Sensitivity and Optimization Tutorial (Pro Mechanica Wildfire 2.0)

Time: 2 hours and 20 minutes

Source: Pro/Mechanica Structure Wildfire: Elements and Applications (Integrated mode) YVES Gagnon, M.A.Sc, SDC Publications (Chapter 1).

Objective:

To demonstrate the use of Pro Mechanica in sensitivity and optimization studies.

Overview:

The purpose of an optimization study is to help the designer in optimizing certain design parameters as a function of known measures, such as Von Mises Stress or maximum displacement for a specific goal (i.e. Minimize total mass). The software can not do all the work in the optimization process; this is only a tool the designer uses to get to the final objective, so that the Designer's knowledge plays a very important roll in any optimization study.

The optimization study (Pro Mechanica) we are going to perform is divided into main phases:

1. **Design Phase I:** In this part a static analysis is created and later is combined with local and global sensitivity studies in Pro Mechanica. In this phase, Pro Engineer Wildfire 2.0 is used to set the Design parameters.

    a. The Main objective of a local sensitivity study is to look at changes of the measures, Von Mises in most cases, for small variations of each design parameter ($^{+}$- 1%) independently.
    b. 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.

2. **Design Phase II:** Completes the optimization of the part according to your design objectives (goals). All parameters are optimized concurrently. A goal could be to minimize the total mass or the total cost of the model

Procedure:

The procedure to carry out the Optimization process is as follows:

1. Create the part in Pro/Engineer
2. Switch to Pro/Mechanica mode and create a shell mid-plane compression idealization (assign material properties)
3. Create the mesh and boundary conditions (loads and constraints)
4. Run a quick check analysis to know if the model converges to a solution with respect to a measure (Von Mises Stress)
5. Set up Design parameters for the preliminary design study
6. Run a local sensitivity study and select parameters that have an effect on the measures predetermined
7. Run a global sensitivity study on selected parameters and find the parameter value (maximum and minimum) that has the greatest effect on the measure
8. Run an optimization study for the above parameters, setting the starting point of the study using the findings of point #7 above. Optimize for your design objective(s). The design objective for this case is to minimize the mass of the plate.

The following figure shows the model that will be used for the study. It's 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 (where the coordinate system is)

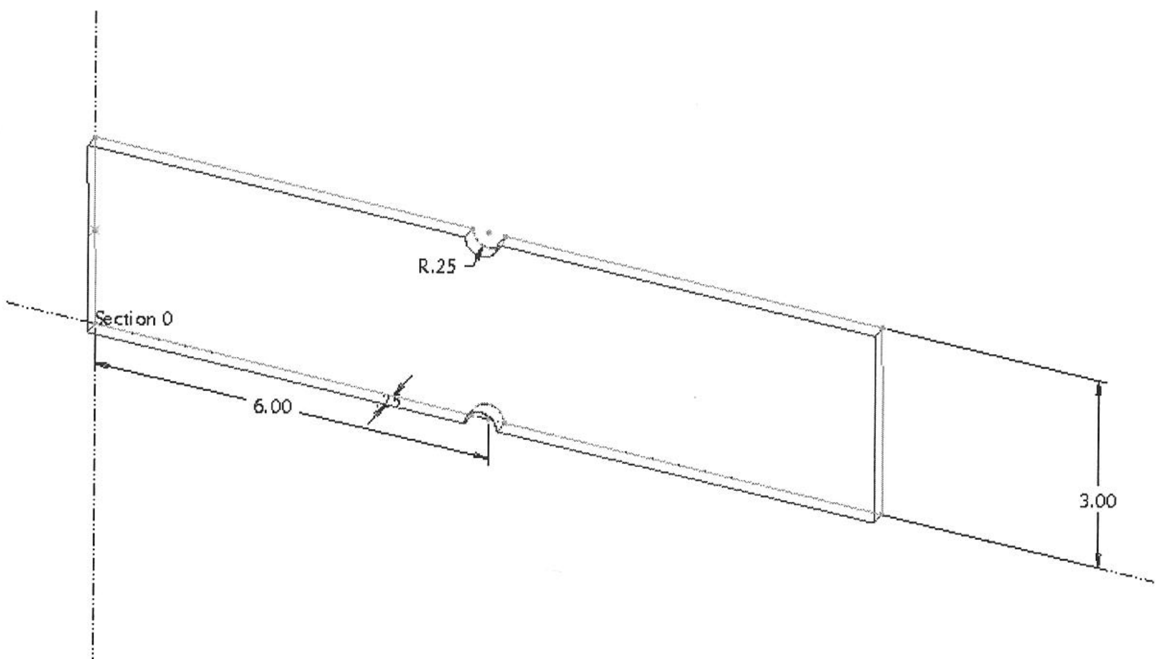


Figure 1 (Notched Cantilever Beam)

The Design objectives are 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.

Please, refer to model represented in Figure 1. It is a plate made of Steel, now, we are going to define the objectives and determine the measure to be studied for the selected design parameters.

Optimization Goal	Measure to be used for Optimization	Design Parameters to be optimized		
Minimize Weight of the part	Von Mises Stresses	**Name**	**Description**	**Initial Value(in)**
		Cut_length	Dimension from left edge to cut	6
		Cut_radius	Notch radius	0.25
		Thickness	Plate Thickness	0.25

Before starting the procedures below, create a directory named **Opt_Study,** using Microsoft Explorer. Copy the part named **Plate_Tutorial** to that directory

Note: **Integrated Mode in Pro/Mechanica**: This mode can be accessed via Applications-Mechanica-Structure-Model. This mode can be used for structural modeling of a part/assembly i.e., define all the simulation modeling entities and prepare the model for finite element analysis.

1. **Start up Pro/E Wildfire 2.0**

Select Start>All Programs>PTC>Pro/Engineer>Pro/Engineer

This procedure to run the software can be different on every computer

Set up your working directory (**File>Set Working directory**)

Select your working directory **Opt_Study,** then click on **Accept**

2. **Open the file Plate_Tutorial**

Once open, the following part show up on your screen

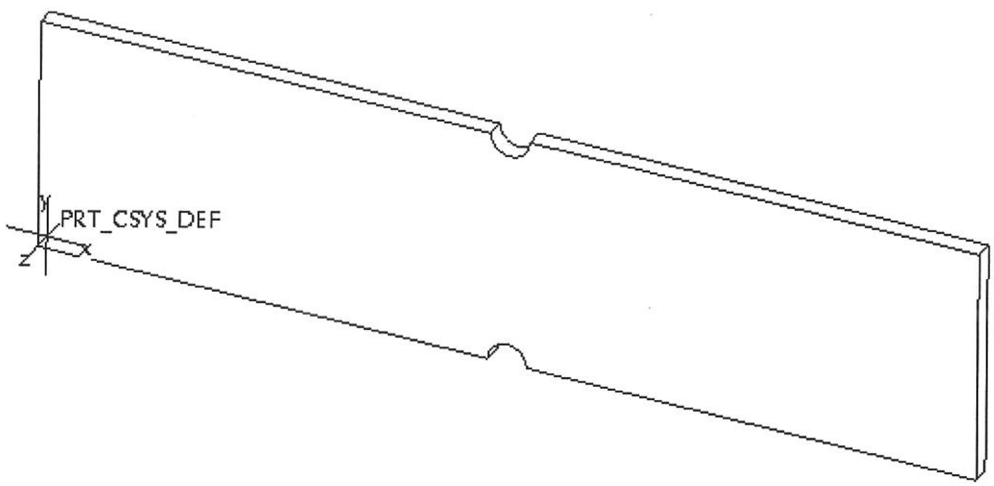
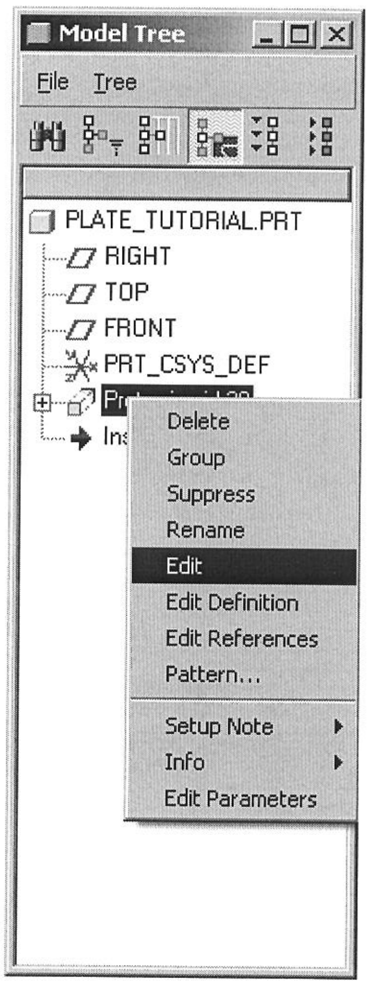


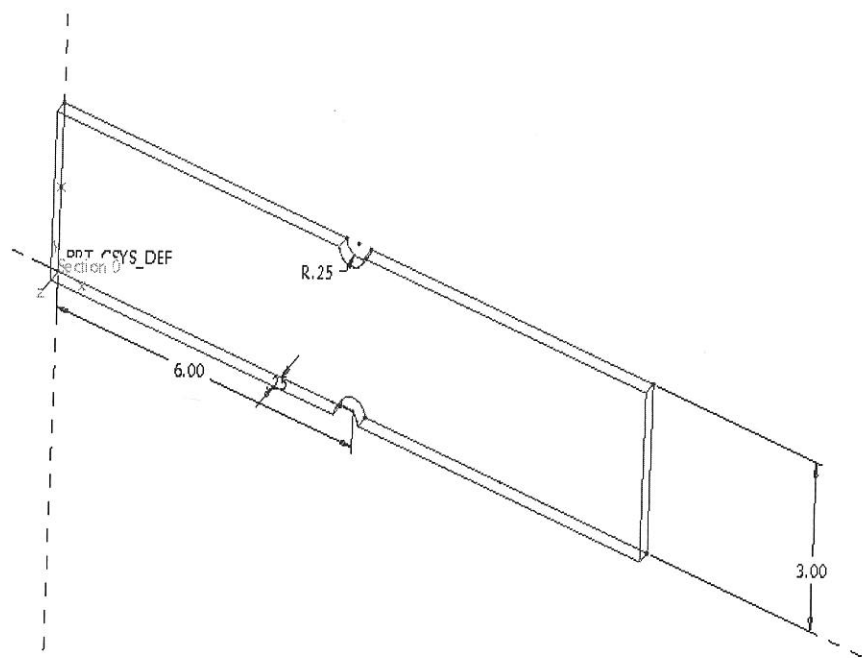
Figure 3

3. Modifying Dimension Cosmetics in Pro/E

We have to modify the dimensions cosmetics in Pro/Engineer for ease of recognizing each optimized design parameter during the process. From the model tree, select **Protrusion id39** (the id number could be different) and click on it with the right button of the mouse, a pop up menu appears, select edit, the dimensions for the feature will show up on the screen

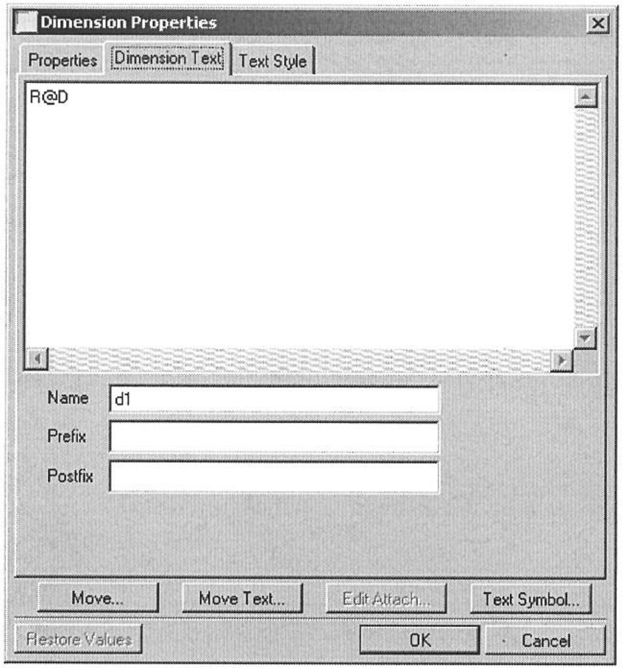
Model Tree



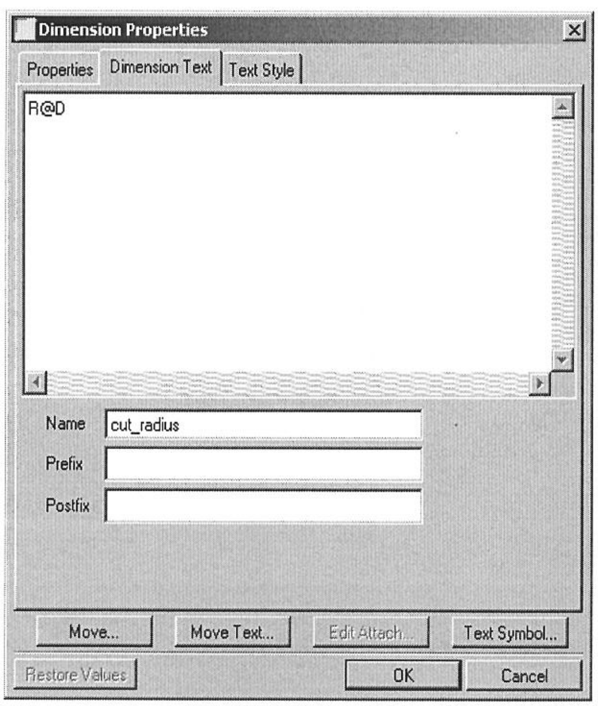


Dimensions on the Model

Select the **R.25** dimension and click on it with the right button of the mouse, and select **properties,** select the Dimension Text Tab

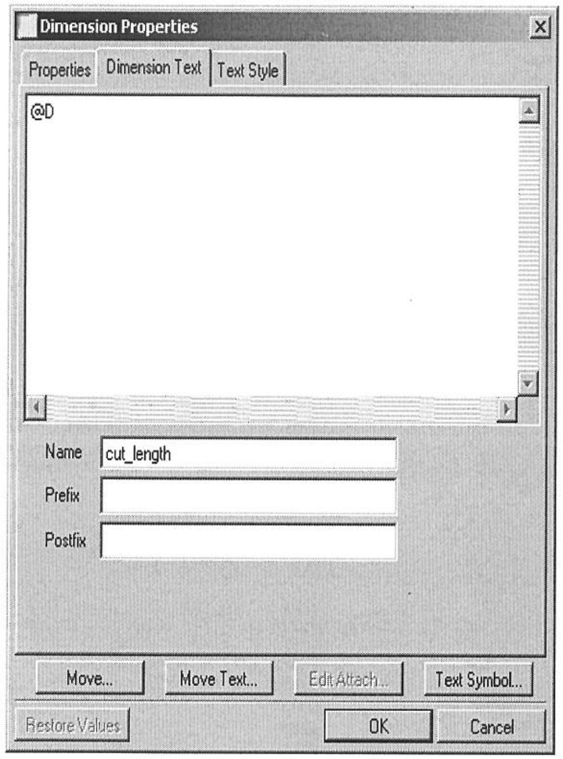


Change the dimension text name to **cut_radius**

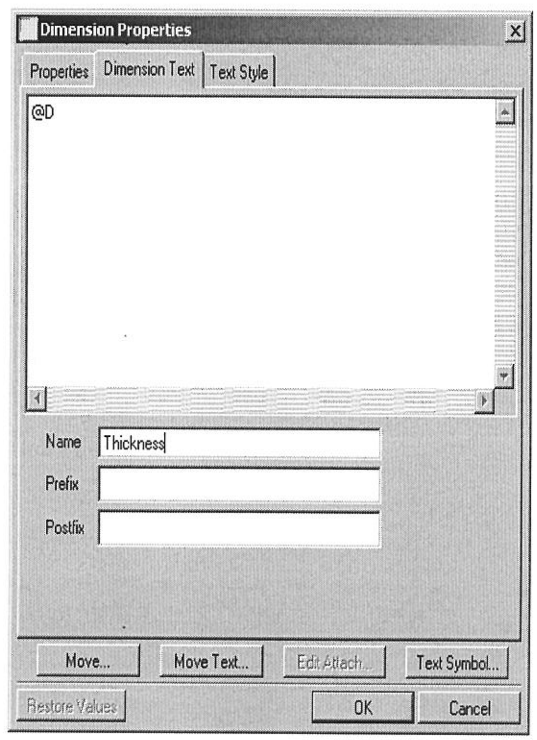


Select **Ok**

Repeat the procedure for the 6.00 inches dimensions and rename it to **cut_length** as shown below



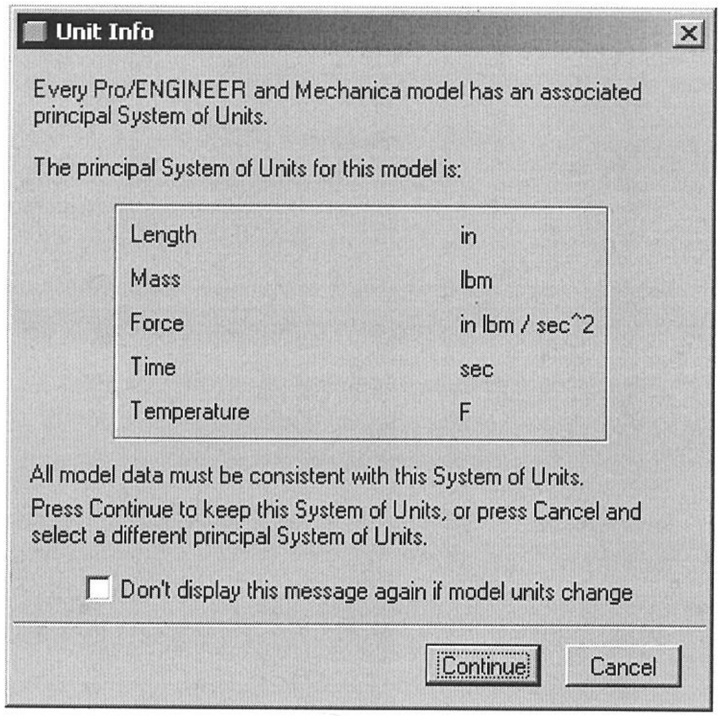
Now, change the thickness dimension



Click **Ok**

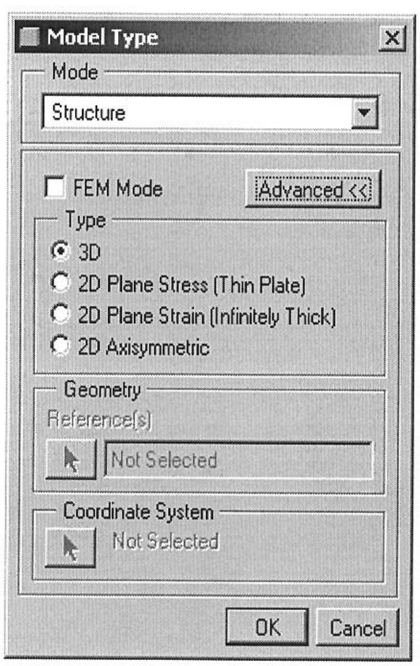
4. Switching to Pro/ Mechanica (Integrated Mode)

Select **Applications>Mechanica**



A dialog box appears indicating the units the software will use to perform the analysis

Click on **Continue**



A new dialog box appears, select structure and 3D analysis
Click on **Ok**

Model Type:- Opens the model type definition dialog form. Lets the user define the model type. Default model type is always 3D. If the model type selected should be 2D, user also has to select the Geometry and the associated coordinate system.

3D
2D Plane Stress
2D Plane Strain
2D Axisymmetric

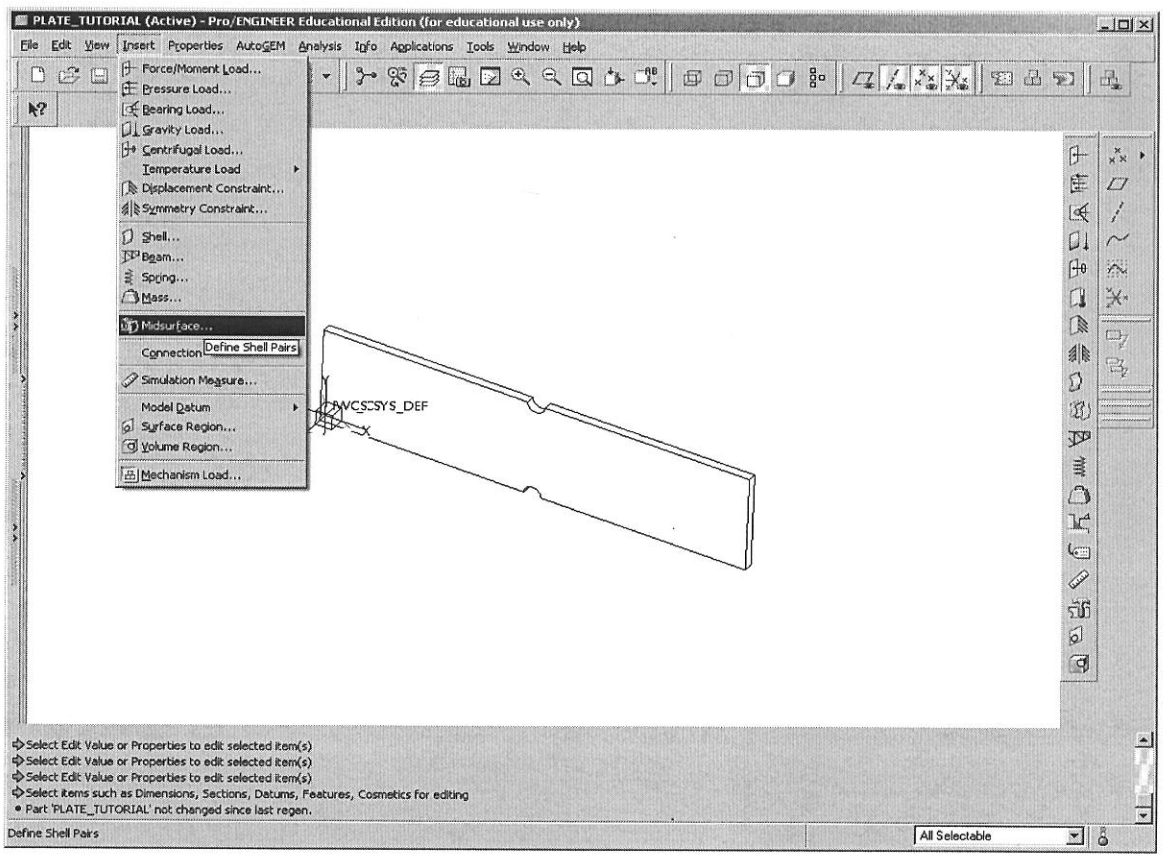
If the model is a 2D model, all geometry, loads and displacements must lie in the xy plane of the Cartesian coordinate system. For 2D axisymmetric models, all coordinates must be positive in X in the XY plane.

A new group of buttons (tool bars) appears on the right of your screen (These are the Pro/Mechanica tool bars)

Idealizations:- The Following idealizations are available in the Integrated Mode – Shells, Beams, Masses, Springs. These idealizations should be used whenever possible as they require less computation time. Another advantage of using these models is that they are easy to model.

For this exercise we are going to create a compressed shell idealization (the thickness of the plate permits to do it).

Select **Insert>Midsurface**

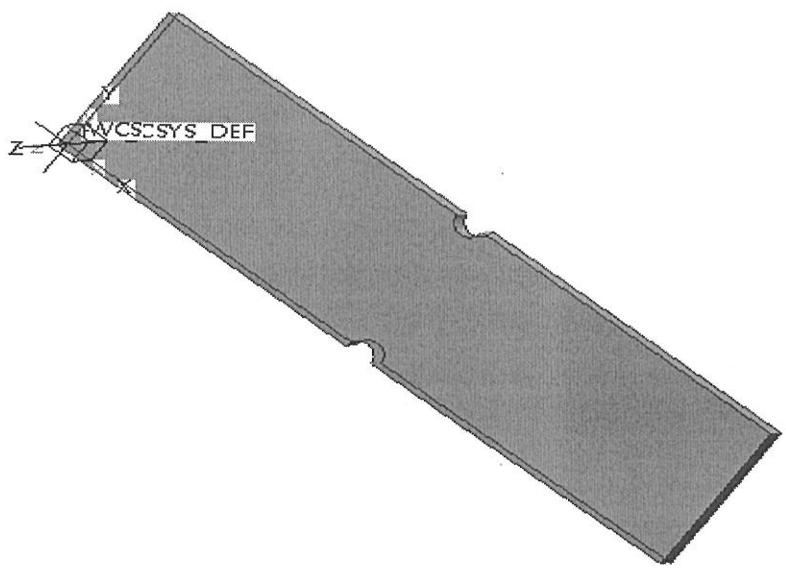


The following pop up menu appears

Select **New**

Another pop up selection box appears

Holding down the Control (CTRL) Key select one of the faces of the plate, and then select the other face

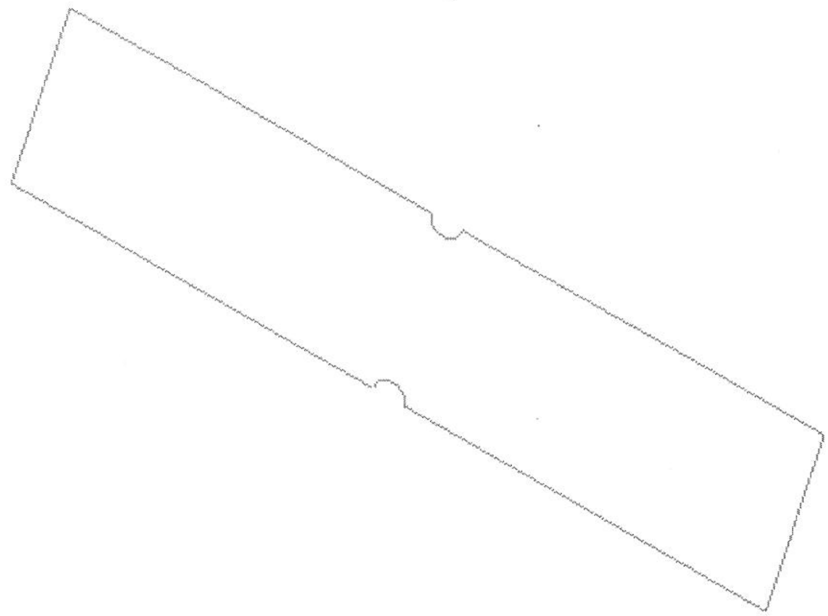


Click on **Ok** twice

Now, click on **Compress**

Click on **Shells only**

Click on **Show compress**



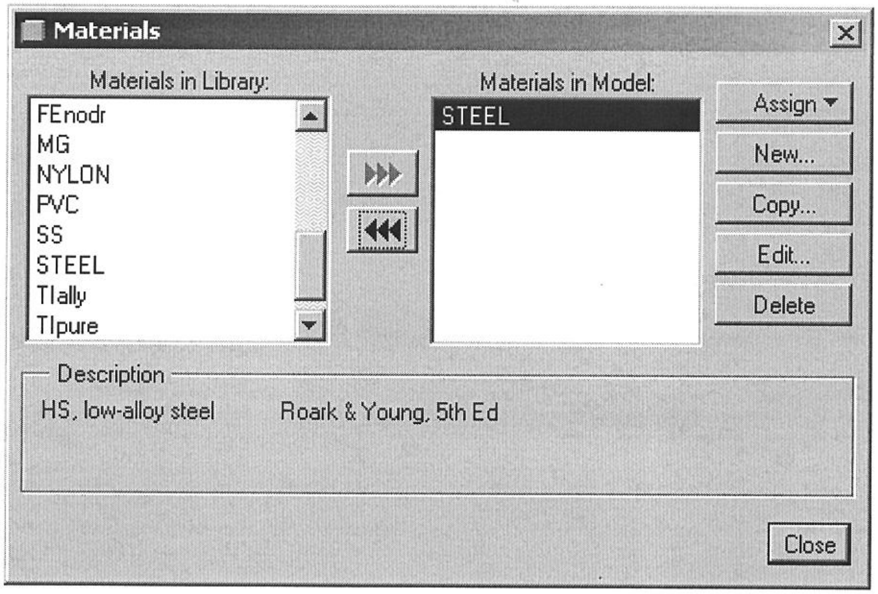
You should see the yellow color contour of the part on the screen as shown above

Select **Done>Done/ Return**

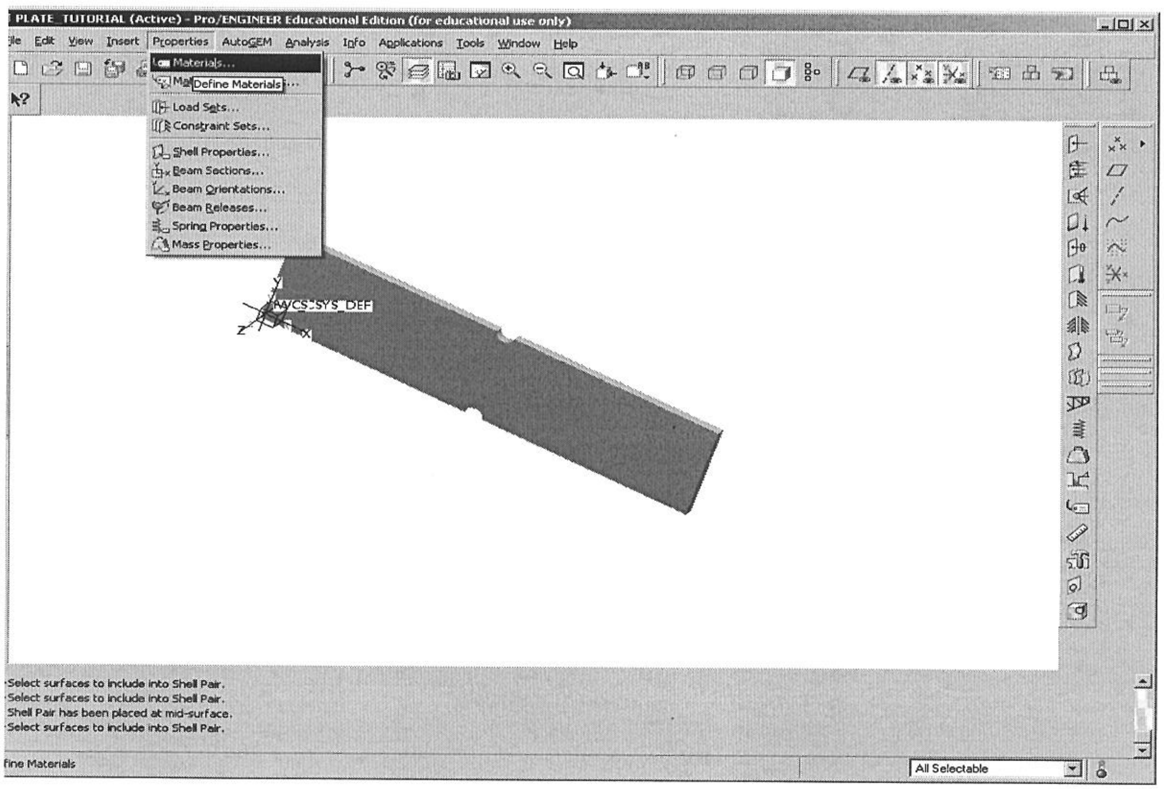
5. Assigning Materials properties to the Model

Materials:- This option selects the user define/select the material for the model. There are many standard materials listed as Materials in Library and the user can assign one of those standard materials to the part(s) and the user can also preview the properties of the standard material by clicking on the "Edit" button on the right hand side. User can also create a material of his/her own choice by clicking on the "New" button on the right hand side.

The plate is made of standard ASTM A-36 Steel (36 ksi yield strength). Select **Properties>Materials (see next page)** and the following window will come up Select **steel** from the left column and click on the arrow to move in under the materials in model column. Click on **Assign>Part** and select the plate. Click on the middle mouse button to accept. Click on **Close**

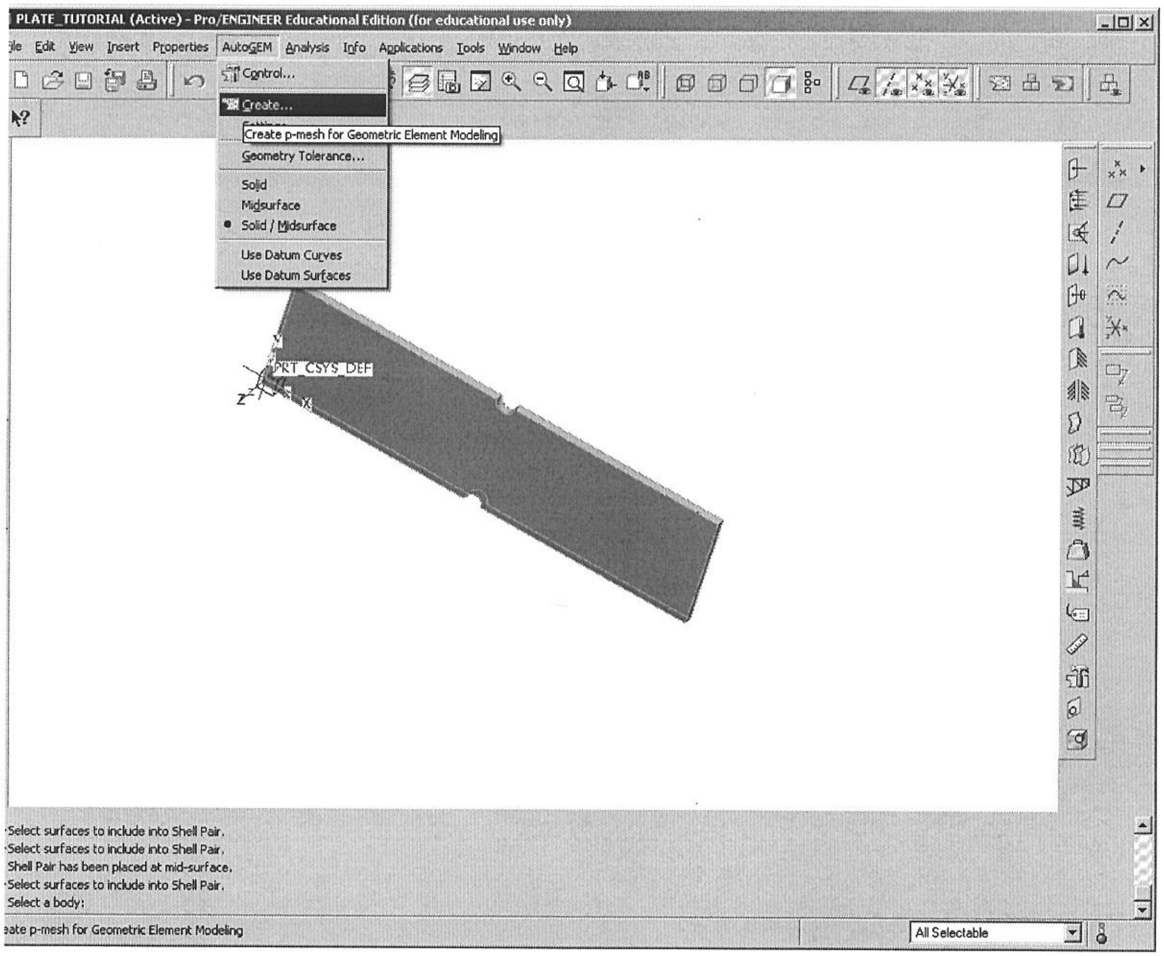


This graphics shows how to assign material properties to the model



6. Creating a Mesh on the Model

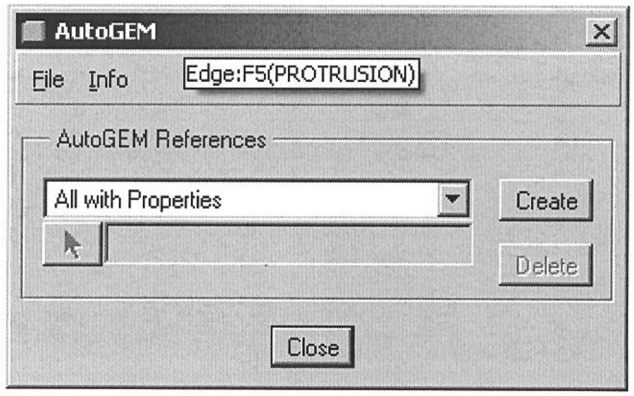
We will keep all default shell elements and settings that AutoGEM will create. Select **AutoGEM >Create**



AutoGEM:- This command can be used to review the mesh before running any analysis. The advantage of using AutoGEM is that if the model fails to create elements after performing AutoGEM operation, we can identify which portion of the model was creating the problem. Please note that it is not mandatory to perform AutoGEM before running an analysis. If AutoGEM is not performed, elements are going to be created during the running of an analysis.

After selecting create the following Selection Box appears

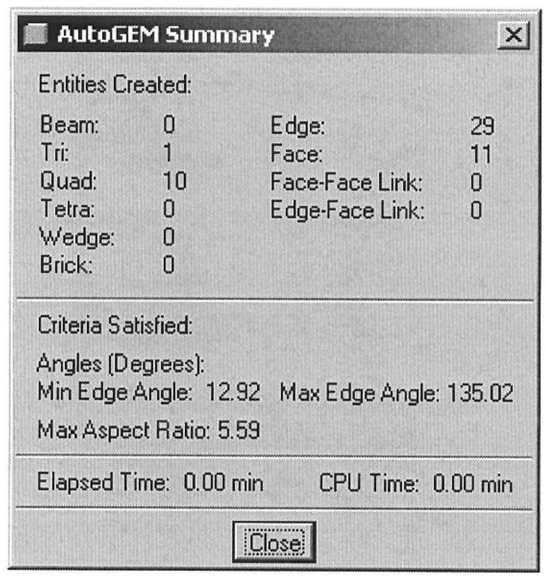
Click on **Create**



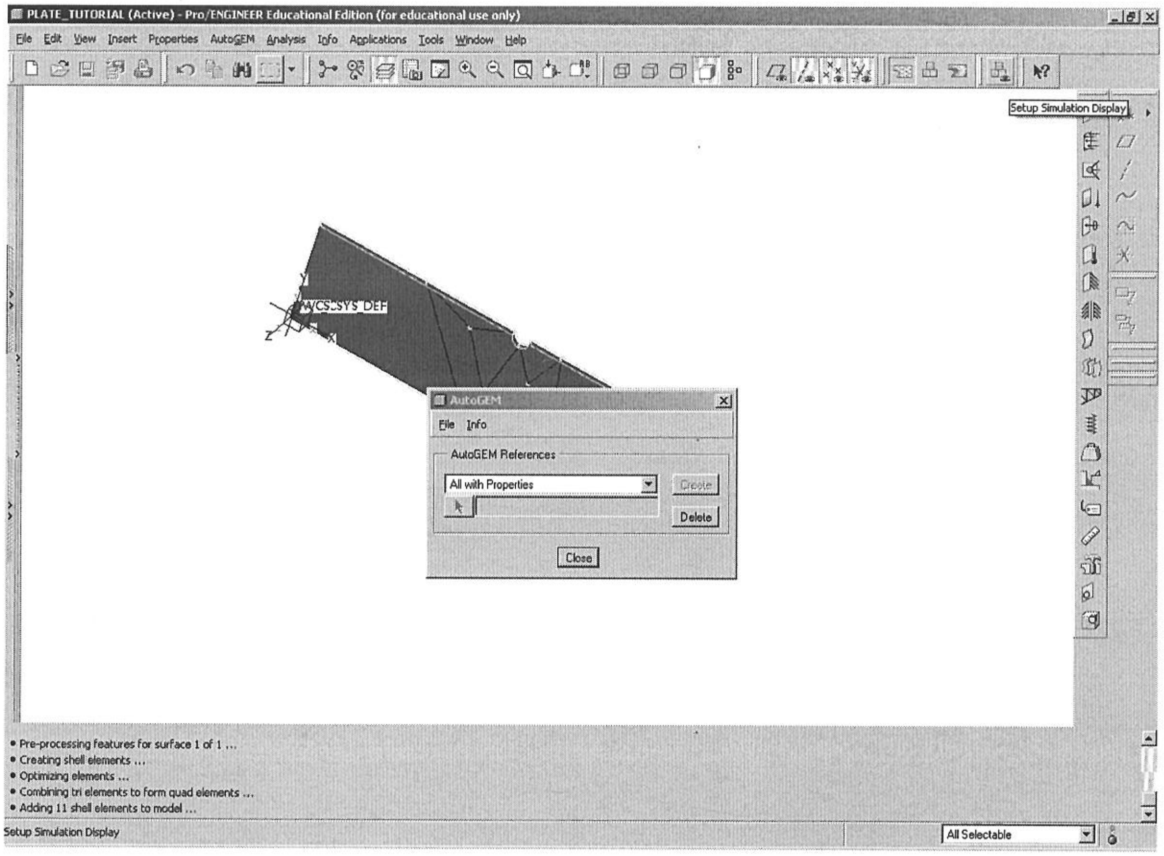
This confirmation of elements box should come up

Note that only one triangular and 10 quadrilateral elements have been created

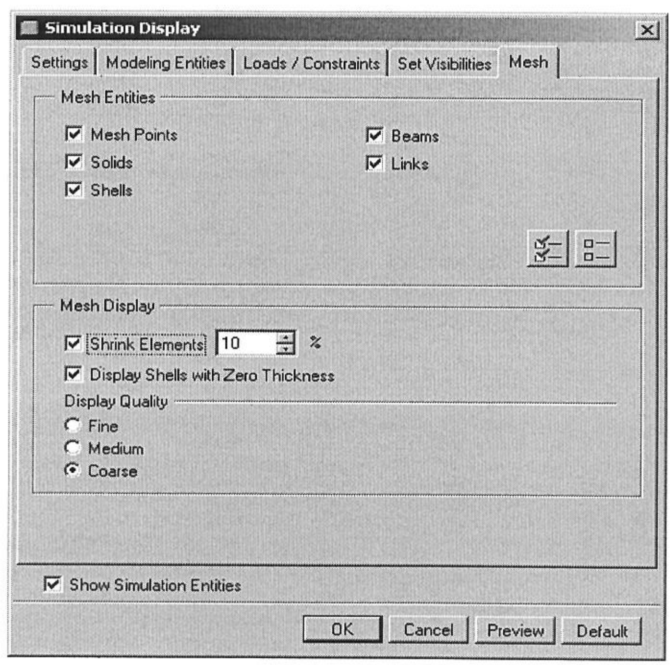
Select **Close**

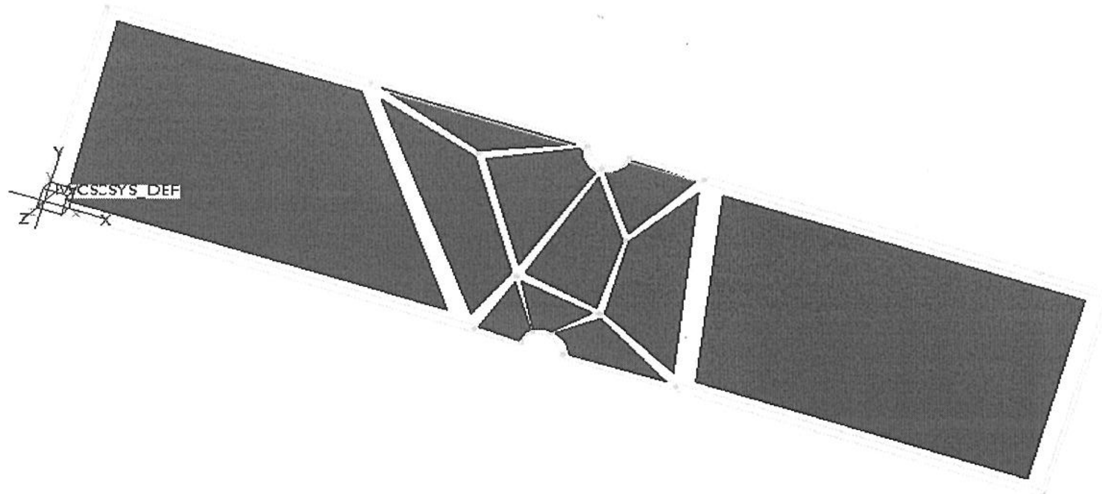


Click on the setup simulation display button on the toolbar



Use the display setting to shrink all elements to 10%, Click on **Preview**

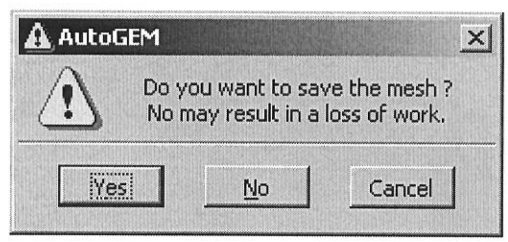




This is the mesh for the model

Now, unselect shrink elements options in the simulation display dialog box and click **Ok,** Click **Close** on the AutoGEM box

Click **Yes** to save the mesh



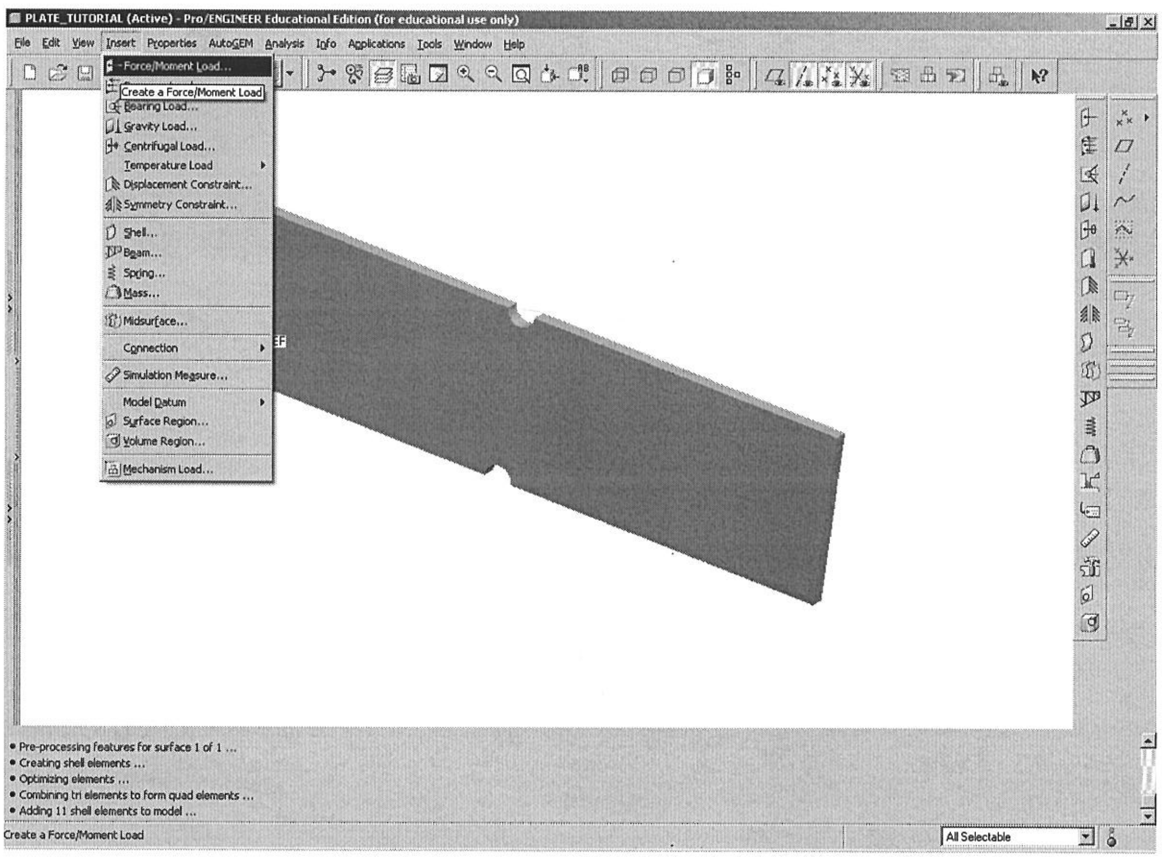
7. Boundary Conditions on the Model

<u>*Constraints/Loads*</u>:- This menu is used to define loads and constraints for the model. Please note that the default coordinate system is WCS and the user has the option to either select a pre-defined coordinate system or define and select a coordinate system on the fly. It is advisable to "preview" loads before hitting OK button just to make sure everything is right.

7.1. Loads

The end load has 500 Lbs in both the X and Y directions. We will set this load as end edge load.

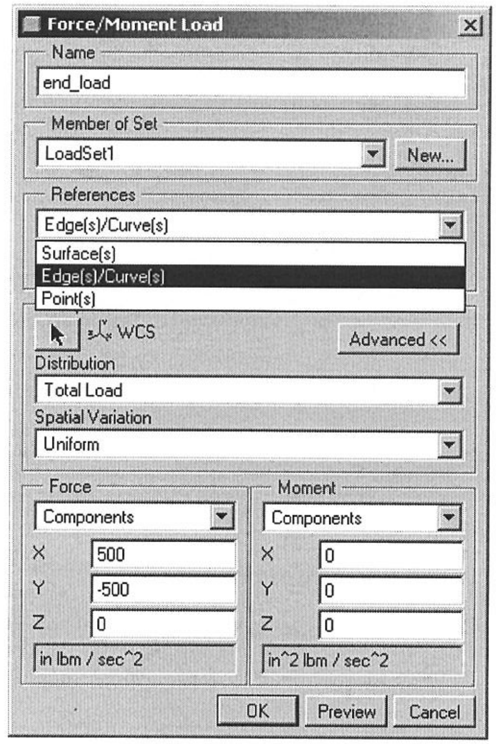
Select **Insert>Force/Moment Load**



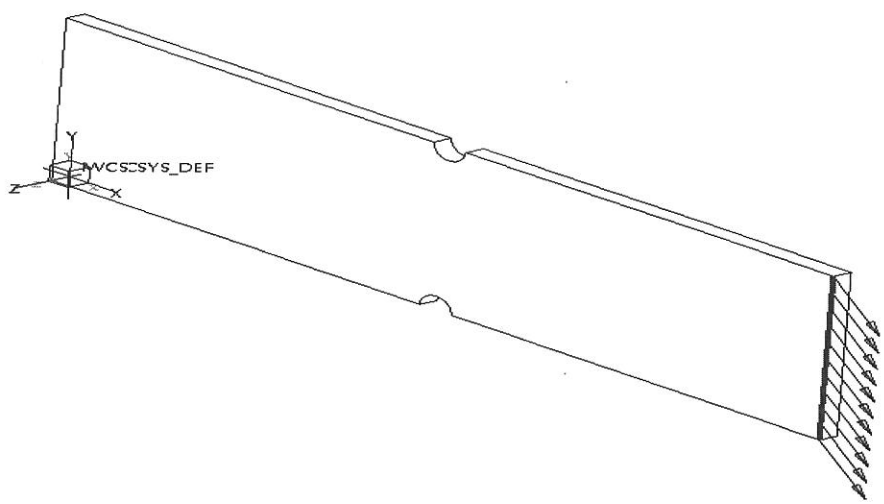
The following Dialog box appears:
Change the name of the constraint set, select Edge/Curve, and use the following force components: 500 and -500 in the X and Y direction
Select the arrow on the references part of the Force/Moment dialog box and select the right vertical edge from the model

Click Preview to see how the constraints are applied on the model

Click **Ok**



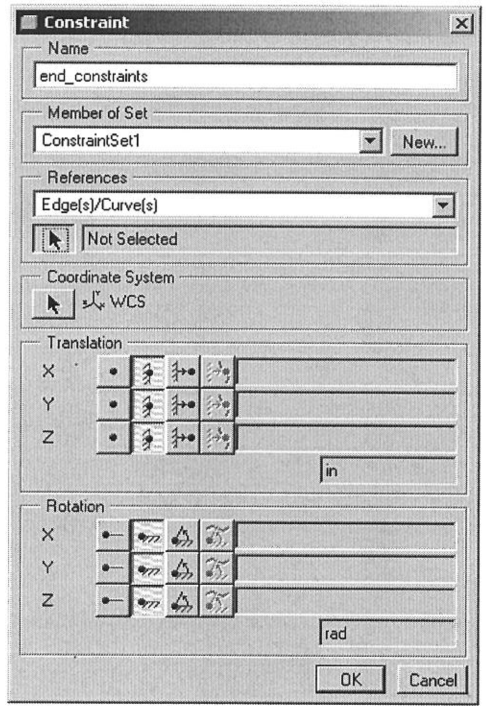
Your model should look like the following graphic



Applied forces diagram

7.2 Constraints

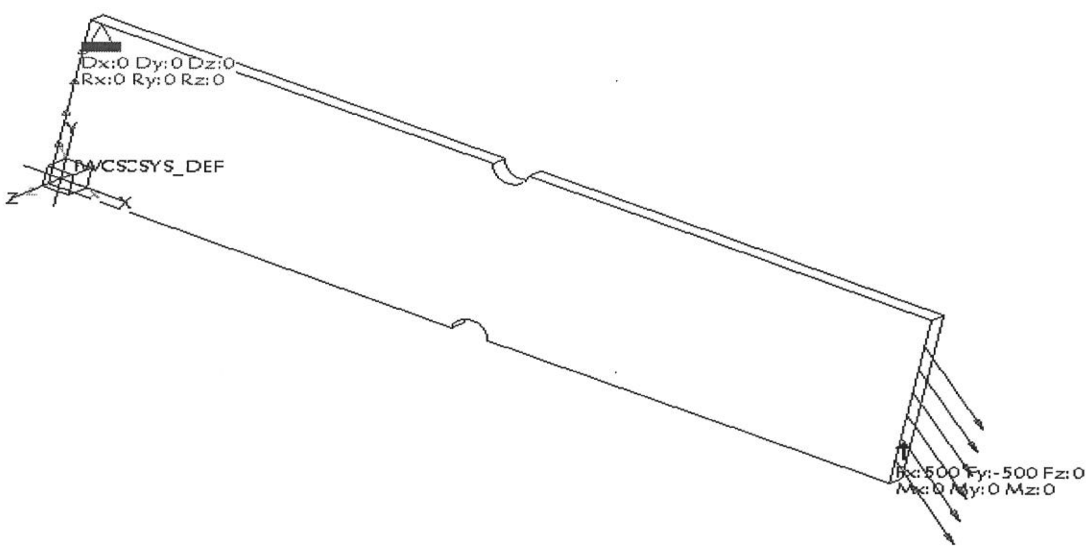
The plate is a cantilever beam with the left end being fixed. To apply the end constraints, select **Insert>Displacement Constraints**



Change the name of constraints, select the arrow on references and select the left vertical edge fro the model.

Keep all DOF fixed (translations and rotations)

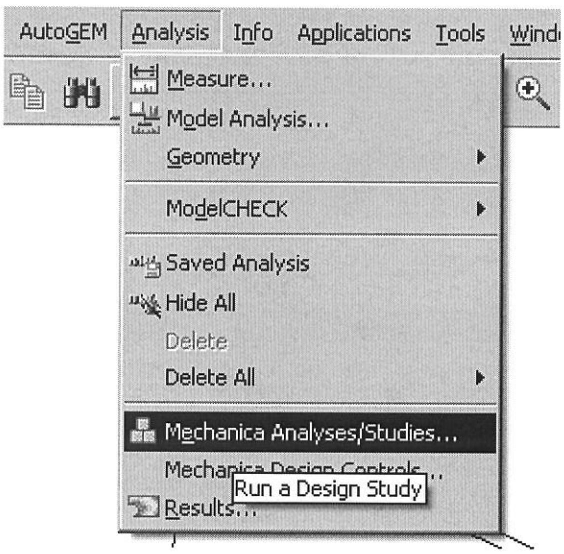
The plate FEA model should look as follow



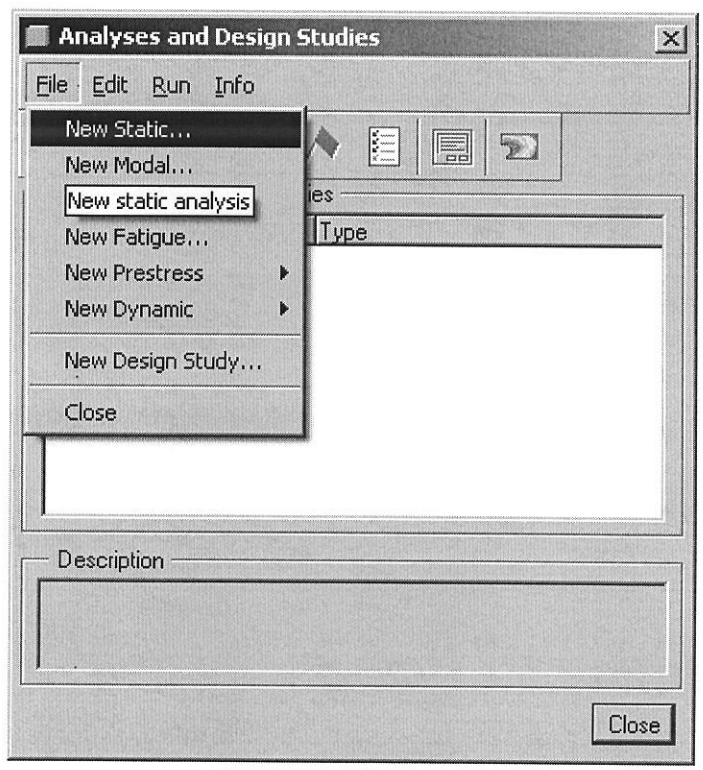
8. Static Analysis

We are going to create a quick check no convergence analysis named **Static_1** to make sure that we can get to a solution. Results are not important at this point, we just want to make sure that the model can get to a solution

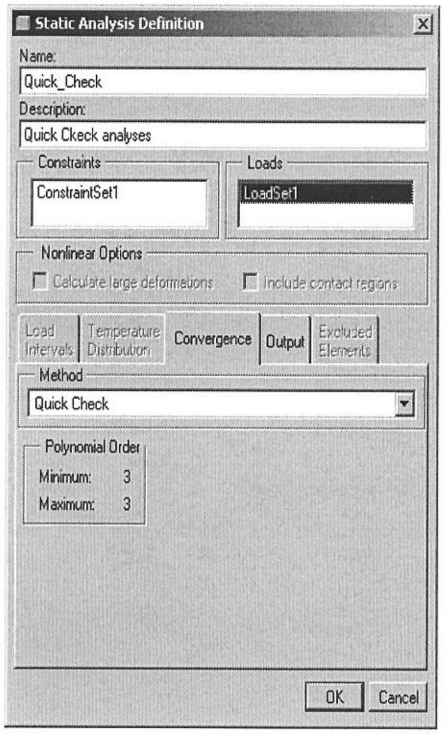
Select Analysis>Mechanica Analyses/ Studies



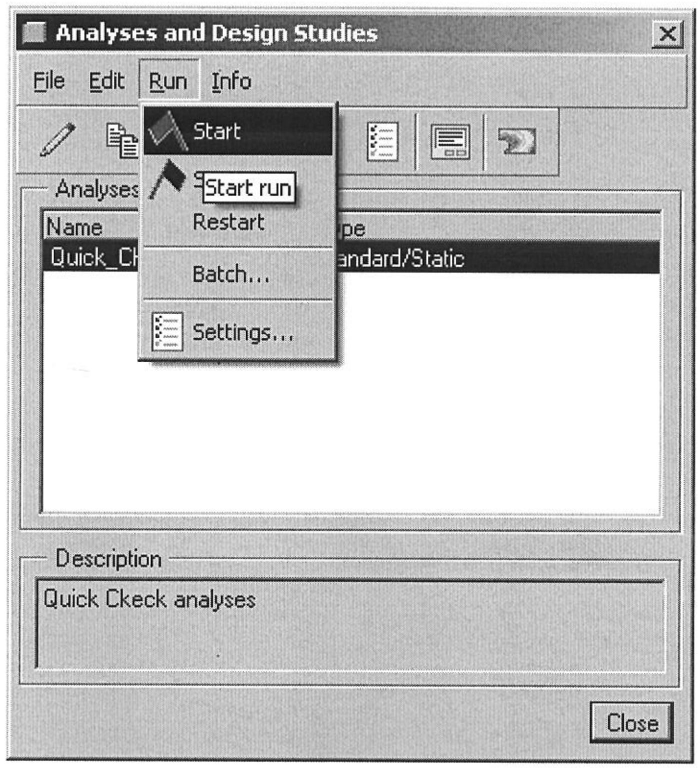
On the Analyses and Design Dialog Box select New Static and enter the information as shown below

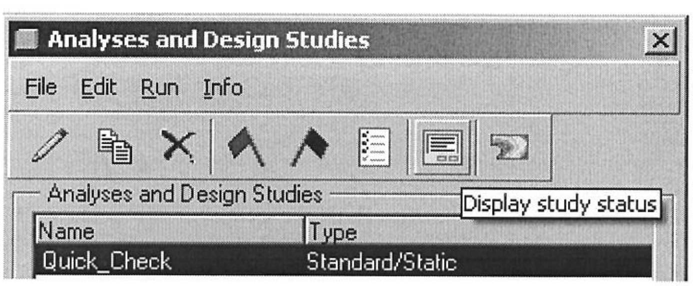


Select **Ok**

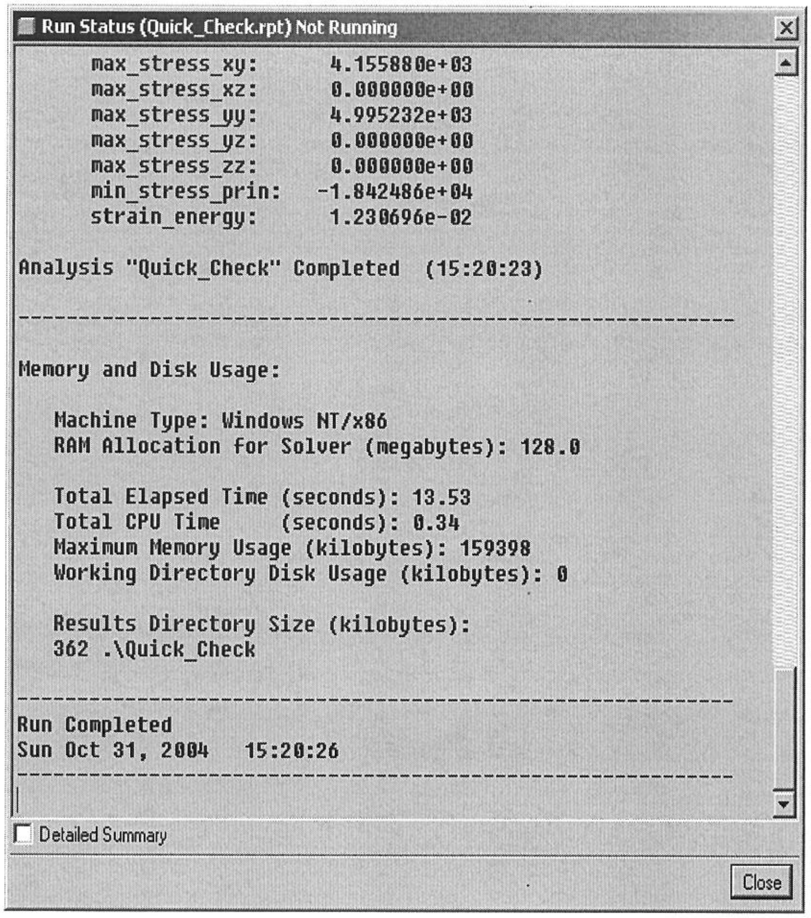


Click on Stat, answer yes to activate the error detection





Click on Display study status to see if we can get a solution

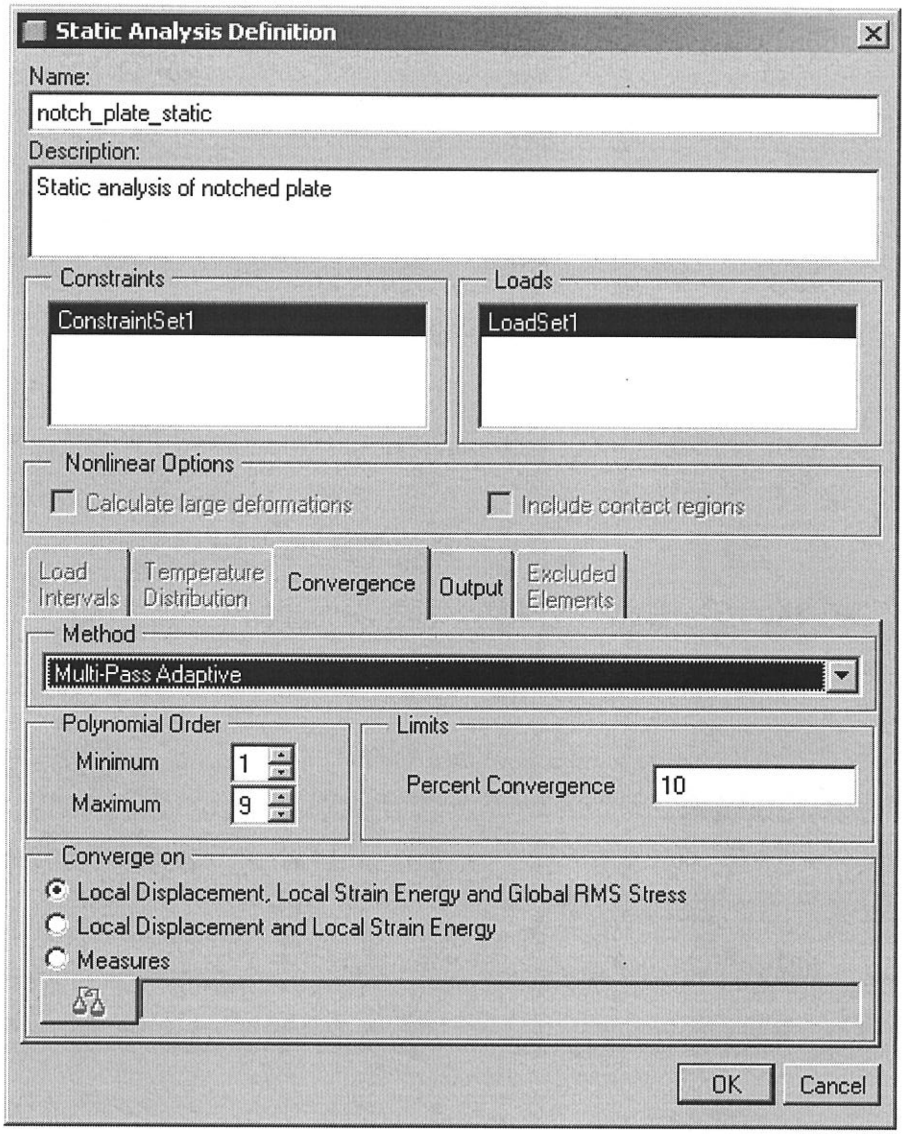


Click **Close**

Now, we are going to create a **Multi Pass Adaptive** convergence

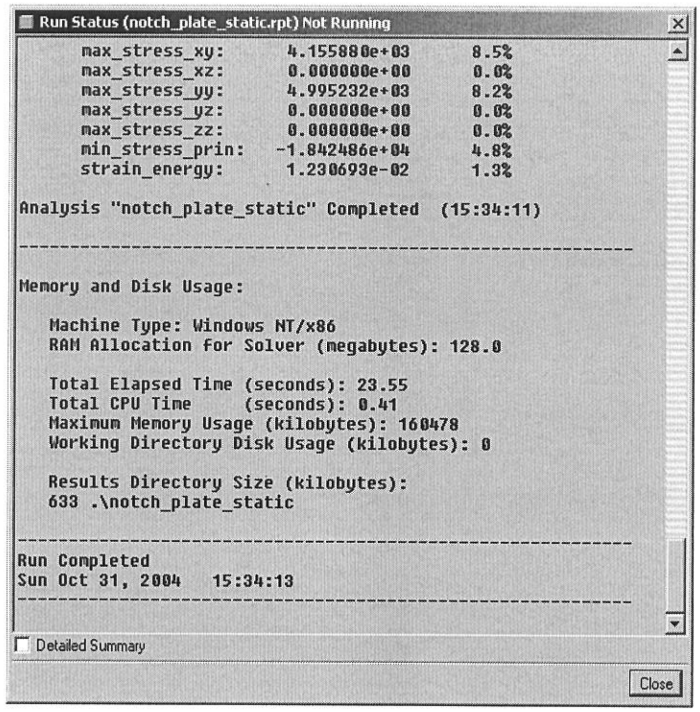
Edit the quick check analysis (right mouse click on it and select edit)

Enter the information as seen in the box below

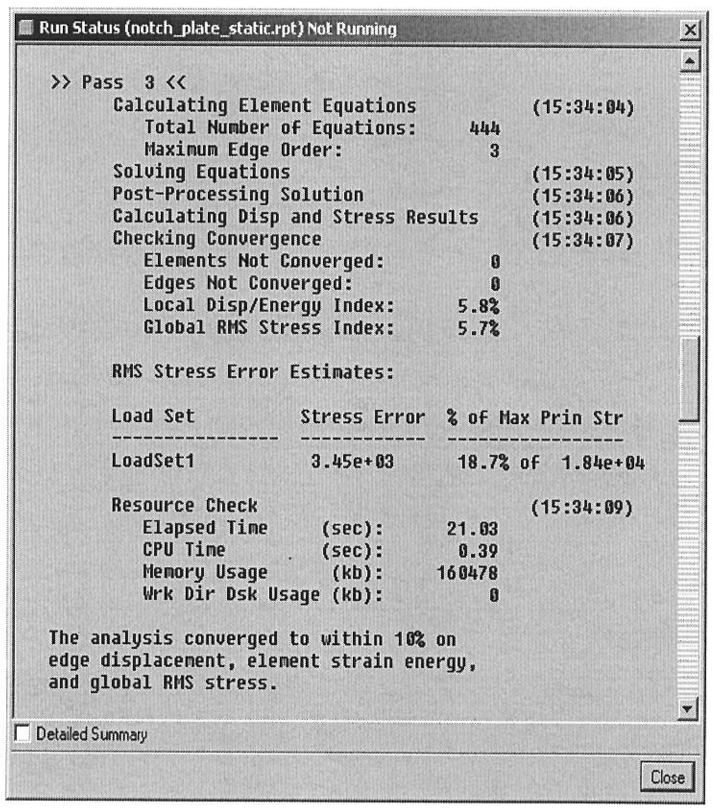


Select **Ok**, then run the analysis. Answer Yes to activate the error detection. Click on Display study status to see if we can get a solution

The solution converges within 10% in 3 passes

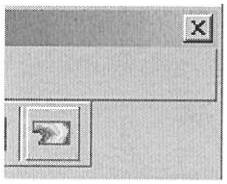


```
Run Status (notch_plate_static.rpt) Not Running
      max_stress_xy:       4.155880e+03       8.5%
      max_stress_xz:       0.000000e+00       0.0%
      max_stress_yy:       4.995232e+03       8.2%
      max_stress_yz:       0.000000e+00       0.0%
      max_stress_zz:       0.000000e+00       0.0%
      min_stress_prin:    -1.842486e+04       4.8%
      strain_energy:       1.230693e-02       1.3%

Analysis "notch_plate_static" Completed  (15:34:11)

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

Memory and Disk Usage:

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

   Total Elapsed Time (seconds): 23.55
   Total CPU Time     (seconds): 0.41
   Maximum Memory Usage (kilobytes): 160478
   Working Directory Disk Usage (kilobytes): 0

   Results Directory Size (kilobytes):
   633 .\notch_plate_static

------------------------------------------------------------
Run Completed
Sun Oct 31, 2004   15:34:13
------------------------------------------------------------
Detailed Summary
Close
```



```
Run Status (notch_plate_static.rpt) Not Running

>> Pass  3 <<
      Calculating Element Equations              (15:34:04)
         Total Number of Equations:     444
         Maximum Edge Order:              3
      Solving Equations                          (15:34:05)
      Post-Processing Solution                   (15:34:06)
      Calculating Disp and Stress Results        (15:34:06)
      Checking Convergence                       (15:34:07)
         Elements Not Converged:          0
         Edges Not Converged:             0
         Local Disp/Energy Index:      5.8%
         Global RMS Stress Index:      5.7%

      RMS Stress Error Estimates:

      Load Set           Stress Error  % of Max Prin Str
      ----------------   ------------  -----------------
      LoadSet1             3.45e+03      18.7% of  1.84e+04

      Resource Check                             (15:34:09)
         Elapsed Time      (sec):       21.03
         CPU Time          (sec):        0.39
         Memory Usage       (kb):      160478
         Wrk Dir Dsk Usage  (kb):           0

The analysis converged to within 10% on
edge displacement, element strain energy,
and global RMS stress.
Detailed Summary
Close
```

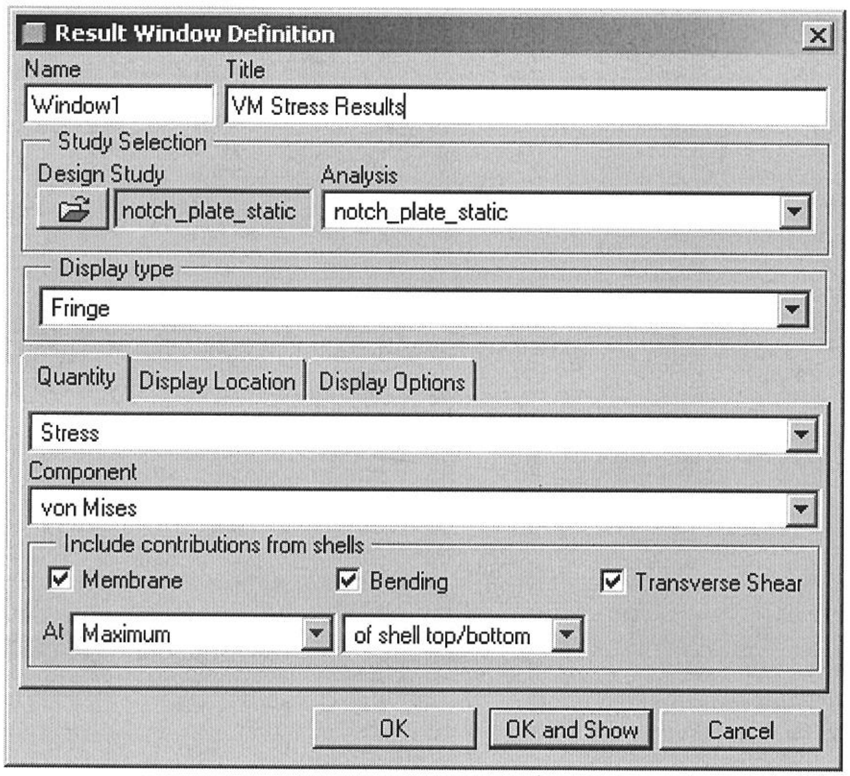
8.1 Results of Static Analysis



Create a Von Mises stress definition sheet as follows:

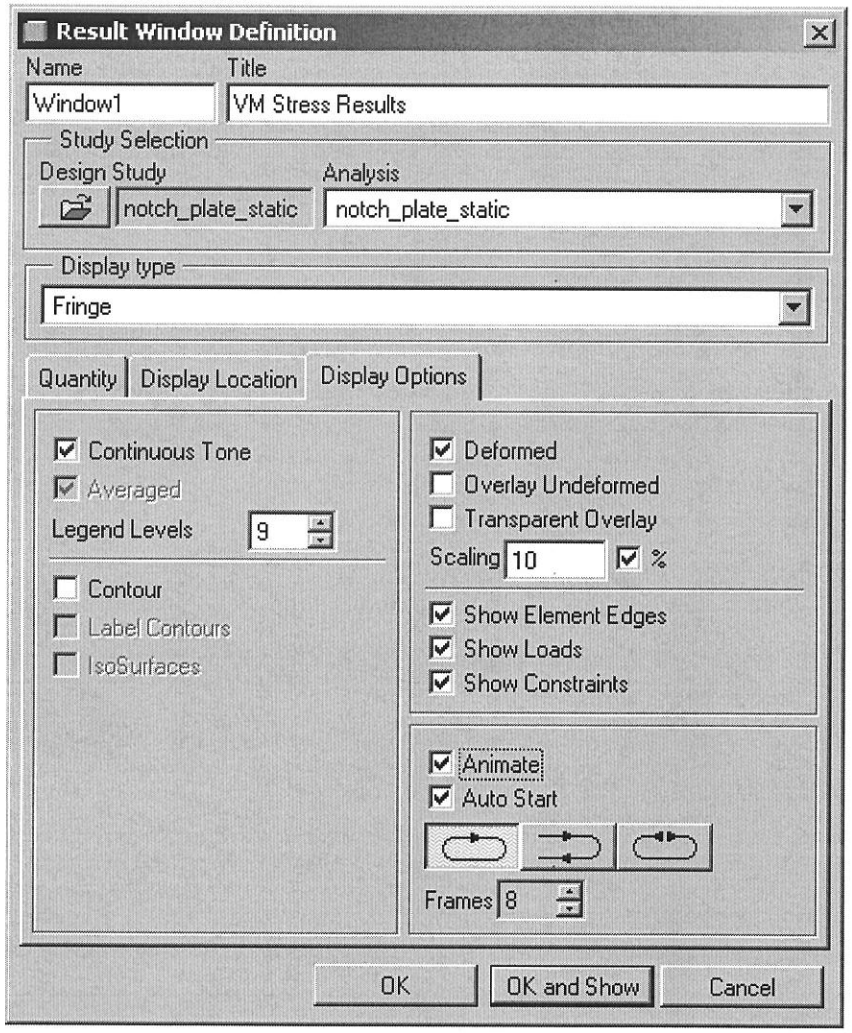
Click on the review result button

Fill out the result window as follows:

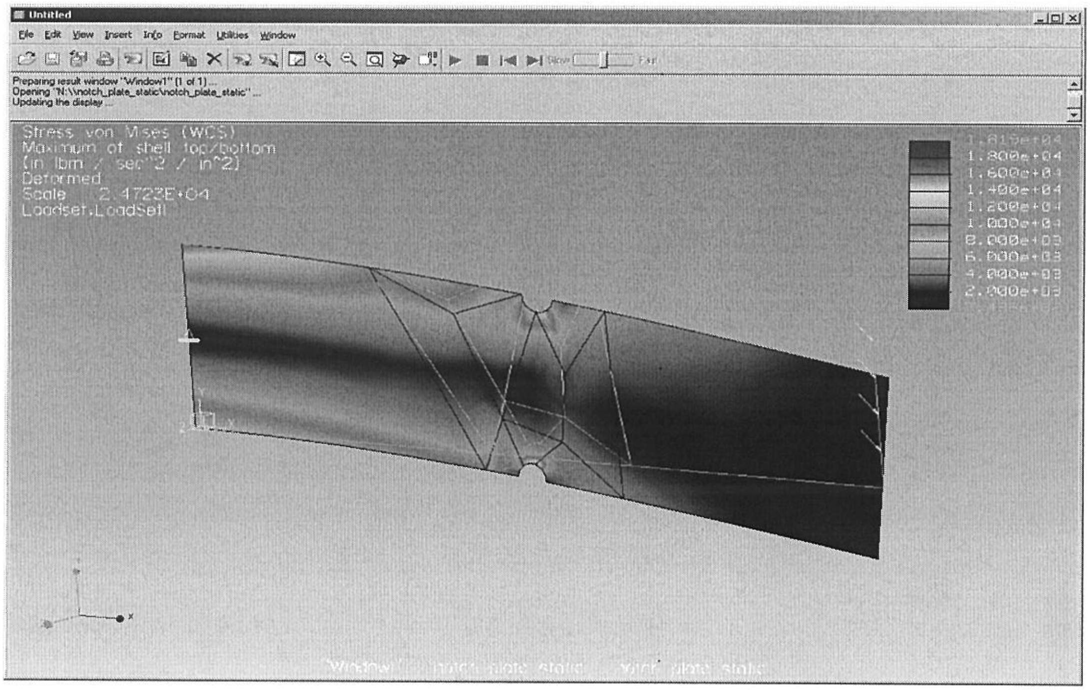


Click on Display Options Tab

Check out the Deformed, animate and continuous Tone boxes



Click **Ok and Show**



Static Analysis Von Mises Stress Results

Select **File>Exit Results>NO**

As we can see in the Von Mises Stress Analysis, the max Stress is 1.82×10^4 Psi, the material has an Sy $=3.6 \times 10^4$, as it is a static analysis we can use a security factor of 1.5, so that our maximum allowed (design) stress should be (36/1.5)-18.2 Ksi.

Then our design stress is 24-18.2 = 5.8 ksi, this is an important design criteria. If this number is grater than 1 and less than 10 ksi, then leaves us some room for weight optimization using this material, otherwise we would have to change the material for another stronger.

8.2 Local Sensitivity Study

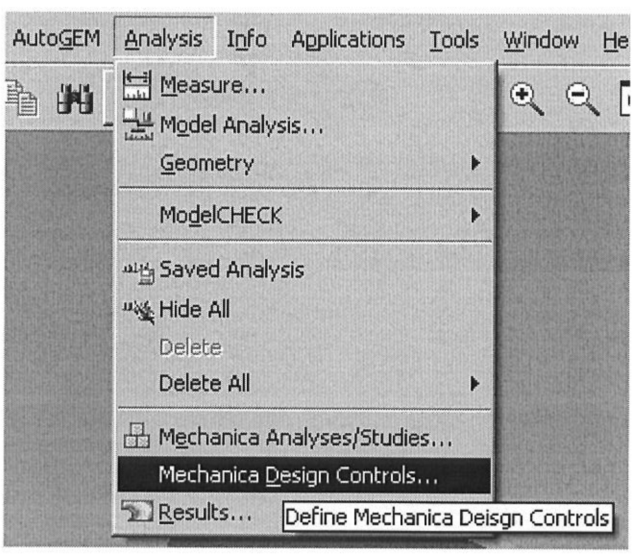
The objective of a local Sensitivity study is to look at small changes of the measures (Von Mises stresses) for small independent variations of each design parameter used (+-1 to 2%).

Our design parameters for this study, as defined earlier are: plate thickness, the cut location with respect to the left edge and the cut radius.

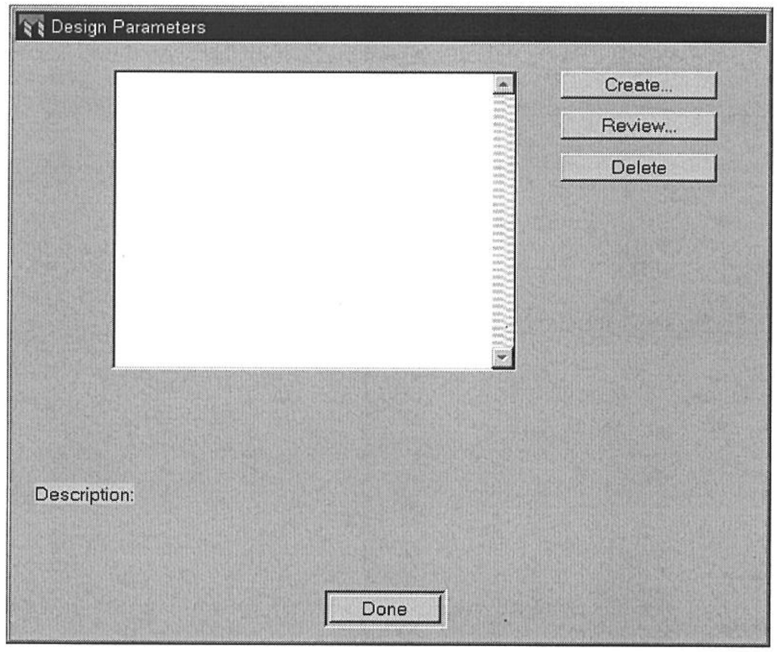
Design Controls:- This command can be used to define design controls which can used later in a design study like global sensitivity, optimization etc. Design parameter can be Dimension/Pro/Engineer parameter/section dimension. Section dimension option can be used only if there some sections already defined in the model like a beam section.

Procedure:

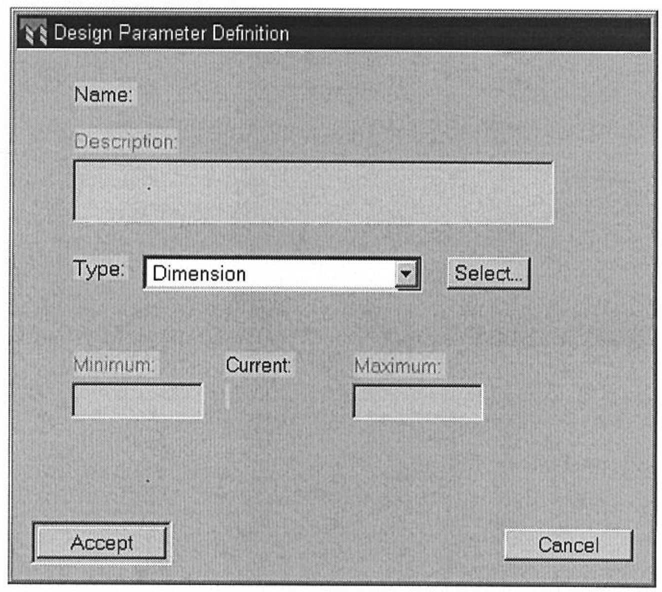
Select: **Analysis>Mechanica>Design Controls**



Select Design Parameters

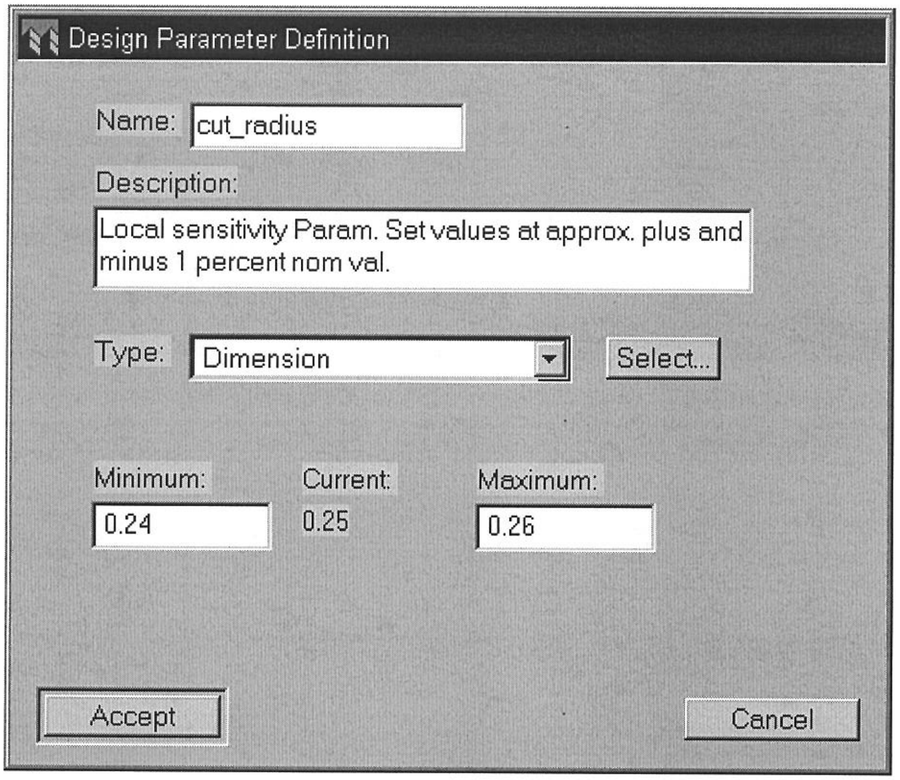


Select: **Create**



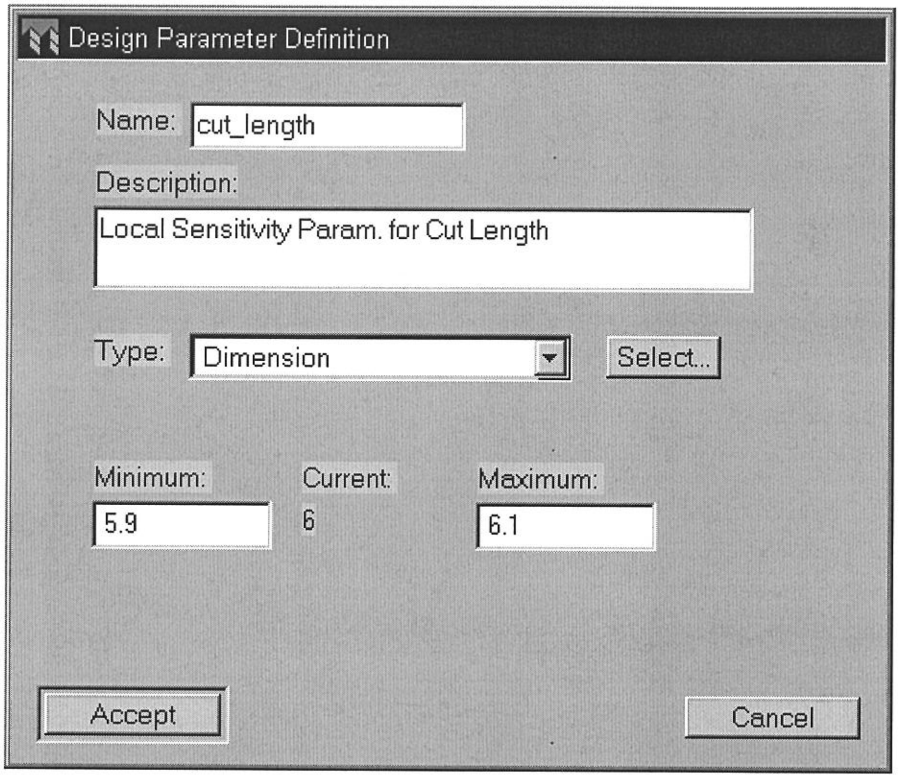
Click on **Select**

Click on the model and select the cut (notch). Select the 0.25 radius of the cut and enter the text shown below.

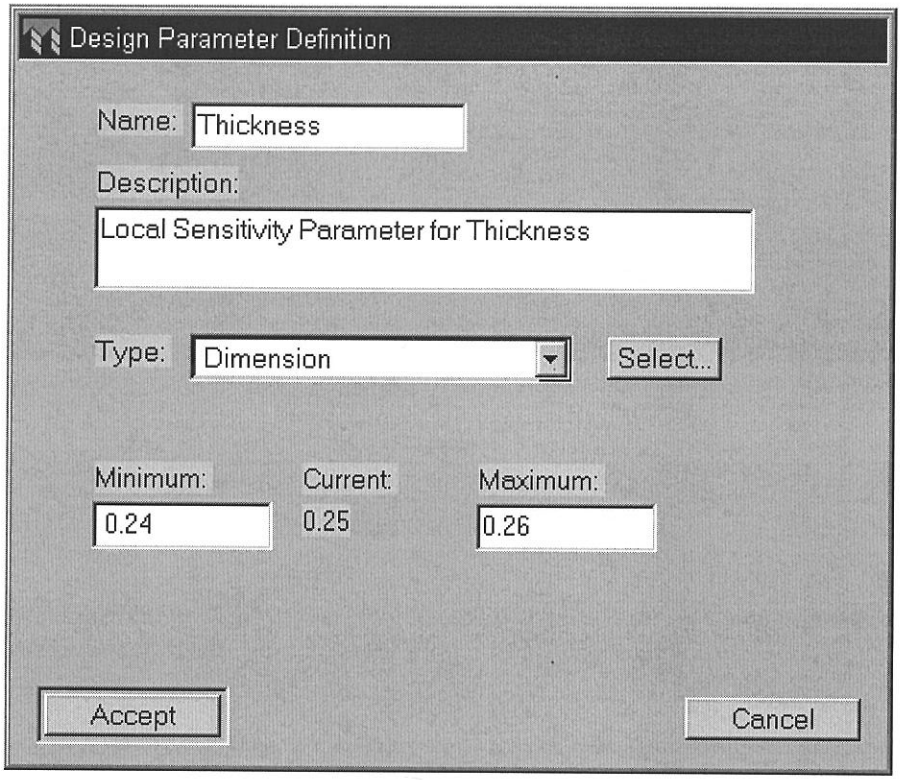


Click on **Accept**

Repeat the same procedure for the cut_length and plate thickness (see next graphics for values to be entered)



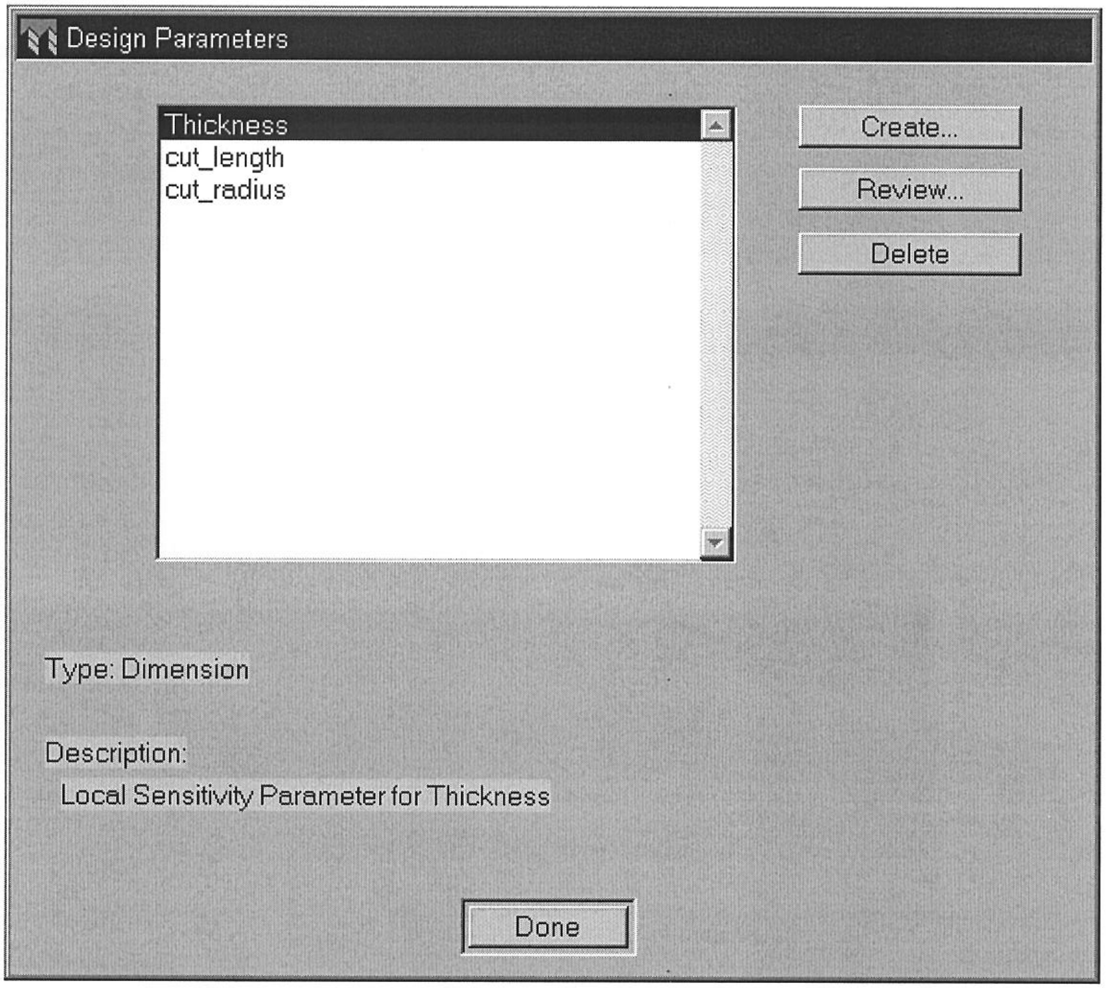
For the cut length, select the 6” dimensions and enter 5.9 and 6.1 for the minimum and maximum values



For the thickness, select the 0.25 inch dimension and enter 0.24 and 0.26 for minimum and maximum values as shown below.

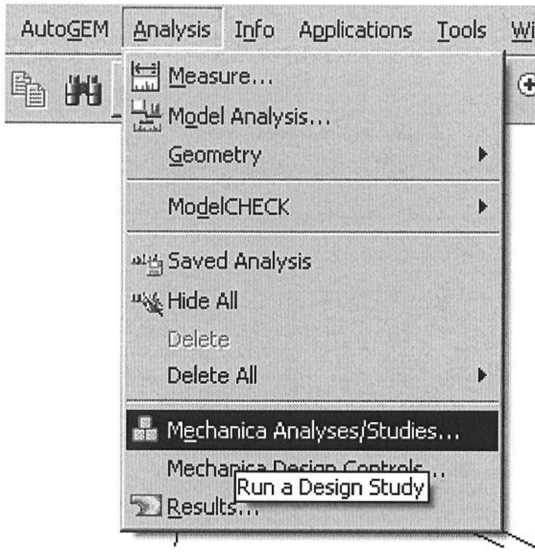
Click on **Accept.**

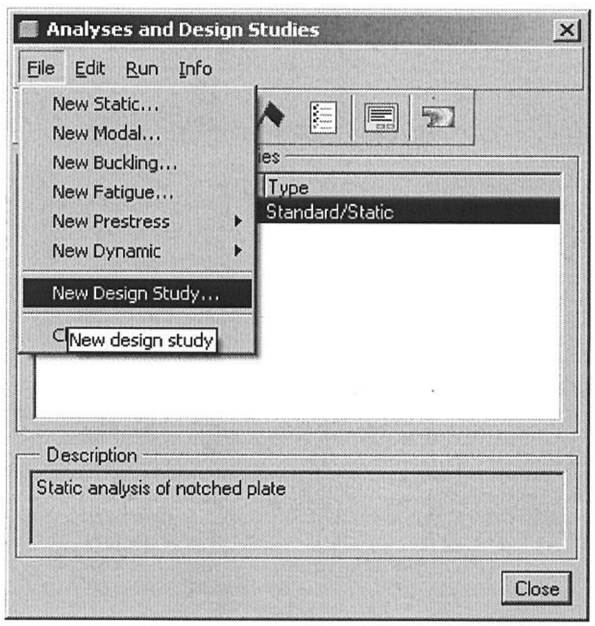
Once the parameters have been defined, the design parameter window should look as follows



Click on **Done>Done/Return**

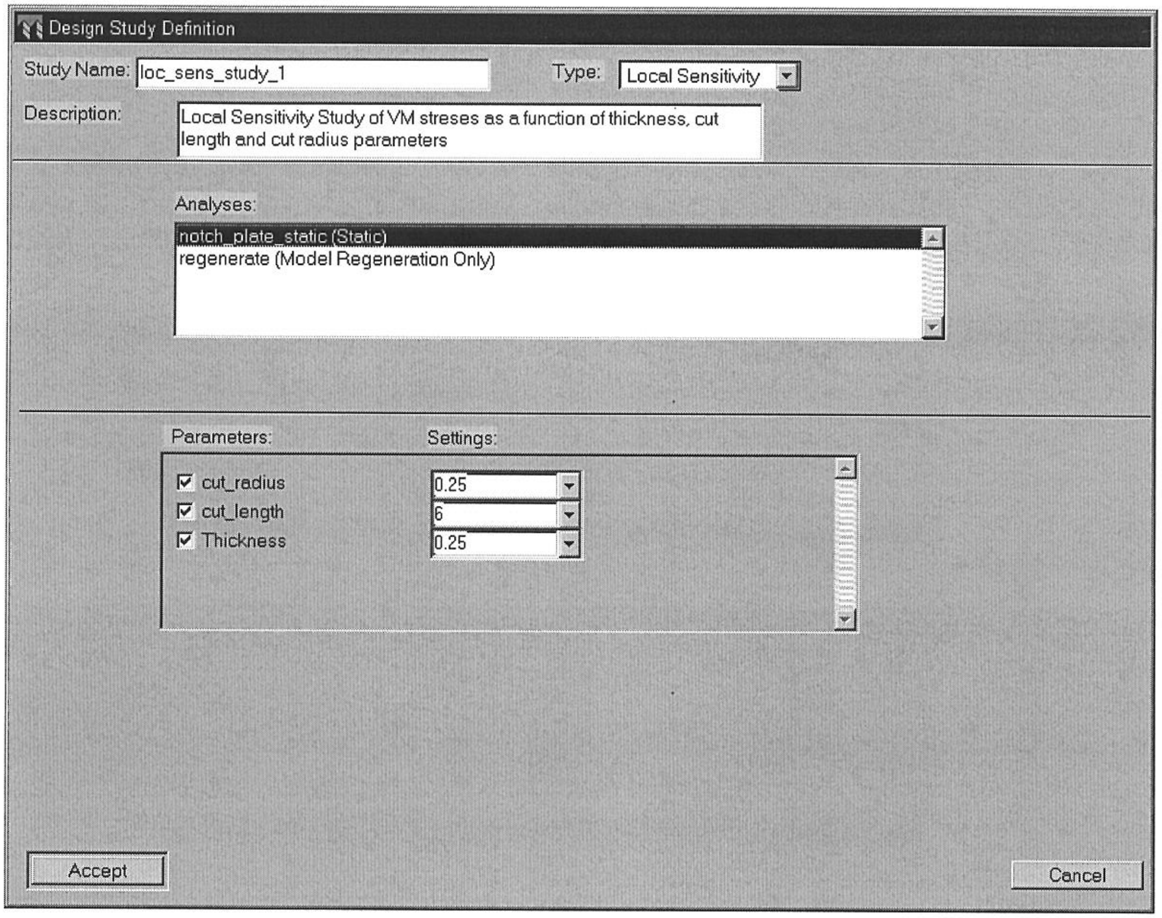
Create a design study:





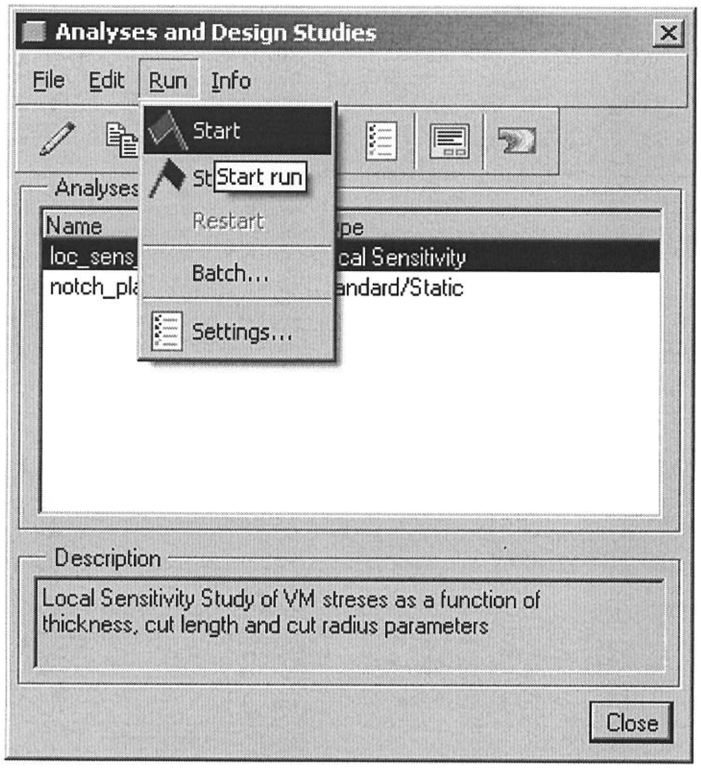
Make the following selection for the study

Select **notch_plate_Static** under Analysis



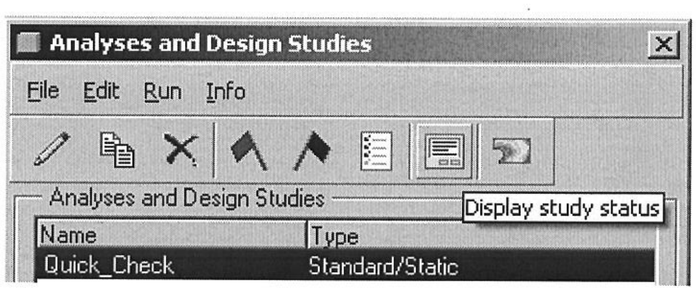
Select **Accept**

Run the local sensitivity study created (with the design study selected) click on **Start**

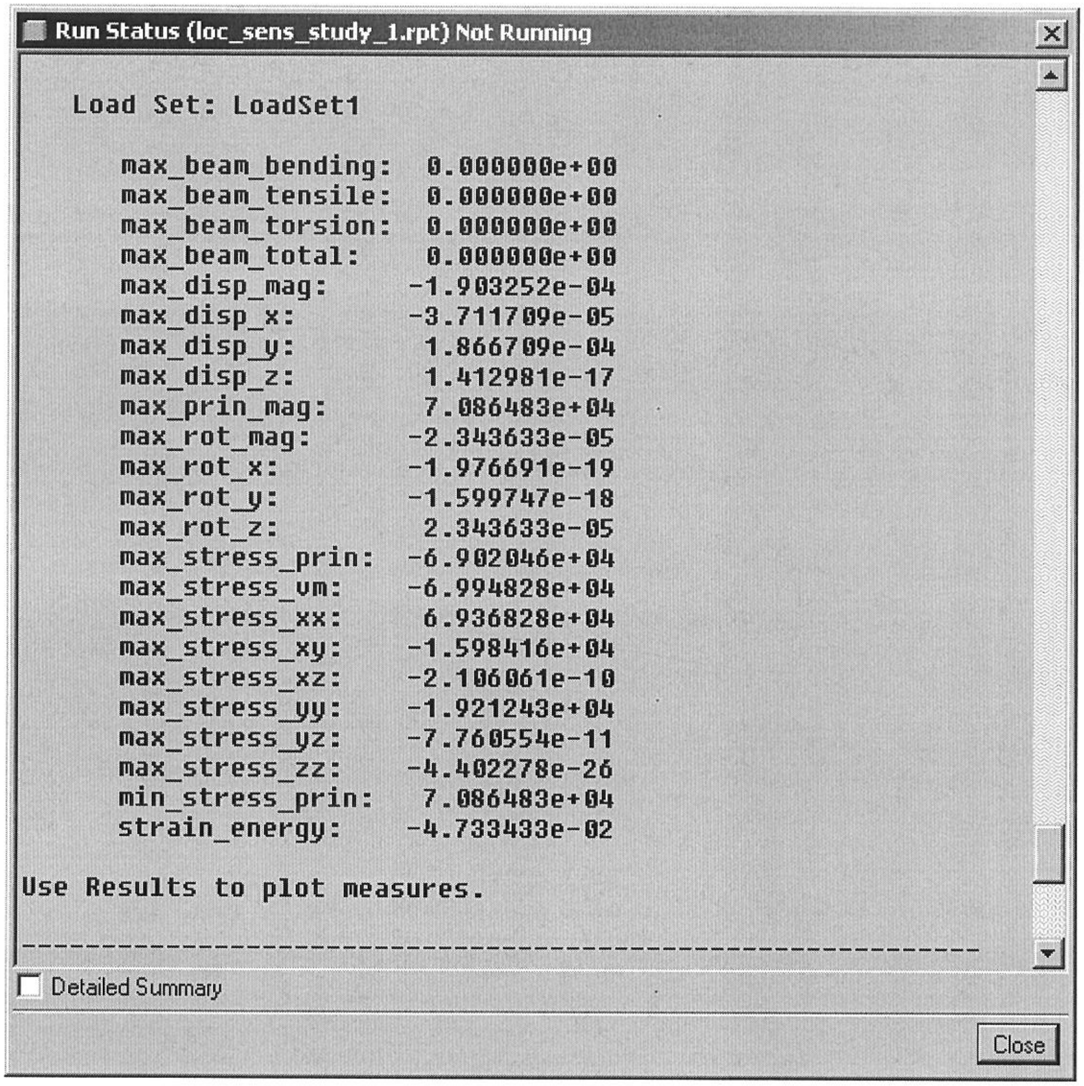


Click on **Yes** to activate the error detection

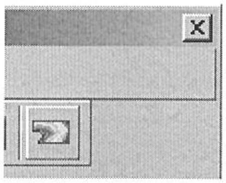
It takes approximately three minutes to complete the analysis. Check the status window to see the results



Final Results



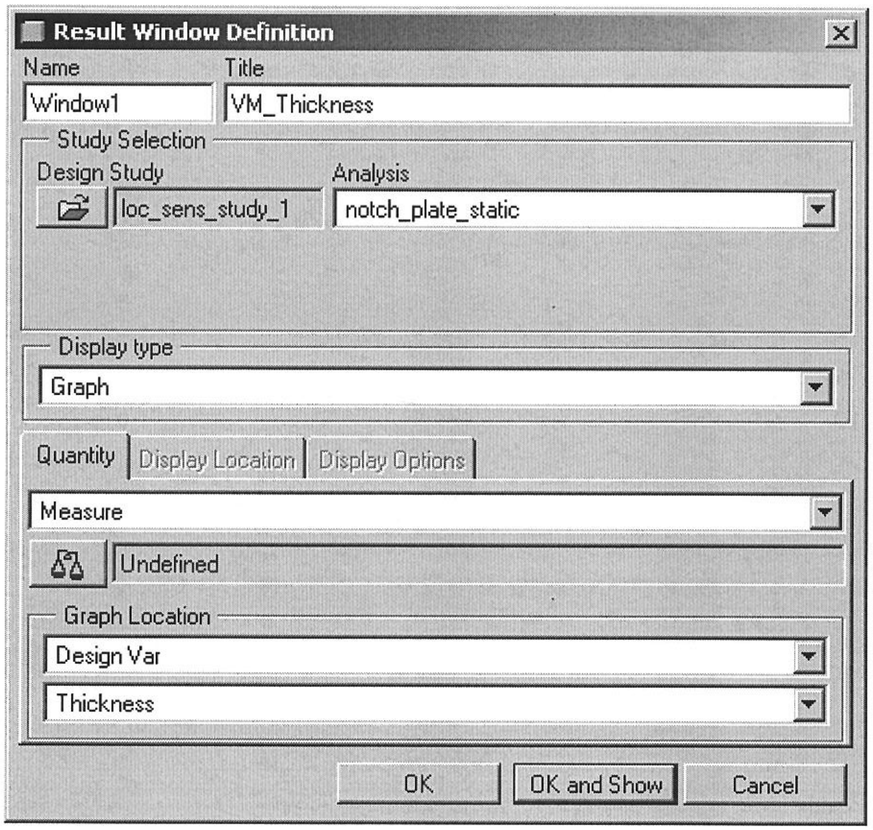
Click: **Close**



Click on the review result button

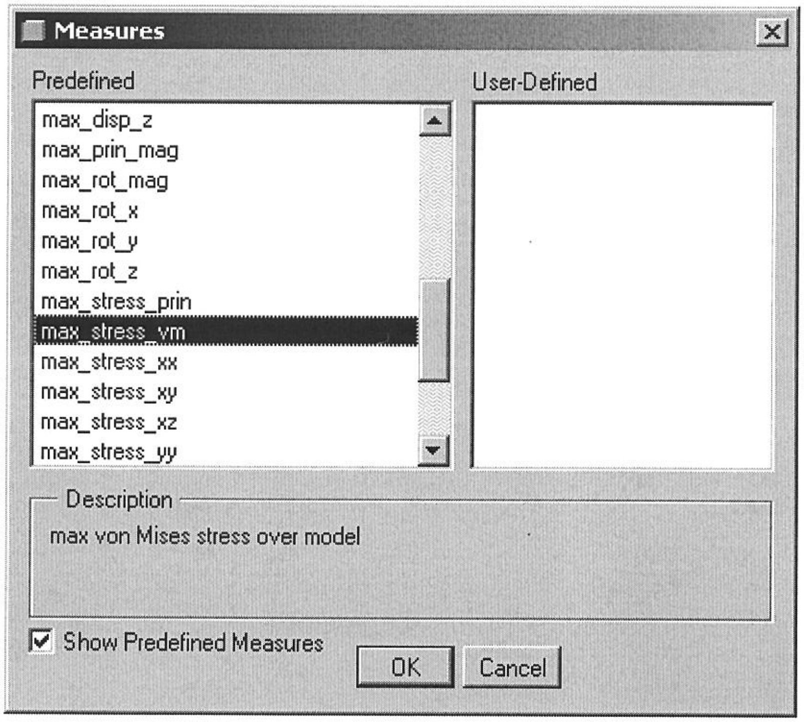
The following box appears

We are going to create results graph by plotting each design parameter versus the Von Mises Stress measure. We are going to create three windows, one for each parameter previously defined. Name the first result window as follows and fill out and make appropriate selections as shown

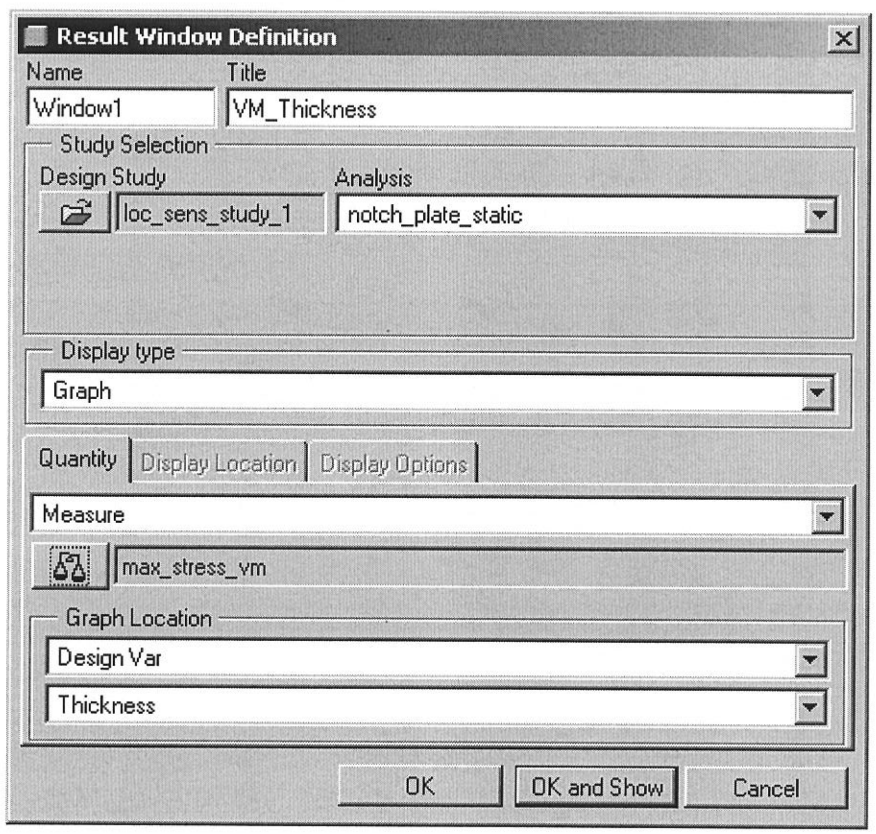


Click on **Ok**

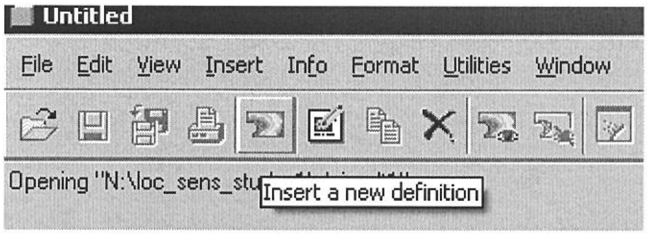
Select max_stress_vm as a measure and select Design Var from the graph location, make sure that the thickness variable has been selected



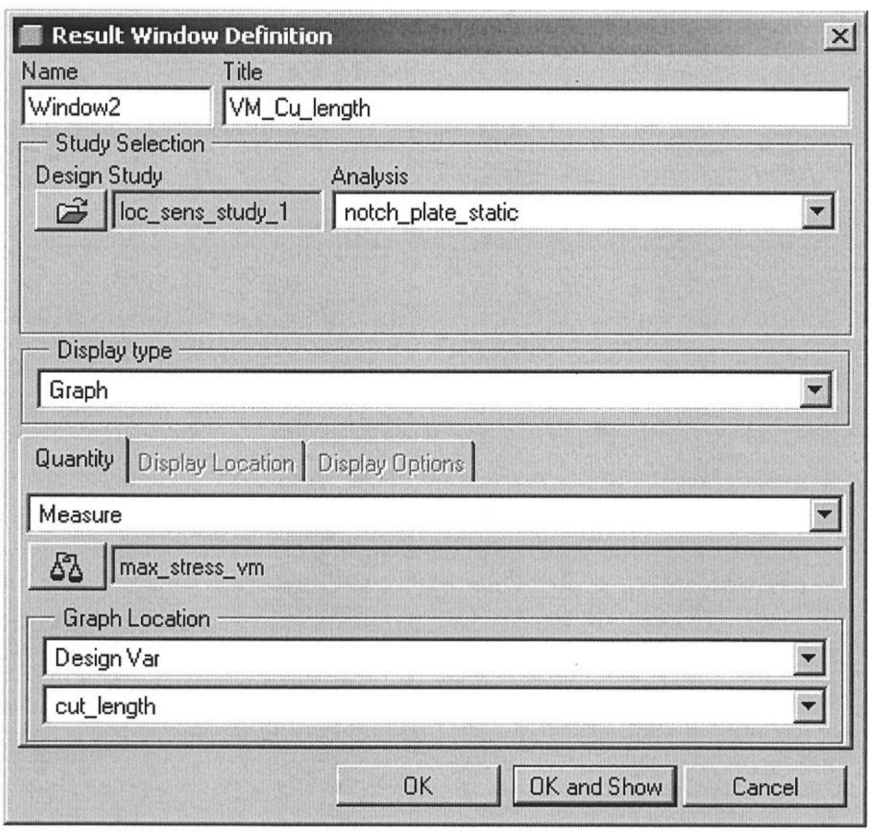
Click **Ok**



Click on the insert a new definition button

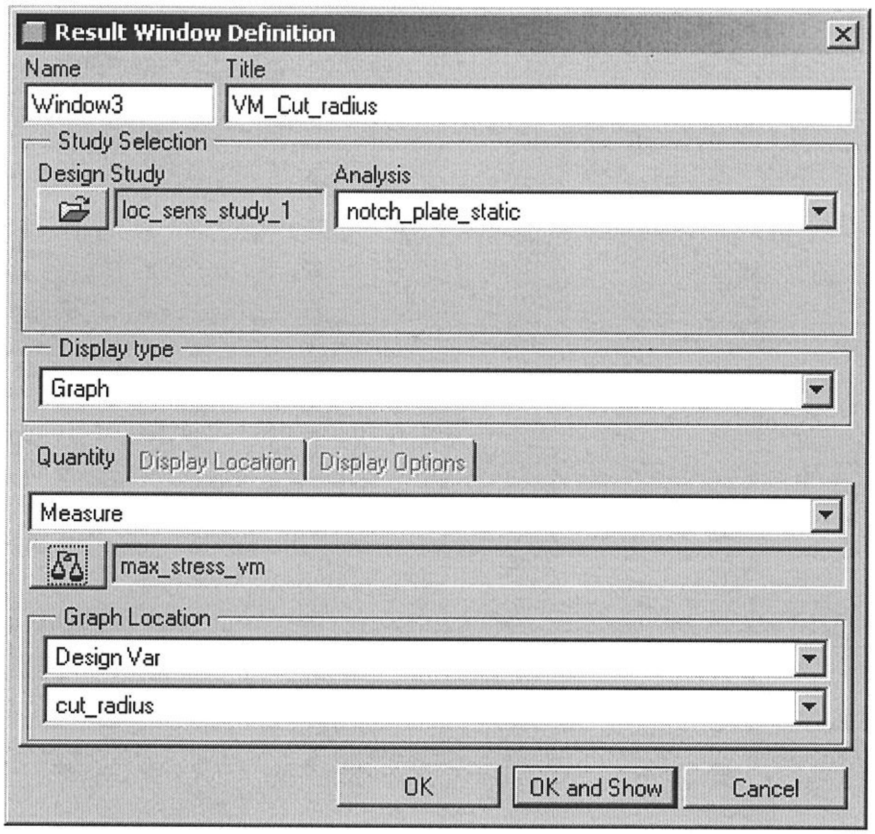


Repeat the procedure to create another window, change the design variable to Cut Length



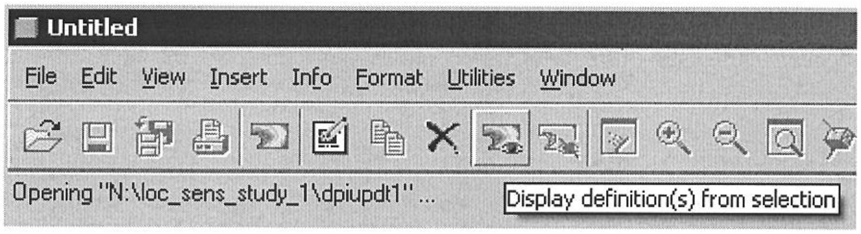
Click **Ok**

Repeat the procedure to create another window, this time change the variable to cut_radius



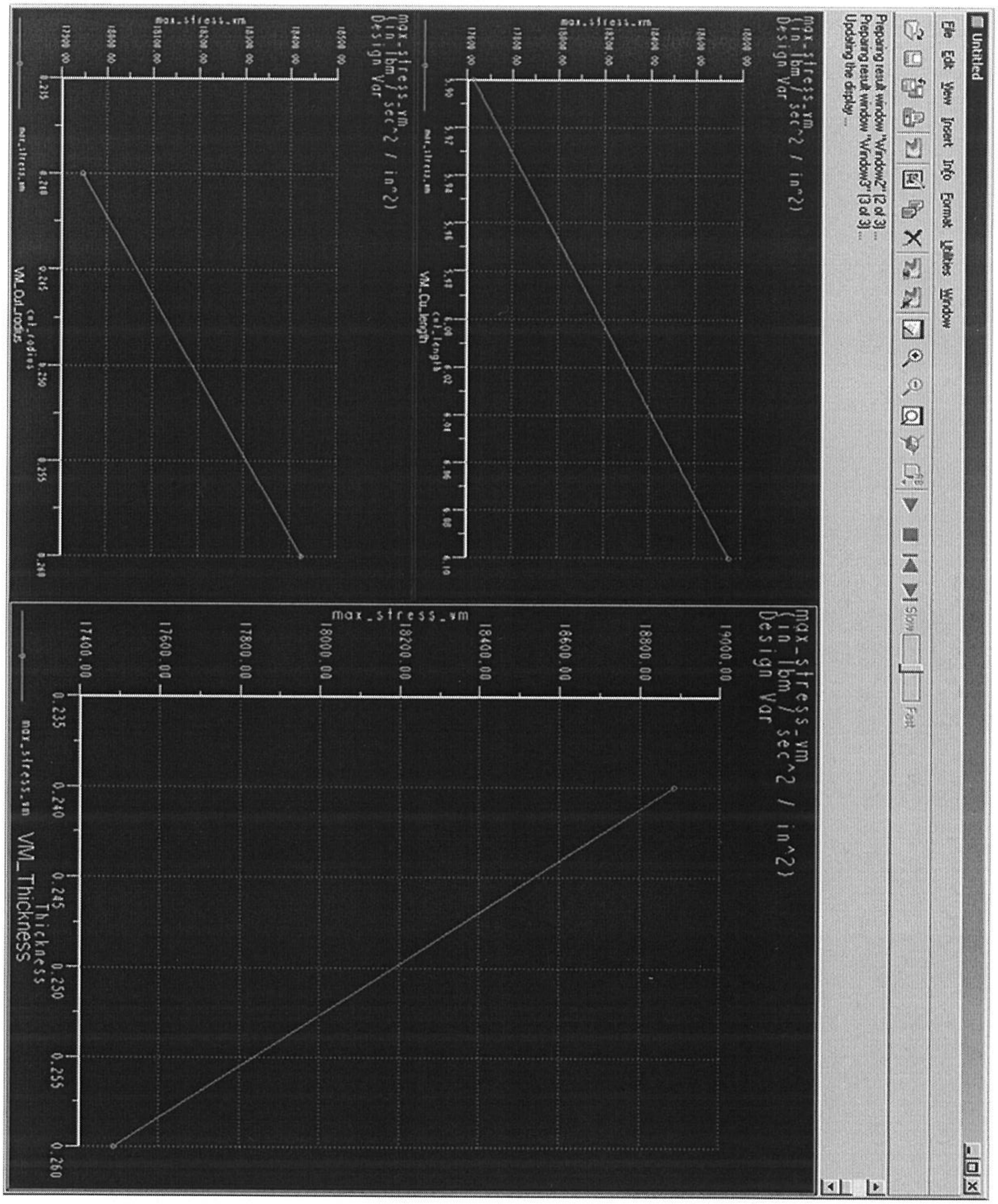
Click **Ok**

Click on the display definition button



Select the three windows by clicking on the left button of the window

the Final result



Local Sensitivity Study Parameters Results

Analyze how sensitive each parameter is to the Von Mises stresses.

The conclusion is:

The VM stress is sensible to all parameters

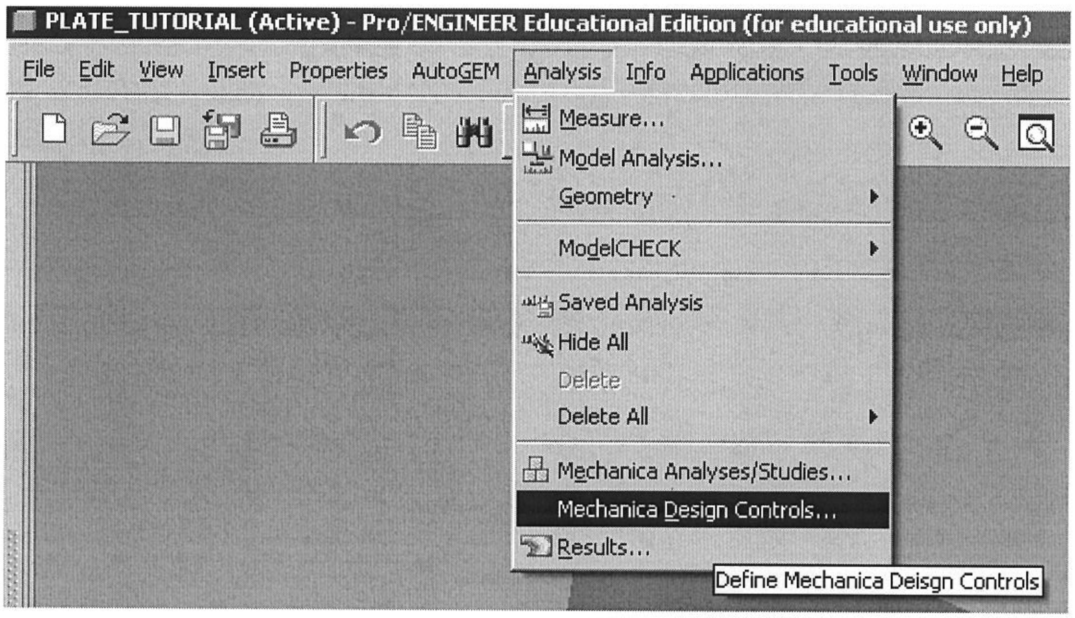
This was the main objective of the local sensitivity study. So we will carry all three parameters into the next phase.

If our study indicates that the Von Mises stress is not sensible to any of our parameters, then the parameter or variable which does not affect the Von Mises stress is not taken into account for the optimization study.

Select: **File>Exit Results>No>Close**

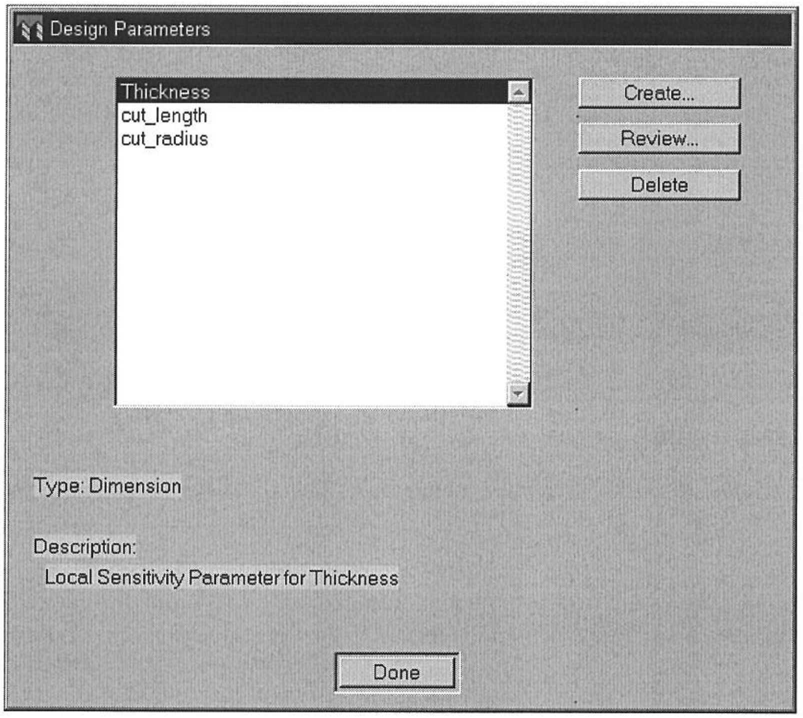
8.3 Global Sensitivity Study

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

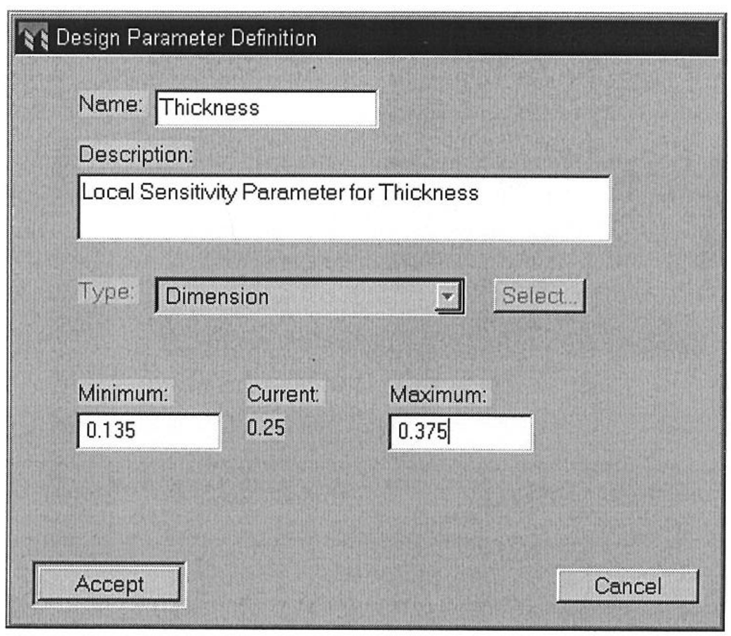


Select: **Analysis>Mechanical Design Controls**

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

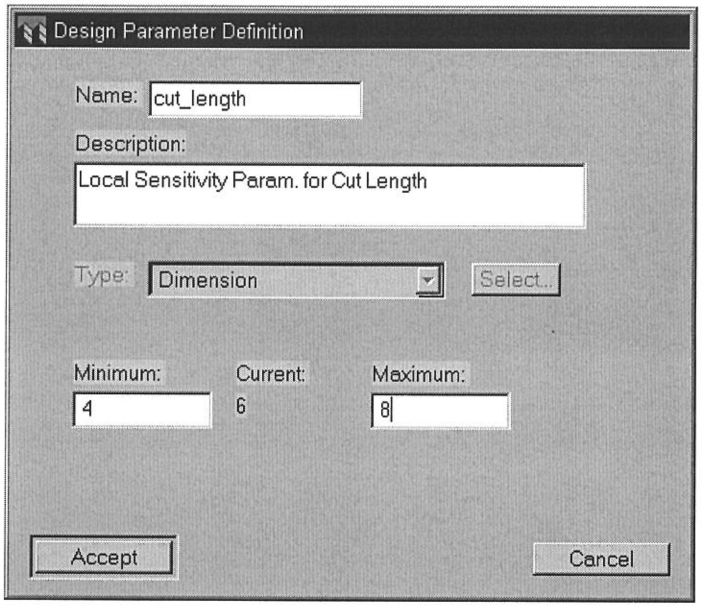


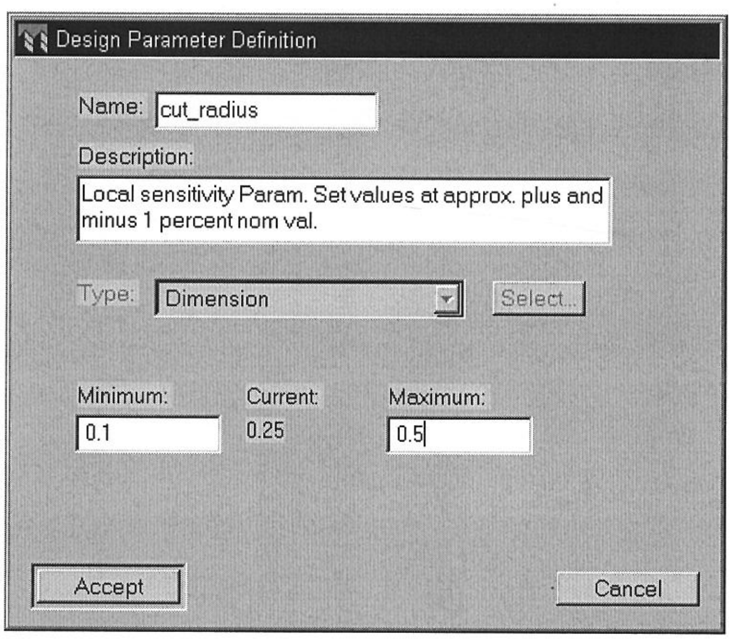
Note that in practice, the designer would set these values according the final dimensions and applications of the model



Click on **Accept**

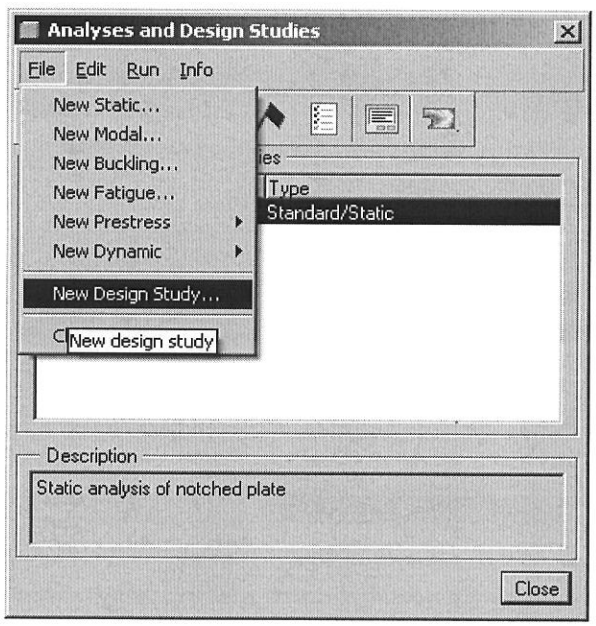
Repeat the procedure with the other two variables

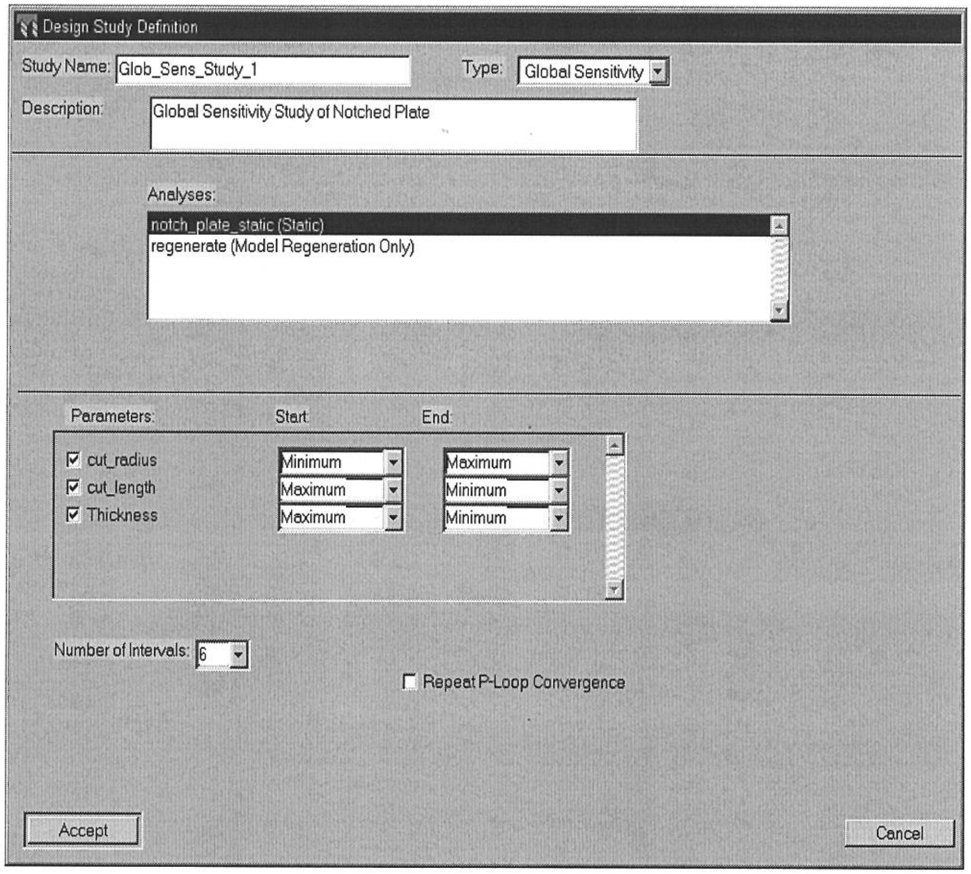




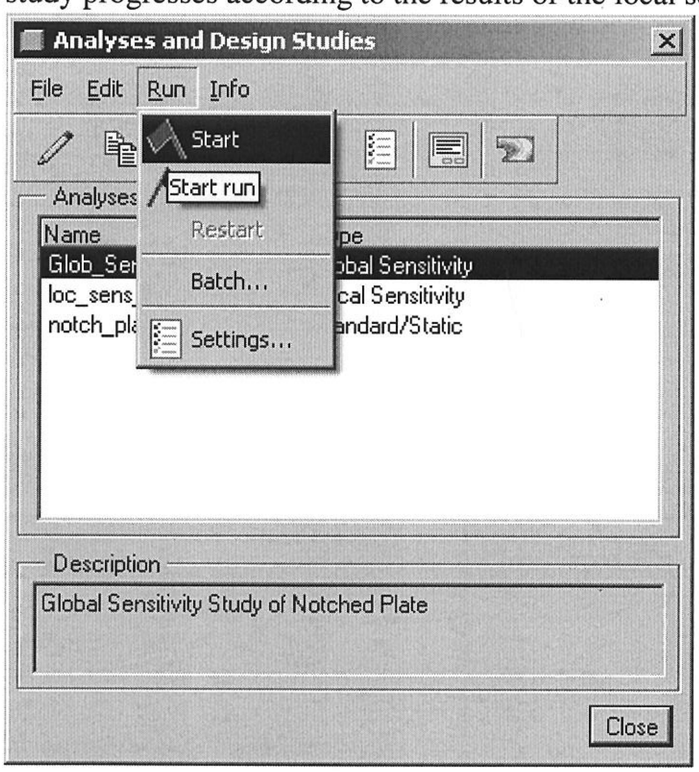
Select: **Accept>Done>Done Return**

Create a design Study Called: **Glob_Sens_Study_1**



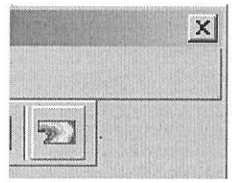


Note that the minimum and maximum values are set to minimize the plate weight as the study progresses according to the results of the local sensitivity study

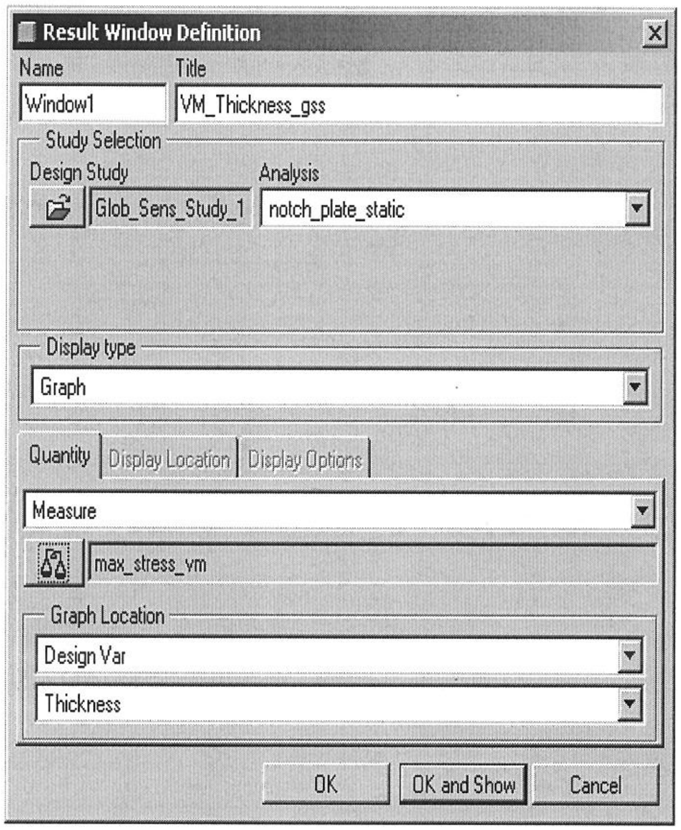


Click on the Status Icon to Check the Iterations

Click on the review result button

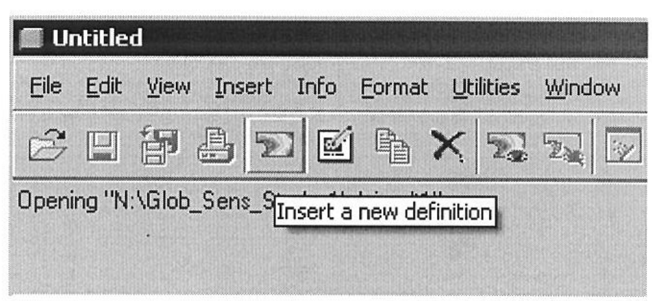


Fill out the dialog boxes with the following data

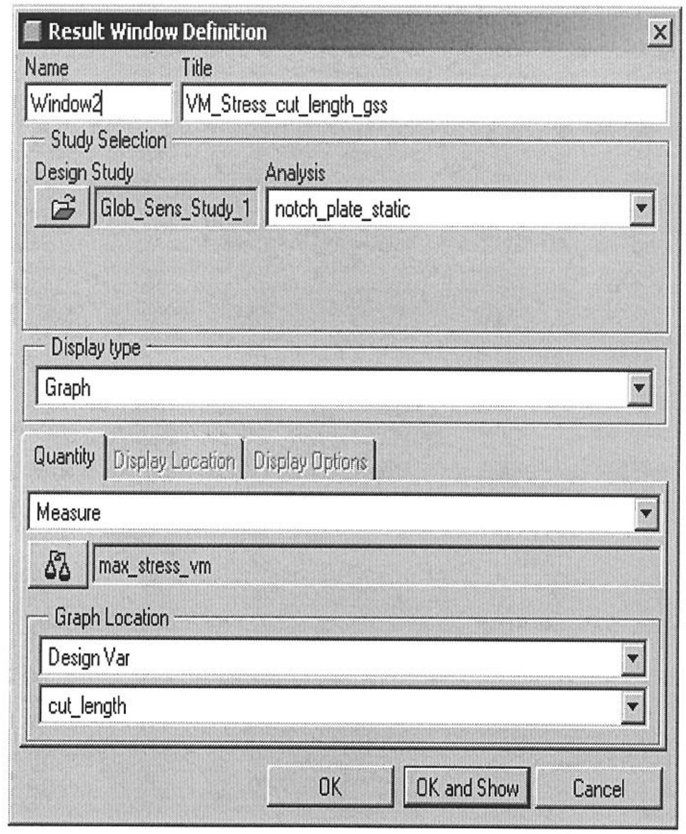


Click **Ok**

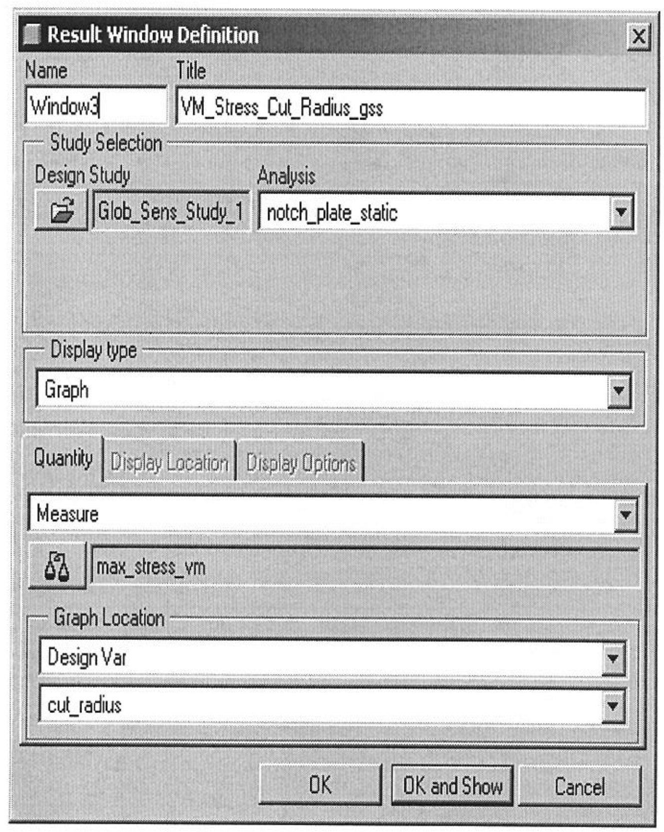
Click on insert new definition



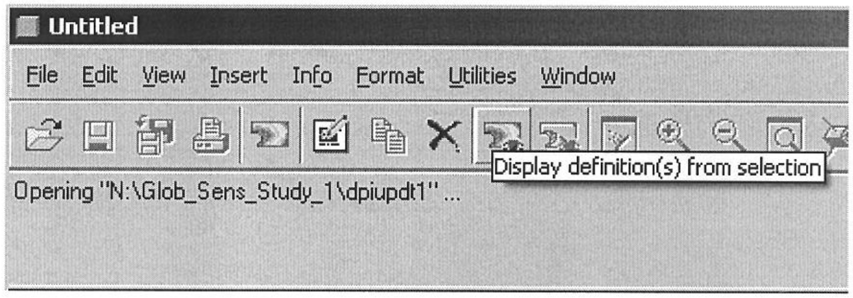
Fill out the dialog box and click **Ok**



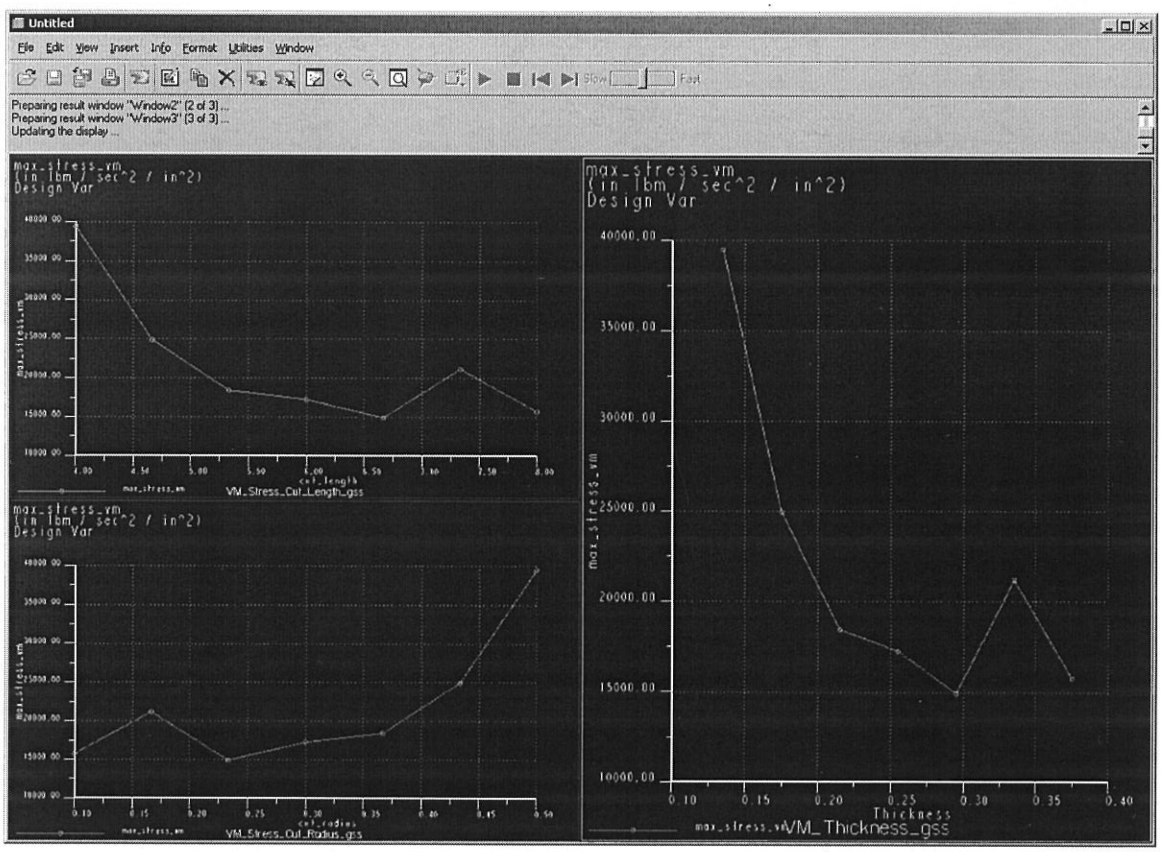
Repeat the procedure for the following box



Select display definition button



Select the four windows and click **OK**



This graphic appears

This study was carried out to find the best combination of parameters that will be taking into account on the final optimization study.

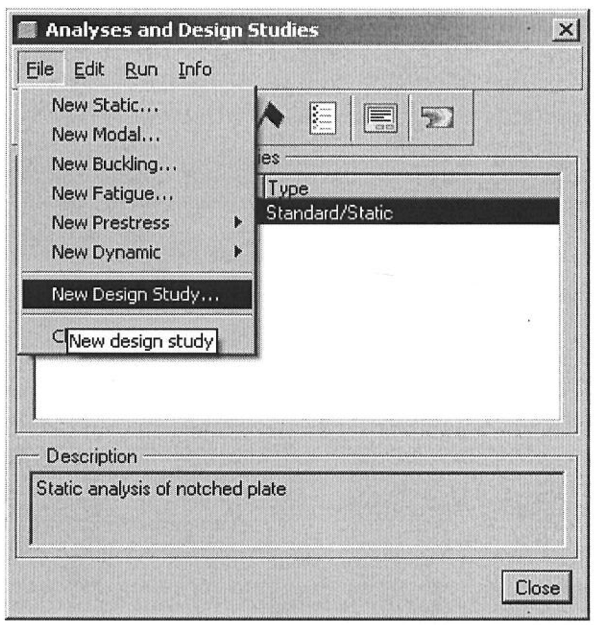
The conclusions we have are:

For minimum VM Stress, we need the following:

-Maximum Thickness
-Maximum Cut length
-Minimum Cut Radius

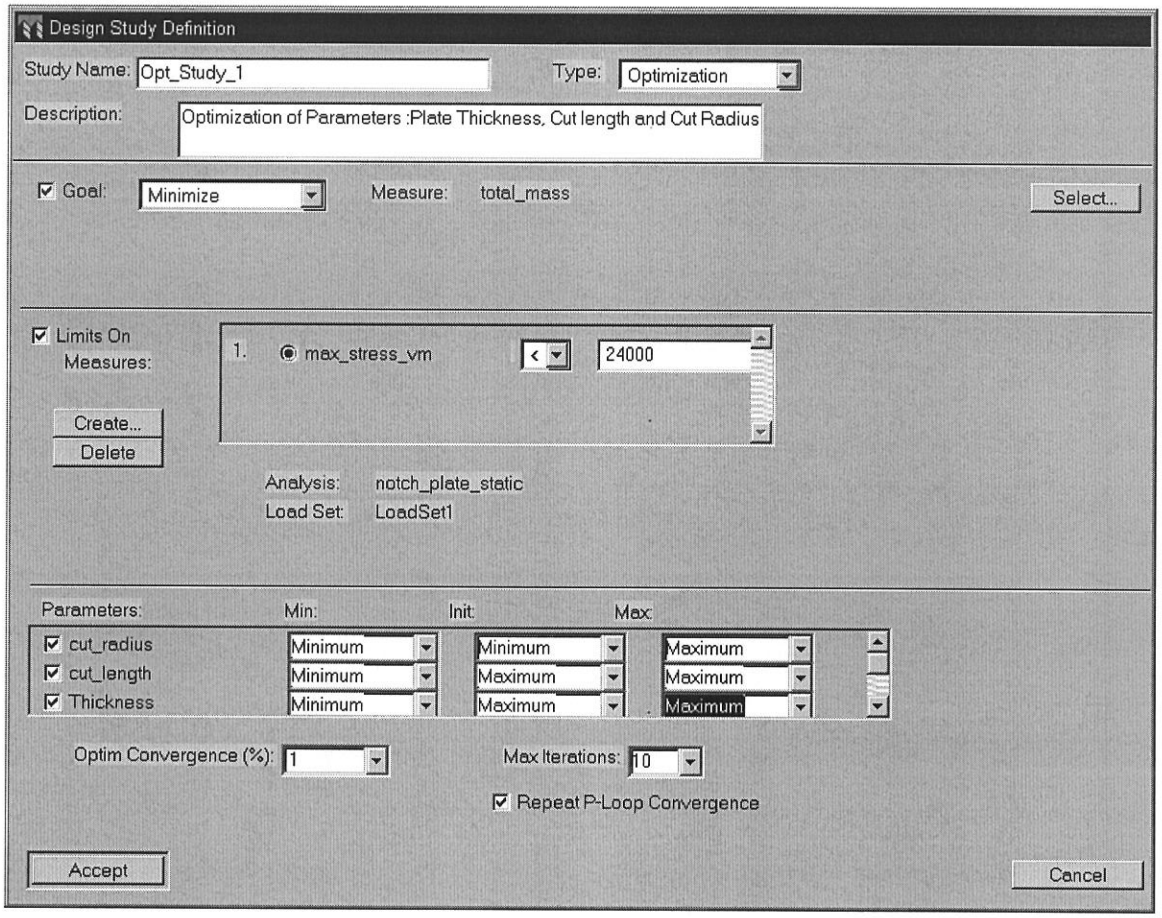
8.3 The Optimization Study on Total Mass

Click **Edit>New Design Study**



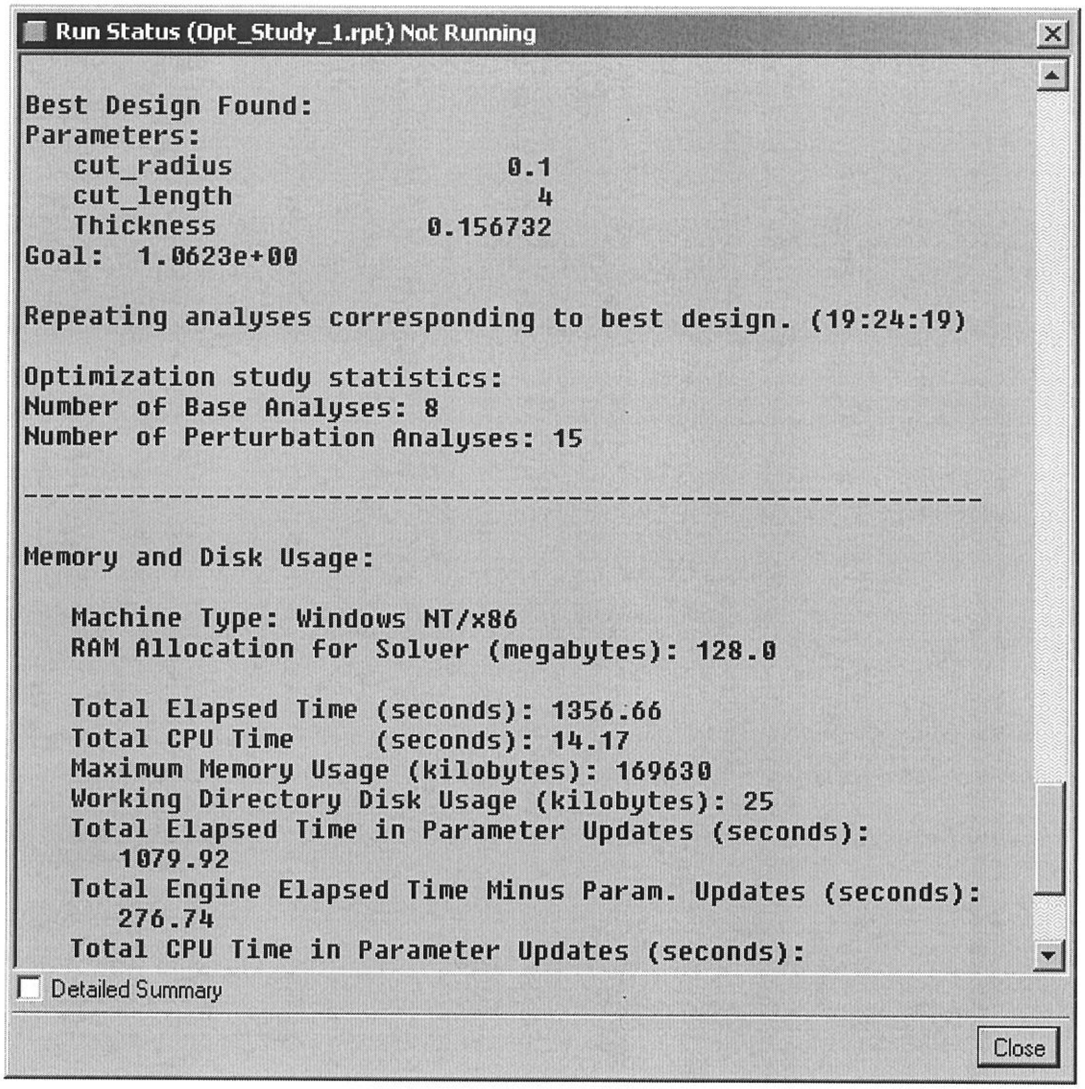
Fill out the blank spaces, click on create and type the right value for the maximum allowed stress, modify the rest of the values if it is necessary (we are going to use 10 iteration to save time, but the minimum suggested is 15)

Click on Accept, and Run the analysis, Click on the status button to check the iteration process



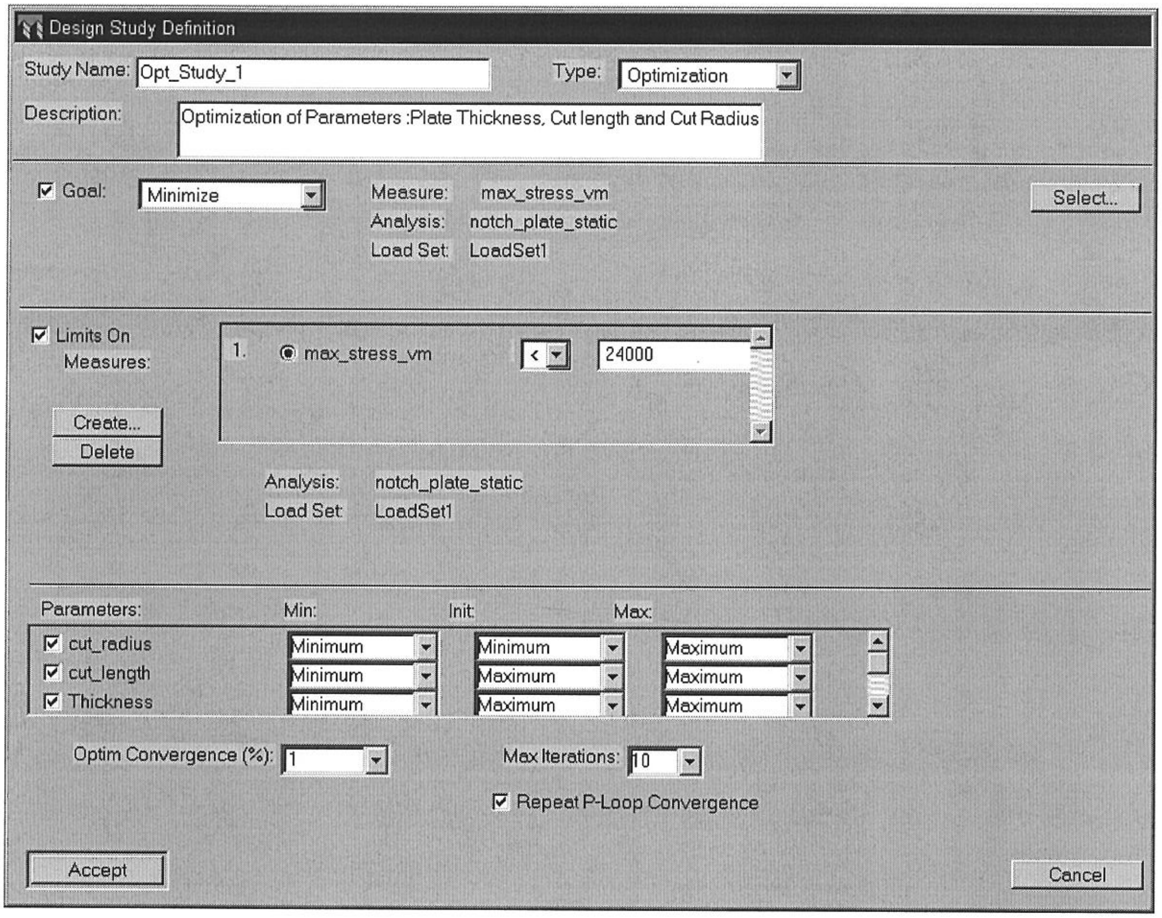
Run the design study it will take between 20 to 30 minutes. Check the status file and compare it to the one shown below.

The final values should be:

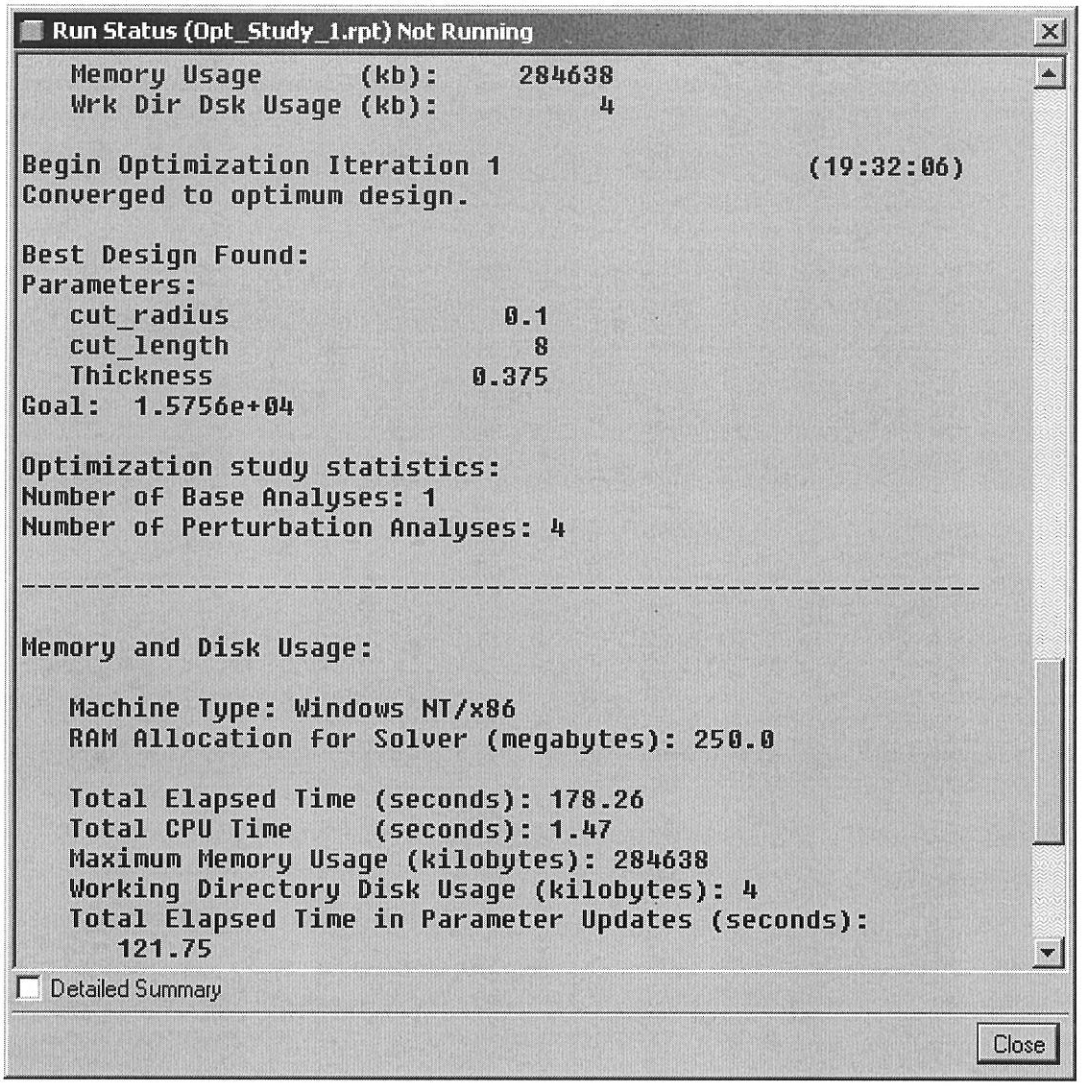


Now, we are going to find the values to minimize the Von Mises Stress

Edit the design Study, and change the study from mass to max_stress_vm (goal)



Accept and Run the Study



The final Values to get the Minimum Von Mises Stress