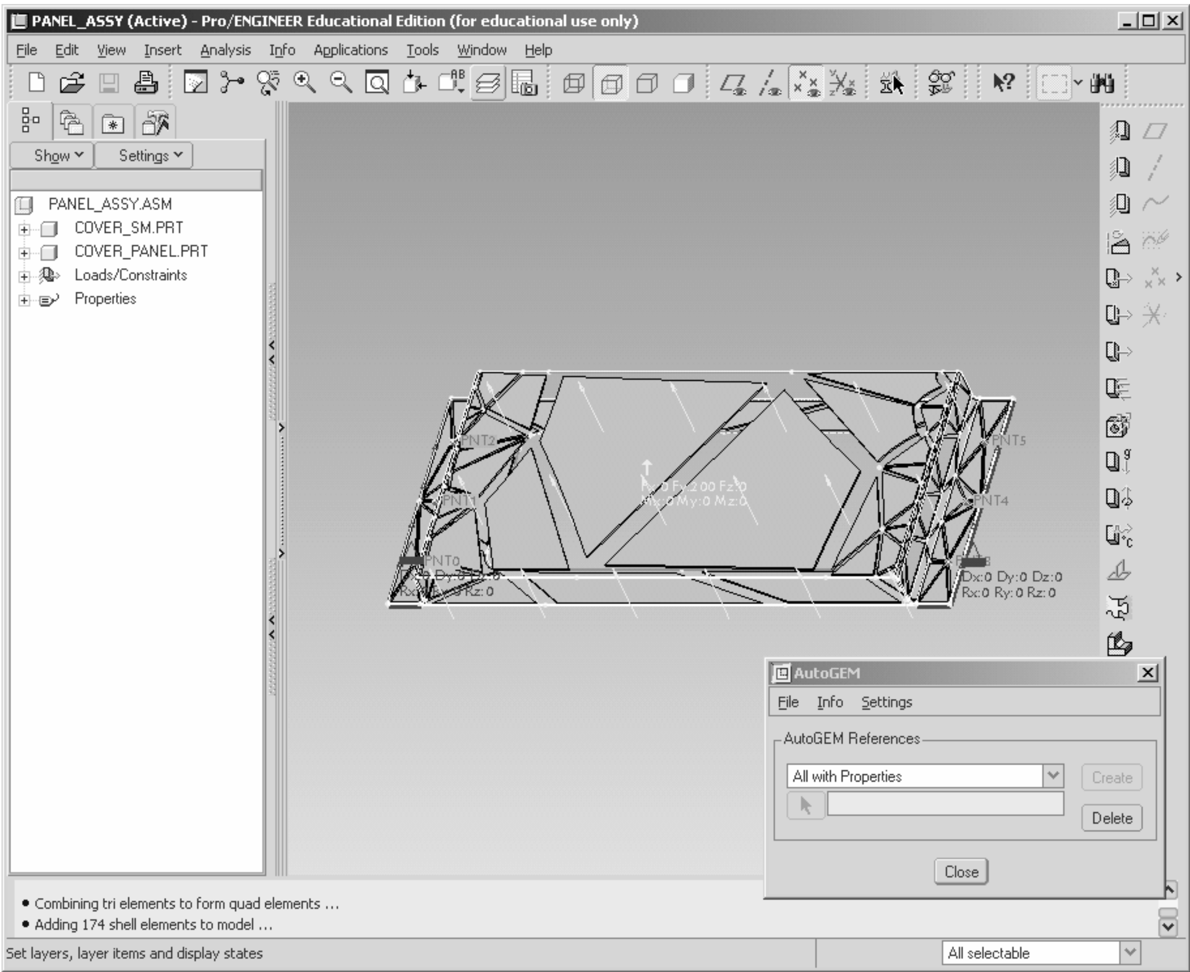
PRO/MECHANICA WILDFIRE 2.0
ELEMENTS AND APPLICATIONS SERIES

PART 2 – STUDIES & CONNECTIONS



Pro/MECHANICA Mesh of a Sheet Metal Weldment

Yves Gagnon, M.A.Sc.
Professor
Mechanical Engineering Technology
Okanagan University College

SDC PUBLICATIONS
Schroff Development Corporation

---

www.schroff.com
www.schroff-europe.com

Estimated time: 2 hours

Exercise 1

Sensitivity and Optimization Studies

Introduction

The purpose of the optimisation study is to help the designer in optimizing predetermined design parameters as a function of known measures, such as Von Mises stress or maximum displacement for a specific goal (i.e. minimise total mass). Your knowledge of stress analysis comes in handy here, as the software cannot do all the work in the optimisation process. There are a few more tasks involved than just clicking the "optimize my model" button. Consequently, the designer has to take the optimization from the start to a certain stage, and then let the software (Pro/MECHANICA®) take over for the final optimization steps.

An optimization study is divided into two main phases:

- **Design phase I** uses a static analysis combined with *local sensitivity* and *global sensitivity* studies in Pro/MECHANICA®. In this phase, the finite element analyst uses Pro/ENGINEER® WILDFIRE to set design parameters. A design parameter is used by the Pro/MECHANICA® studies as a variable for measures. The measures also have to be determined by the analyst.

  1. The main objective of a local sensitivity study is to look at changes of the measures (most likely Von Mises stress) for small variations of each design parameter (± 1%) independently.
  2. The objective of the global sensitivity study is to look at the variations of all parameters, within their respective range, into each step of the process as defined by the user.

- **Design phase II** completes the optimization of the part according to your design objectives (goals). All parameters are optimised concurrently. A goal could be to minimise the total mass or the total costs of a model. There are also other goals that can be defined in the *optimization study*.

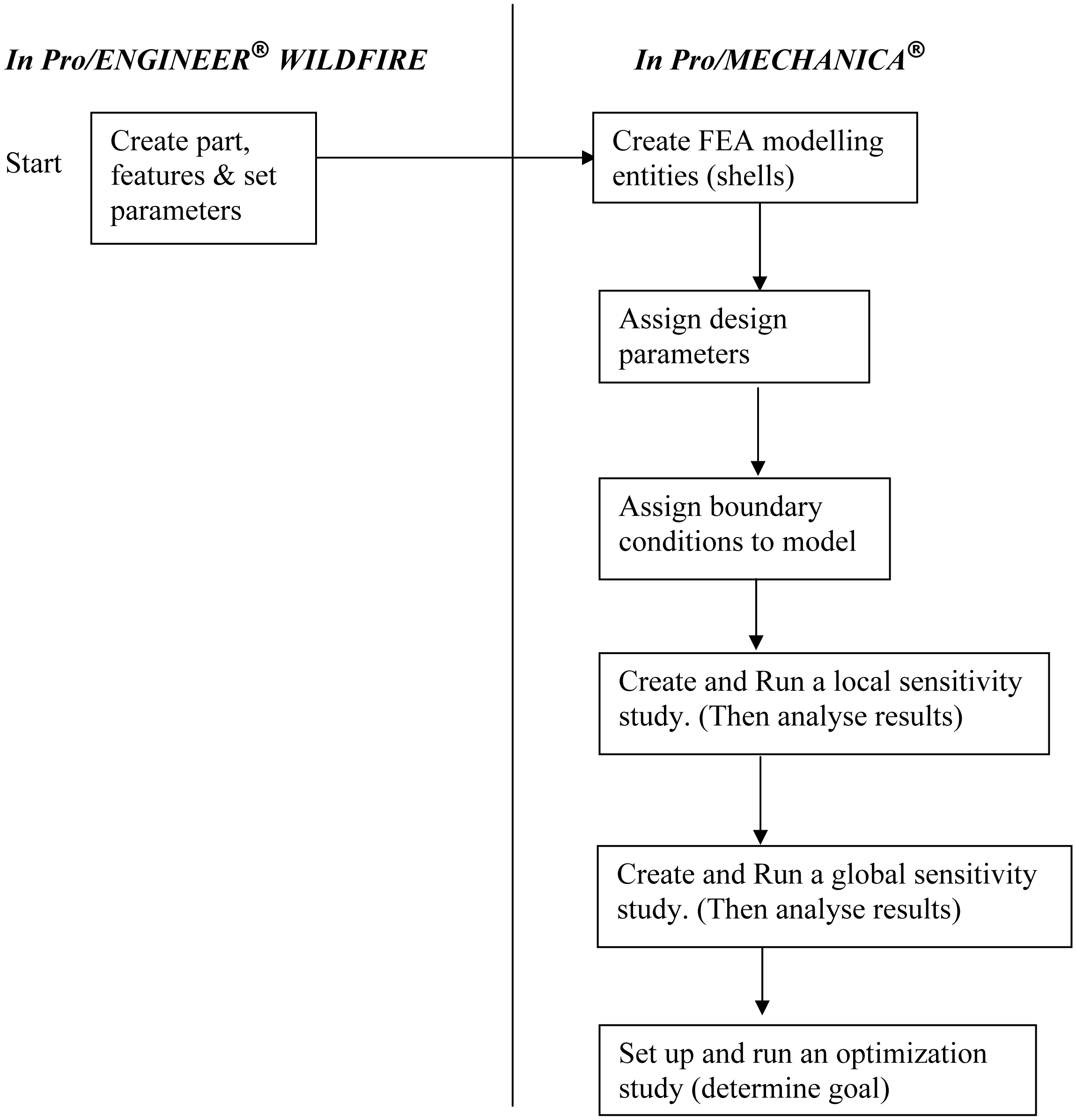


Figure 1: Overview of Design Optimization Study

As you can appreciate, an optimization study is a lot of work. So let's get right to it.

Overview

This exercise will demonstrate the know-how and uses of sensitivity, global and optimization studies in Pro/MECHANICA®. A simple plate with two circular notches will be used for the model. A shell mid-plane compression idealisation will be used to model the plate for FEA and a combination of triangular and quadrilateral shell elements will be used for the mesh. The procedure is as follows:

1. Create (Open) the part in Pro/ENGINEER® WILDFIRE and create the shell mid-plane compression idealisation in Pro/MECHANICA®;
2. Create the mesh and boundary conditions (loads and constraints);
3. Run an analysis (quick check) to make sure that the model converges to a solution; while establishing a preliminary goal, with respect to a measure (Von Mises stress);
4. Set up design parameters for the preliminary design study;
5. Run a local sensitivity study and select parameters that have an effect on the measures pre-determined;
6. Run a global sensitivity study on selected parameters and find the parameter value (maximum or minimum) that has the greatest effect on the measure;
7. Run an optimization study for the above parameters, setting the starting point of the study using the findings of point #6 above. Optimize for your design objective(s). The design objective for this case is to minimise the mass of the plate.

The following figure shows the model that will be used for the study. It is a simple plate with two notched cuts located 6 in. from the left end. The loads are applied at the tip of the right end of the plate, while the left end is constrained.

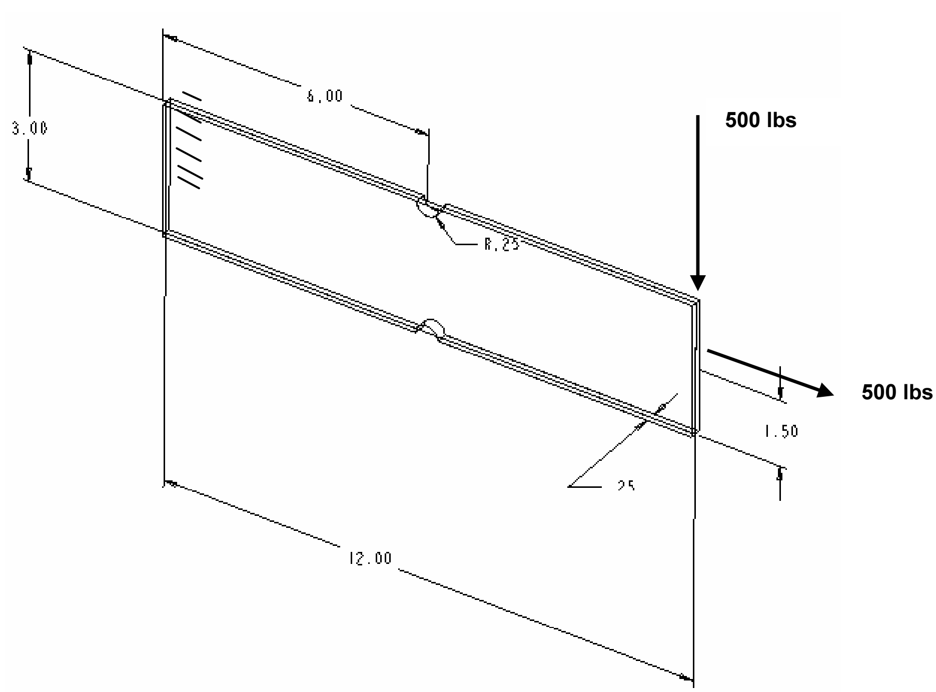


Figure 2: Notched Cantilever Plate

Our design objective is to optimize the location and radius of the notch with respect to the left end of the plate. We will also look at plate thickness and optimize all three parameters above for the total mass, using the Von Mises stresses as the measure.

Procedure

Phase 1: Preliminary Optimization Work

Please refer to the model represented in Figure 2. It is a flat bar made of steel. Even though a simple model, it will illustrate the entire procedure of the optimization study. First, it is important to understand what our objectives are for the optimization study, as it may help you understand better each step involved in the process. Consequently, let's define our objectives and determine the measure to be studied for the selected design parameters. Table 1 gives an overview of the objectives for this study.

TABLE 1: OPTIMIZATION OBJECTIVES, MEASURES AND DESIGN PARAMETERS

Optimization Goal	Measure to be used for optimization	Design Parameters to be optimized		
		Name	**Description**	**Initial value (in.)**
Minimise weight of the part	Von Mises stress	Cut_length	Dimension from left edge to cut	6
		Cut_radius	Notch radius	.25
		thickness	Plate thickness	.25

It is important to keep the above objective, design parameters and measure(s) in mind as we move through the procedures of the optimization study.

Part A: Building the Model and the Design Parameters

Before you start the procedures below, create a directory named 'opt_study' using Microsoft Explorer®. Move the part named notch_flatbar.prt to that directory.

1. Start up Pro/ENGINEER® WILDFIRE

Details on how to do this are different from system to system. For a typical Windows platform, select **Start > Programs > Proewildfire**.
(It takes 30 seconds approximately for Pro/ENGINEER® to start.)

Set up your working directory. **(File > Set Working Directory)**
Select your working directory 'opt_study,' then click on **Accept**.

2. Open the Following Part: notched_flatbar

Once open, the following part should appear on your screen.

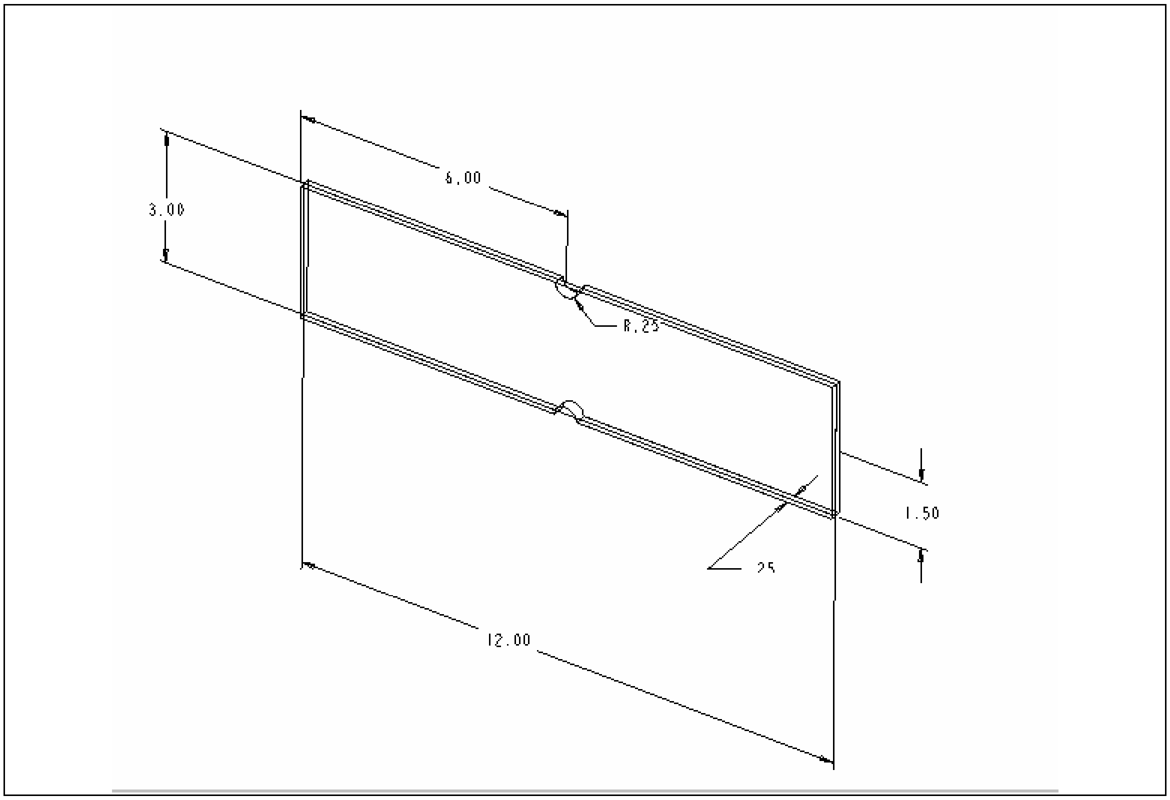


Figure 3: Notch_flatbar.prt (bar dimensions: 3″ X 12″ X 1/4″)

3. Modifying Dimension Cosmetics

Before we go to Pro/MECHANICA®, we will modify the dimension cosmetics in Pro/ENGINEER® WILDFIRE for ease of recognising each optimized design parameter during the process. From the model tree, select NOTCH_CUT and right mouse click on it to select **Edit**. At this time the dimensions for the feature should show up on the screen. From the filter at the bottom right corner of the screen, select **Dimensions**.

Select the **R.25** dimension and right mouse click on it to select **Properties**. From the window that comes up (see Figure 4), change the dimension text name to **cut_radius**.

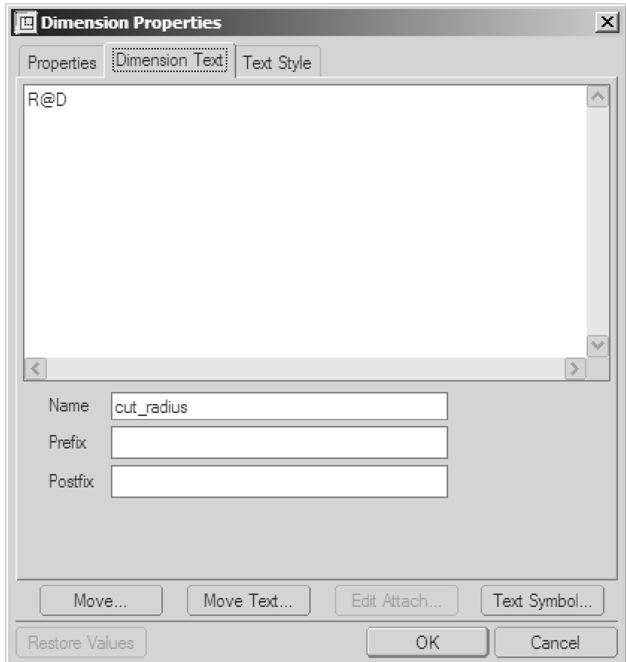


Figure 4

Select **OK**. Repeat same procedure as above for the 6.00 dimension. This time change the dimension text name to **cut_length** as shown below.

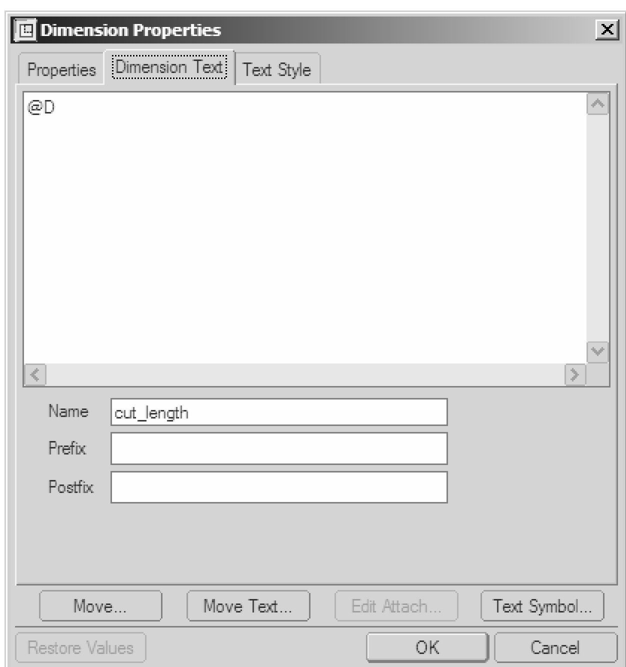


Figure 5

Select **OK**. Edit the feature **FLAT_BAR** by selecting it from the model tree, right click on it and select **Edit**. Change the **.25** dimension text name as shown in Figure 6 to thickness.

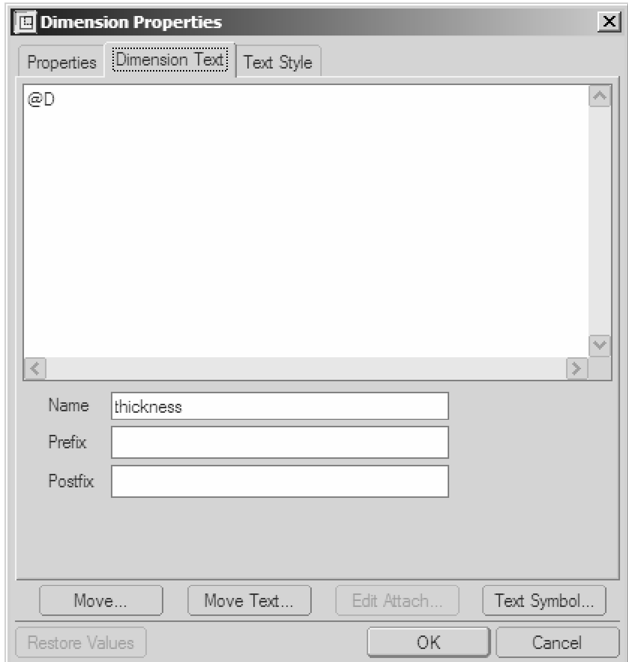


Figure 6

Select **OK**.

This completes the Pro/ENGINEER® WILDFIRE portion of the exercise. Next we go to Pro/MECHANICA®.

4. Creating Shell Mid-surface Compression

Select **Applications > Mechanica > Continue**. Then from the right menu select **Structure > Model > Idealizations > Shells > Midsurface > New**.

Holding down the CTRL key, select one of the plate faces, and then select the other face (then release the CTRL key and middle button mouse click). To make sure that the compression of the shells works, select **Compress > Shells Only > Show Compress**, and you should see the yellow colour contour of the part on the screen as shown in Figure 7.

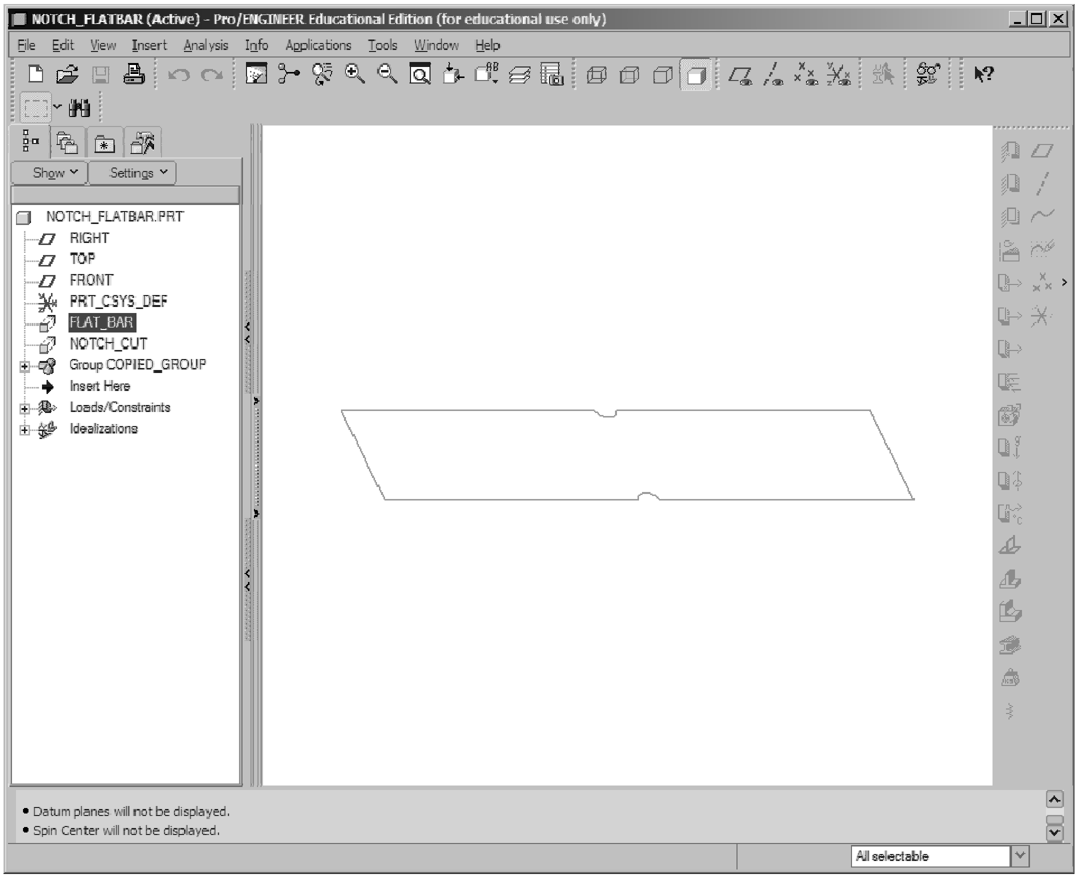


Figure 7: Notched plate shell model

Select **Done-Return** > **Done Return**.

5. Assign Materials

The plate is made of standard ASTM A-36 steel (36 ksi yield strength). Select **Materials** and the following window will come up.

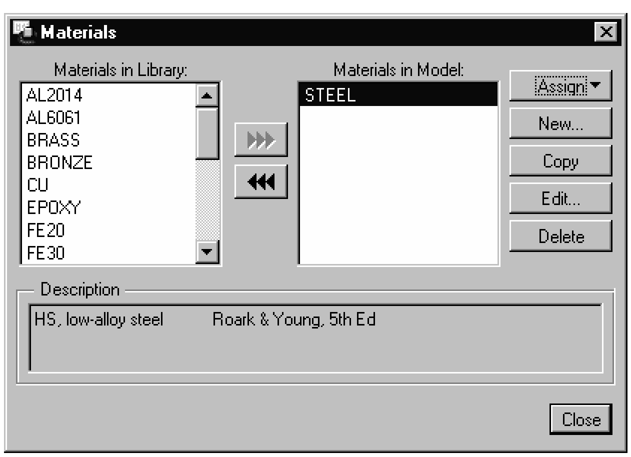


Figure 8

Select **Steel** from the left column and click on the arrow to move in under the *Materials in Model* column. Click on **Assign > Parts** and select the plate. Click on the middle mouse button to accept. Click on **Close**.

6. Meshing of the Part

We will keep the all default shell elements and settings that AutoGEM will create. Select **AutoGEM** from the right menu, the following dialog box comes up on your screen.

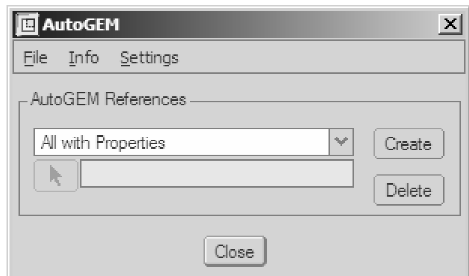


Figure 9

Select C**reate**. The following confirmation of elements box should come up after about five seconds.

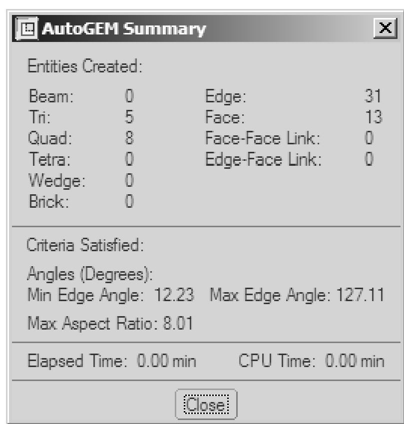


Figure 10

Note that five triangular and eight quadrilateral elements have been created. Select **Close**. Use the display setting to shrink all elements to ten. Your window should then look as follows:

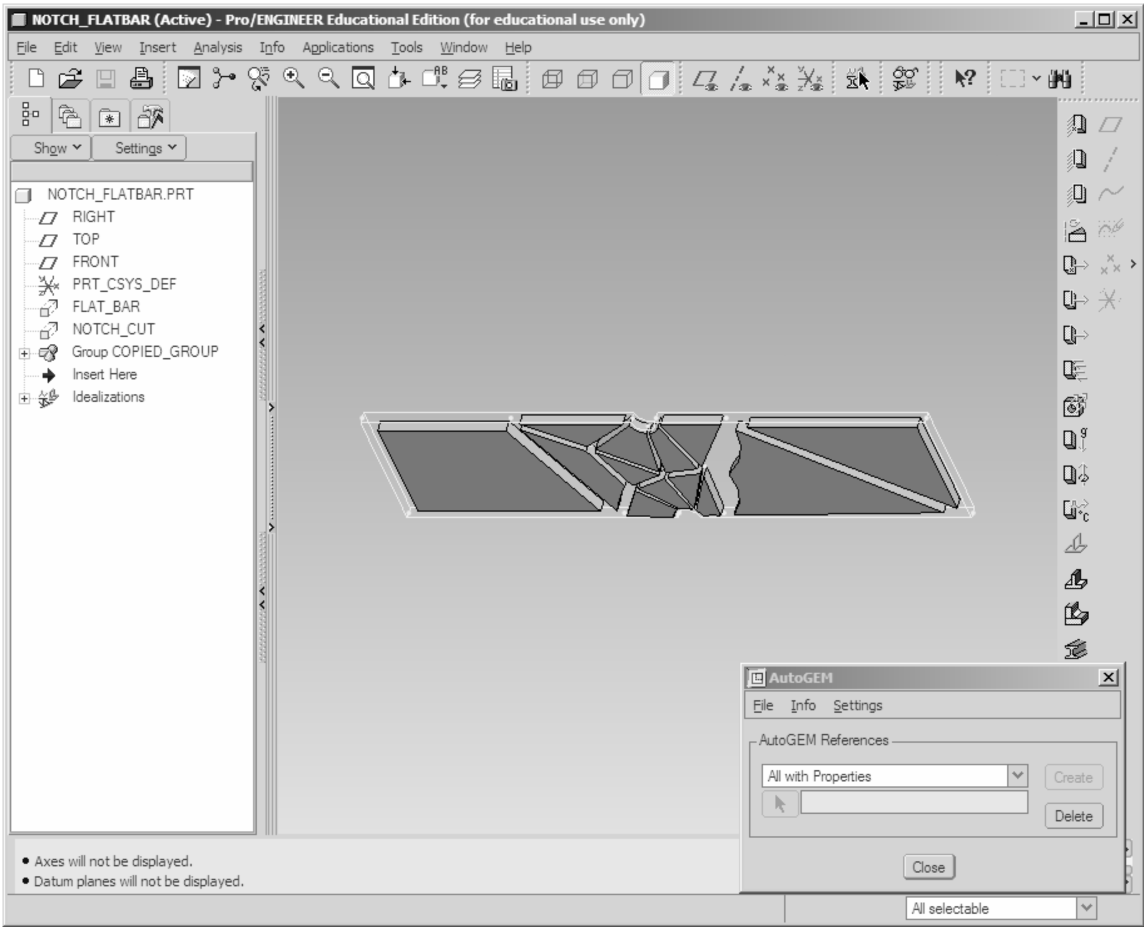
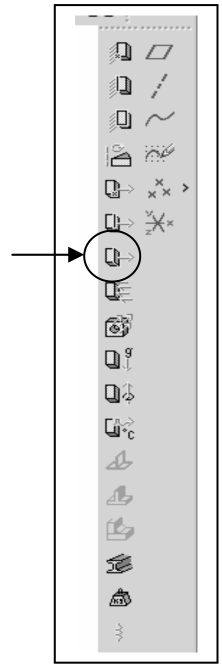


Figure 11: Meshed Notched Plate

The mesh looks adequate for further analysis. Select **Close** and answer **Yes** to saving the mesh.

7. Assignment of Boundary Conditions to the Model

7.1 Loads



The end load has 500 lbs in both the X and Y directions, as shown in an earlier figure. We will set this load as and edge load. Create the following **Curve/Edge** loads and constraints on the part.

Select **Loads > New > Edge-Curve** or click on the corresponding icon. Enter information as seen below.

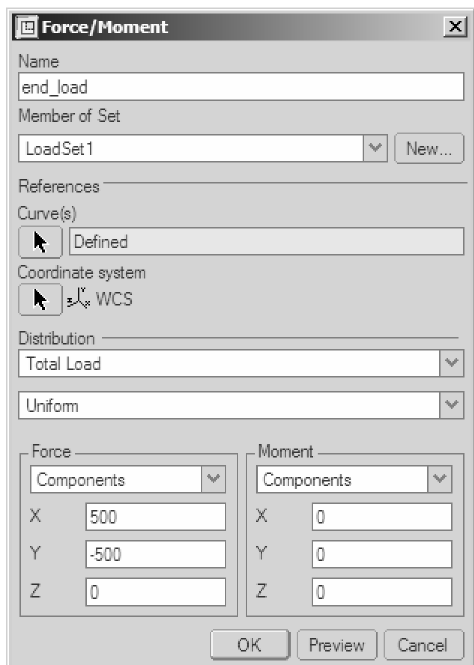
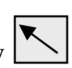


Figure 12: Force/Moment Definition Box

Note: For reference-curves, **click on the arrow**  and select the right vertical edge from the model.

Select **OK**. Set simulation display to **Tails Touching.** The model should now appear as shown in Figure 13.

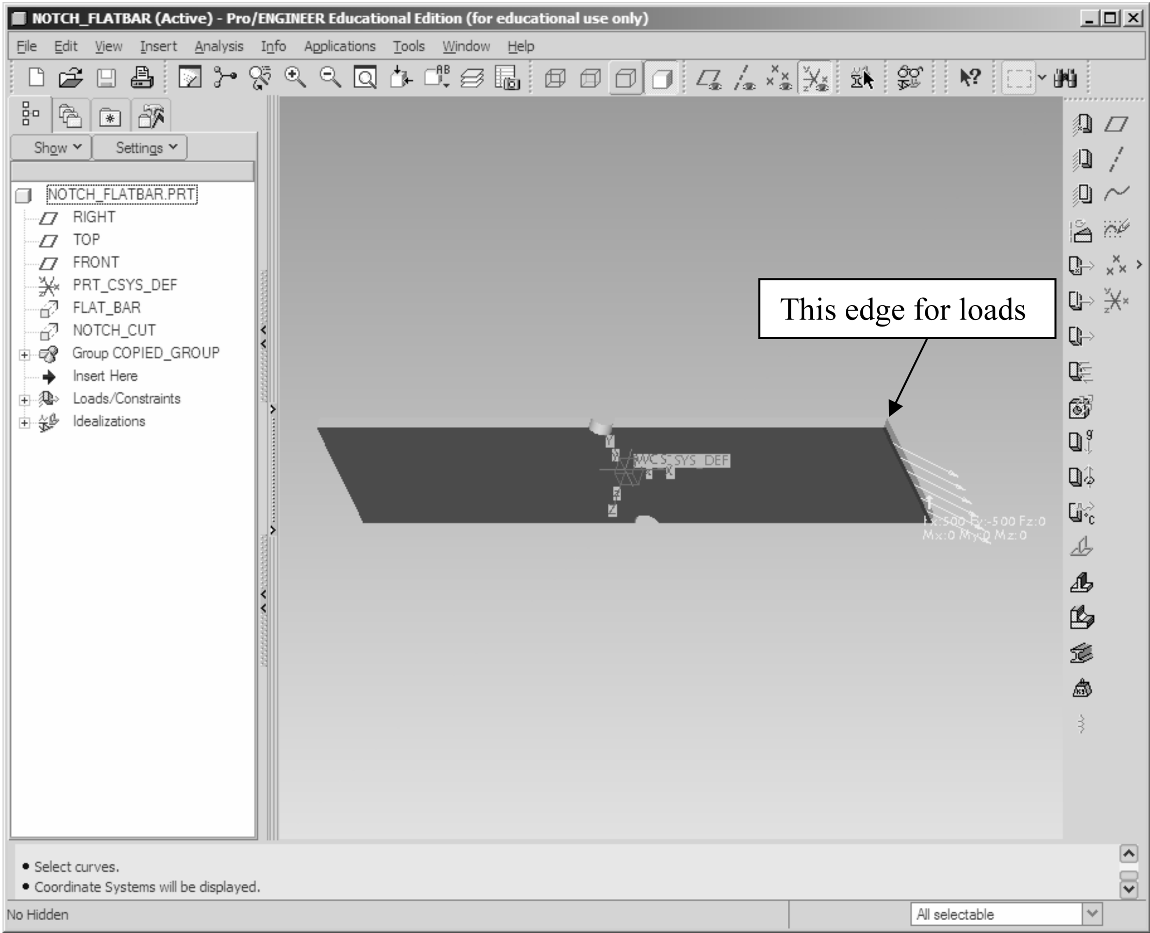
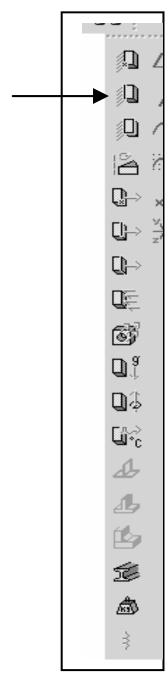


Figure 13: Notched Plate showing end load

7.2 Constraints

The plate is basically a cantilever beam with the left end being fixed. To apply the end constraints, select **Model > Constraints > New > Edge-Curve** or click on the corresponding icon.



Fill in and select the appropriate information from the dialog box below.

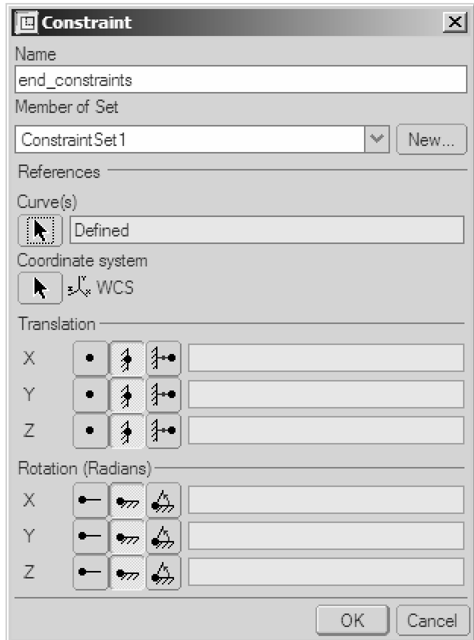
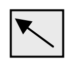


Figure 14: Constraint Definition Box

Note: For reference-curves, **click on the arrow**  and select the left vertical edge from the model.

Keep all DOF fixed (translations and rotations). Select **OK**. The plate FEA model should now look as follows:

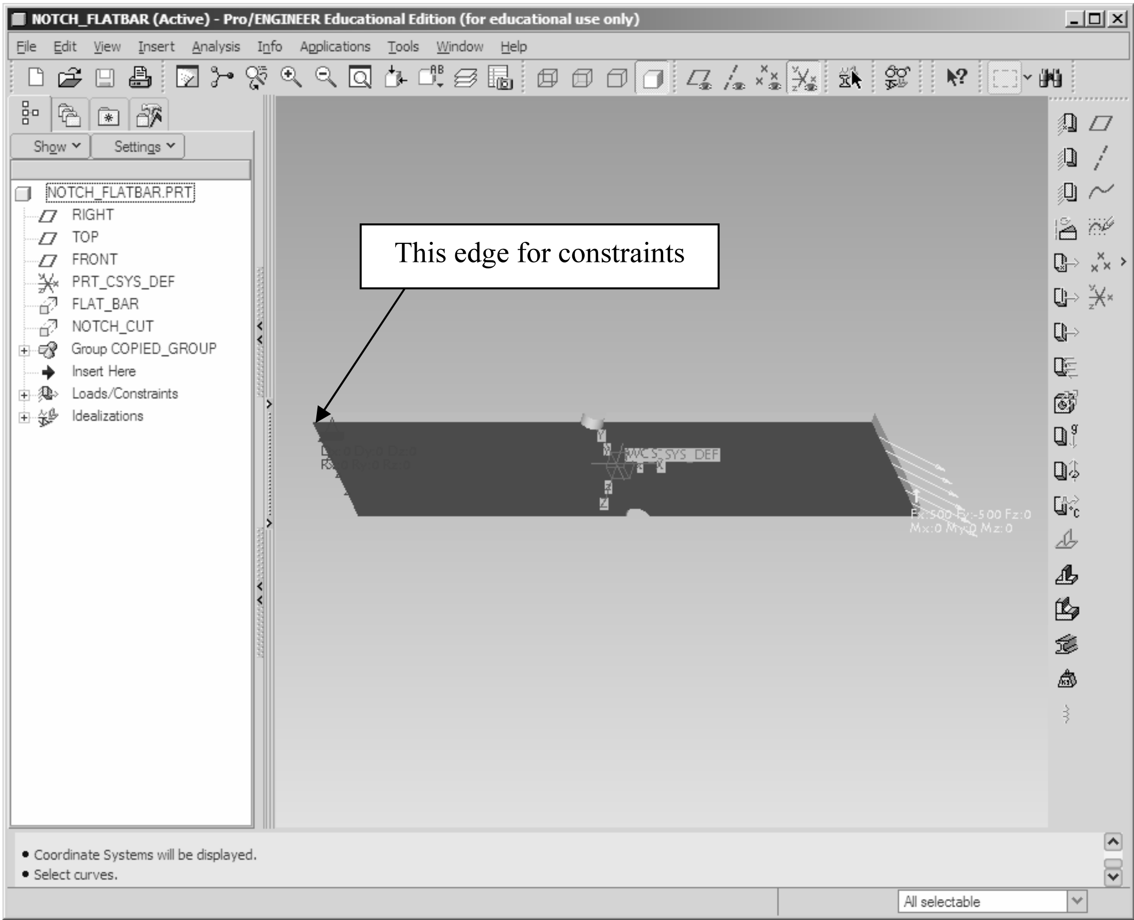


Figure 15: Boundary conditions of the notched plate.

8. Create the Static Analysis

8.1 Quick Check

Create and run a quick check-no convergence analysis named ***static_1*** (see from below) to make sure that you can get to a solution. Select **Analyses/studies** so the corresponding dialog box comes up. Then **File > New Static** and enter information as shown in Figure 16.

Figure 16

Select **OK**. Then select the analysis and right mouse click on it to select **Start**. Answer **Yes** to "**Do you want error detection?**" Right mouse click to **Status** and make sure that you can get to a solution. Once the run is completed, double-check your analysis results with the figure below:

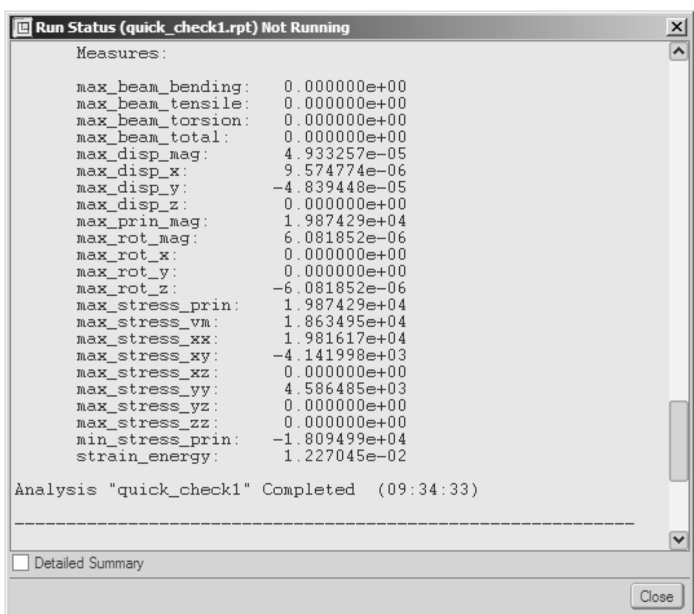


Figure 17: Status summary File (quick check)

Select **Close**.

STOP

*You will know that the model completed the solution by seeing **Run Completed** at the bottom of the summary file*. Results are not important at this point, we just want to make sure that the model can get to a solution.

8.2 Multi-pass Adaptive Convergence

Next edit the quick check analysis by right mouse clicking on it and selecting **Edit**. Enter the information as seen in the box below.

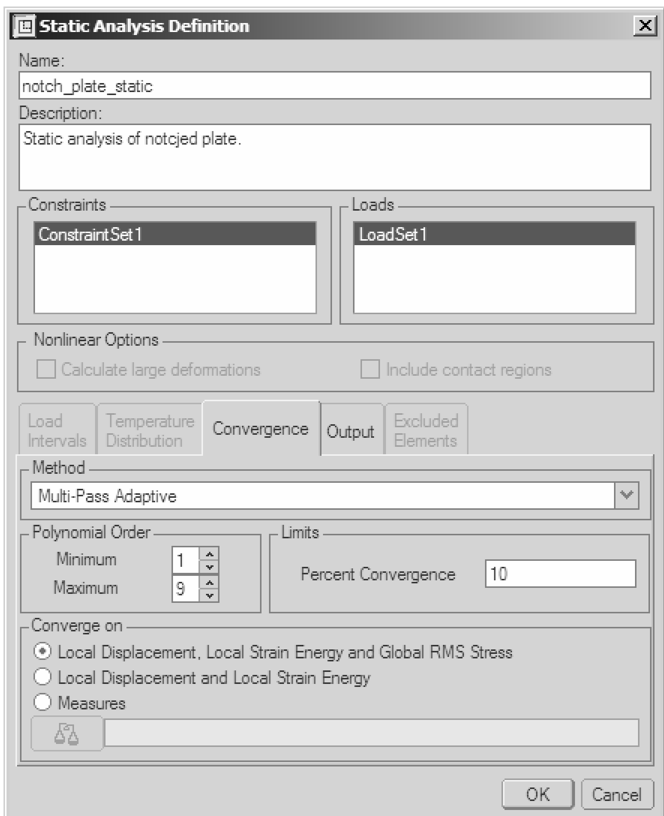


Figure 18: Static Analysis Definition (Multi-Pass Adaptive)

Select **OK** then **Run the analysis.** Answer **Yes** to "**Do you want error detection?**" Select **Status** and make sure that you can get to a solution by showing Run Completed at the bottom of the status window.

The solution converges within 10 percent in three passes. Check your status window with the one shown in Figure 19.

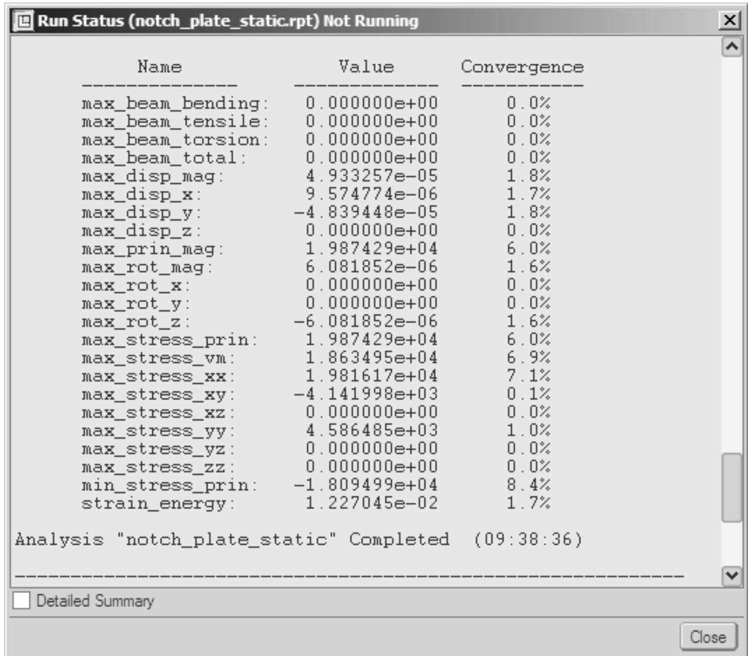


Figure 19: Status Summary File

8.3 Results of Static Analysis

Check the results and record the max Von Mises stress off the summary window. The analysis should converge after six passes and the VM stress results should be around 1.86×10^4 psi Select **Close > Close**. Select **Results** and create the maximum Von Mises stress definition sheet as follows:

Click on this icon named "**Insert a new definition**."

Figure 20

Fill out result definition window as follows:

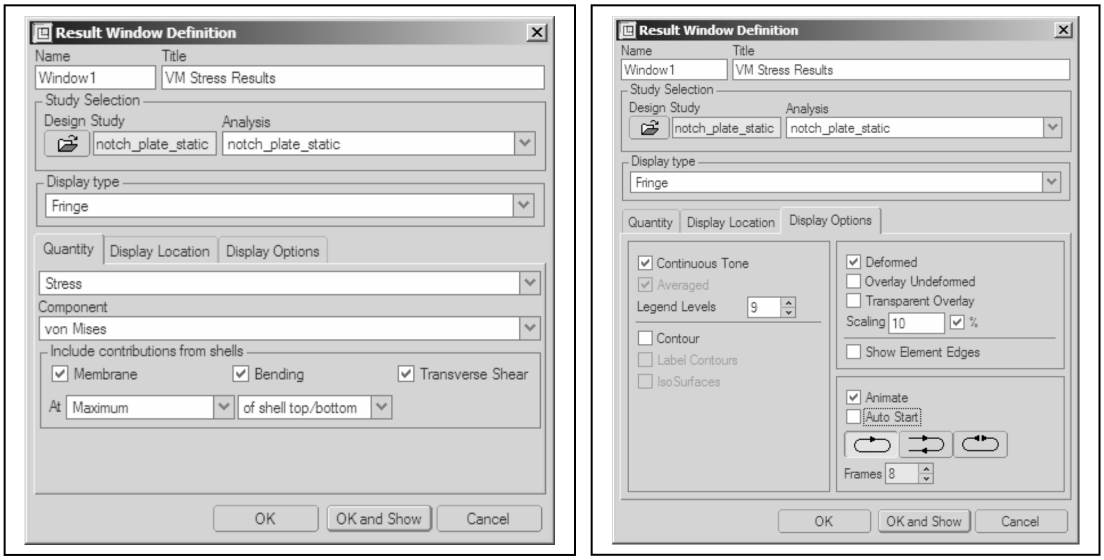


Figure 21: a) Quantity Tab b) Display Options Tab

Select **OK and Show**. Animate the part to make sure that it behaves as we would expect it to, given the current loads and constraints.

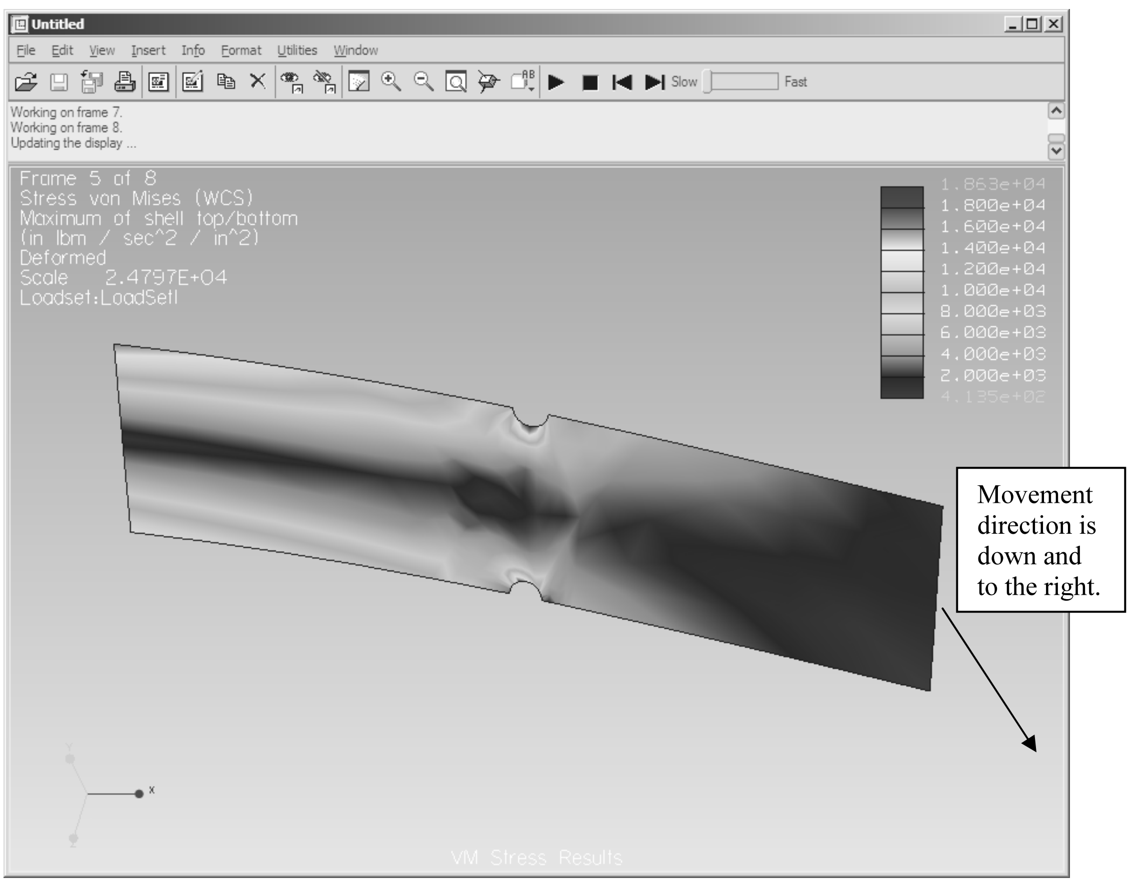


Figure 22: Static Analysis Von Mises Stress results

Once you are satisfied, Select **File > Exit Results > No**.

STOP

Important: We must make sense of the above results before we go to the optimization. Ask yourself the following question:

- Are the maximum VM stresses above the material design stress that we want to use? Say (S_y/1.5) for static? If they are, there is no need really for an optimization study to minimise weight right?

But let's say that we want to use steel with (S_y = 36 ksi).
Then our design stress is 36/1.5 = 24 ksi.

The difference in value of (24 ksi - our results 18.6 = 5.4 ksi) is an important design criteria. If this number is greater than 1 and less than 10 ksi, then this leaves us some room for weight optimization using this material, without wasting our time.

Note that phase 1 of the optimization demands a lot out of the user (from knowledge and experience). The software cannot make the preliminary design decisions for us. For instance, we may want to change the thickness of the plate, say 5/16″ and rerun the analysis to give us some more room to maneuver from an optimization standpoint. For the sake of this example, let's keep the thickness at 1/4″.

Procedure

Part B: Local Sensitivity Study

Recall:

- The objective of a local sensitivity study is to look at small changes of the measures (in our case Von Mises stresses) for small *independent* variations of each design parameters used ($\pm$ 1 to 2%).
- Our design parameters for this study, as defined earlier in Pro/ENGINEER® WILDFIRE are plate thickness, the cut location with respect to the left edge and the cut radius.

1. From the Pro/MECHANICA® menu select **Model > Dsgncontrols > Design Params > Create** and select the type **Dimension** as shown below.

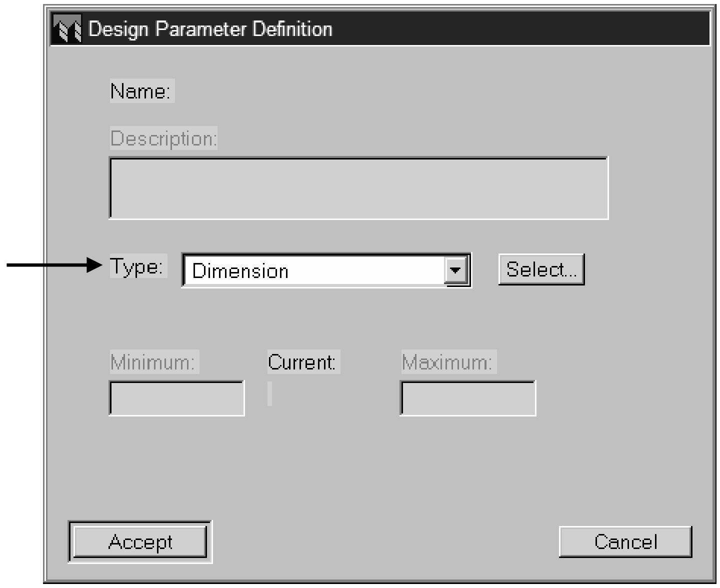


Figure 23

2. Click on **Select** and click on the cut (notch). Select the **.25″** radius of the cut and enter the text shown below. Note the values entered for the minimum and maximum are set at approximately plus and minus a few percent only of the nominal value as *we are only interested at this stage in the effect of the small changes of the parameter on VM stress or how 'sensitive' the parameter is to the measure.*

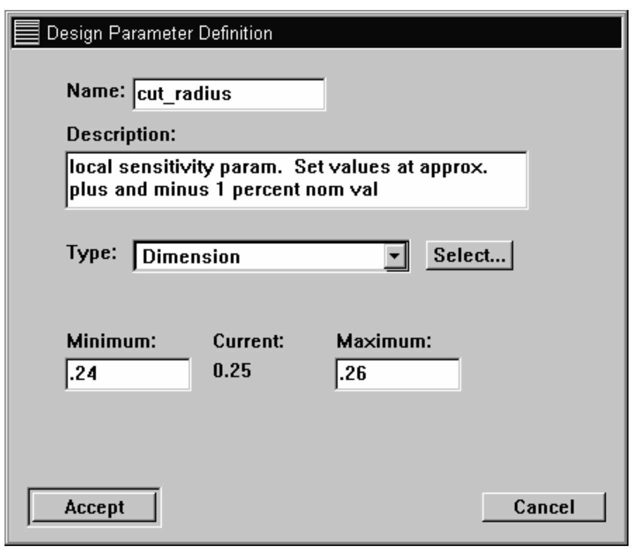


Figure 24: Design Parameter 1 Definition

Click on **Accept**.

3. Repeat same procedure for the cut_length and plate thickness (see below for values to be entered).

- For cut length, select the 6″ dimension and **enter 5.9 and 6.1 for the minimum and maximum values** as shown below.

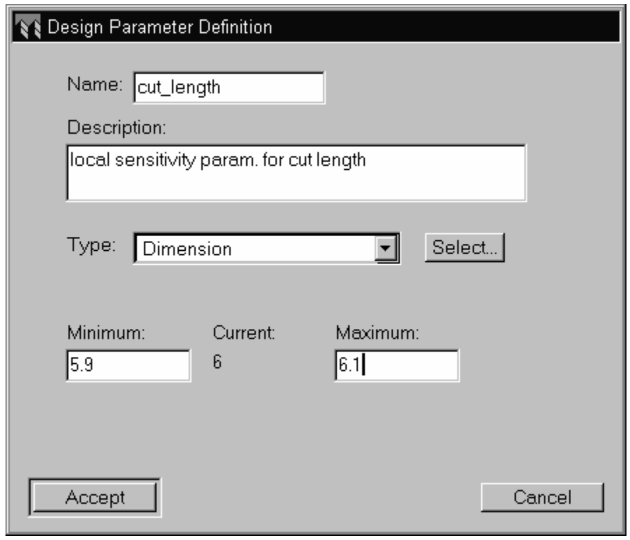


Figure 25: Design Parameter 2 Definition

- For the thickness, select the .25 inch dimension and **enter .24 and .26 for minimum and maximum values** as shown below.

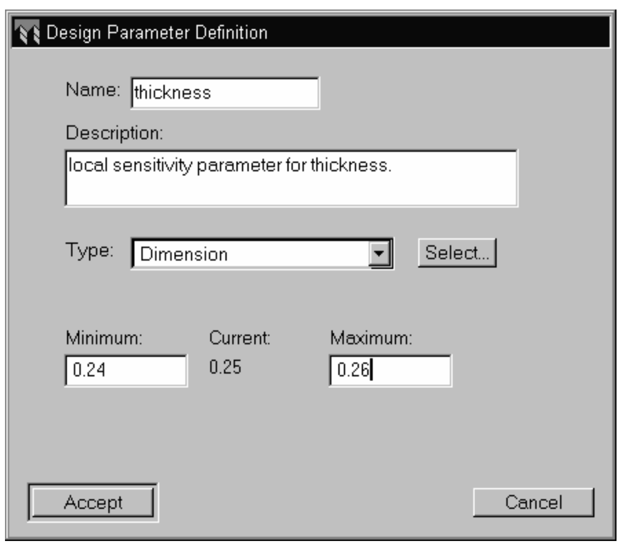


Figure 26: Design Parameter 3 Definition

Click on **Accept** once all parameters have been defined. The design parameters window should look as shown in Figure 27.

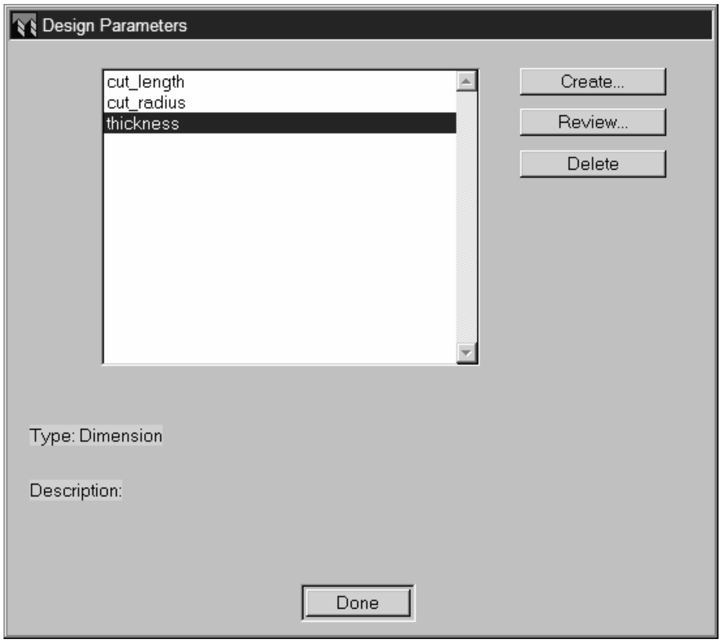


Figure 27

Click on **Done** when completed.

4. To make sure of the effect of our parameters. Play around with the S**hape Animate** under the **dsgncontrols** menu. Select the **Parameter Cut Length** and change the minimum and maximum values to 4 and 8 (as seen below).

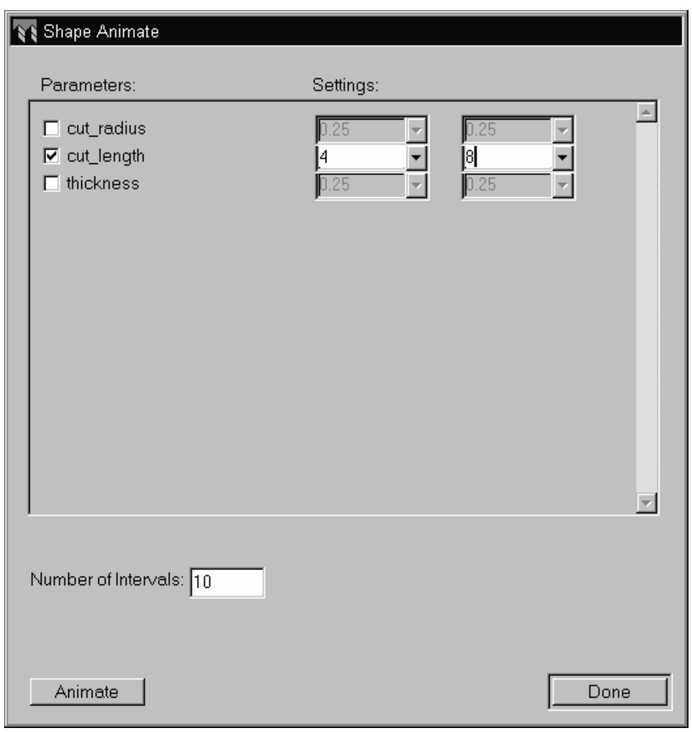


Figure 28: Shape Animation Definition Window

5. Click on **Animate** and answer **Yes** to all the prompts. At the same time, check the movement of the feature on the screen. Repeat procedure for each parameter independently or animate as a group.

6. Once completed with the animation of the parameters, select **Done** from the animate window.

7. Create a design study as shown below. Select **Analyses/Design Studies**. Select **File > New Design Study**. Make the following selections for the study:

   - Select ***local sensitivity study*** from the analysis menu and name it: ***loc_sens_stud1***.
   - Check off all three parameters previously defined.
   - For settings, select the **nominal dimension value** for each of our previously defined parameter in Pro/ENGINEER® WILDFIRE i.e. 6″, .25″ and .25″.
   - ***Select static_1*** under analysis.

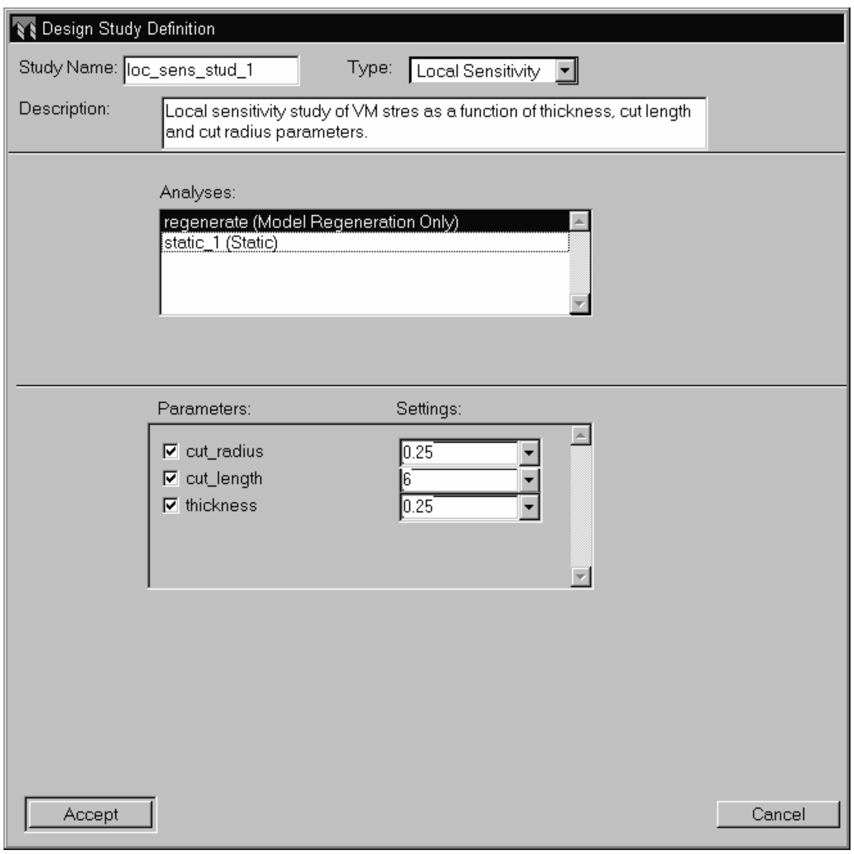


Figure 29: Design Study Definition Window

Select **Accept**.

8. Run the local sensitivity created above. With the design study selected, click on **Run**.

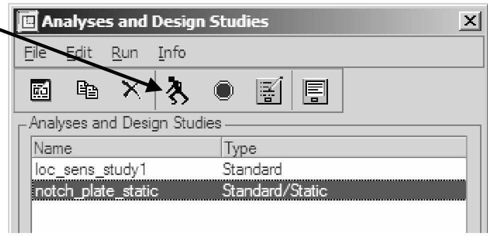


Figure 30

Answer **Yes** to "**Do you want error detection?**" It takes approximately two to three minutes for the analysis to complete. Verify your status window to the one below shown below:

Begin Sensitivity Analysis, Parameter: cut_radius

```
Static Analysis "notch_plate_static":

  Convergence Method: Multiple-Pass Adaptive
  Plotting Grid:     4

      Calculating Element Equations          (19:29:54)
        Total Number of Equations:     492
        Maximum Edge Order:             3
      Solving Equations                     (19:29:58)
      Post-Processing Solution                (19:29:59)
      Calculating Disp and Stress Results     (19:30:04)

      RMS Stress Error Estimates:

      Load Set          Stress Error     % of Max Prin Str
      -------------     ---------------  ------------------------
      LoadSet1          4.27e+03         21.2% of  2.02e+04

      Resource Check                        (19:30:06)
        Elapsed Time    (sec):    482.15
        CPU Time        (sec):      6.95
        Memory Usage     (kb):    167396
        Wrk Dir Dsk Usage (kb):        0

Analysis "notch_plate_static" Completed  (19:30:06)

Derivatives of Measures for Analysis: notch_plate_static
With Respect to Parameter: cut_radius

    total_cost:       0.000000e+00
    total_mass:      -1.132648e-01
    com_x:            2.960725e-08
    com_y:           -5.787104e-09
    com_z:            0.000000e+00
    inertia_xx:      -2.033609e-01
    inertia_xy:       3.912722e-08
    inertia_xz:       0.000000e+00
    inertia_yy:      -3.683823e-03
```

```
    inertia_yz:       0.000000e+00
    inertia_zz:      -2.070448e-01

  Load Set: LoadSet1

    max_beam_bending:  0.000000e+00
    max_beam_tensile:  0.000000e+00
    max_beam_torsion:  0.000000e+00
    max_beam_total:    0.000000e+00
    max_disp_mag:      1.432914e-05
    max_disp_x:        3.515730e-06
    max_disp_y:       -1.391126e-05
    max_disp_z:        0.000000e+00
    max_prin_mag:      2.998970e+04
    max_rot_mag:       2.394071e-06
    max_rot_x:         0.000000e+00
    max_rot_y:         0.000000e+00
    max_rot_z:        -2.394071e-06
    max_stress_prin:   2.998970e+04
    max_stress_vm:     2.950060e+04
    max_stress_xx:     2.971161e+04
    max_stress_xy:    -1.168241e+04
    max_stress_xz:     0.000000e+00
    max_stress_yy:    -2.327032e+02
    max_stress_yz:     0.000000e+00
    max_stress_zz:     0.000000e+00
    min_stress_prin:  -2.324522e+04
    strain_energy:     3.496831e-03
```

Begin Sensitivity Analysis, Parameter: cut_length

```
Static Analysis "notch_plate_static":

  Convergence Method: Multiple-Pass Adaptive
  Plotting Grid:      4

      Calculating Element Equations           (19:54:04)
        Total Number of Equations:     492
        Maximum Edge Order:              3
      Solving Equations                       (19:55:07)
      Post-Processing Solution                (19:55:45)
      Calculating Disp and Stress Results     (19:56:41)

      RMS Stress Error Estimates:

      Load Set        Stress Error       % of Max Prin Str
      -------------   ---------------    ------------------------
      LoadSet1        4.34e+03           21.9% of  1.98e+04

      Resource Check                          (19:58:47)
        Elapsed Time     (sec):    2200.55
        CPU Time         (sec):      10.02
        Memory Usage      (kb):     168484
        Wrk Dir Dsk Usage (kb):          0

Analysis "notch_plate_static" Completed  (19:59:49)

Derivatives of Measures for Analysis: notch_plate_static
With Respect to Parameter: cut_length

    total_cost:       0.000000e+00
```

```
    total_mass:       6.043610e-10
    com_x:           -5.484068e-03
    com_y:           -1.227412e-09
    com_z:            0.000000e+00
    inertia_xx:      -4.448353e-09
    inertia_xy:      -8.720819e-09
    inertia_xz:       0.000000e+00
    inertia_yy:      -1.387973e-04
    inertia_yz:       0.000000e+00
    inertia_zz:      -1.388018e-04

  Load Set: LoadSet1

    max_beam_bending:  0.000000e+00
    max_beam_tensile:  0.000000e+00
    max_beam_torsion:  0.000000e+00
    max_beam_total:    0.000000e+00
    max_disp_mag:     -5.465288e-07
    max_disp_x:       -2.288639e-08
    max_disp_y:        5.525955e-07
    max_disp_z:        0.000000e+00
    max_prin_mag:     -2.787973e+03
    max_rot_mag:       3.250554e-08
    max_rot_x:         0.000000e+00
    max_rot_y:         0.000000e+00
    max_rot_z:        -3.250554e-08
    max_stress_prin:  -2.787973e+03
    max_stress_vm:    -2.936599e+03
    max_stress_xx:    -2.902010e+03
    max_stress_xy:     4.588746e+02
    max_stress_xz:     0.000000e+00
    max_stress_yy:     4.885327e+01
    max_stress_yz:     0.000000e+00
    max_stress_zz:     0.000000e+00
    min_stress_prin:   4.651636e+03
    strain_energy:    -1.340397e-04
```

Begin Sensitivity Analysis, Parameter: thickness

```
Static Analysis "notch_plate_static":

  Convergence Method: Multiple-Pass Adaptive
  Plotting Grid:      4

      Calculating Element Equations          (20:07:32)
        Total Number of Equations:     492
        Maximum Edge Order:              3
      Solving Equations                      (20:07:40)
      Post-Processing Solution               (20:07:42)
      Calculating Disp and Stress Results    (20:07:43)

      RMS Stress Error Estimates:

      Load Set        Stress Error        % of Max Prin Str
      ------------    ---------------     ------------------------
      LoadSet1        4.18e+03            21.9% of  1.91e+04

      Resource Check                         (20:07:45)
        Elapsed Time     (sec):    2741.65
        CPU Time         (sec):      12.50
        Memory Usage      (kb):     168484
```

```
        Wrk Dir Dsk Usage (kb):        0

Analysis "notch_plate_static" Completed  (20:07:47)

Derivatives of Measures for Analysis: notch_plate_static
With Respect to Parameter: thickness

    total_cost:       0.000000e+00
    total_mass:       1.012423e+01
    com_x:           -3.509118e-14
    com_y:           -2.488438e-15
    com_z:            0.000000e+00
    inertia_xx:       7.526698e+00
    inertia_xy:       2.972349e-08
    inertia_xz:       0.000000e+00
    inertia_yy:       1.221562e+02
    inertia_yz:       0.000000e+00
    inertia_zz:       1.296829e+02

  Load Set: LoadSet1

    max_beam_bending:  0.000000e+00
    max_beam_tensile:  0.000000e+00
    max_beam_torsion:  0.000000e+00
    max_beam_total:    0.000000e+00
    max_disp_mag:     -1.897407e-04
    max_disp_x:       -3.682606e-05
    max_disp_y:        1.861326e-04
    max_disp_z:        0.000000e+00
    max_prin_mag:     -7.643958e+04
    max_rot_mag:      -2.339174e-05
    max_rot_x:         0.000000e+00
    max_rot_y:         0.000000e+00
    max_rot_z:         2.339174e-05
    max_stress_prin:  -7.643958e+04
    max_stress_vm:    -7.167290e+04
    max_stress_xx:    -7.621604e+04
    max_stress_xy:     1.593076e+04
    max_stress_xz:     0.000000e+00
    max_stress_yy:    -1.764033e+04
    max_stress_yz:     0.000000e+00
    max_stress_zz:     0.000000e+00
    min_stress_prin:   6.959613e+04
    strain_energy:    -4.719402e-02

Use Results to plot measures.

------------------------------------------------------------

Memory and Disk Usage:

  Machine Type: Windows NT/x86
  RAM Allocation for Solver (megabytes): 128.0

  Total Elapsed Time (seconds): 2776.18
  Total CPU Time     (seconds): 13.40
  Maximum Memory Usage (kilobytes): 168484
  Working Directory Disk Usage (kilobytes): 0
  Total Elapsed Time in Parameter Updates (seconds):
    1420.12
  Total Engine Elapsed Time Minus Param. Updates (seconds):
    1356.06
```

Total CPU Time in Parameter Updates (seconds):
1.58
Total Engine CPU Time Minus Param. Updates (seconds):
11.82

Results Directory Size (kilobytes):
386 .\loc_sens_study1

Run Completed
Sat Apr 05, 2003 20:08:23

Figure 31: Status Summary File for local sensitivity study (partial)

9. Select **Close > Close**. Select **Results** and create the maximum Von Mises stress definition sheet as follows:

Click on this icon named "**Insert a New Definition**."

Figure 32

Let's create results graph plotting each design parameter versus the Von Mises stress measure. Under results, **create three windows**, one for each parameter defined. Name the first result window the following: Vm_thickness. Fill out and make the appropriate selections as shown.

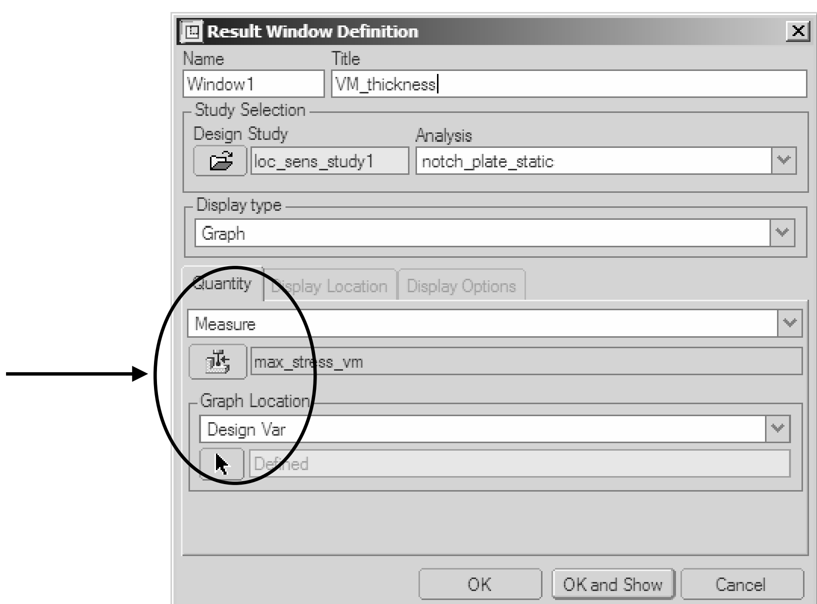


Figure 33: Result Window Definition (Thickness)

10. Next, select **maxvmstress** as a **measure** and select **Design Var** from the Graph location. Click on the **Select (colored icon)** from the quantity row and select **max_stress_vm** as shown below:

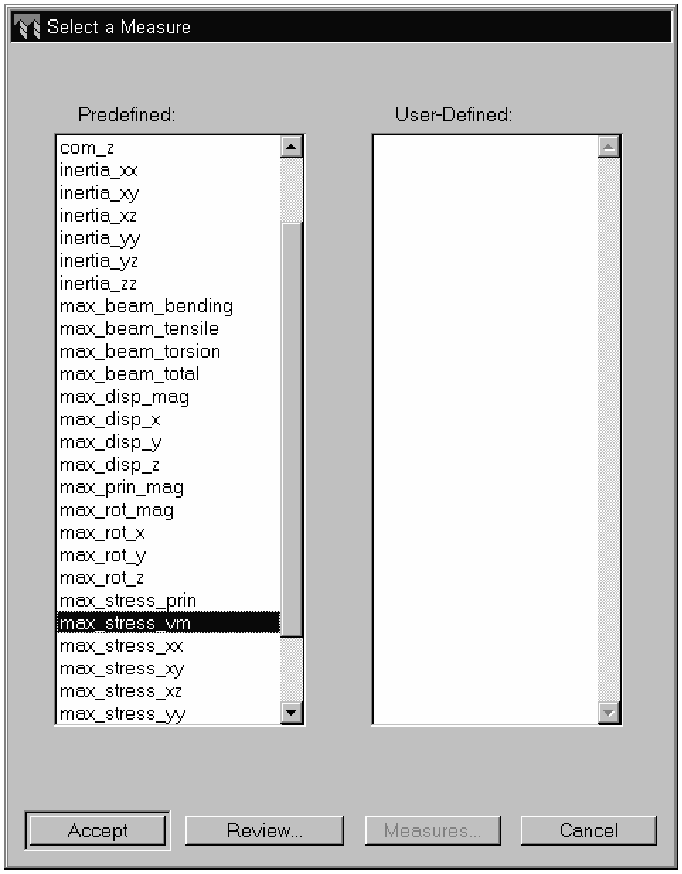


Figure 34

Select **Accept**. Click on the **Arrow Icon** and select the parameter thickness as shown:

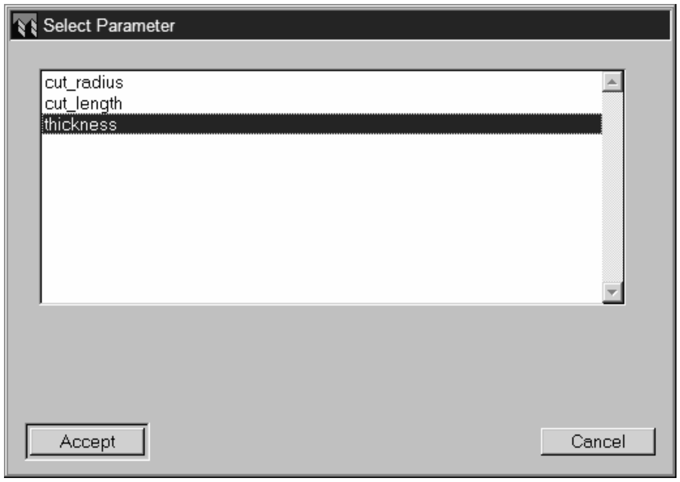


Figure 35

Select **Accept and OK**. Repeat the procedure above to create two additional result graphs as shown below.

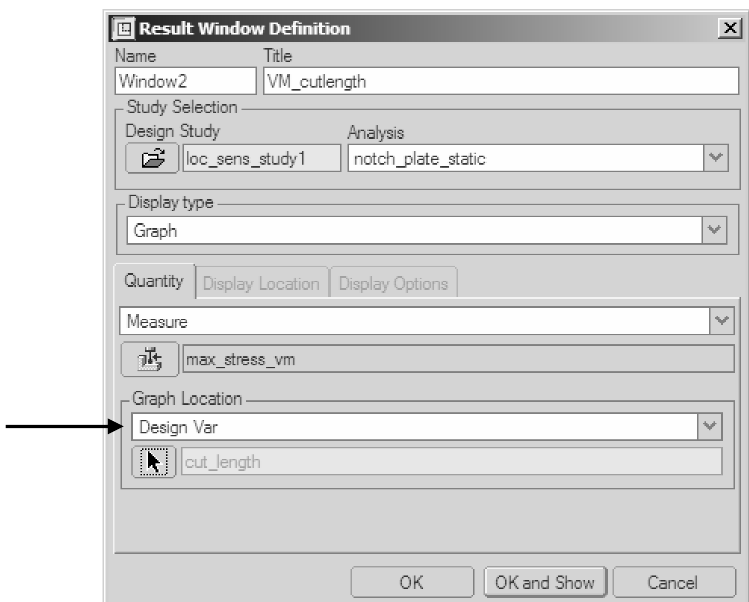


Figure 36: Result Window Definition (Cut length)

Select **OK**. Create a new result window as shown:

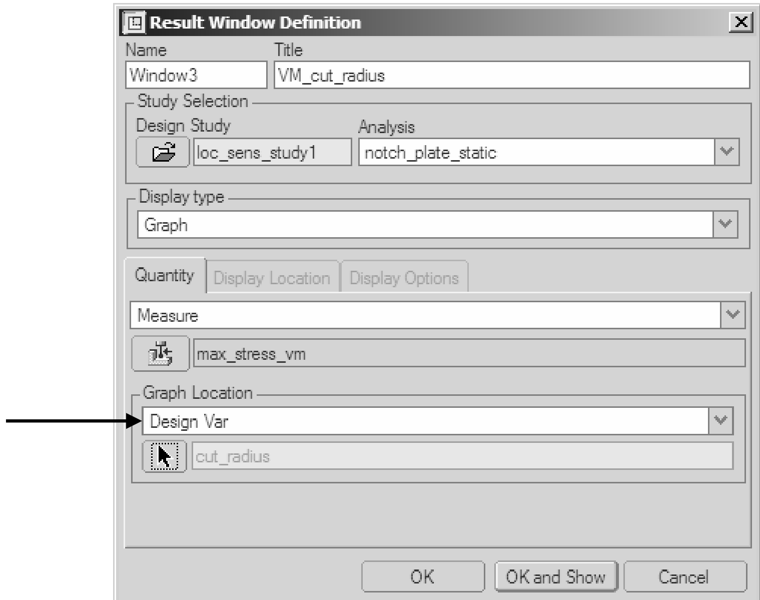


Figure 37: Result Window Definition (Cut radius)

Select **OK**. Select **View > Display** and select all three windows as shown:

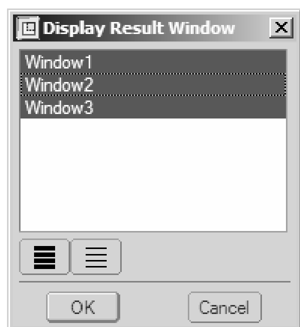


Figure 38

Select **OK**. The three result graphs should come up on your screen.

11. Look at the three graphs in Figure 39. To understand the results, look at the variation of the VM stress versus each parameter. Establish the relationship between each measure and the Von Mises stresses.

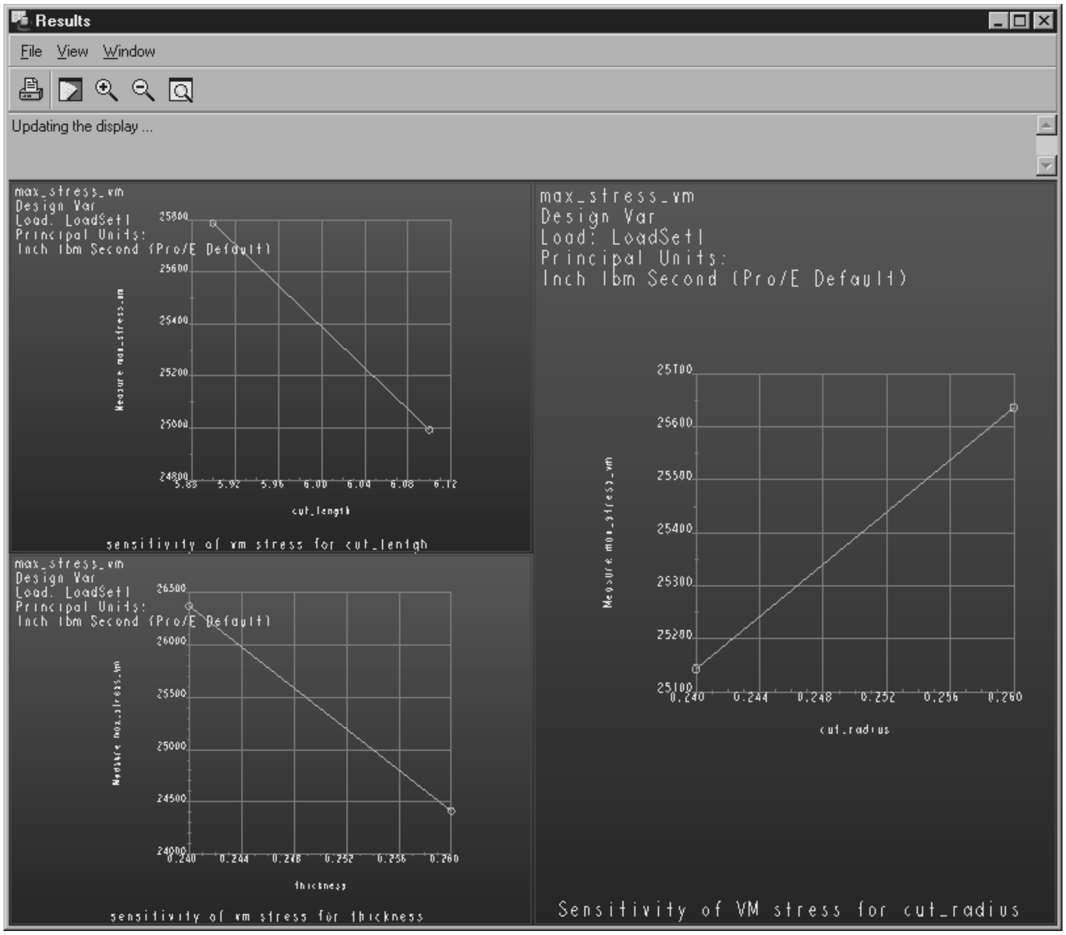


Figure 39: Local Sensitivity Study Parameter Results

12. Analyse how sensitive each parameter is to the Von Mises stresses. With the results shown, you should come to the following conclusions:

That the VM stress is sensitive to all parameters.

This was the main objective of the local sensitivity study. We will carry all three parameters into the next phase. Please note that should the VM stress not be sensitive to one of the parameters, we would not carry it on any further in the optimization study. Select **File > Exit Results > No**.

Procedure

Part C: Global Sensitivity Study

Recall:

The objective of the global sensitivity study is to look at the variations of all parameters into each step of the process as defined by the user.

1. In Pro/MECHANICA®, go back to **Dsgn Controls > Design Parameters** and the following window should come up on your screen.

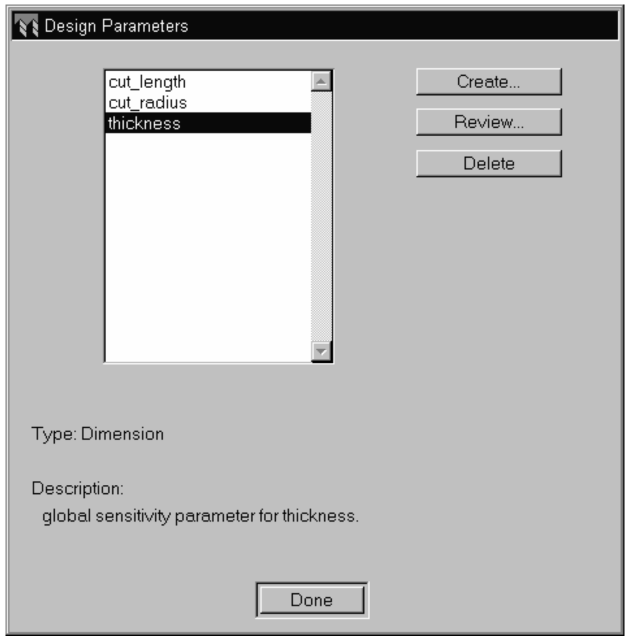


Figure 40

Select **Review**. Review each parameter and **change the minimum and maximum values** to the following magnitudes.

Note: In practice, *the designer would set these values as he has an understanding of the model by now.*

To modify the parameter cut length, select **cut_length > Review**. The following window will come up:

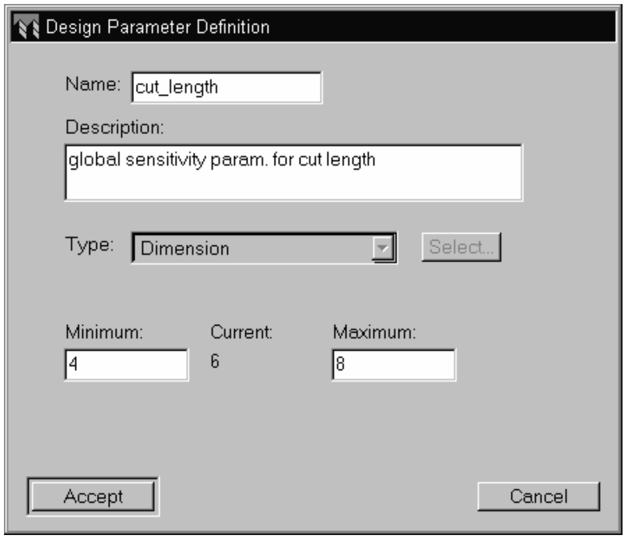


Figure 41

Modify minimum values to 4 and 6 as shown. Click on **Accept**. Repeat procedure for the other design parameters, values to be entered are shown in the following table:

TABLE 2: DESIGN PARAMETERS

Design parameter	**Minimum**	**Current**	**Maximum**
Cut_length (in.)	4	6	8
Thickness (in.)	.135	.25	.375
Cut_radius (in.)	.1	.25	.5

Click on **Done** when you have completed all design parameters.

2. Next, create a design study called:***glob_sens_stud1***. Select **Analyses/Design Studies** and fill out the design study sheet as shown with the following settings:

- Select ***global sensitivity*** under type.
- Under analysis, select anlys_1 (static).
- For the parameters: set as seen on the figure below.
- Set the number of intervals to six for this analysis.
- Do not check repeat P-loop convergence for this analysis (saves time).

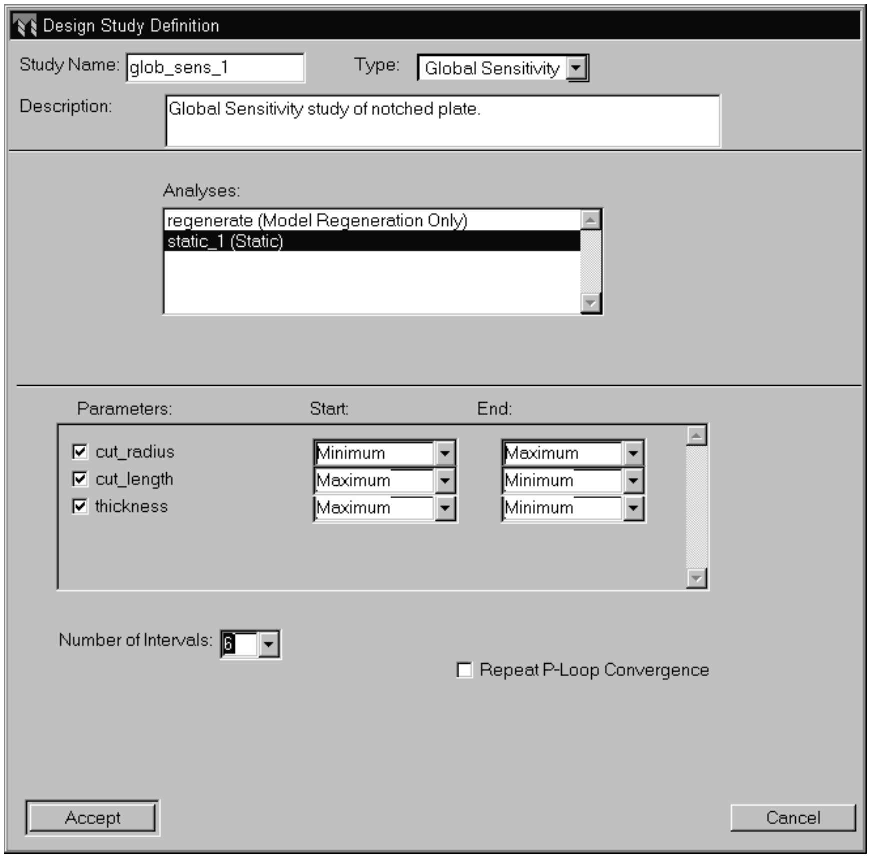


Figure 42

Note that the minimum and maximum values are set to *minimize the plate weight as the study progresses* according to the results of the local sensitivity study: loc_sens_stud1. Select **Accept**.

3. Run the design study by selecting the design study and click on **Run**. (Answer **Yes** to "**Do you want error detection?**") It takes approximately five to six minutes. Take a well-deserved break. If you want to follow the analysis, click on the status icon:

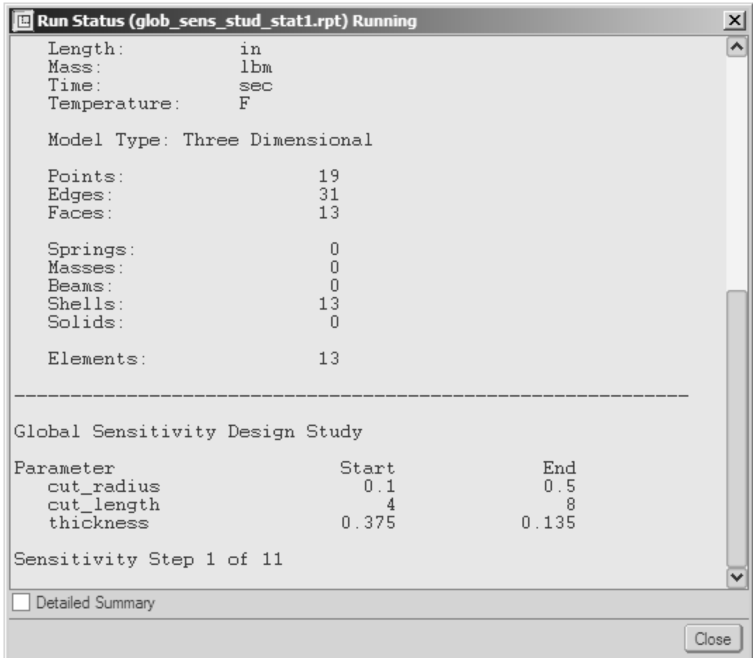


Figure 43: Status Summary File (global sensitivity study)

Once the analysis is completed, check the last step (step 11) with the partial status file below:

```
Sensitivity Step 11 of 11

Parameters:
  cut_radius          0.5
  cut_length            8
  thickness         0.135

Static Analysis "notch_plate_static":

  Convergence Method: Multiple-Pass Adaptive
  Plotting Grid:     4

  Convergence Loop Log:                (21:49:36)

  >> Pass  1 <<
      Calculating Element Equations          (21:49:36)
        Total Number of Equations:    330
        Maximum Edge Order:            1
      Solving Equations                  (21:49:37)
      Post-Processing Solution              (21:49:37)
      Calculating Disp and Stress Results     (21:49:37)
      Checking Convergence                  (21:49:37)
        Elements Not Converged:       51
        Edges Not Converged:        107
        Local Disp/Energy Index:    100.0%
        Global RMS Stress Index:    100.0%
```

```
      Resource Check                  (21:49:37)
        Elapsed Time    (sec):   2951.92
        CPU Time        (sec):     59.03
        Memory Usage     (kb):    166628
        Wrk Dir Dsk Usage (kb):     2048

  >> Pass  2 <<
      Calculating Element Equations        (21:49:38)
        Total Number of Equations:    966
        Maximum Edge Order:             2
      Solving Equations               (21:49:38)
      Post-Processing Solution             (21:49:38)
      Calculating Disp and Stress Results    (21:49:38)
      Checking Convergence                 (21:49:39)
        Elements Not Converged:        33
        Edges Not Converged:          106
        Local Disp/Energy Index:    100.0%
        Global RMS Stress Index:     67.3%
      Resource Check                  (21:49:39)
        Elapsed Time    (sec):   2953.28
        CPU Time        (sec):     60.14
        Memory Usage     (kb):    166628
        Wrk Dir Dsk Usage (kb):     2048

  >> Pass  3 <<
      Calculating Element Equations        (21:49:39)
        Total Number of Equations:   1734
        Maximum Edge Order:             3
      Solving Equations               (21:49:39)
      Post-Processing Solution             (21:49:40)
      Calculating Disp and Stress Results    (21:49:40)
      Checking Convergence                 (21:49:41)
        Elements Not Converged:         0
        Edges Not Converged:            0
        Local Disp/Energy Index:      7.6%
        Global RMS Stress Index:      4.5%
      RMS Stress Error Estimates:
      Load Set       Stress Error     % of Max Prin Str
      ------------   ------------     -----------------------
      LoadSet1        3.74e+03     12.0% of  3.12e+04
      Resource Check                  (21:49:41)
        Elapsed Time    (sec):   2955.73
        CPU Time        (sec):     61.43
        Memory Usage     (kb):    166628
        Wrk Dir Dsk Usage (kb):     2048
  The analysis converged to within 10% on
  edge displacement, element strain energy,
  and global RMS stress.
Analysis "notch_plate_static" Completed  (21:49:42)
Completed Global Sensitivity Study
Use Results to plot measures.
-----------------------------------------------------------
Memory and Disk Usage:
  Machine Type: Windows NT/x86
  RAM Allocation for Solver (megabytes): 128.0
  Total Elapsed Time (seconds): 2956.72
  Total CPU Time     (seconds): 61.84
  Maximum Memory Usage (kilobytes): 166628
  Working Directory Disk Usage (kilobytes): 2048
  Total Elapsed Time in Parameter Updates (seconds):
    2647.64
  Total Engine Elapsed Time Minus Param. Updates (seconds):
```

```
    309.08
  Total CPU Time in Parameter Updates (seconds):
    4.88
  Total Engine CPU Time Minus Param. Updates (seconds):
    56.96
  Results Directory Size (kilobytes):
  789 .\glob_sens_stud_stat1
  Maximum Data Base Working File Sizes (kilobytes):
  2048 .\glob_sens_stud_stat1.tmp\kel1.bas
------------------------------------------------------------
Run Completed
Sat Apr 05, 2003   21:49:43
------------------------------------------------------------
```

Figure 44: Status Summary File (Partial) – Global Sensitivity Study

Select **Close** > **Close**.

Results

Select **Results** and create the maximumVon Mises stress definition sheet as follows:

Click on this icon named '**Insert a new definition**.'

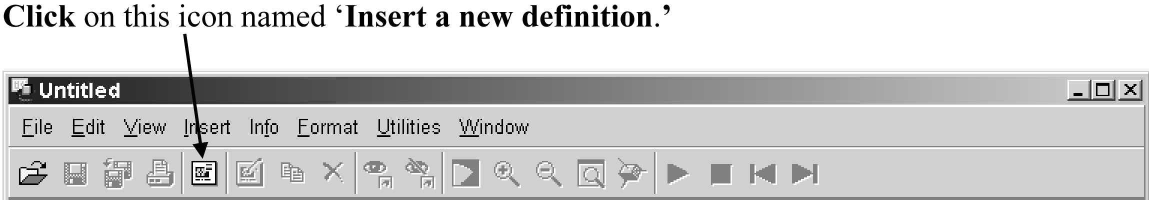


Figure 45

Create and show the design parameter graphs, similar to the ones for the local sensitivity study. The three result windows are presented below.

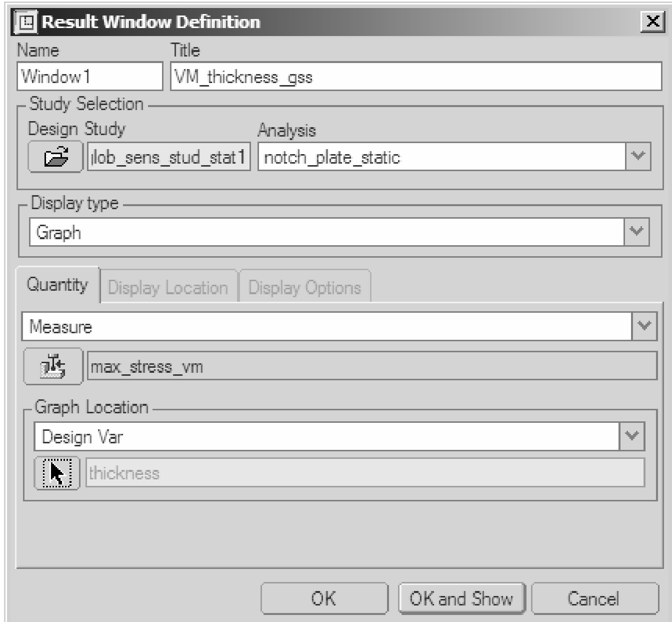


Figure 46: Result Window Definition (Thickness)

The first result window is titled: vm_thickness_gss. Fill out the definition sheet as shown above, then select **OK**. Create a second result definition window as follows.

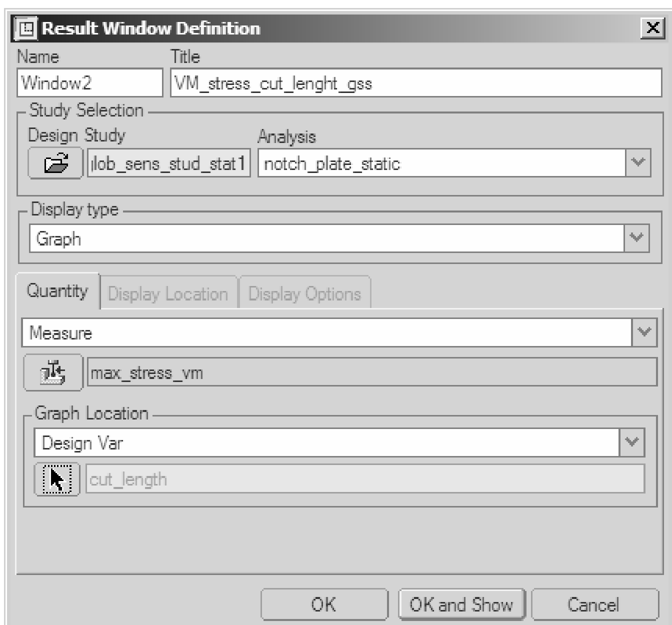


Figure 47: Result Window Definition (Cut length)

The second result window is titled: vm_stress_cut_length_gss. Fill out definition sheet as shown above, then select **OK**. Similarly, create a third definition window titled: VM_cut_radius_gss. Select the radius as the design variable and select **OK**. Select **View > Display** and select all three windows as in Figure 48.

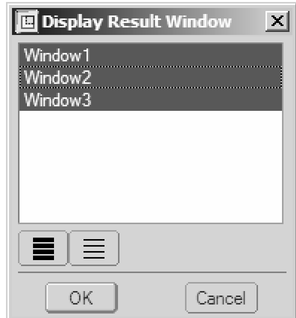


Figure 48

Select **OK**. The three result graphs should come up on your screen as shown below.

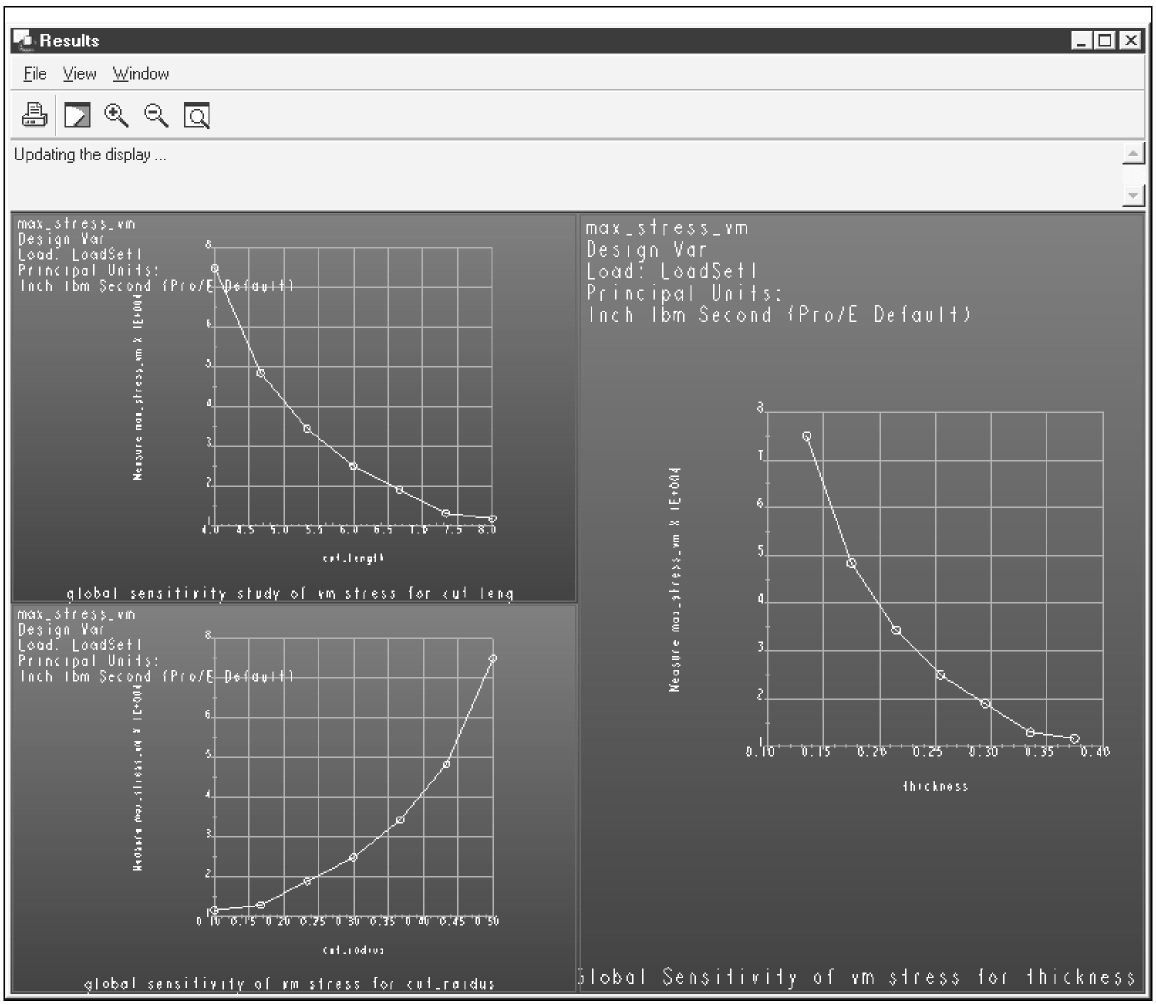


Figure 49: Local Sensitivity Study Results

The intent here is to find the best combination of parameters that will be carried on to the final optimization study. What conclusions do you come to with regards to each design parameter as a function of our measure (the Von Mises Stress)?

Your conclusion looking at the results above should be the following. That for minimum VM stress, we need the following:

- ***maximum thickness***,
- ***maximum cut length and;***
- ***minimum cut radius***.

I know, these results may be evident to us as strength of material experts. But in practice of other design projects, it may not be so easy to get to them. We suggest carrying this procedure for any project regardless of your expertise and confidence, especially for larger assemblies and design parameters that the effects on the measure are unknown. This will minimise the optimization time in all cases!

Phase 2: The Optimization Study on Total Mass

We now can move to the final phase of optimization. Let's review what we have done so far:

1. With our knowledge of the model and boundary conditions, we determined the parameters that we wish to optimize on for minimum weight design.
2. We ran a static analysis to determine if our material of choice actually will fit the application and the study.
3. We ran a local sensitivity study on each parameter to see the effects of VM stress as the parameter is modified slightly;
4. We ran a global sensitivity study (minimum and maximum values set by the designer) to look at the best possible combination of parameters that will lead to our optimized design.
5. We then concluded by selecting parameter values that will minimize the VM stress.

Now, let's perform the steps for the final optimization.

Procedure

1. Create a design study of type: optimization study named: ***opt_study_1***. Select **Design Studies** and make the following settings (see procedures on figure below).

   Select **Design Study** and enter the information as shown in Figure 50.

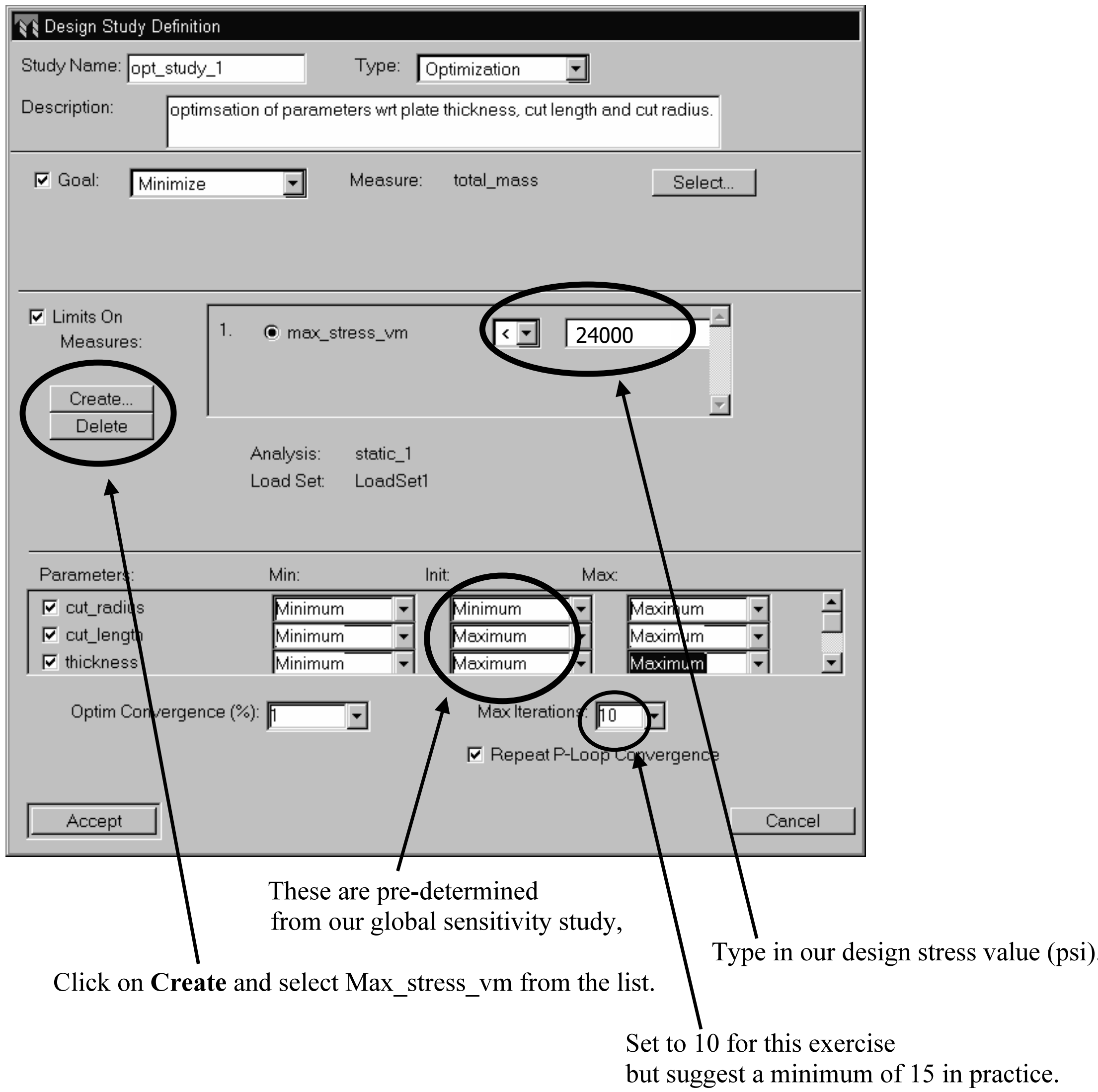


Figure 50: Optimization Study Definition

2. Run the design study. This may approximately 20 to 30 minutes. The study should go through five optimization passes. Check the status file and compare it to the one shown below. Locate the following results on your status file and write them down here:

   - Goal (Minimum mass of the model): __________
   - Optimized thickness: __________
   - Optimized cut_length: __________
   - Optimized cut_radius: ___________

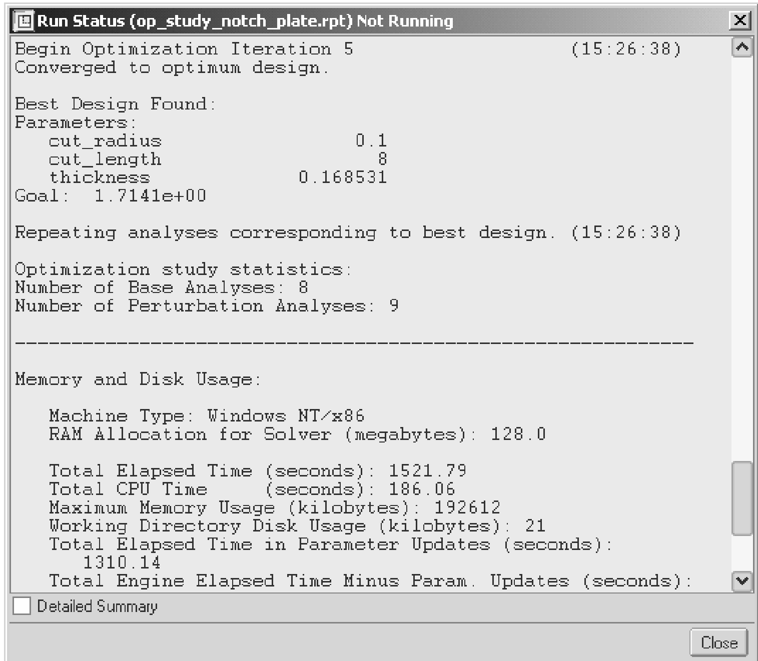


Figure 51: Status Summary File (partial) – Optimization Study

Select **Close** > **Close**.

Results

We can create and plot the graphs to visualise the different parameters as a function of mass and Von Mises stresses. Though we will only show a few result windows for this part of the exercise, there are a lot for the finite element analyst to play around with in this section.

Select **Results.** Once the empty window comes up, click on the icon named **'Insert a new definition**.'

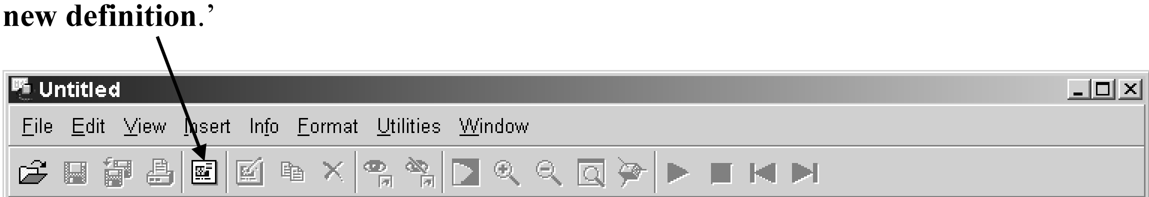


Figure 52

Create the two windows shown in Figures 53 and 54.

From the result window, click on **Select**; and select the ***max_vm stress*** as a measure as seen below:

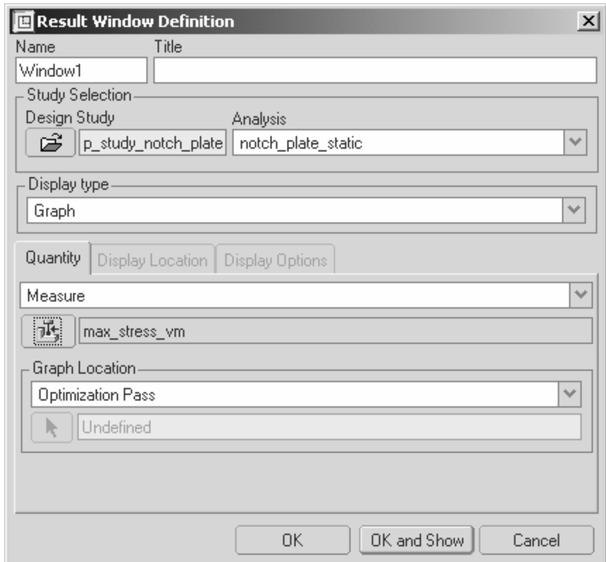


Figure 53: Result Window1 Definition (optimization study)

Select **OK**. Insert the following definition window.

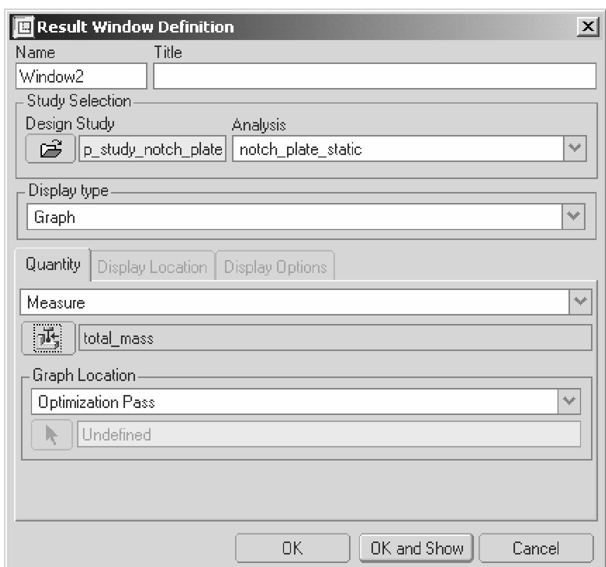


Figure 54: Result Window1 Definition (optimization study)

Repeat the same procedure to get the results for total mass. Select **OK**. Select **View >Display** and select both windows as shown below**:**

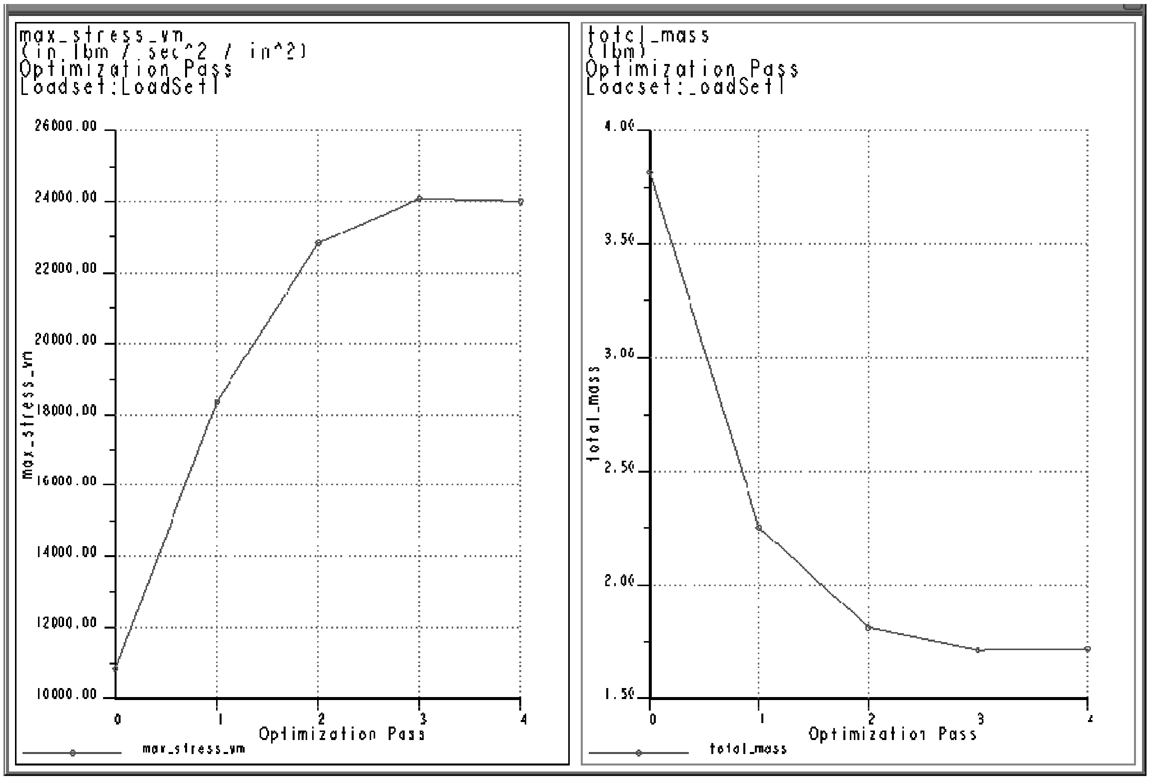


Figure 55: Result Graphs (optimization study)

Conclusion

First, congratulations on your first optimization study! I hope you realise that this is a lot of work. We have reduced the weight of the plate to a minimum with the parameters that we wanted to measure, which should lead to reduced costs.

Look at the optimization passes above. After the first pass, Pro/MECHANICA® starts moving towards our limit on the VM stress. The reason that so few passes were needed is that we did the right work prior to the optimization. All the preliminary work that we performed with the local and global sensitivity studies lead us to the general area of optimization, and then we used the optimization design study to refine it to perfection using the least amount of optimization pass.

Project 1

Problem Definition

We would like to optimise the sheet metal bracket in Table 3 in order to minimise weight and cost.

TABLE 3: OPTIMIZATION PROJECT OBJECTIVES

Optimization Objective	Measure to be used for analysis of parameter effect on part	Parameters to be used for optimisation study
Minimise mass of the part	Von Mises stress	3. Material Thickness and; 4. Bend radius

The model is built from two features (you can omit the holes shown in the figures below). It is simply a swept protrusion which can be built as shown in Table 4.

TABLE 4: BRACKET MODEL FEATURES

Feature #	Description	Reference figure
1	Datum Curve (sketched)	P1
2	Thin swept protrusion	P2
3	Hole Pattern	P4

Boundary Conditions

Flat vertical flange surfaces (back): fixed in all DOF.

Load (1000 lbs): Applied on curve surface (top of hat section) though the centre point.

Hint: In load definition form, select Load through a point from load type.

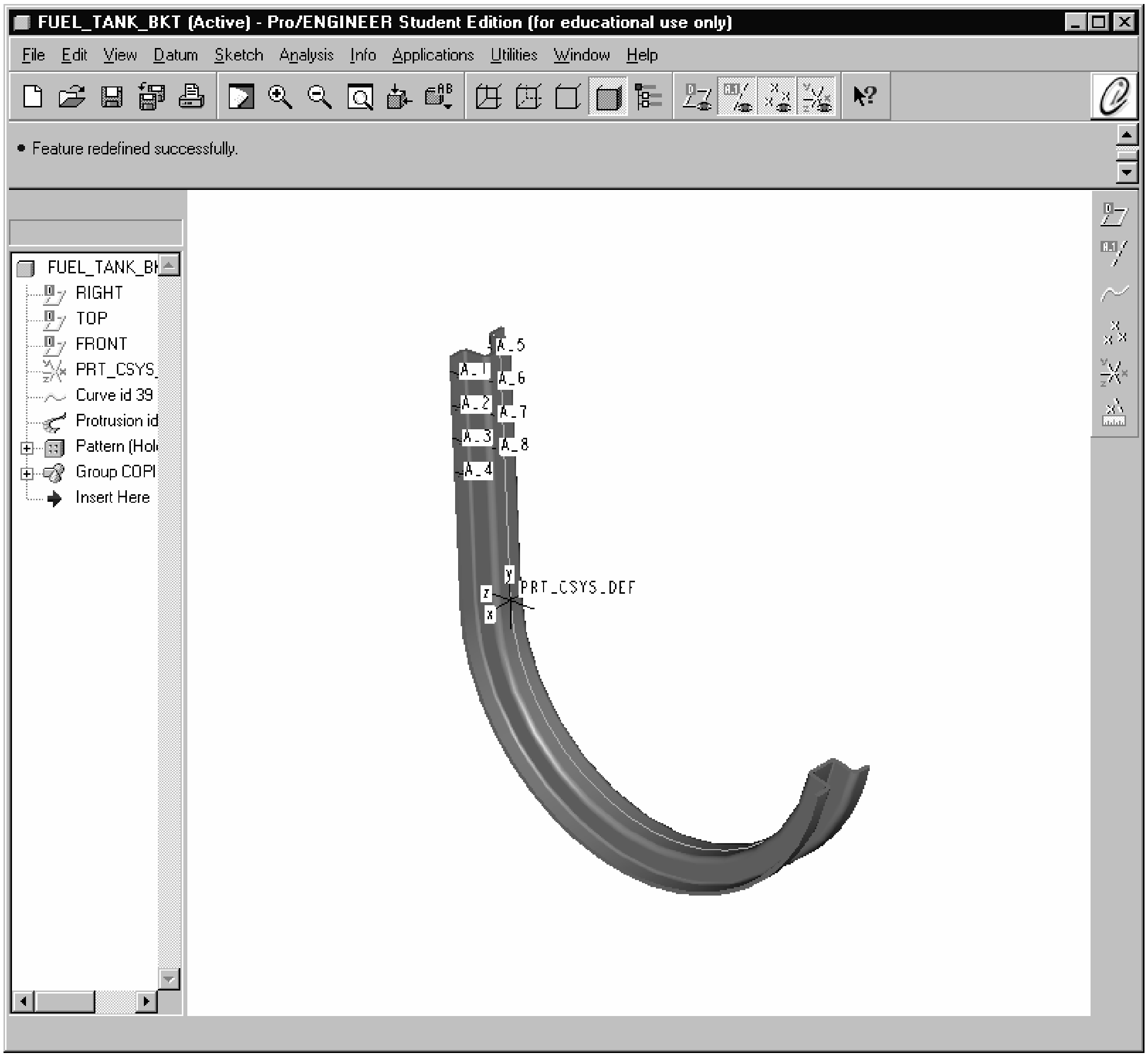


Figure P1: Suspension bracket (3-D view)

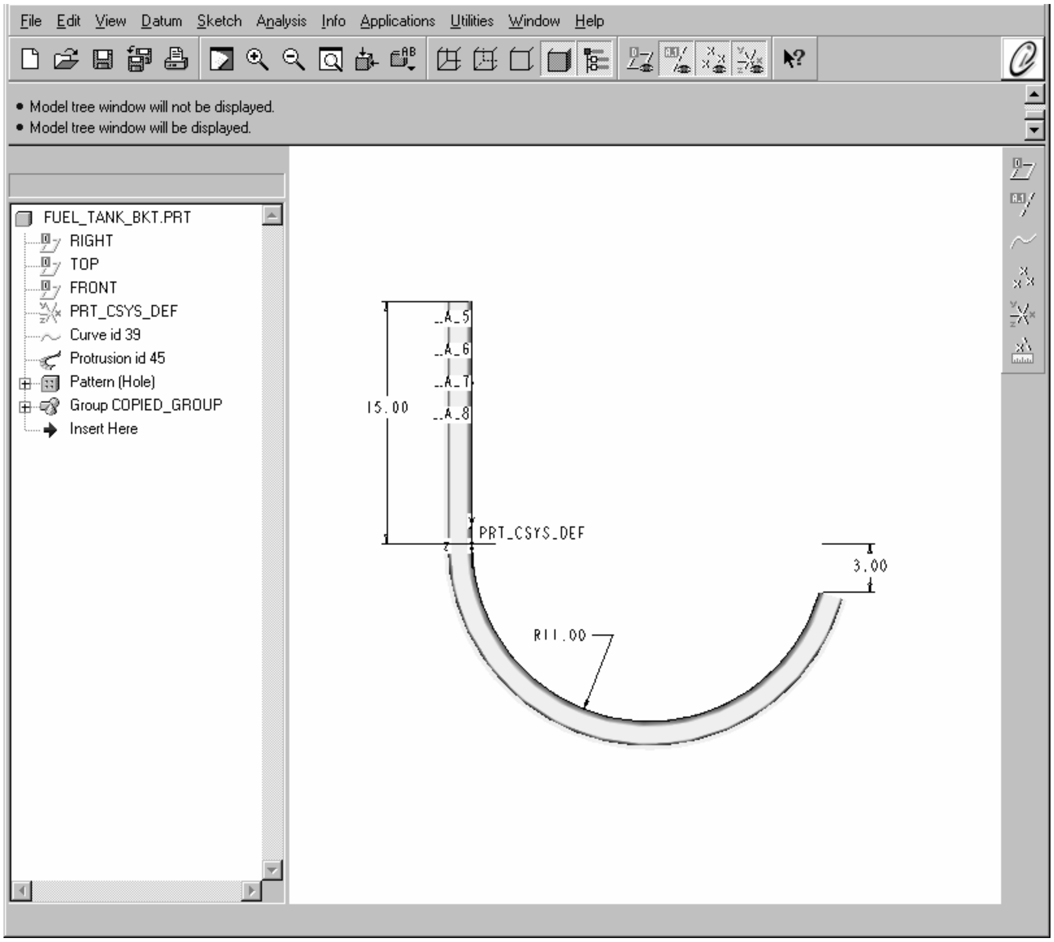


Figure P2: Dimensions of datum curve (2-D)

Cross Section

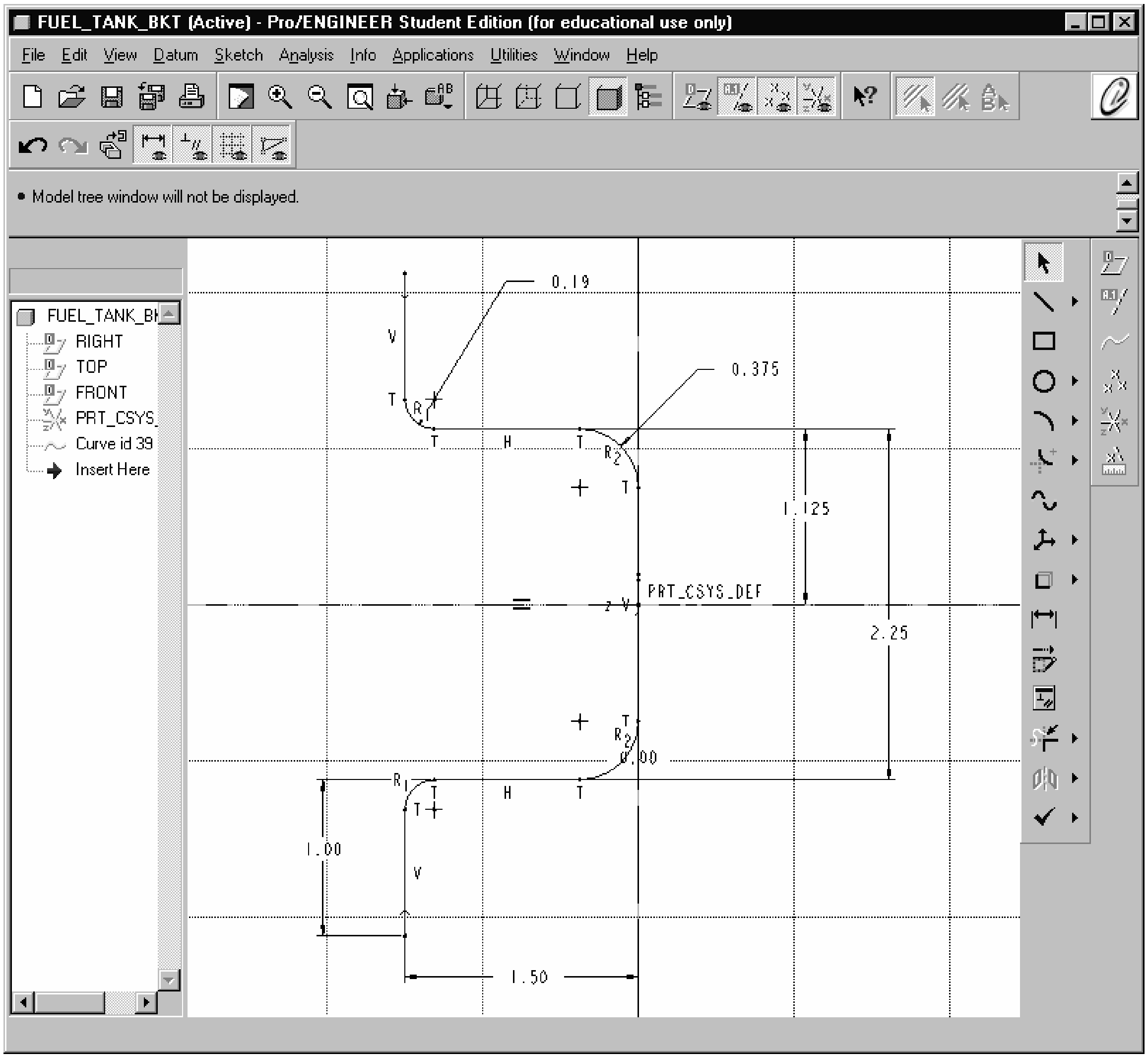


Figure P3: Cross-section of suspension bracket - Thickness and BR are: 3/16″

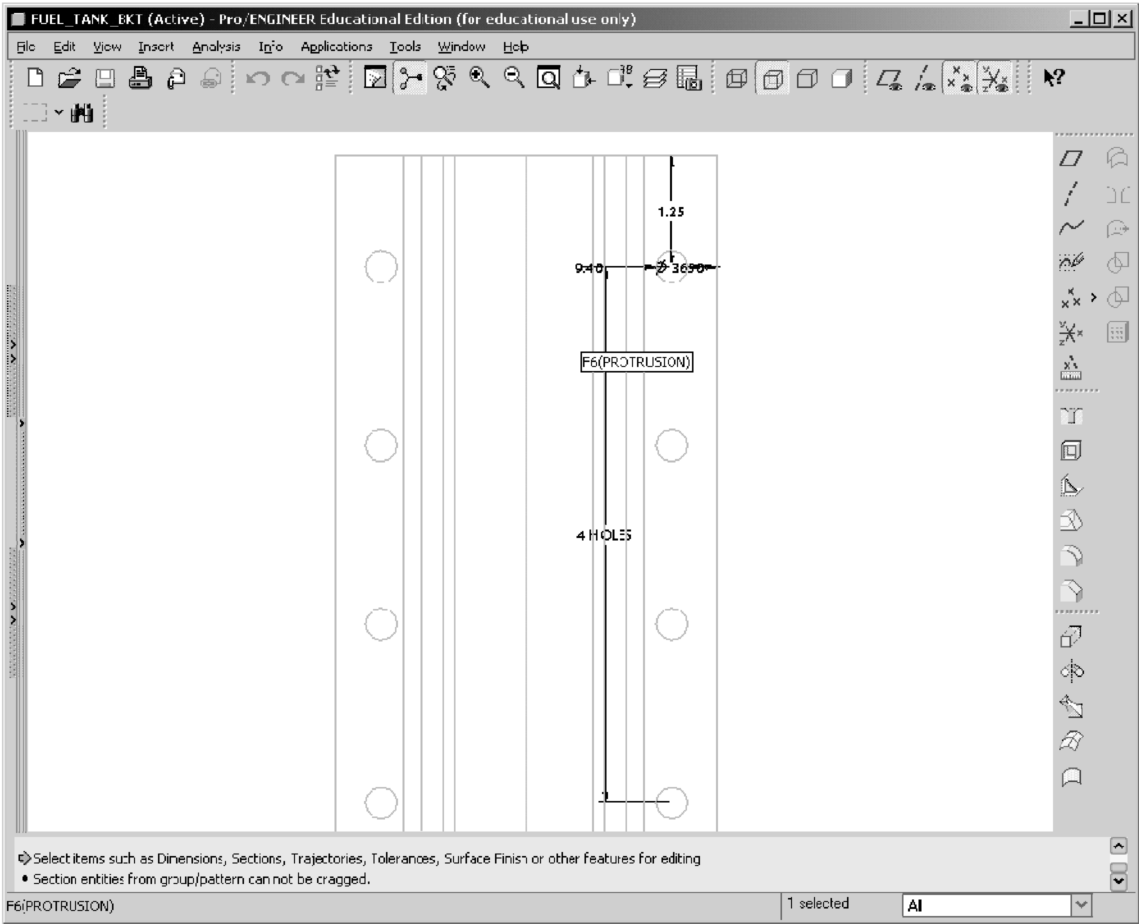


Figure P4: Hole Pattern – Holes (.36 in. diameter) are two inches apart (vertically)

Objective: Optimize combined material thickness and bend radius for a design stress of 34 ksi.

Answer: (after "manually" rounding up to nearest nominal values)
Thickness: .25″
Bend Radius: 5/16″

End of Exercise