column so that the design limit looks like max_stress_vm < 2500.

By following the same procedure in the sensitivity study, define two design variables (height and diameter). For diameter, use minimum (0.5), initial (0.75), and maximum (0.9). For height, use minimum (1.85), initial (2), and maximum (3).

The screenshot shows the 'Optimization Study Definition' dialog box. The 'Name' field is 'Optimization'. The 'Type' is 'Optimization'. The 'Goal' is 'Minimize' with the measure 'total_mass'. The 'Design Limits' section contains one limit: 'max_stress...' with a value of '2500' and units 'lbf / in^2'. The 'Analysis' is 'PlaneStress' and the 'Loadset' is 'LoadSet1'. The 'Variables' section contains two variables: 'Diameter' with current value 0.75, minimum 0.5, initial 0.75, maximum 0.9, and units 'in'; and 'Height' with current value 2, minimum 1.85, initial 2, maximum 3, and units 'in'. The 'Options...' button is visible at the bottom right, along with 'OK' and 'Cancel' buttons.

Measure	Value	Units
max_stress...	< 2500	lbf / in ²

Variable	Current	Minimum	Initial	Maximum	Units
Diameter	0.75	0.5	0.75	0.9	in
Height	2	1.85	2	3	in

3. Launch the optimization program

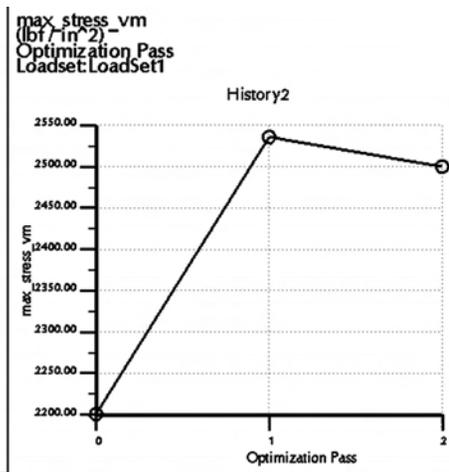
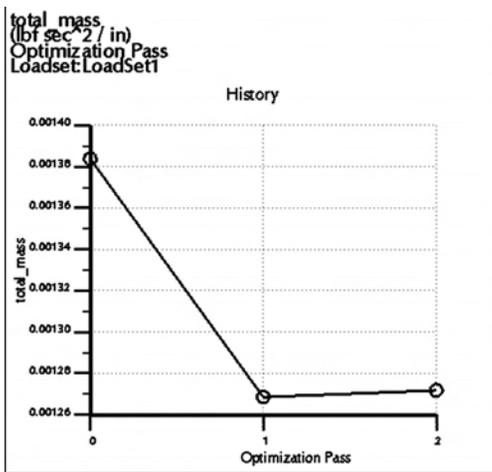
In Analysis and Design Studies window, click **Start Run** icon to run sensitivity analysis.

4. Plot the design history

When analysis is finished, choose **Review Results** icon in the Analysis and Design Study window.

In Result Window Definition window, type in “Optimization” for Name. Note that Optimization and PlaneStress are already chosen for Design Study and Analysis, respectively. Select **Graph** in the Display type section and **Measure** in the Quantity section. Using the Measure icon below Measure, choose total_mass as the quantity of plot. Click **OK and Show** to plot the history of the total mass.

In order to insert maximum stress history, choose **Insert > Result window** from the main menu. Result Window Definition window will appear. Type in “History2” for Name and choose Optimization in Design Study section. Note that PlaneStress will be automatically added in Analysis because it is the only analysis in Optimization. Choose **Graph** in the Display type section and **Measure** in the Quantity section. Using the Measure icon below Measure, choose max_stress_vm as the quantity of plot. Click **OK and Show** to plot the history of the total mass.



In the history figure, the goal (total mass) is reduced significantly at the first design cycle. However, as can be found in the stress history, the new design slightly violates the stress constraints (it is greater than 2500 psi!). Thus, the optimization program slightly modifies the designs twice so that the total mass cannot be reduce further without violating the stress constraint, which is by definition an optimum design.