Sensitivity and Optimization Tutorial

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)

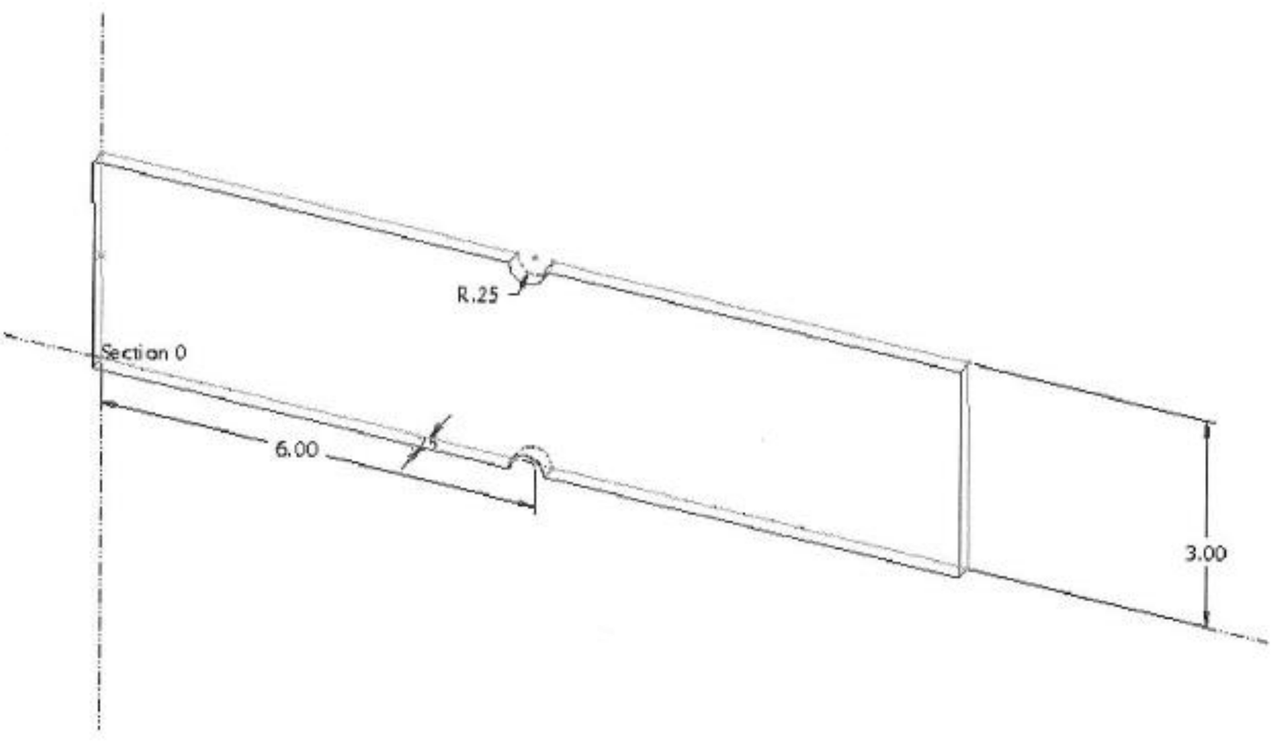


Figure 1 (Notched Cantilever Beam)

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.

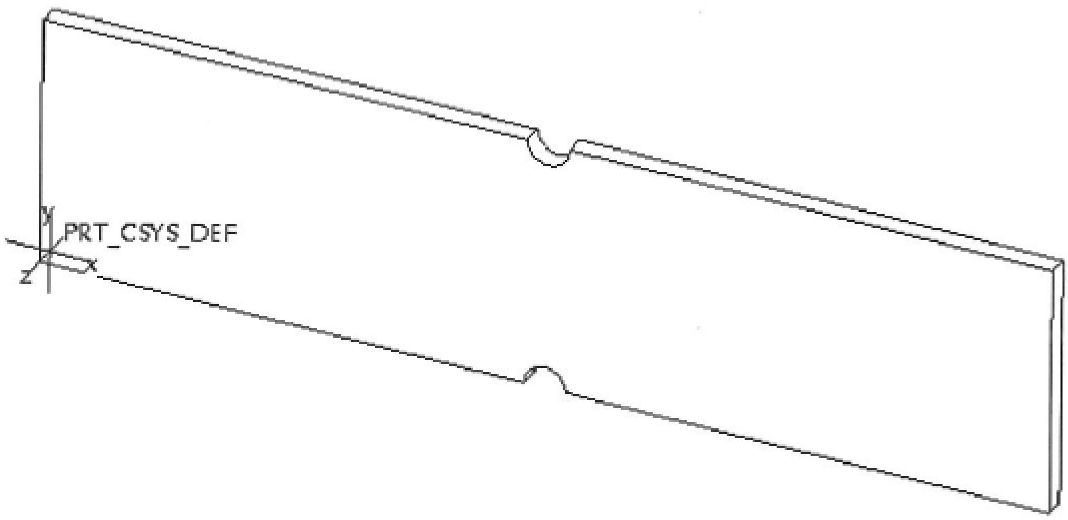
Optimization Goal	Measure to be used for Optimization	Design Parameters to be optimized		
Minimize Weight of the part	Von Mises Stresses	**Name**	**Description**	**Initial Value(in)**
		Cut_length	Dimension from left edge to cut	6
		Cut_radius	Notch radius	0.25
		Thickness	Plate Thickness	0.25

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.

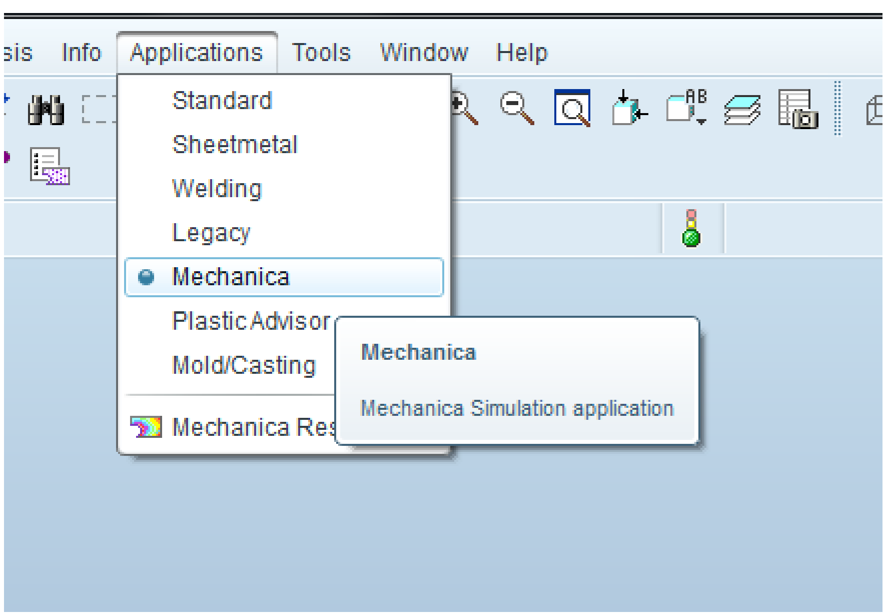
1. **Open the file Plate_tutorial**

Once open, the following part show up on your screen

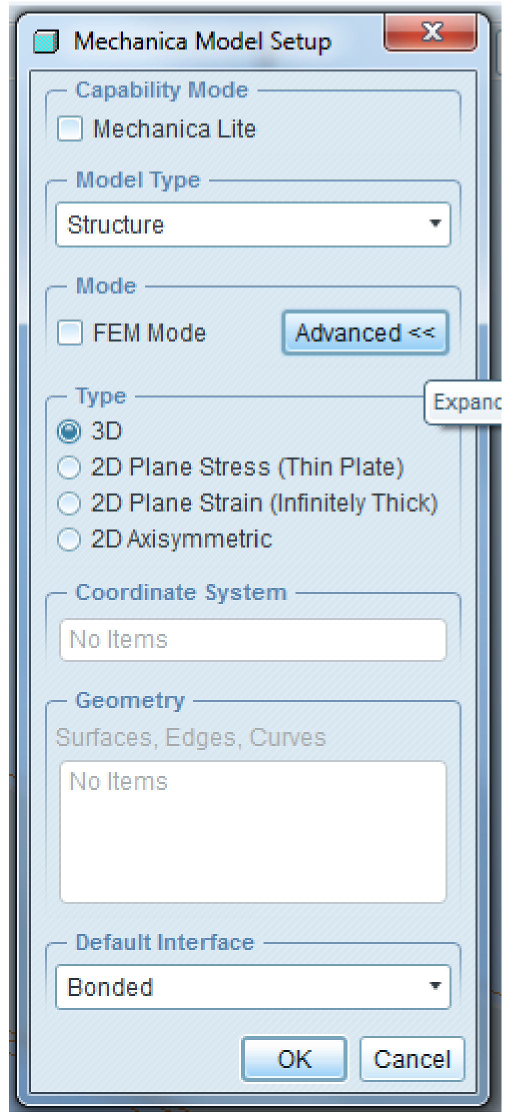


2. **Switching to Pro/Mechanica (Integrated Mode)**

Select **Applications>Mechanica**

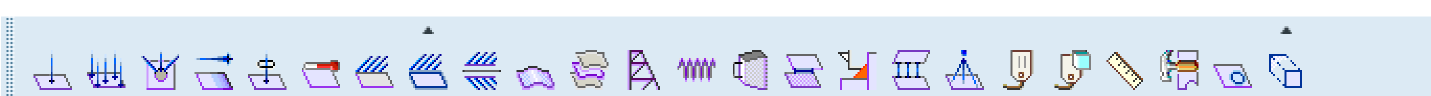


A new dialog box appears, select structure and 3D analysis
Click on **Ok**



<u>*Model Type*</u>:- 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)



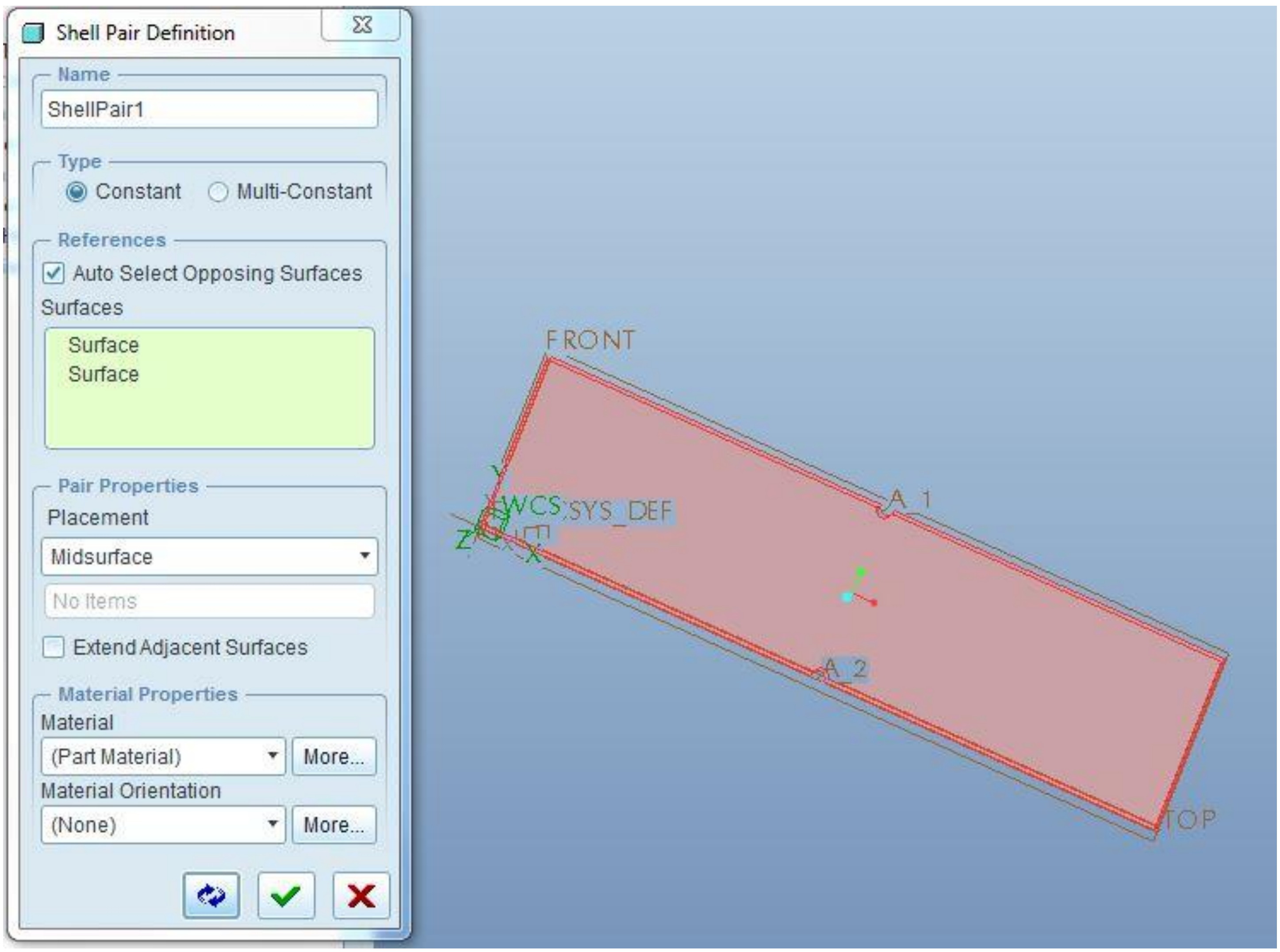
Idealizations:- The Following idealizations are available in the Integrated Mode – Shells, Beams, Masses, Springs. These idealizations should be used whenever possible as they require less computation time. Another advantage of using these models is that they are easy to model.

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

Select **Insert>Midsurface**

Select one of the surface of the plate, the other one will be selected automatically.

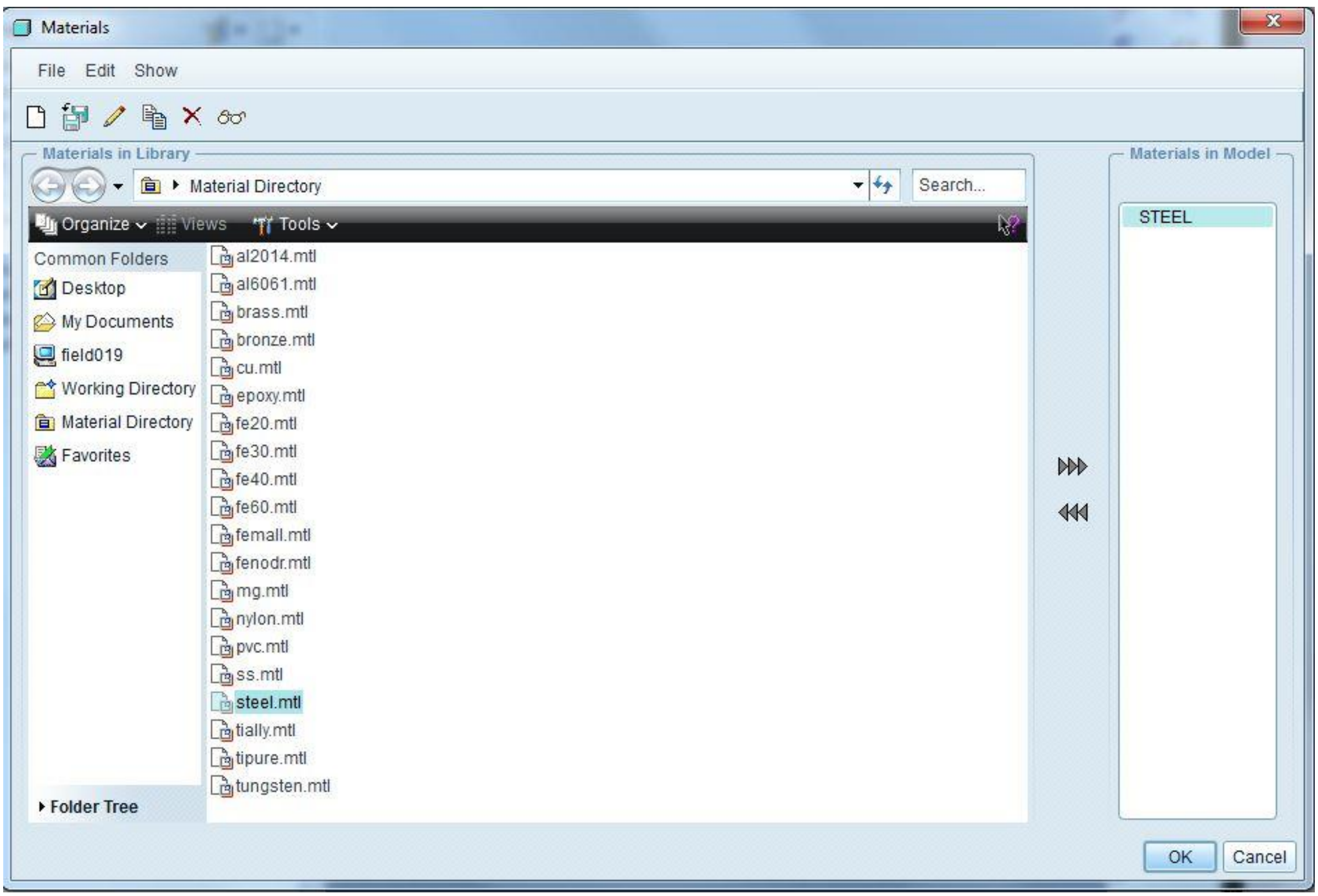
Click on **Close**.



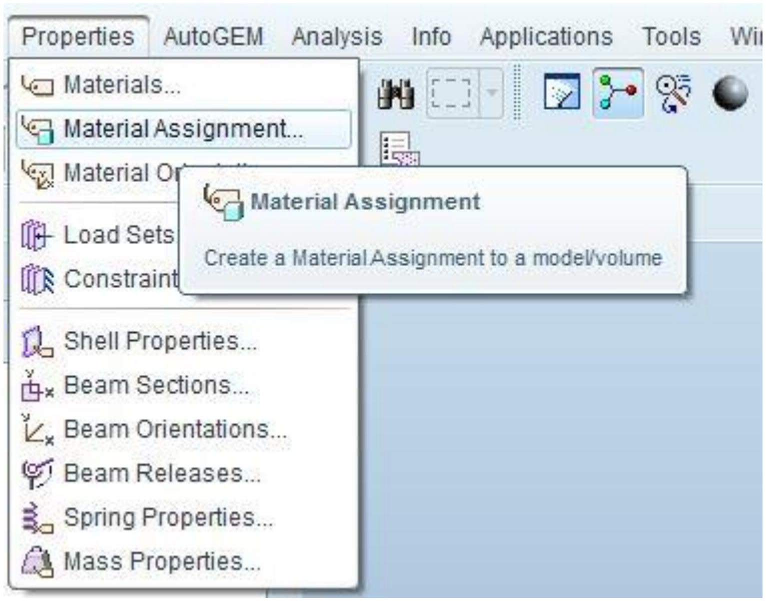
3. Assigning Materials Properties to the Model

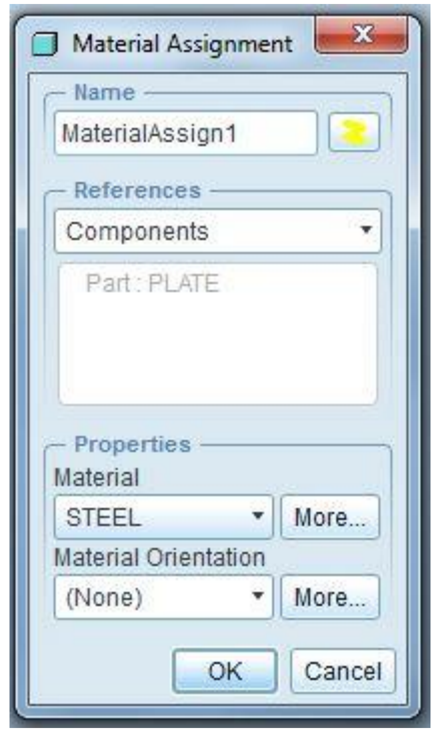
Materials:- This option selects the user define/select the material for the model. There are many standard materials listed as Materials in Library and the user can assign one of those standard materials to the part(s) and the user can also preview the properties of the standard material by clicking 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.



Select Properties/Material Assignment

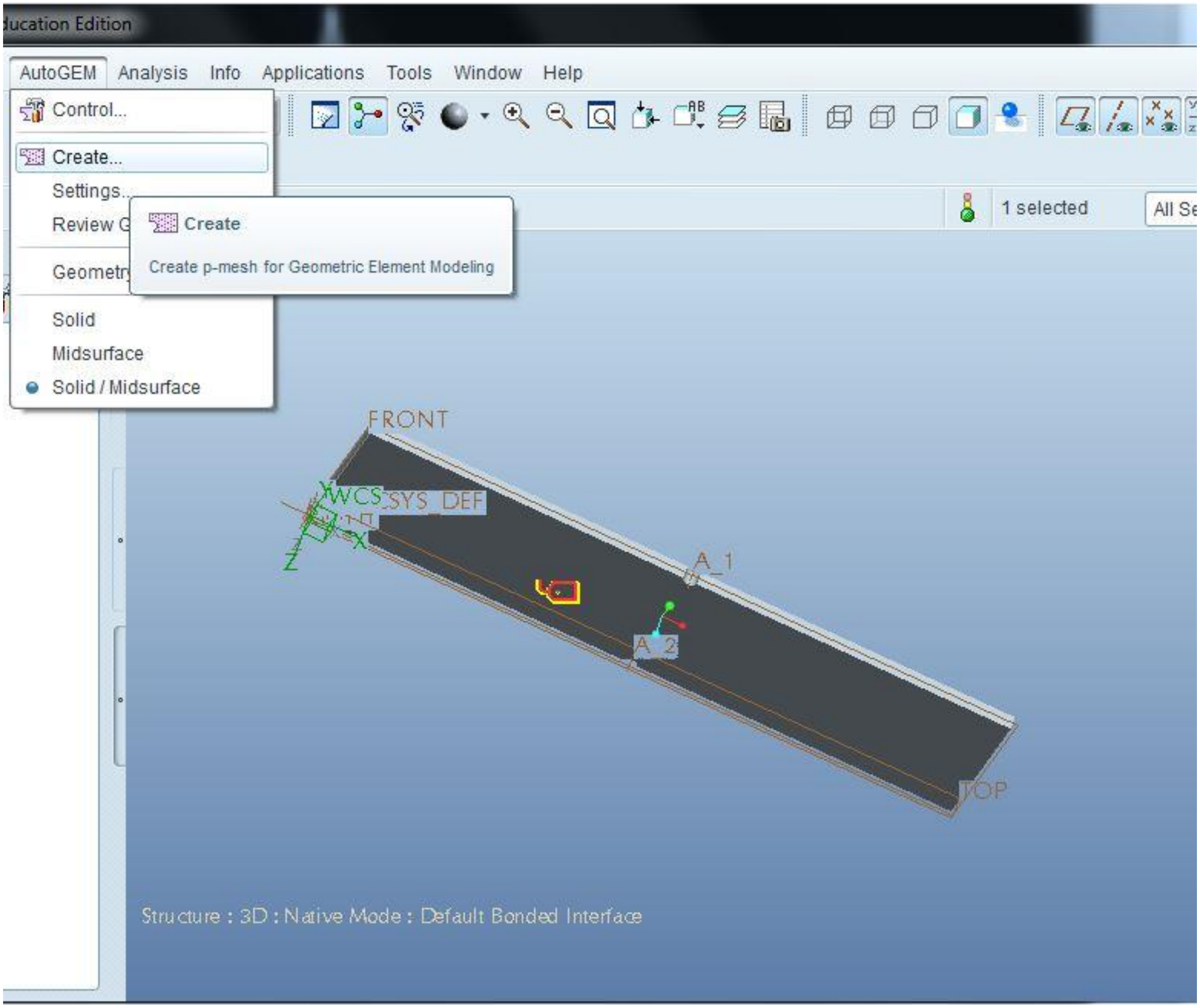




Click “OK”.

4. Creating a Mesh on the Model

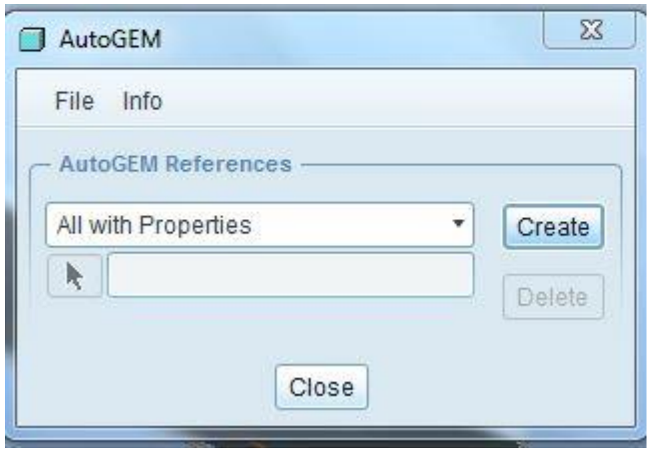
We will keep all default shell elements and settings that AutoGEM will create. Select **AutoGEM >Create**



__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.

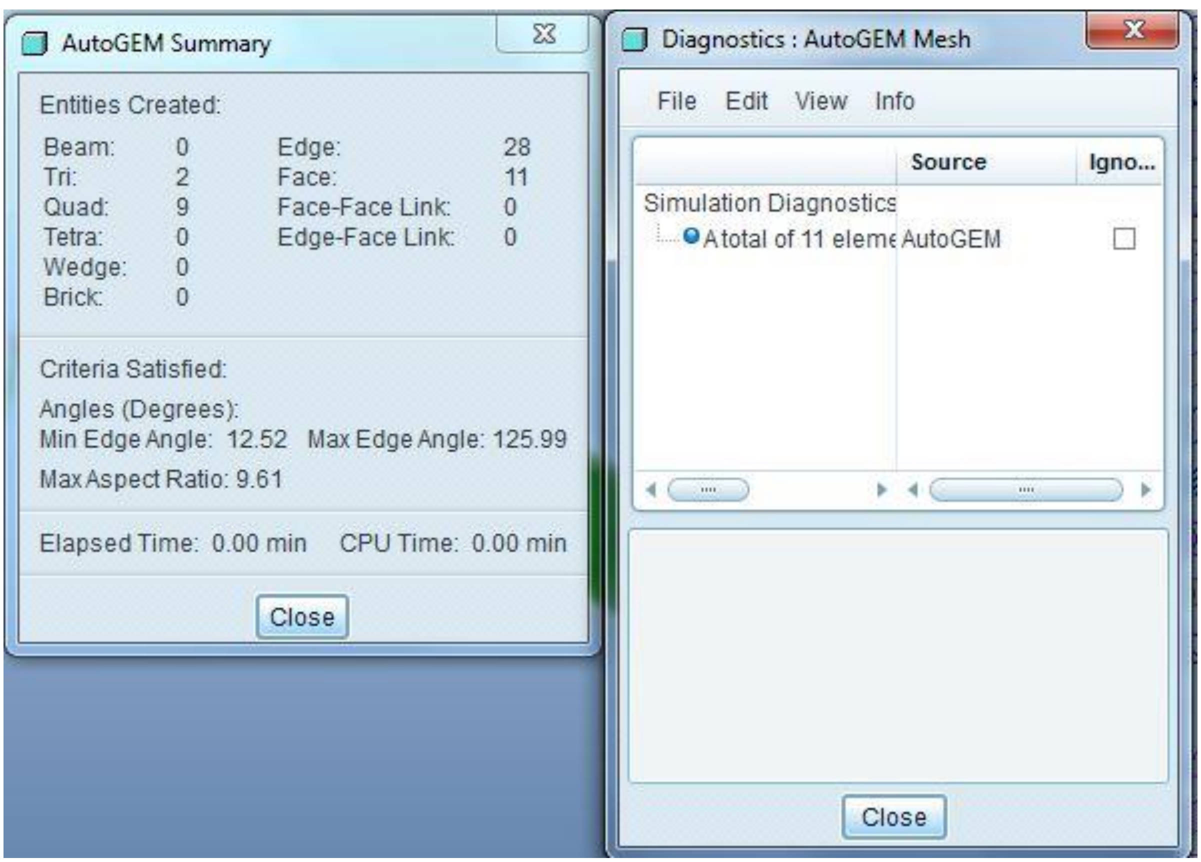
After selecting create the following Selection Box appears

Click on **Create**

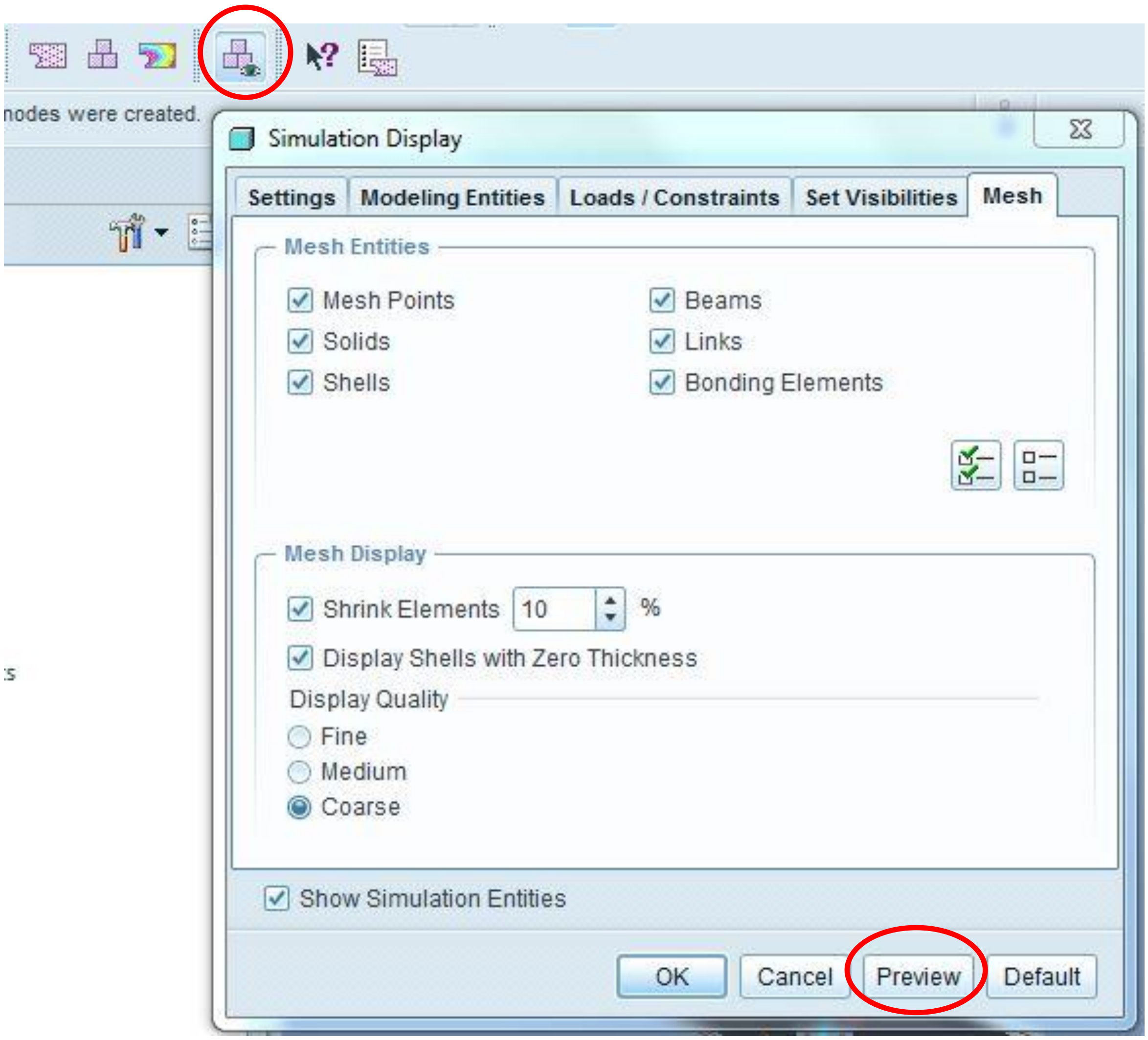


This confirmation of the elements box should come up.

Select **Close**



Click on the setup simulation display button on the toolbar

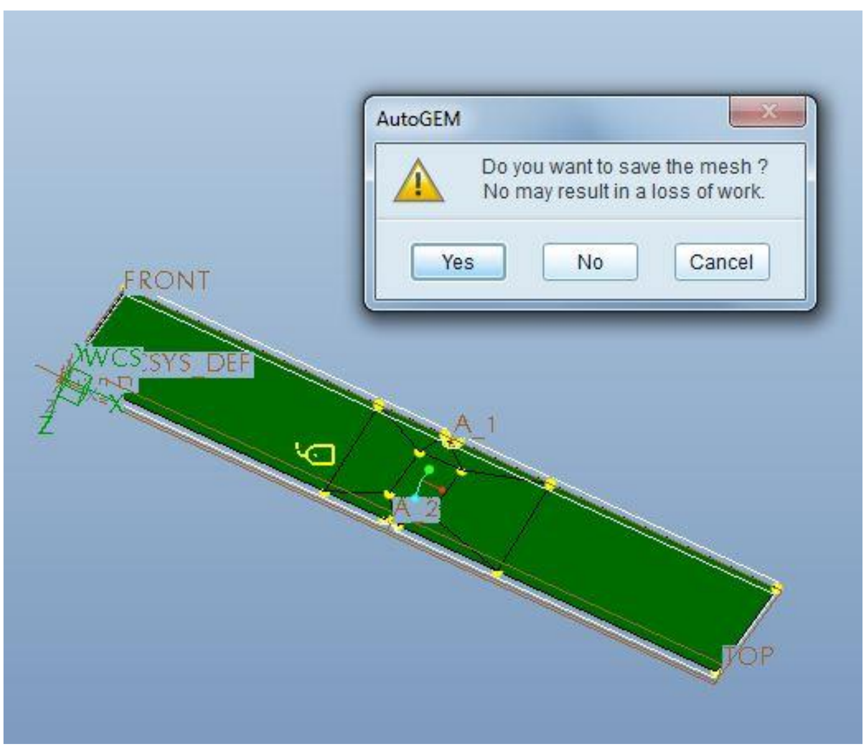


Use the display seeting to shrink all elements to 10%. Click on **Preview**.

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



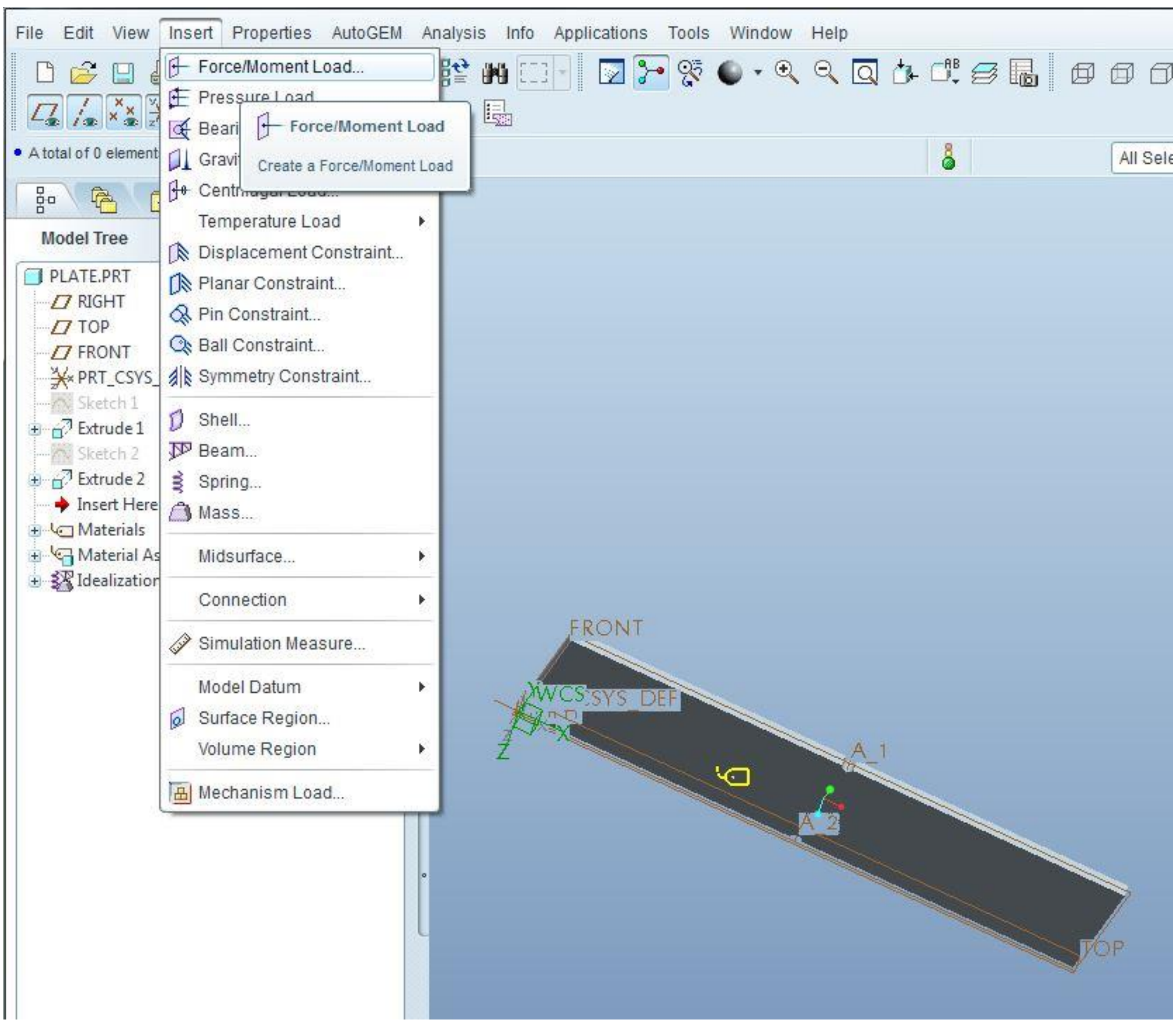
5. Boundary Conditions on the Model

Constraints/Loads:- 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.

5.1 Loads

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

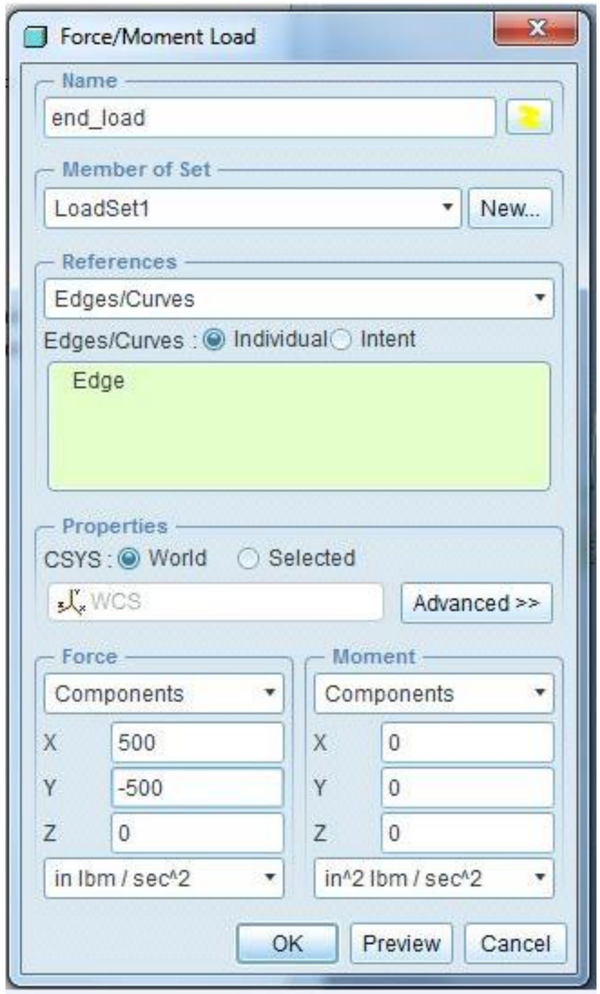
Select **Insert>Force/Moment Load**



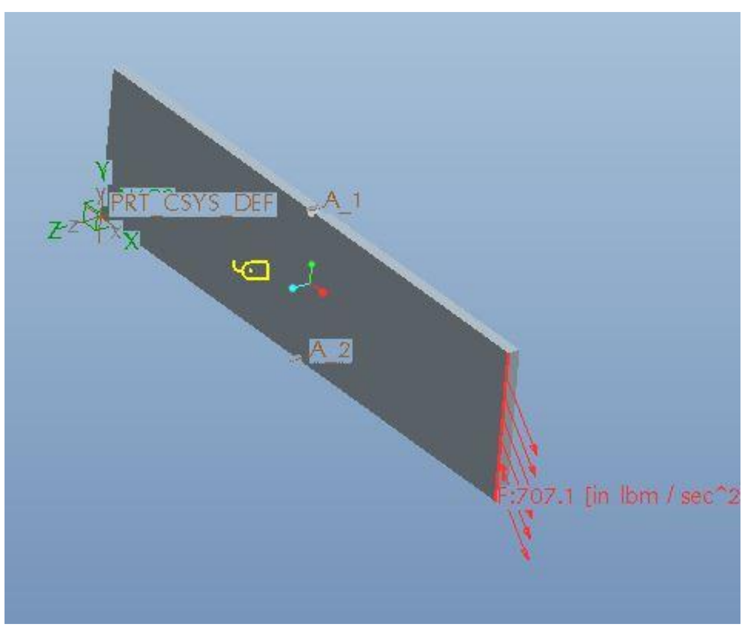
The following Dialog box appears:
Change the name of the constraint set, select Edge/Curve, and use the following force components: 500 and -500 in the X and Y direction
Select the arrow on the references part of the Force/Moment dialog box and select the right vertical edge from the model

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

Click **Ok**

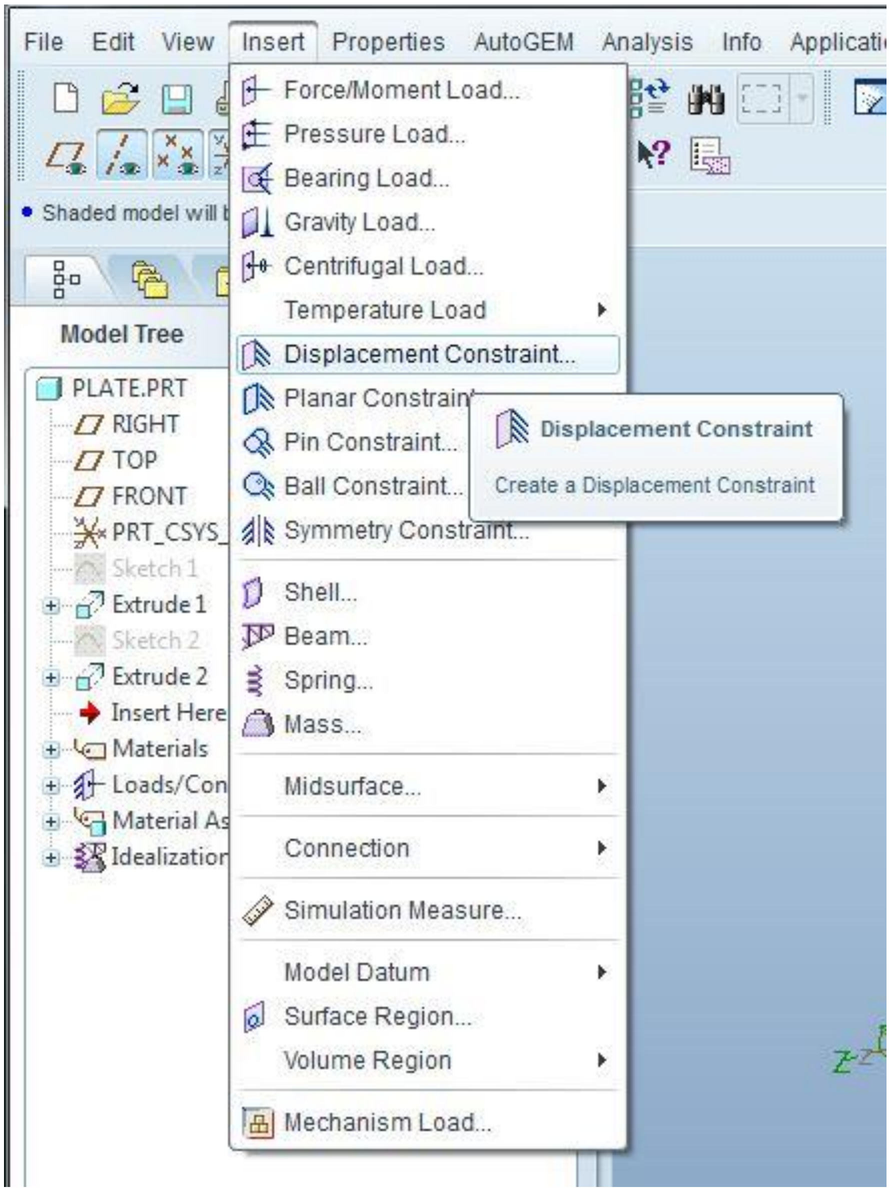


Your model should look like the following graphic

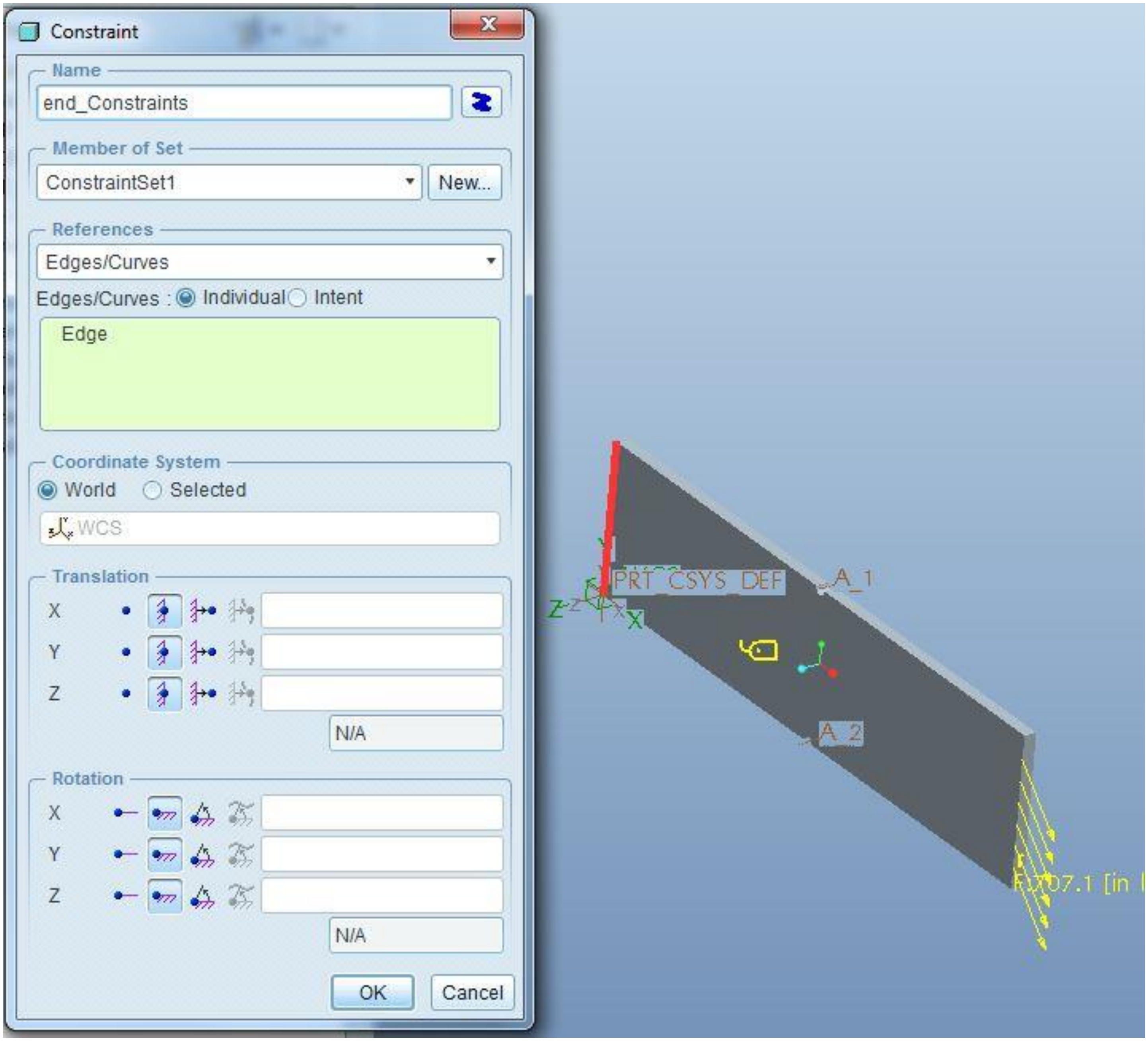


5.2 Constrains

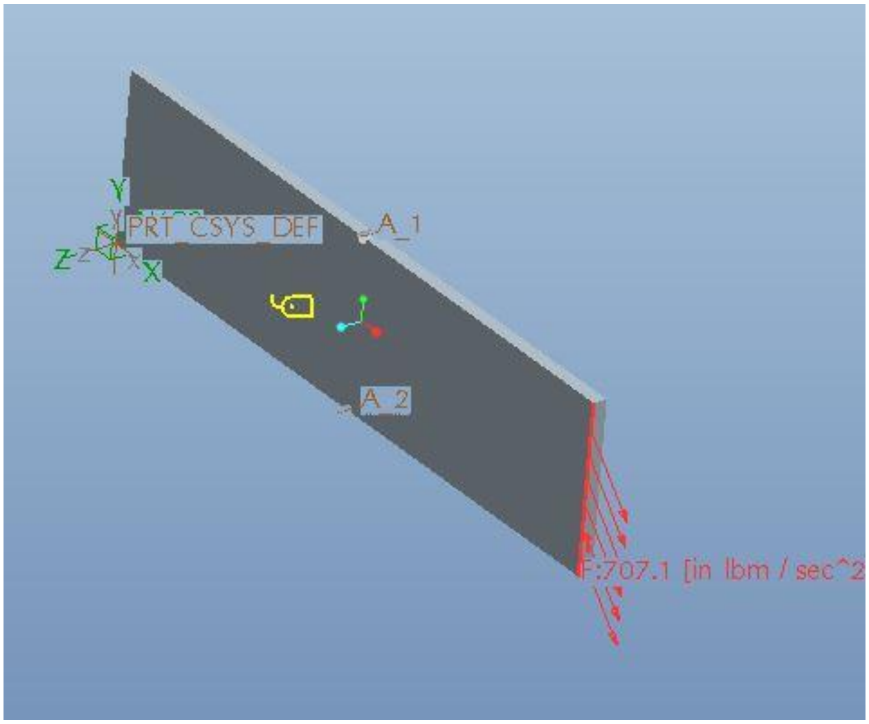
The plate is a cantilever beam with the left end being fixed. To apply the end constraints, select **Insert>Displacement Constraints**



Change the name of constrains, select the edges/Curves on reference and select the left vertical edge of the model



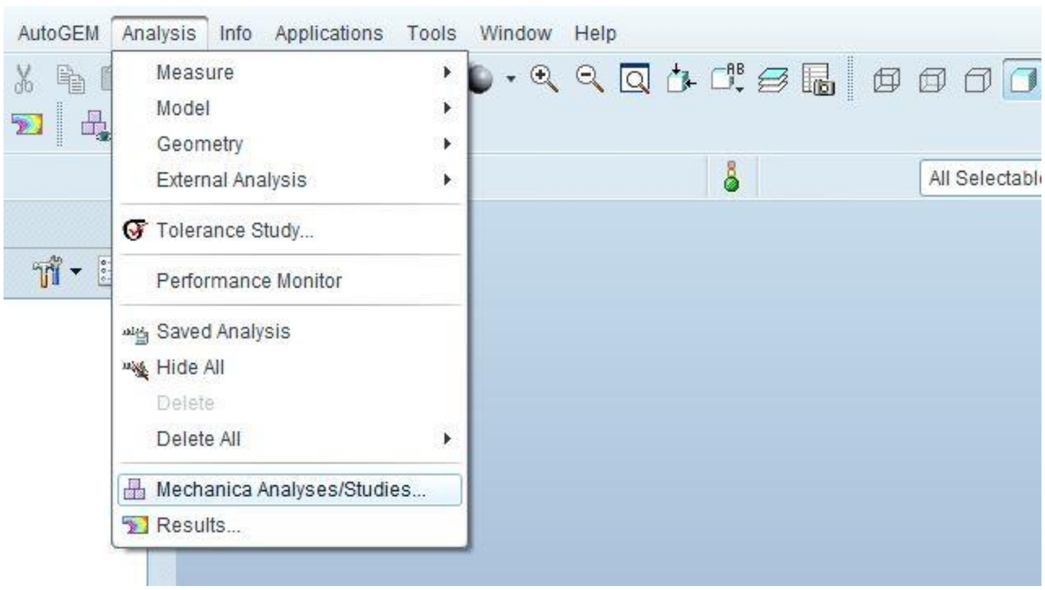
The plate FEA model should look as follow



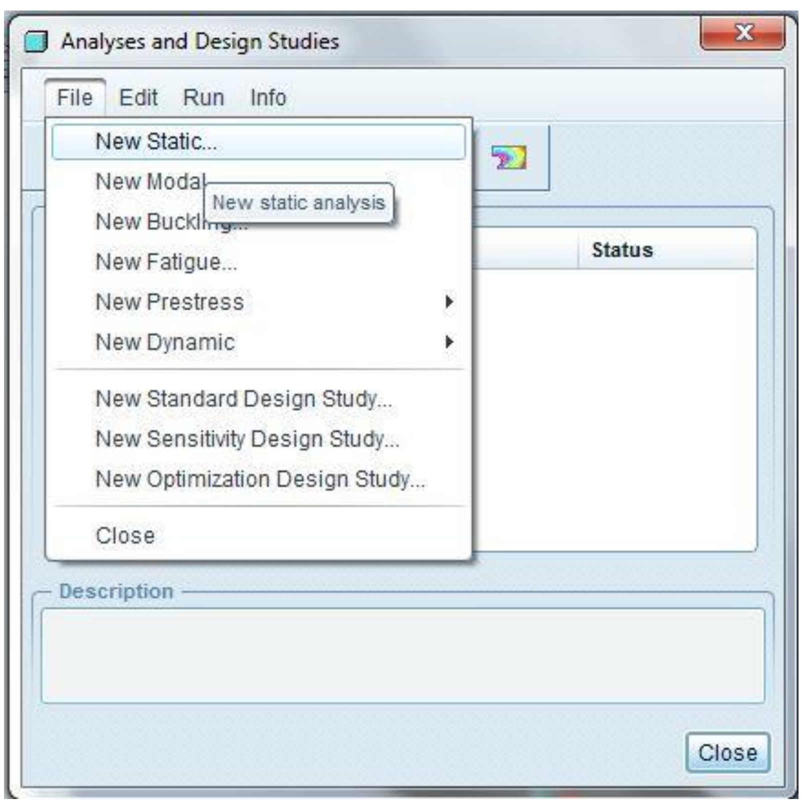
6. Static Analysis

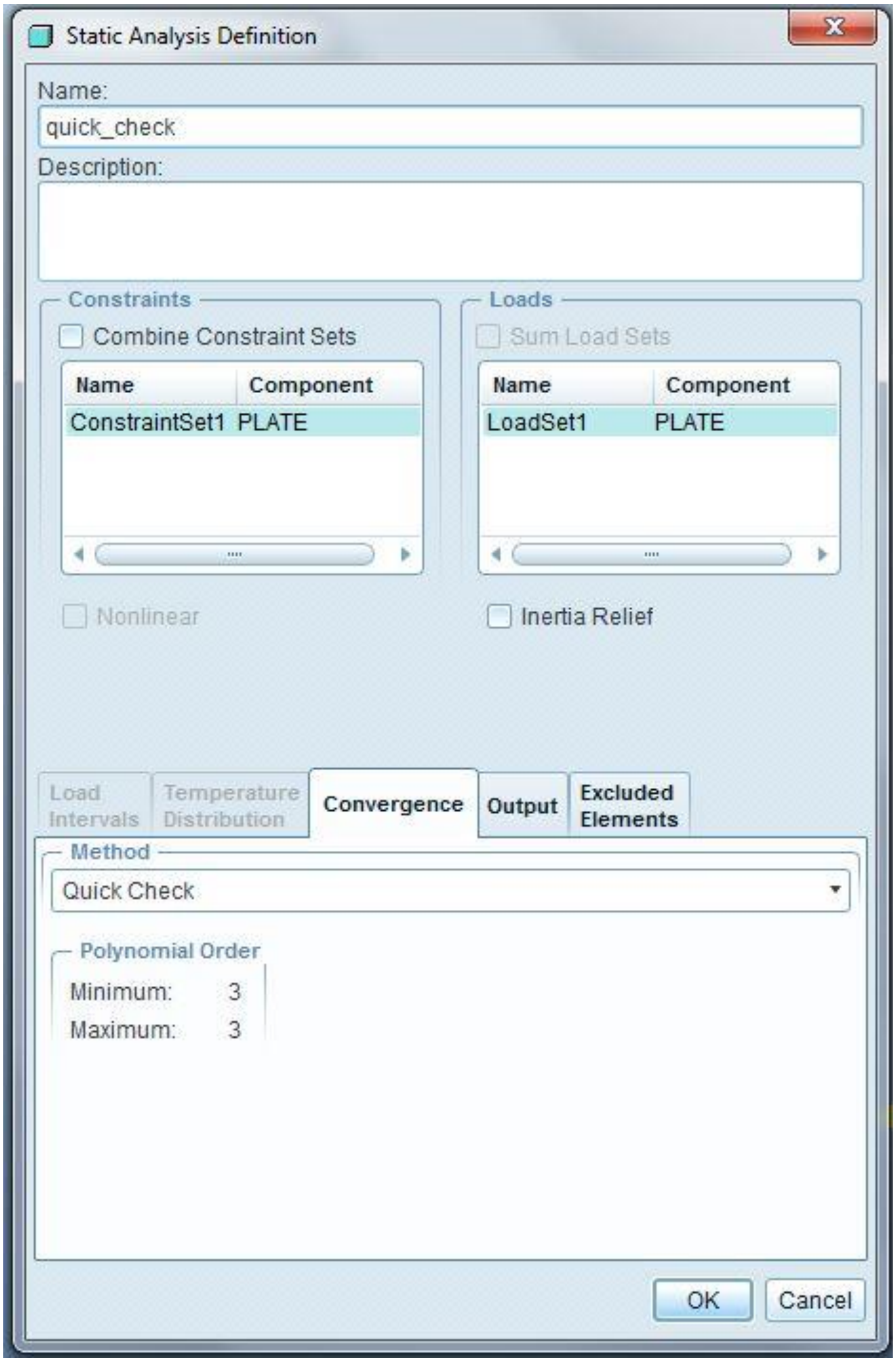
We are going to create a quick check no convergence analysis named **Static_1** to make sure that we can get to a solution. Results are not important at this point, we just want to make sure that the model can get to a solution

Select Analysis>Mechanica Analyses/ Studies



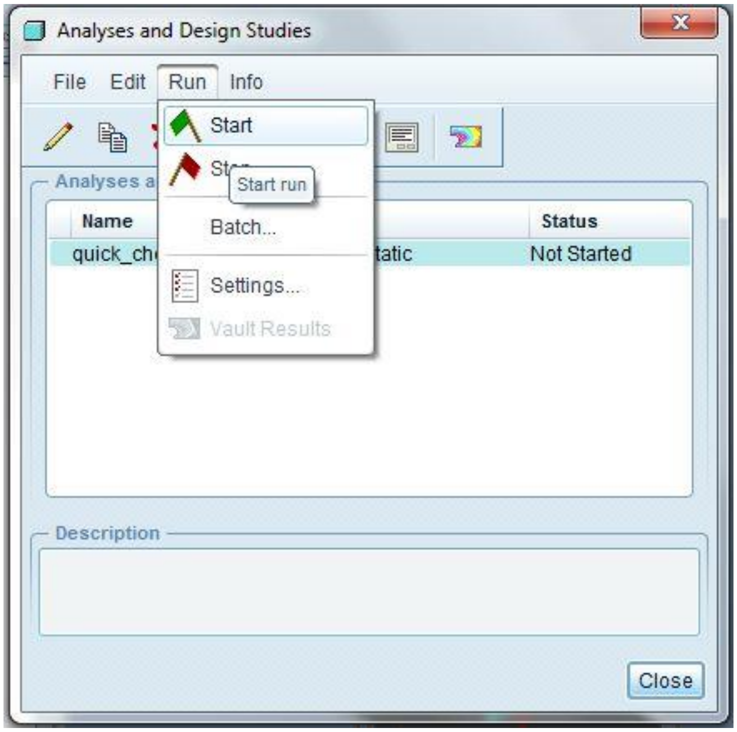
Select New Static and enter the information as shown below



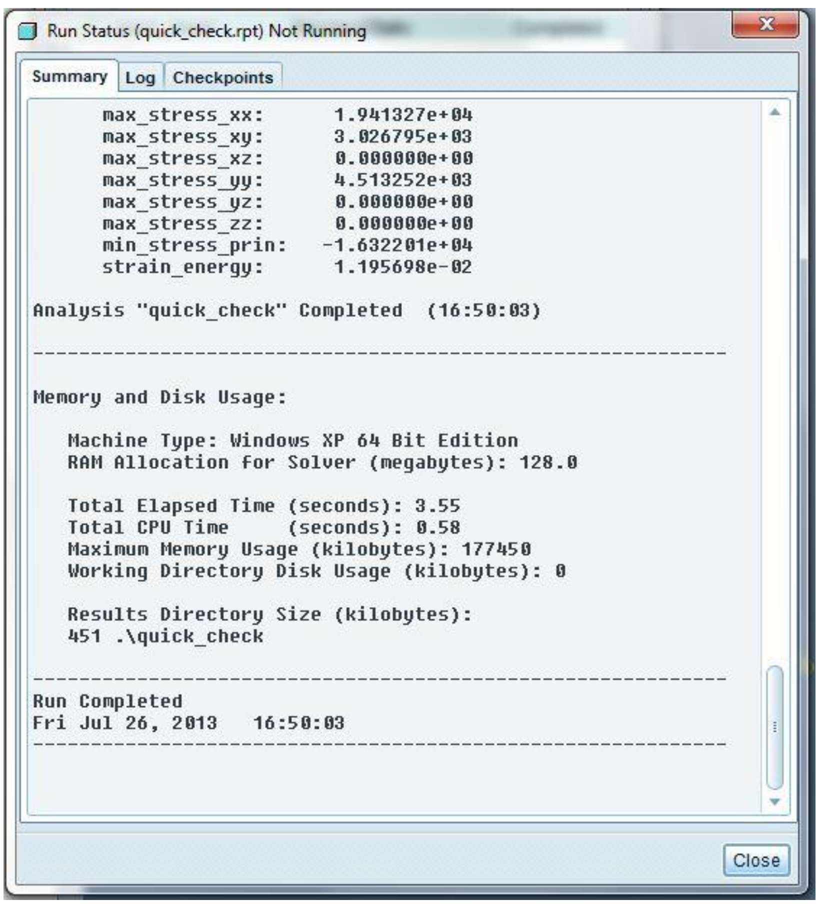

Static Analysis Definition
Name:
quick_check
Description:
Constraints
Combine Constraint Sets
Name
Component
ConstraintSet1
PLATE
Loads
Sum Load Sets
Name
Component
LoadSet1
PLATE
Nonlinear
Inertia Relief
Load Intervals
Temperature Distribution
Convergence
Output
Excluded Elements
Method
Quick Check
Polynomial Order
Minimum: 3
Maximum: 3
OK
Cancel

Select **OK**

Click in **Start**, answer yes to activate the error dection



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

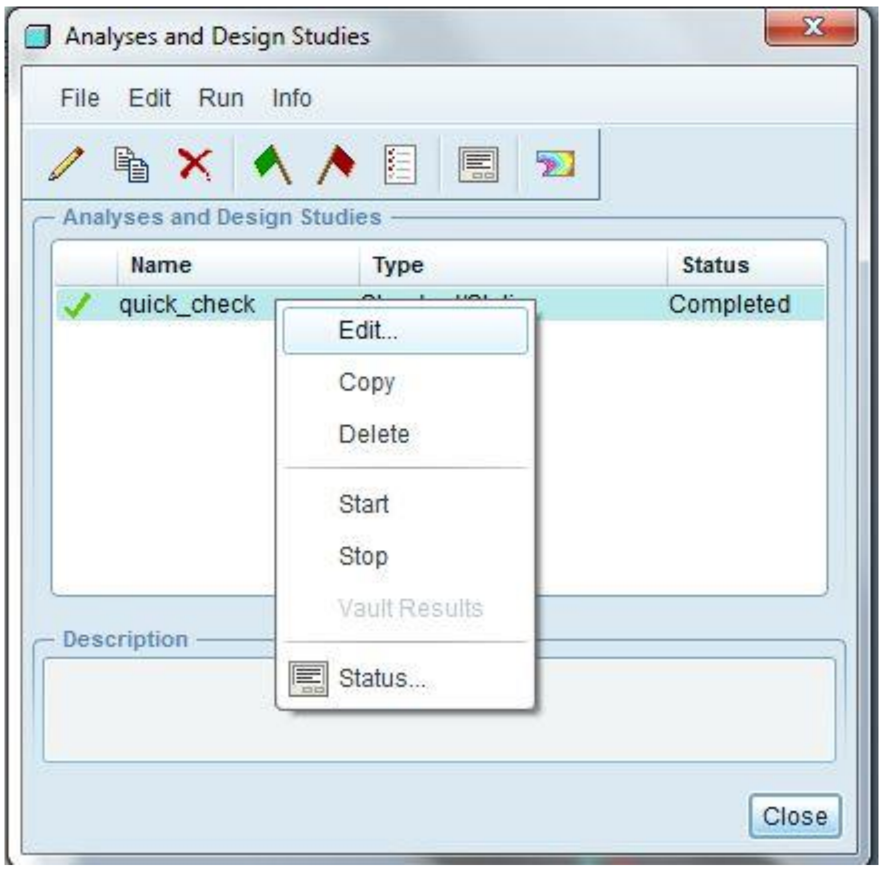


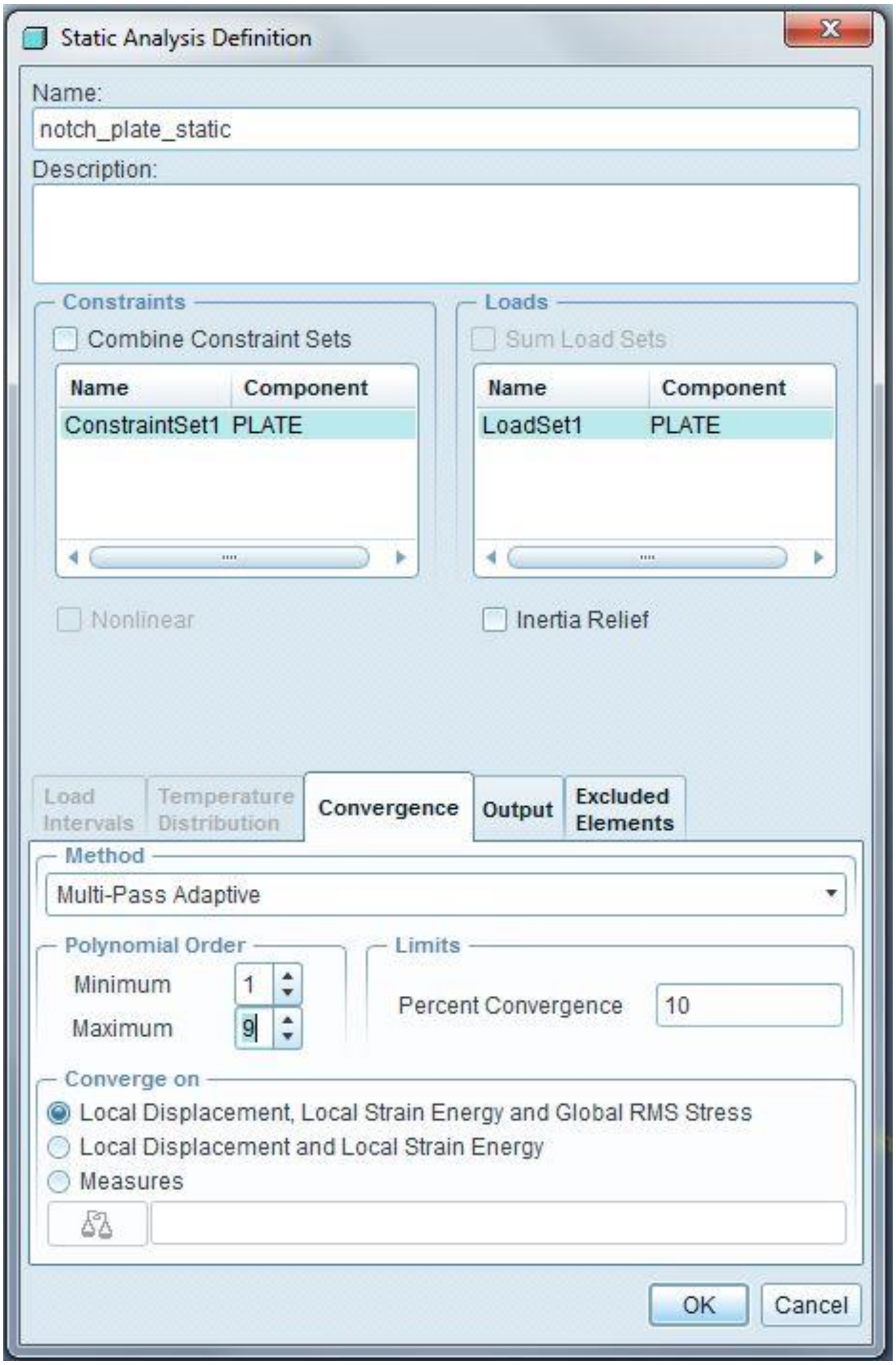
Click **Close**

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

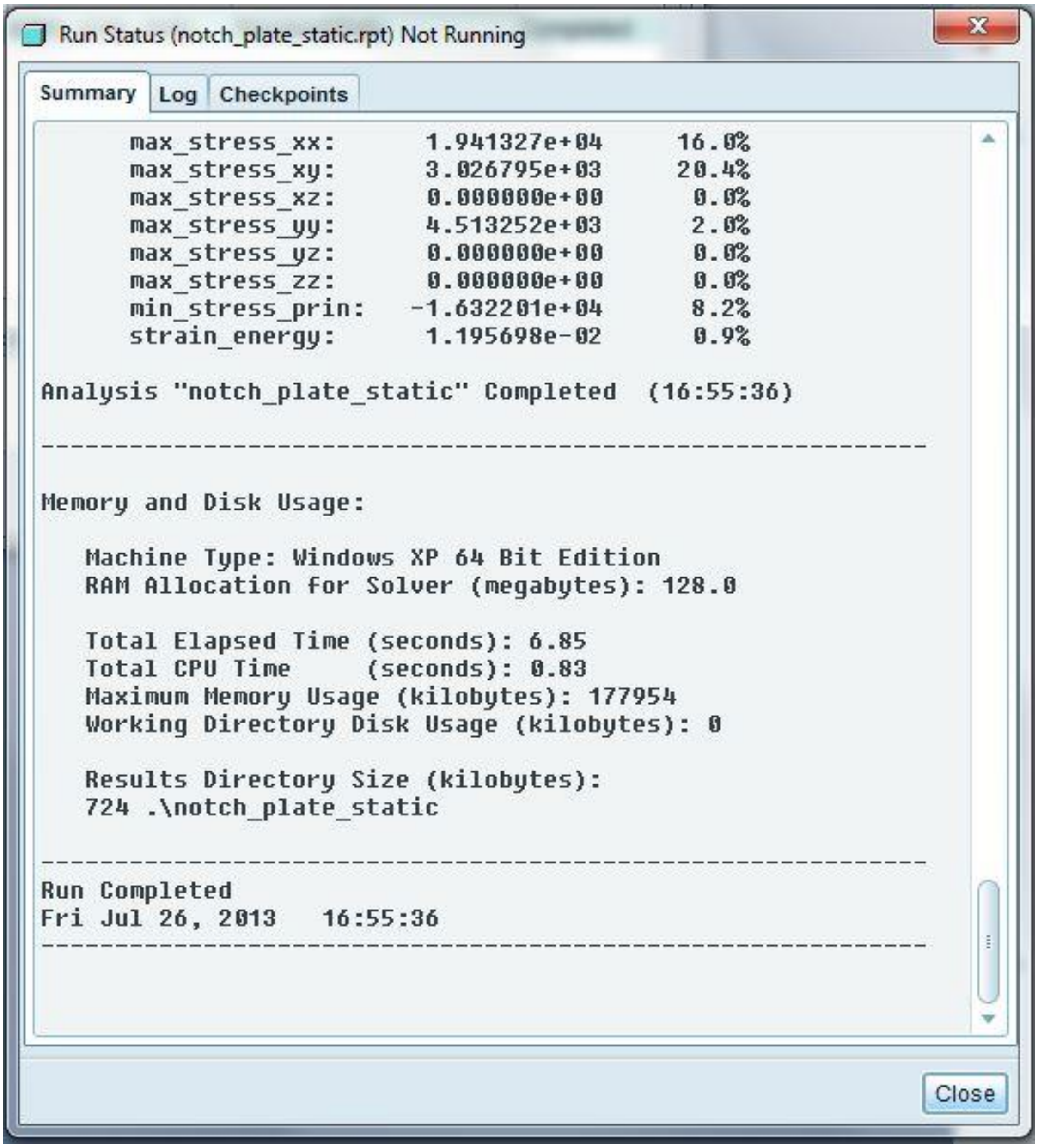
Edit the quick check analysis (right mouse click on it and select edit)

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

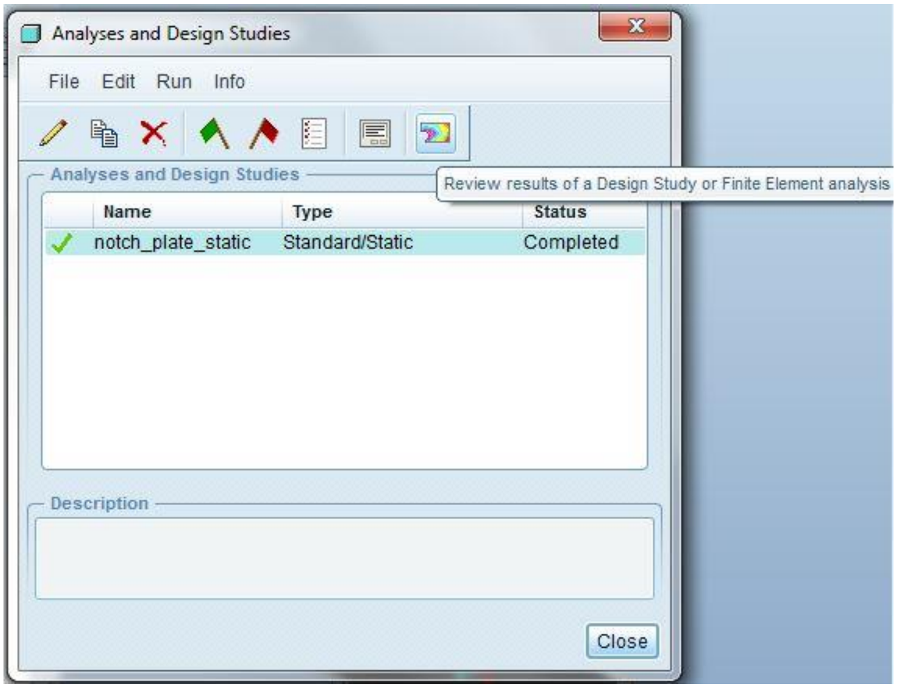


Run Status (notch_plate_static.rpt) Not Running
Summary
Log
Checkpoints
max_stress_xx: 1.941327e+04 16.0%
max_stress_xy: 3.026795e+03 20.4%
max_stress_xz: 0.000000e+00 0.0%
max_stress_yy: 4.513252e+03 2.0%
max_stress_yz: 0.000000e+00 0.0%
max_stress_zz: 0.000000e+00 0.0%
min_stress_prin: -1.632201e+04 8.2%
strain_energy: 1.195698e-02 0.9%
Analysis "notch_plate_static" Completed (16:55:36)
Memory and Disk Usage:
Machine Type: Windows XP 64 Bit Edition
RAM Allocation for Solver (megabytes): 128.0
Total Elapsed Time (seconds): 6.85
Total CPU Time (seconds): 0.83
Maximum Memory Usage (kilobytes): 177954
Working Directory Disk Usage (kilobytes): 0
Results Directory Size (kilobytes):
724 .\notch_plate_static
Run Completed
Fri Jul 26, 2013 16:55:36
Close

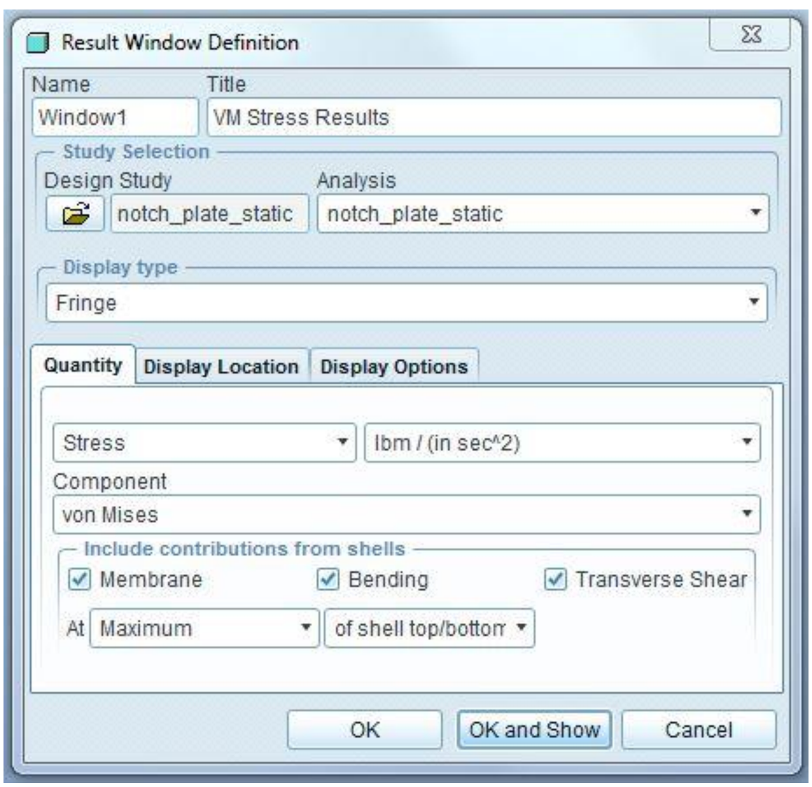
6.1 Results of Static Analysis

Create a Von Mises stress definition sheet as follows:

Click on the review result button

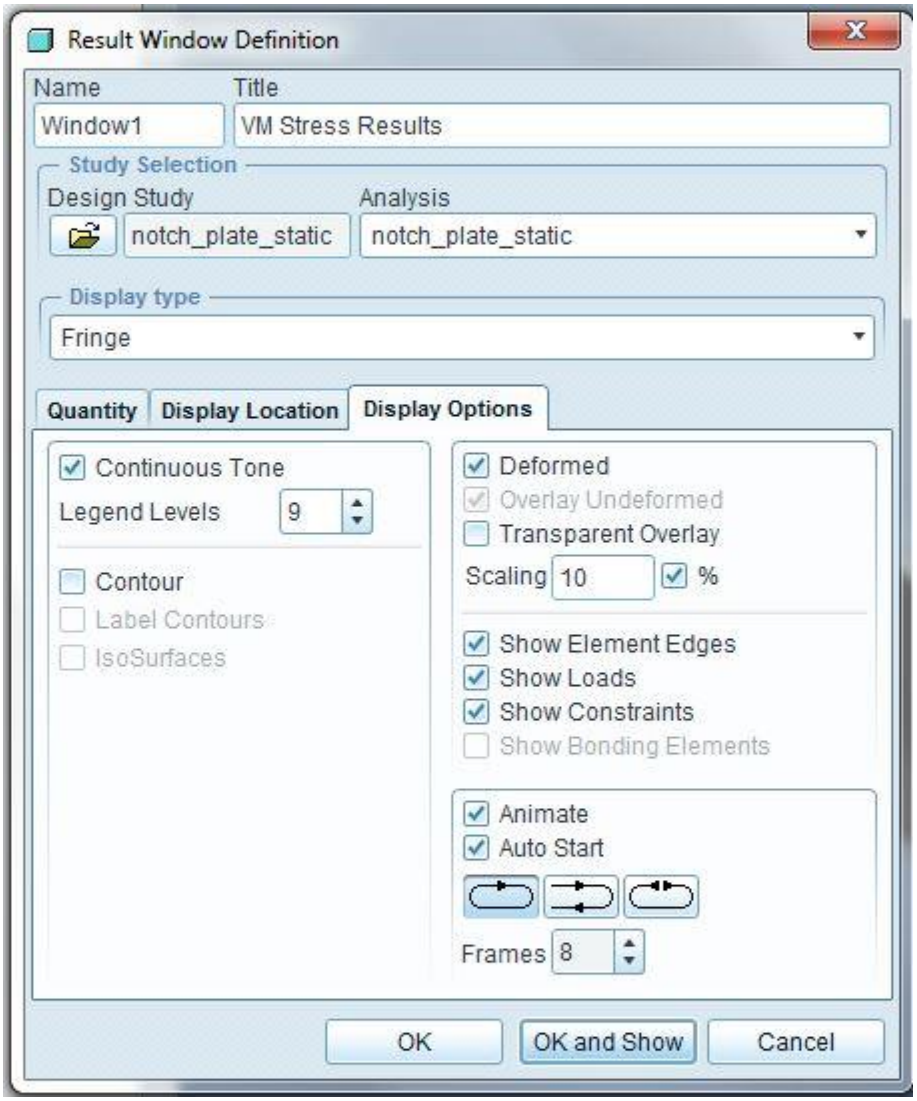


Fill out the result window as follows:

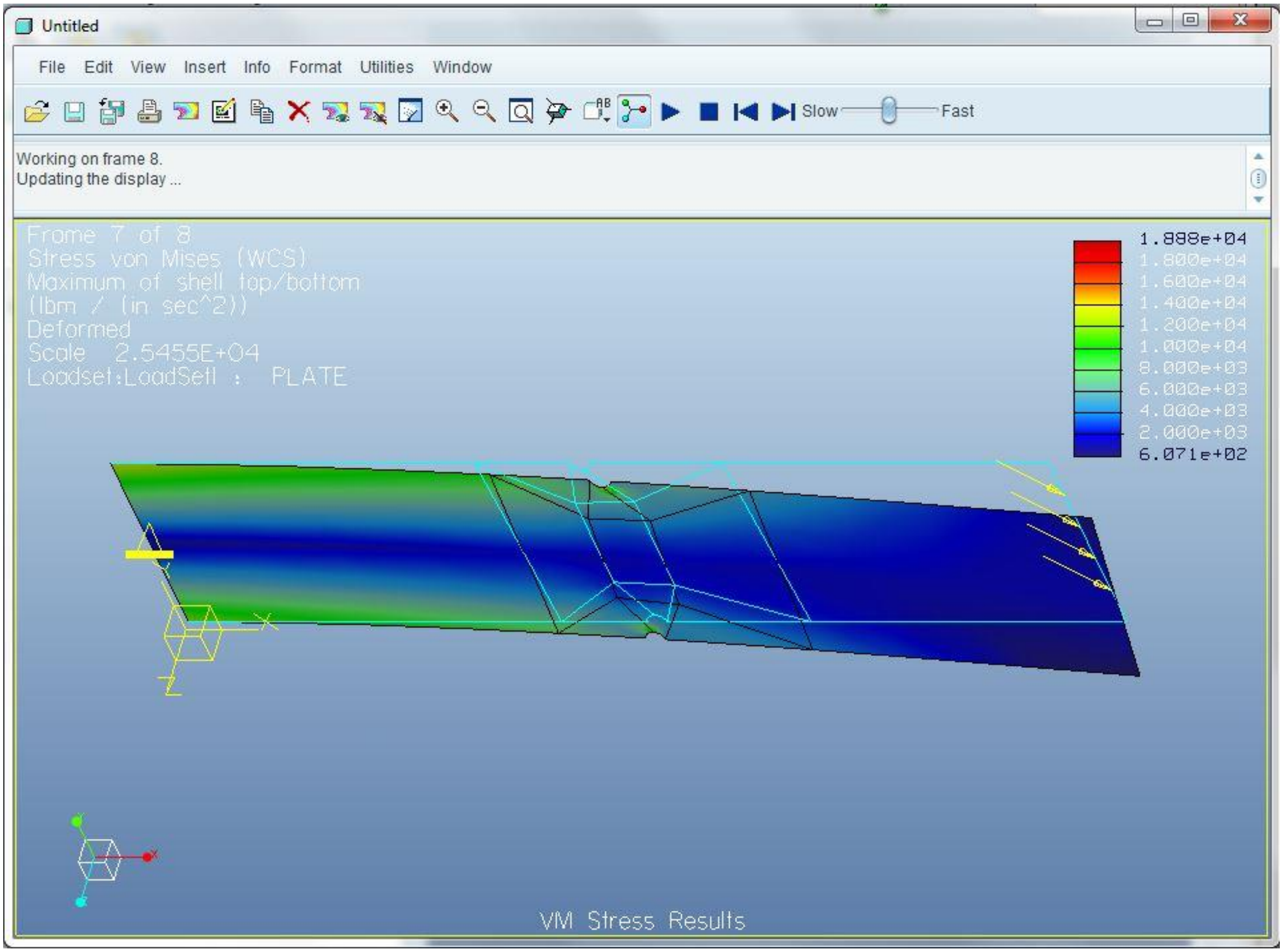


Click on Display Options Tab

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 $x10^4$ Psi, the material has an Sy =3.6 $x10^4$, 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.

6.2 Local Sensitivity Study

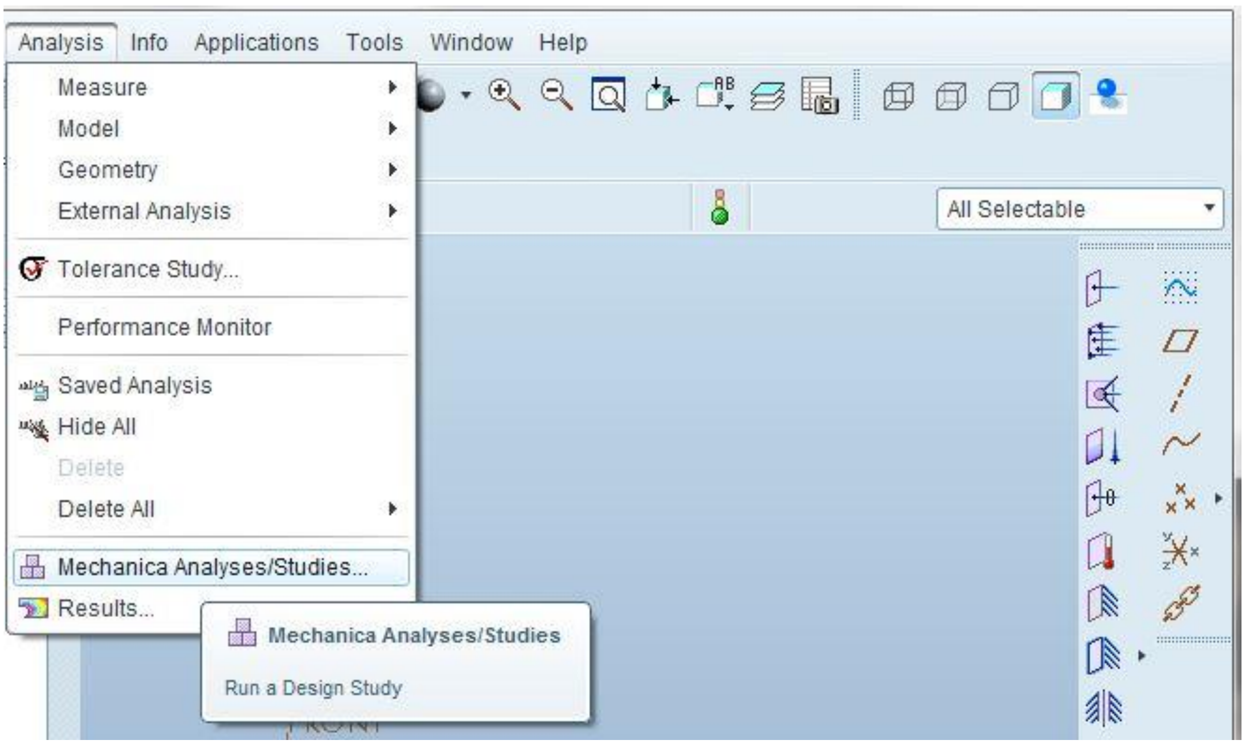
The objective of a local Sensitivity study is to look at small changes of the measures (Von Mises stresses) for small independent variations of each design parameter used (+-1 to 2%).

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.

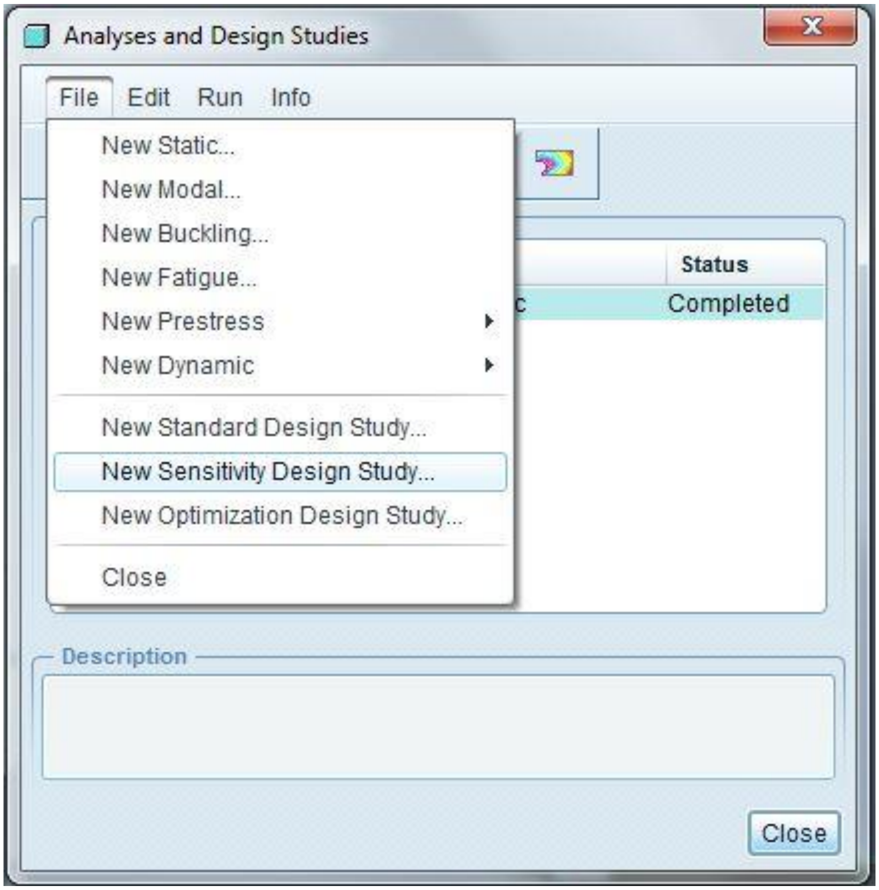
Design Controls:- This command can be used to define design controls which can used later in a design study like global sensitivity, optimization etc. Design parameter can be Dimension/Pro/Engineer parameter/section dimension. Section dimension option can be used only if there some sections already defined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

Procedure:

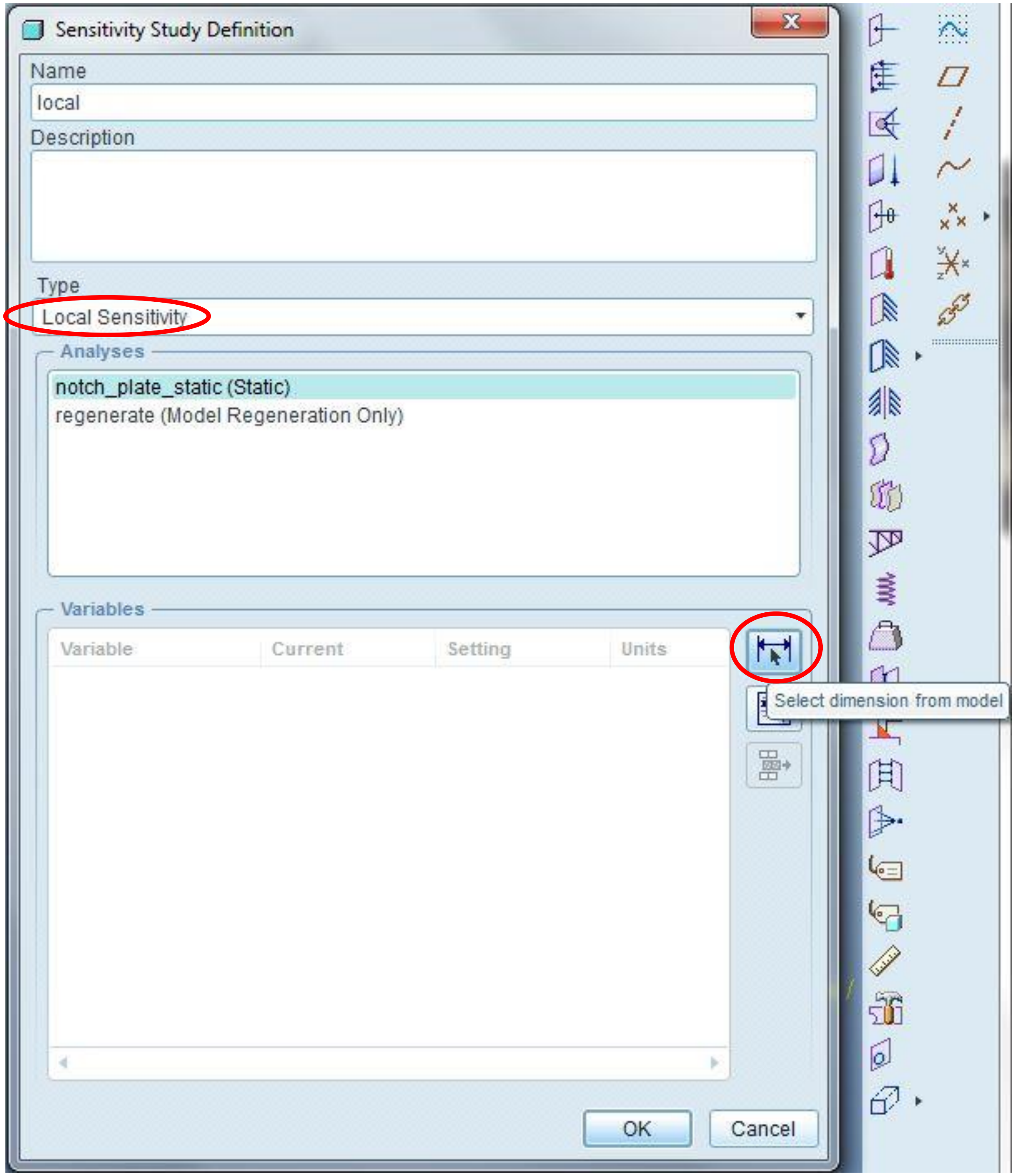
Select: **Analysis>Mechanica Analyses/Studies**



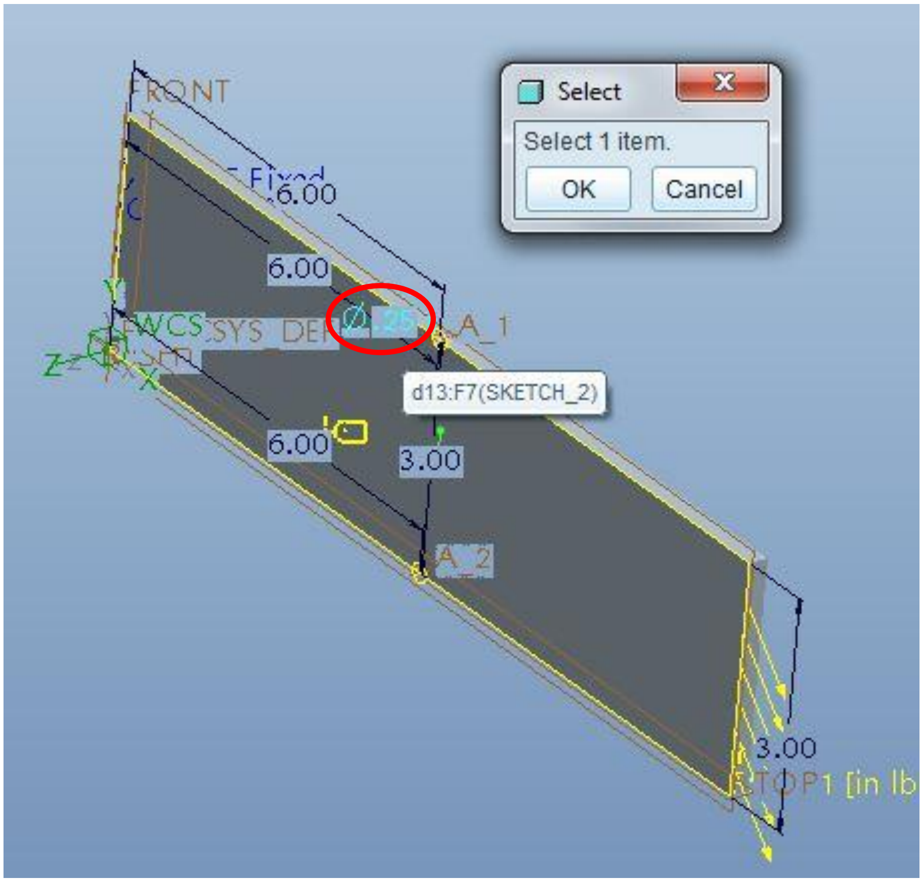
File>New Sensitivity Design Study



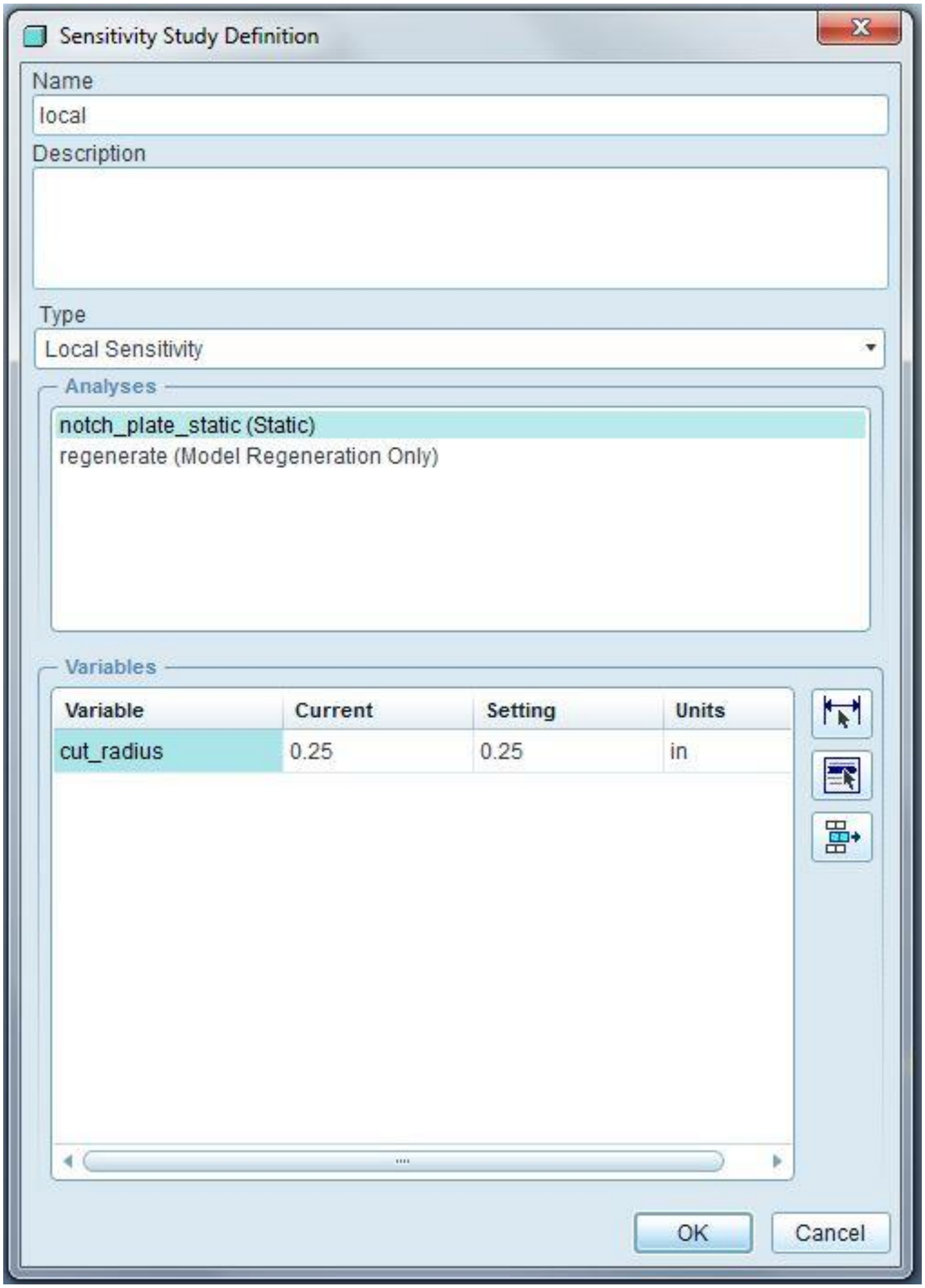
Define study name>Type Local Sensitivity>Select dimension from model



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

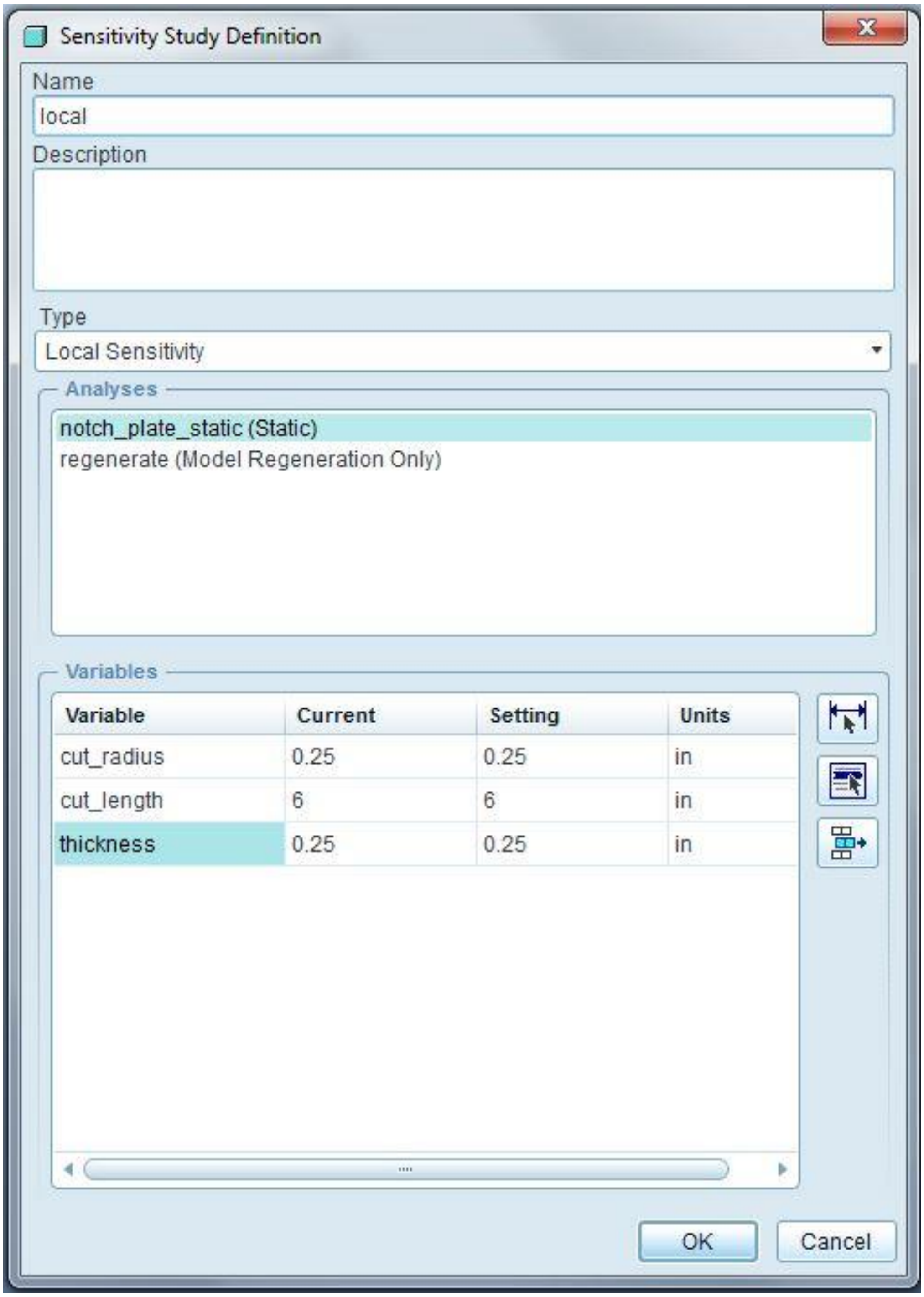


Click **OK**



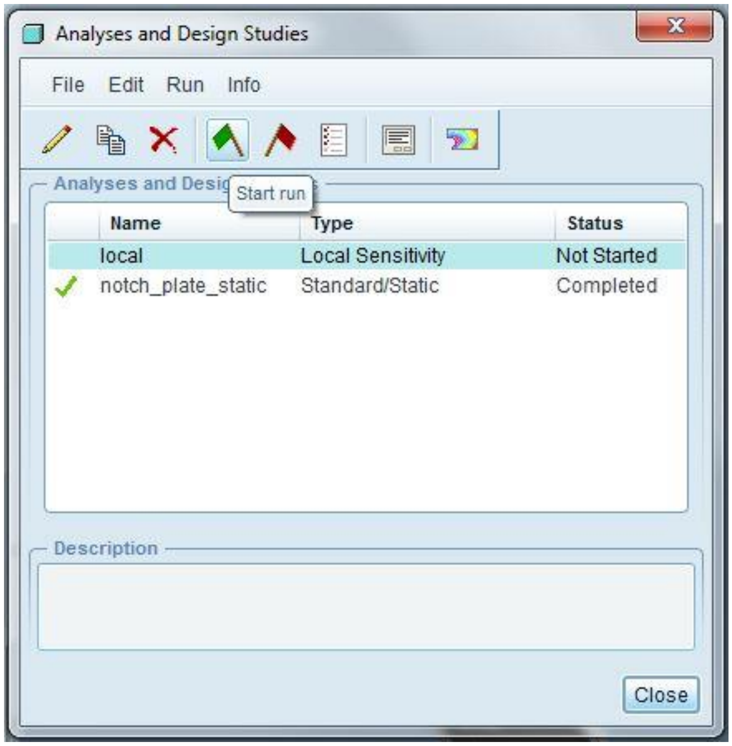
Sensitivity Study Definition
Name
local
Description
Type
Local Sensitivity
Analyses
notch_plate_static (Static)
regenerate (Model Regeneration Only)
Variables
Variable
Current
Setting
Units
cut_radius
0.25
0.25
in
OK
Cancel

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

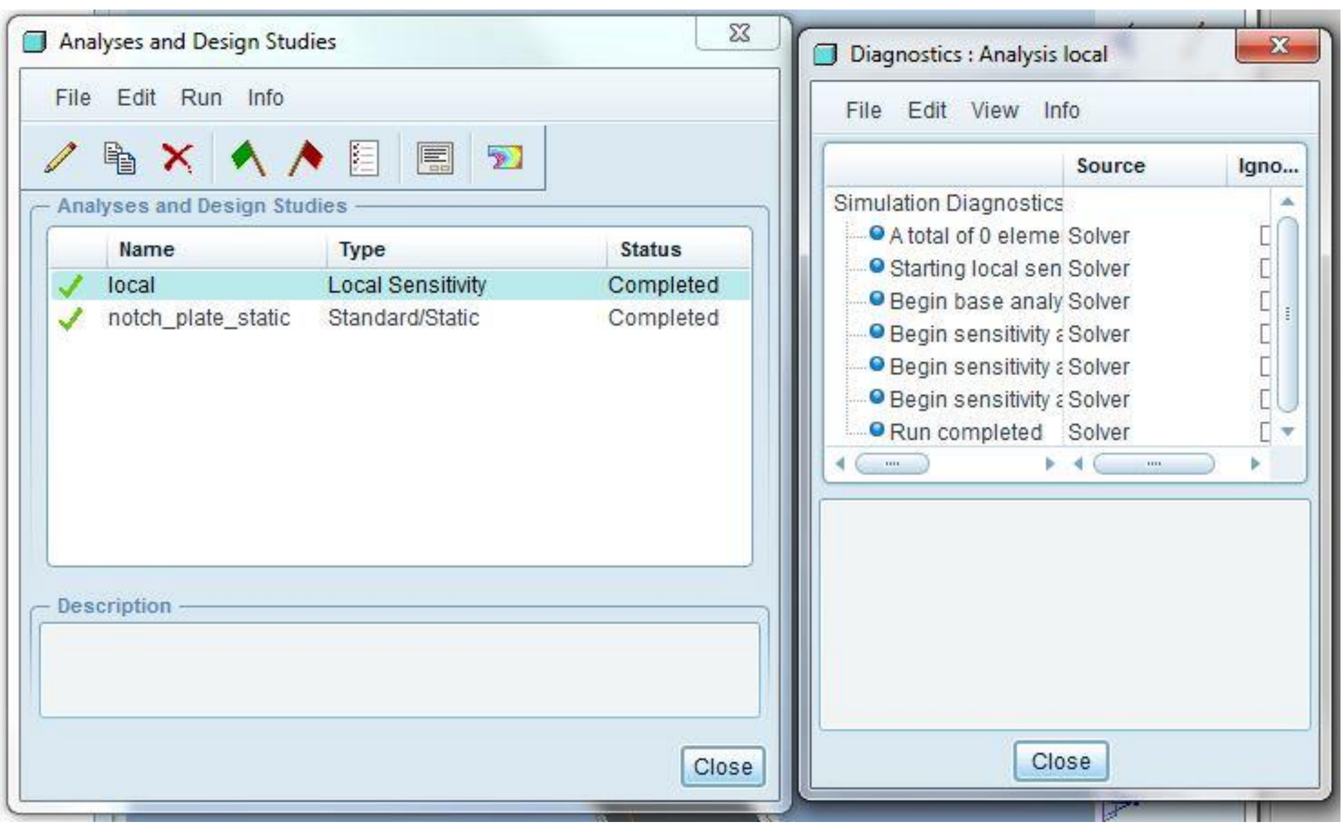


Click on **OK**

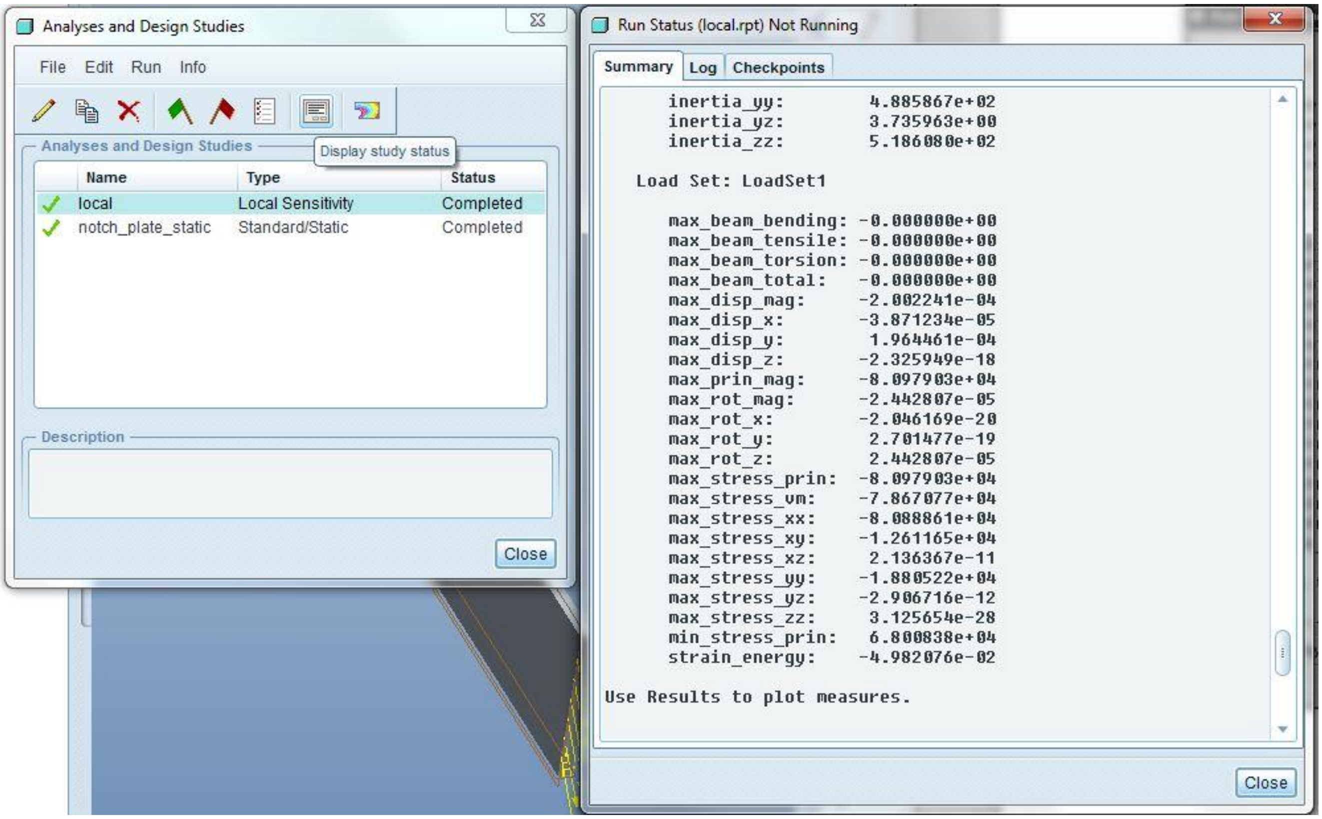
Run the local sensitivity study created (with the design study selected) click on **Start**



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

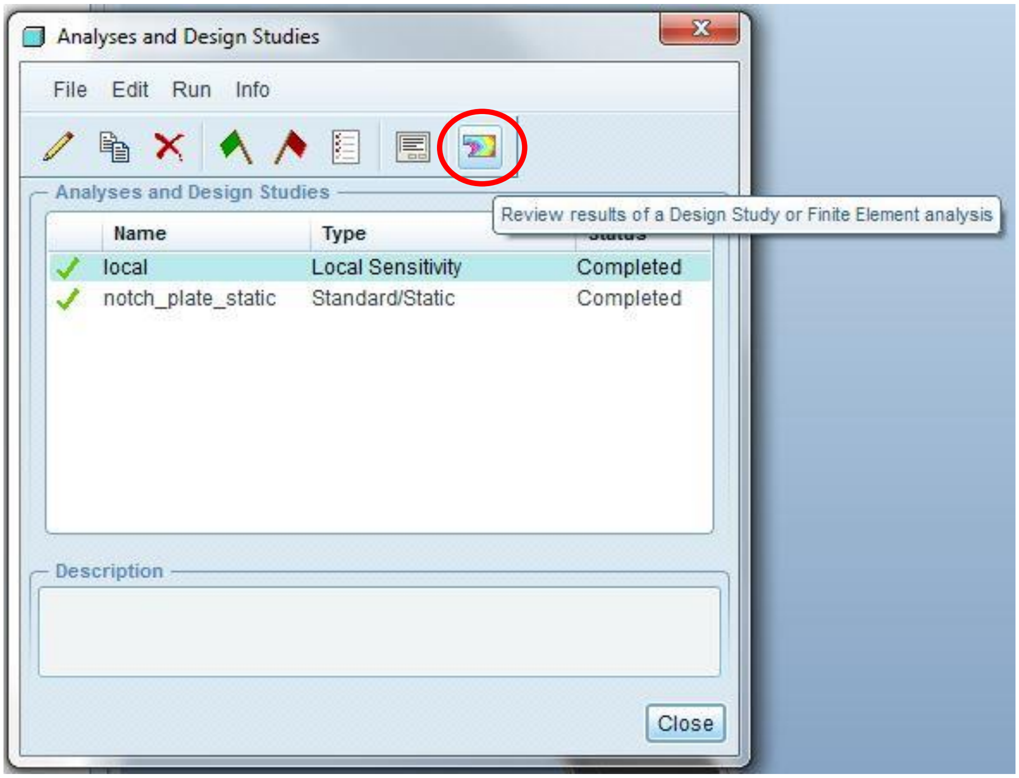


It takes approximately three minutes to complete the analysis. Check the status window to see the results



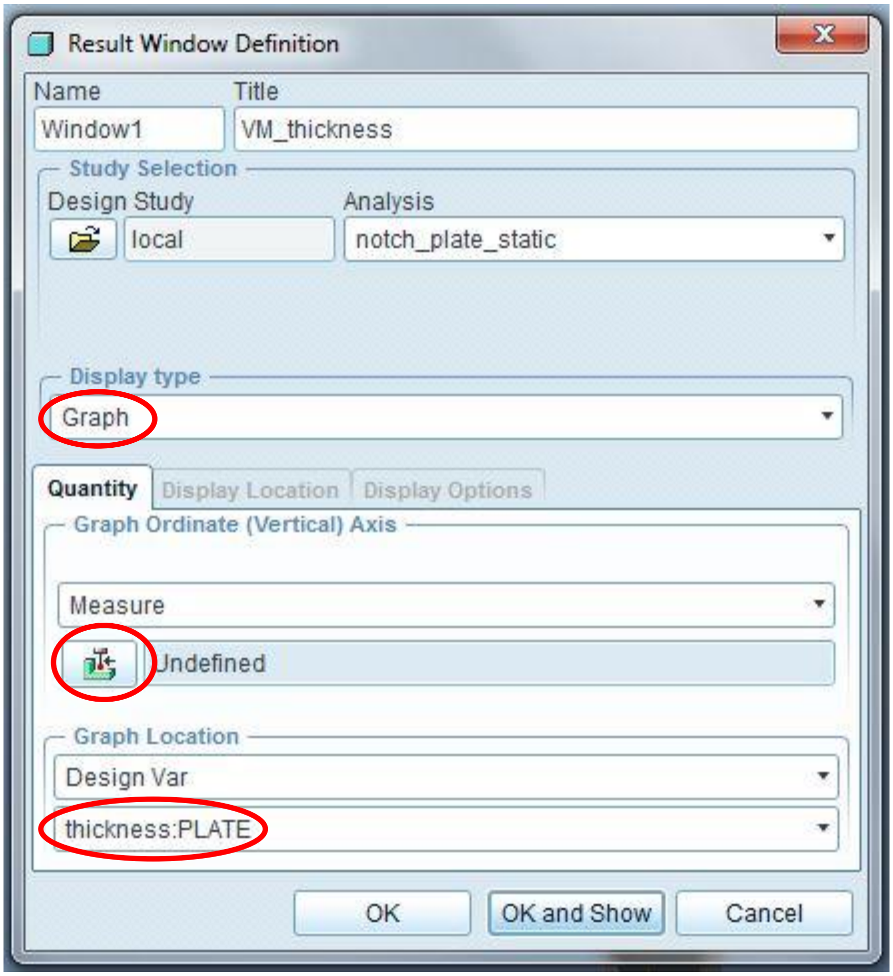
Click **Close**

Click on the review result button

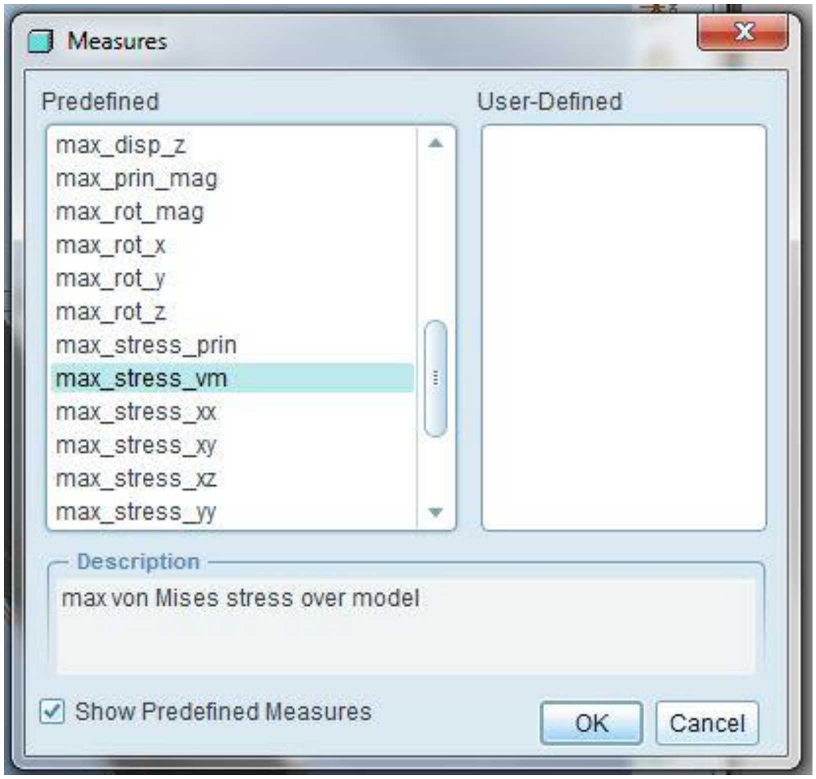


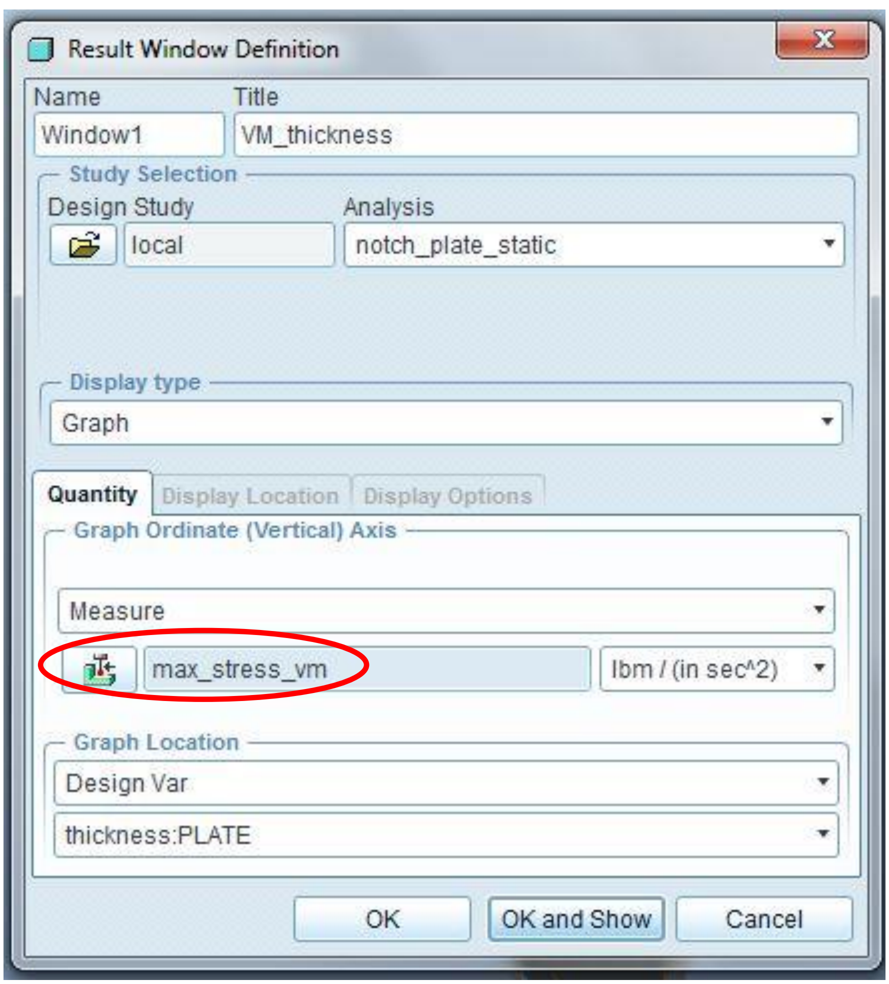
The following box appears

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

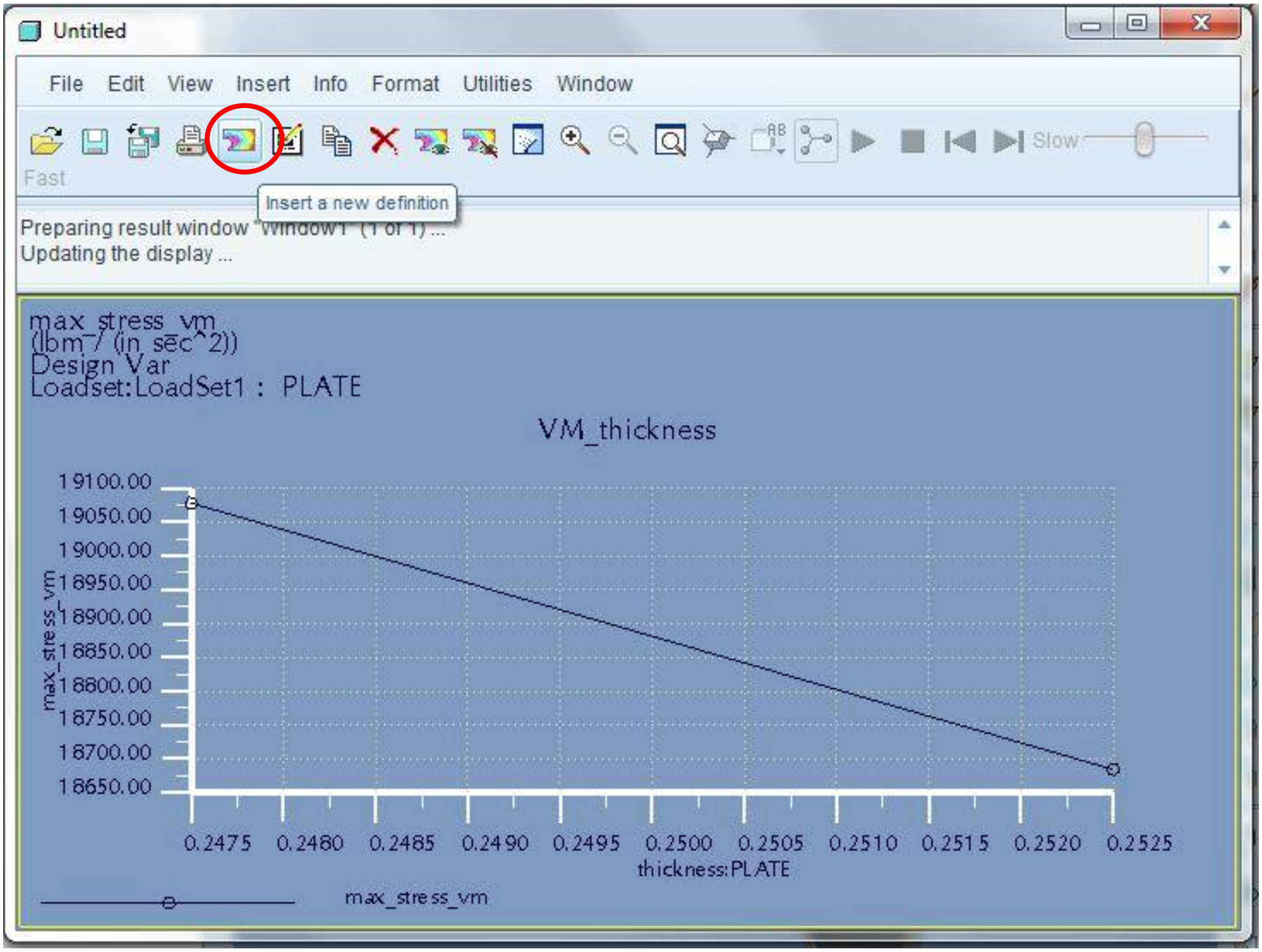


Select max_stress_vm as a measure and select Design Var from the graph location, make sure that the thickness variable has been selected


Measures
Predefined
User-Defined
max_disp_z
max_prin_mag
max_rot_mag
max_rot_x
max_rot_y
max_rot_z
max_stress_prin
max_stress_vm
max_stress_xx
max_stress_xy
max_stress_xz
max_stress_yy
Description
max von Mises stress over model
Show Predefined Measures
OK
Cancel

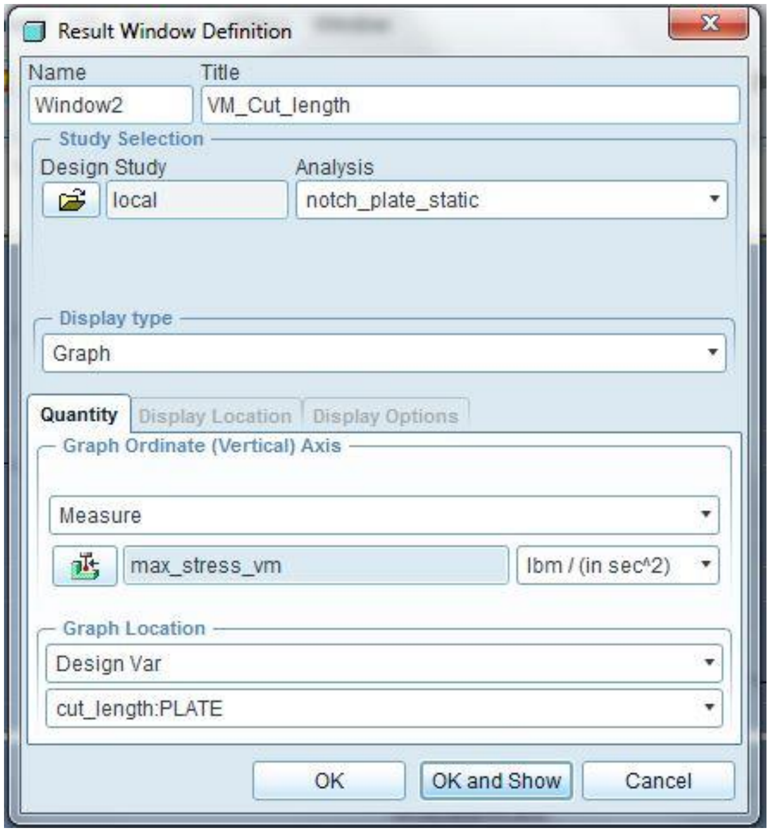

Result Window Definition
Name
Title
Window1
VM_thickness
Study Selection
Design Study
Analysis
local
notch_plate_static
Display type
Graph
Quantity
Display Location
Display Options
Graph Ordinate (Vertical) Axis
Measure
max_stress_vm
lbm / (in sec^2)
Graph Location
Design Var
thickness:PLATE
OK
OK and Show
Cancel

Click on **OK and Show**



Click on the insert a new definition button

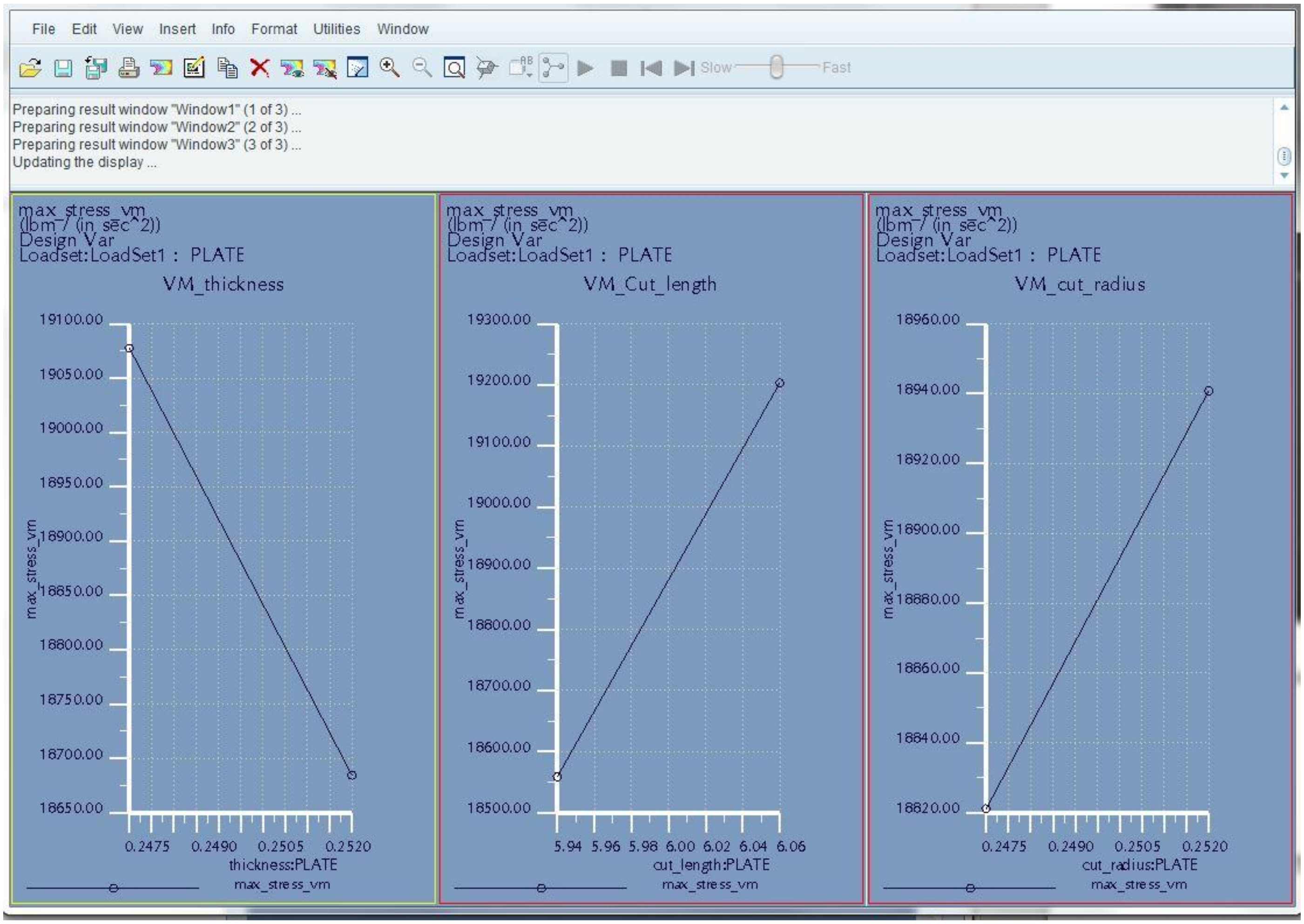
Repeat the procedure to create another window, change the design variable to Cut Length



Repeat the procedure to create another window, change the design variable to Cut_radius

Click on OK and Show

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.

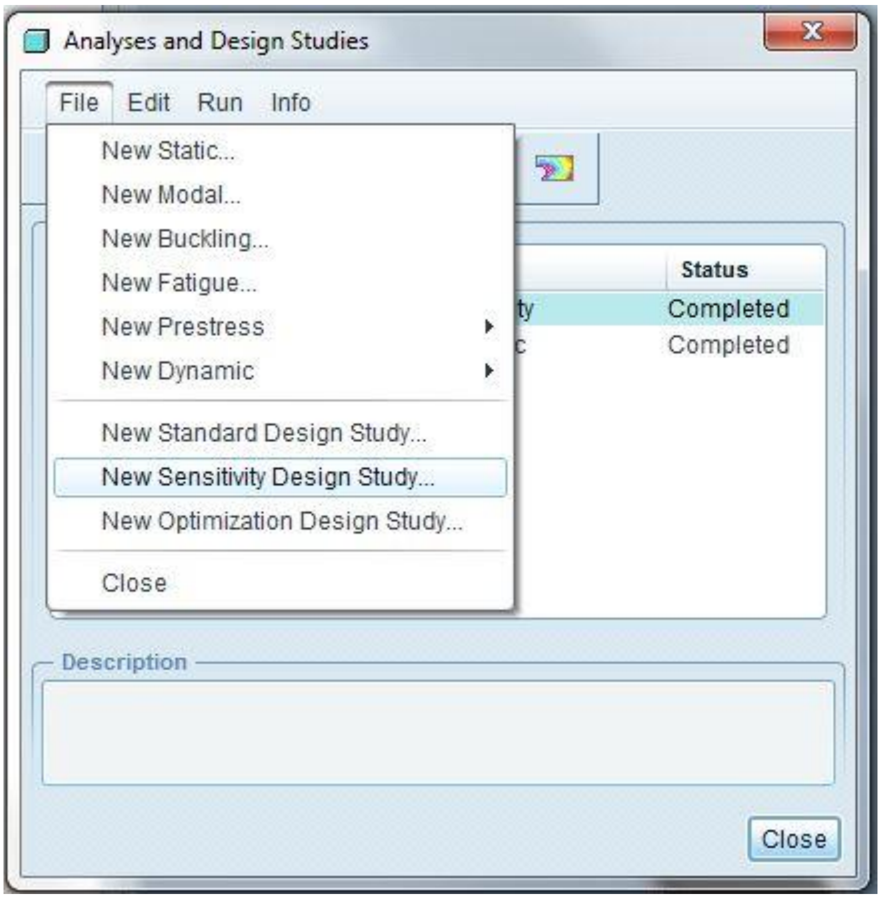
Select: **File>Exit Results>No>Close**

6.3 Global Sensitivity Study

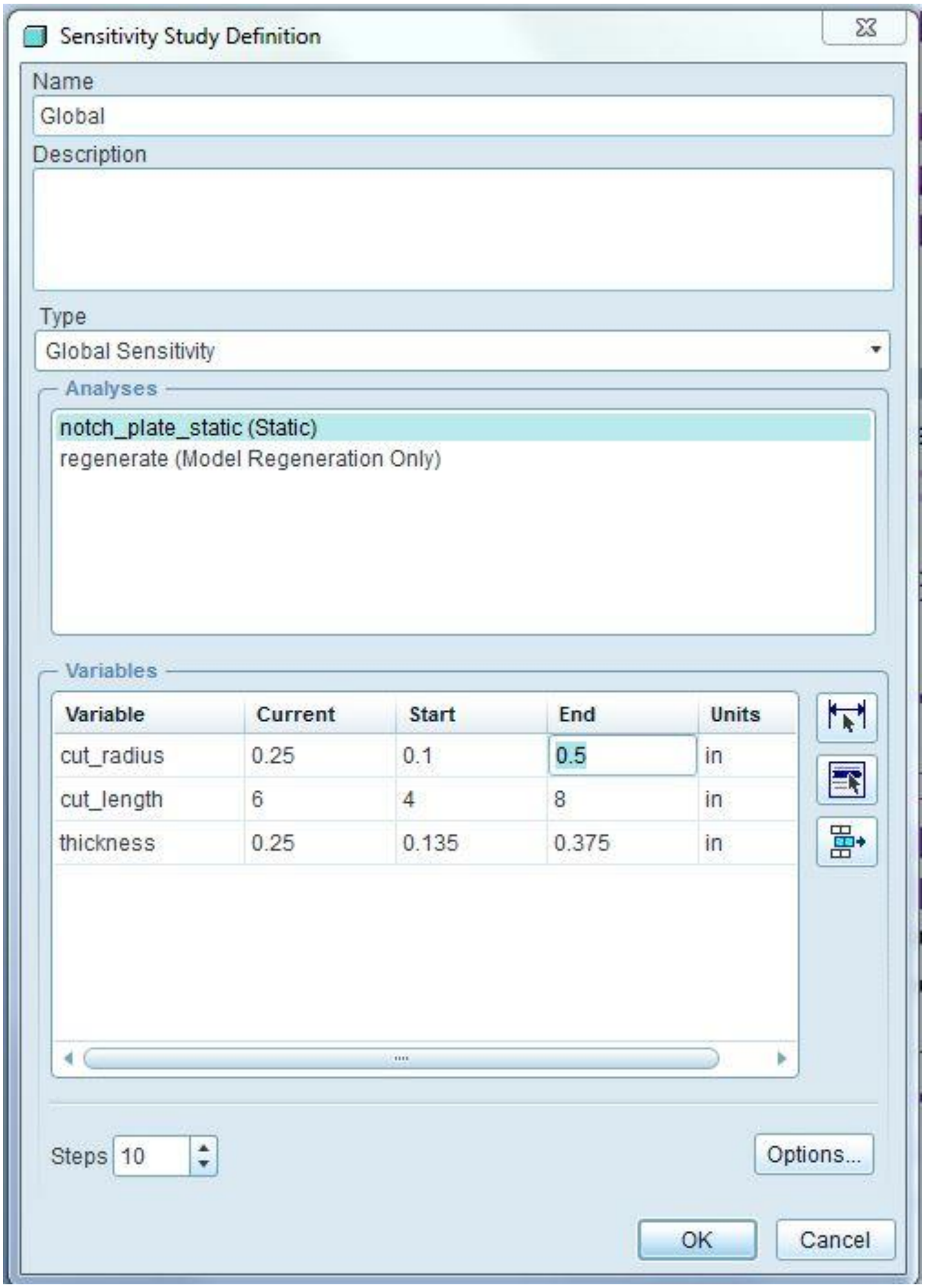
The objective of the global sensitivity study is to look at the variations of all parameters into each step of the process as defined by the user

Procedure:

File>New Sensitivity Design Study

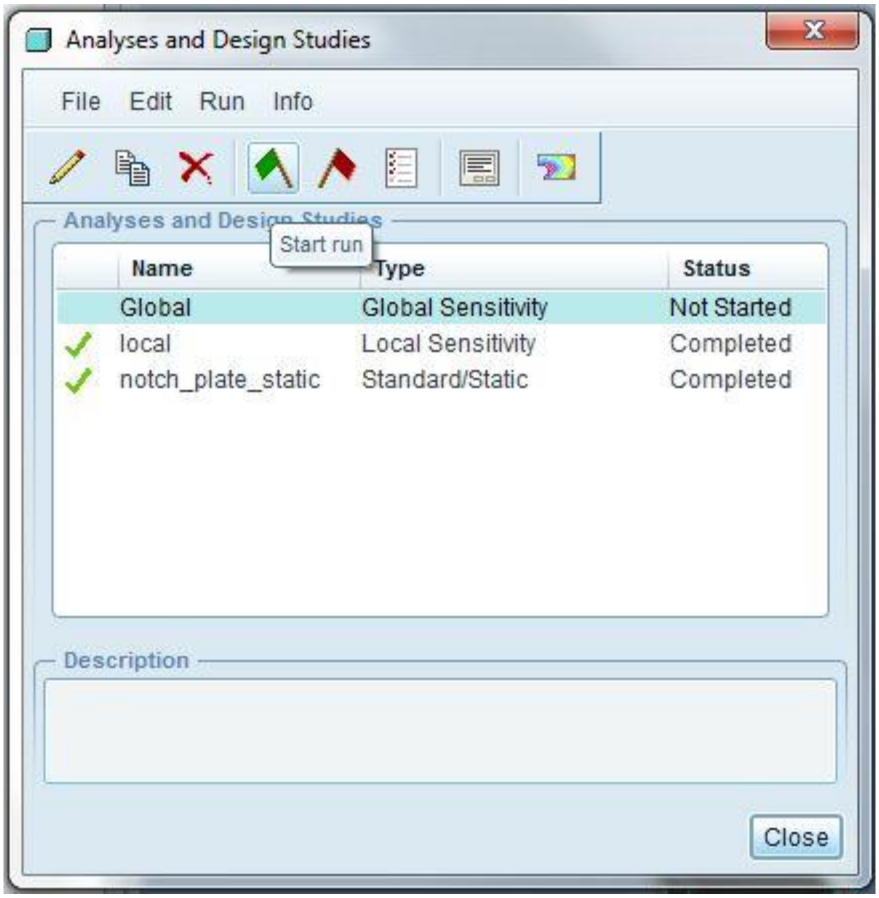


Define study name>Type **Global Sensitivity**>Select dimension from model>Select the three variables from the model>Define the start and end values of each variable as below



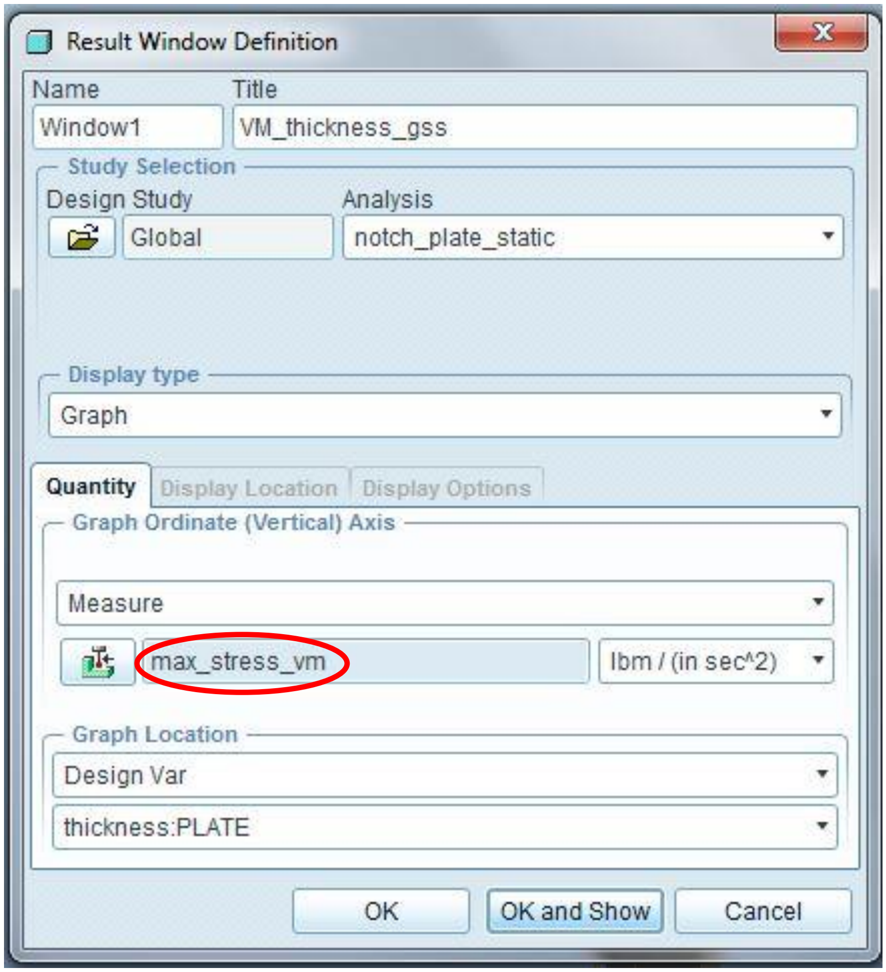
Click on **OK**

Run the global sensitivity study



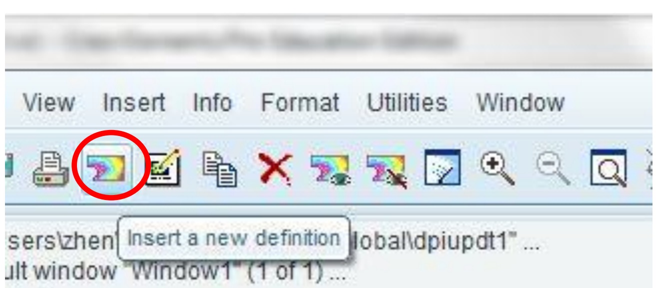
Click on Review result button

Fill out the dialog boxes with following data



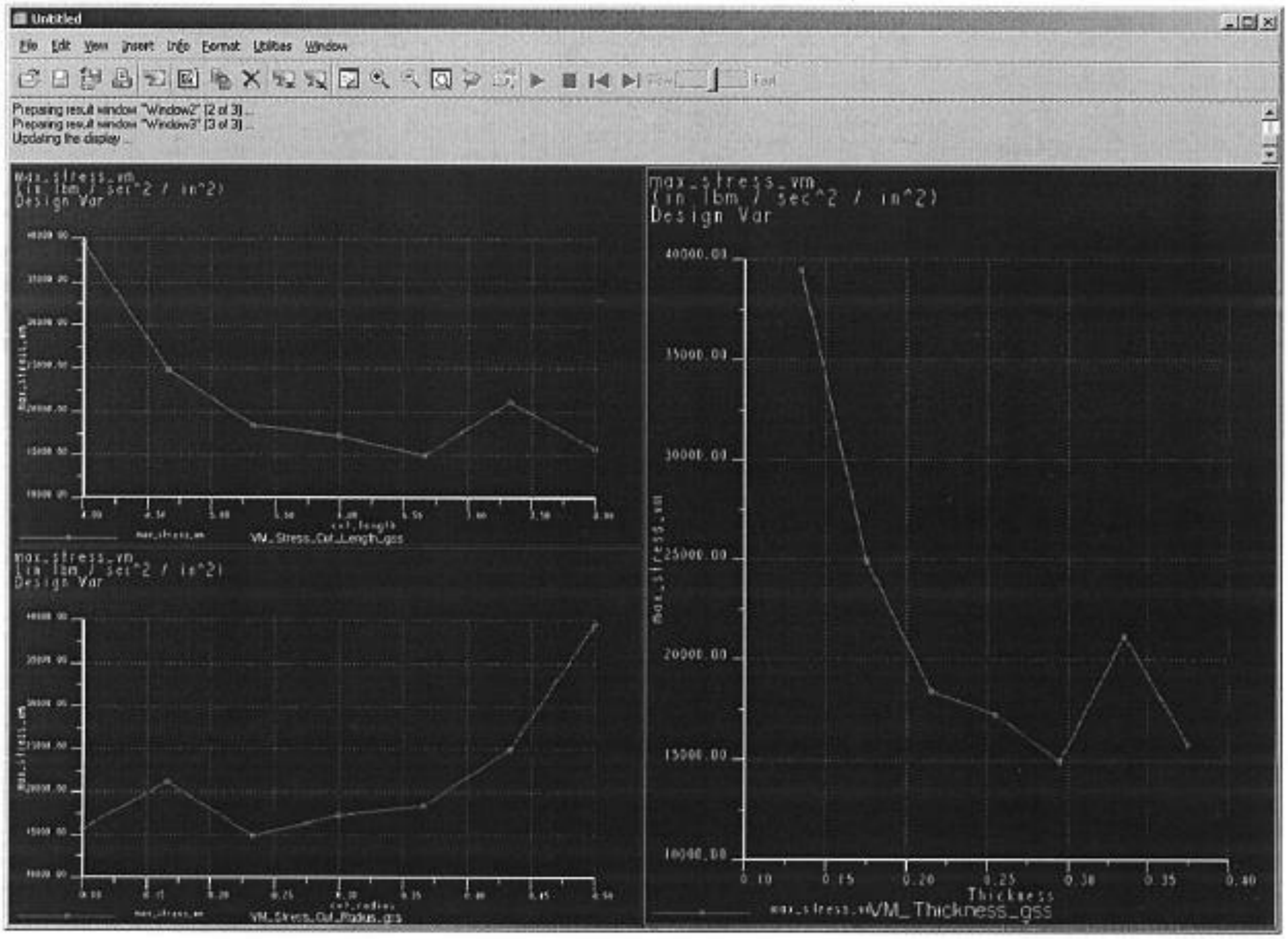
Click on **Ok and show**

Click on the insert a new definition button



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

Click on **Ok and show**



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

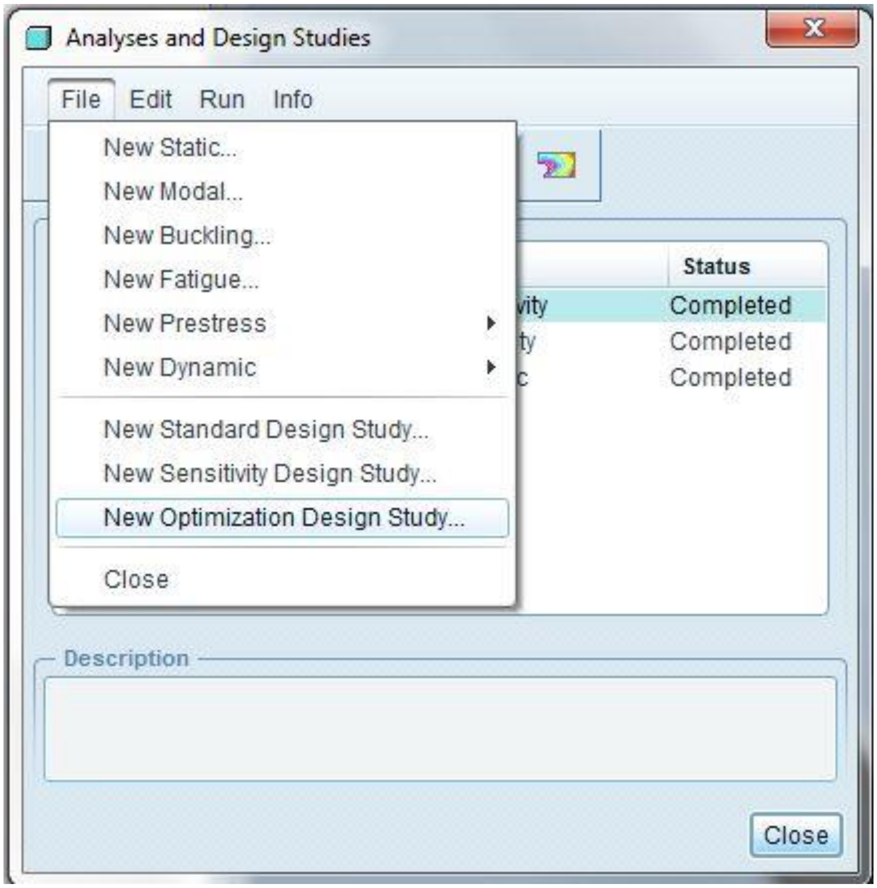
The conclusions we have are:

For minimum VM Stress, we need the following:

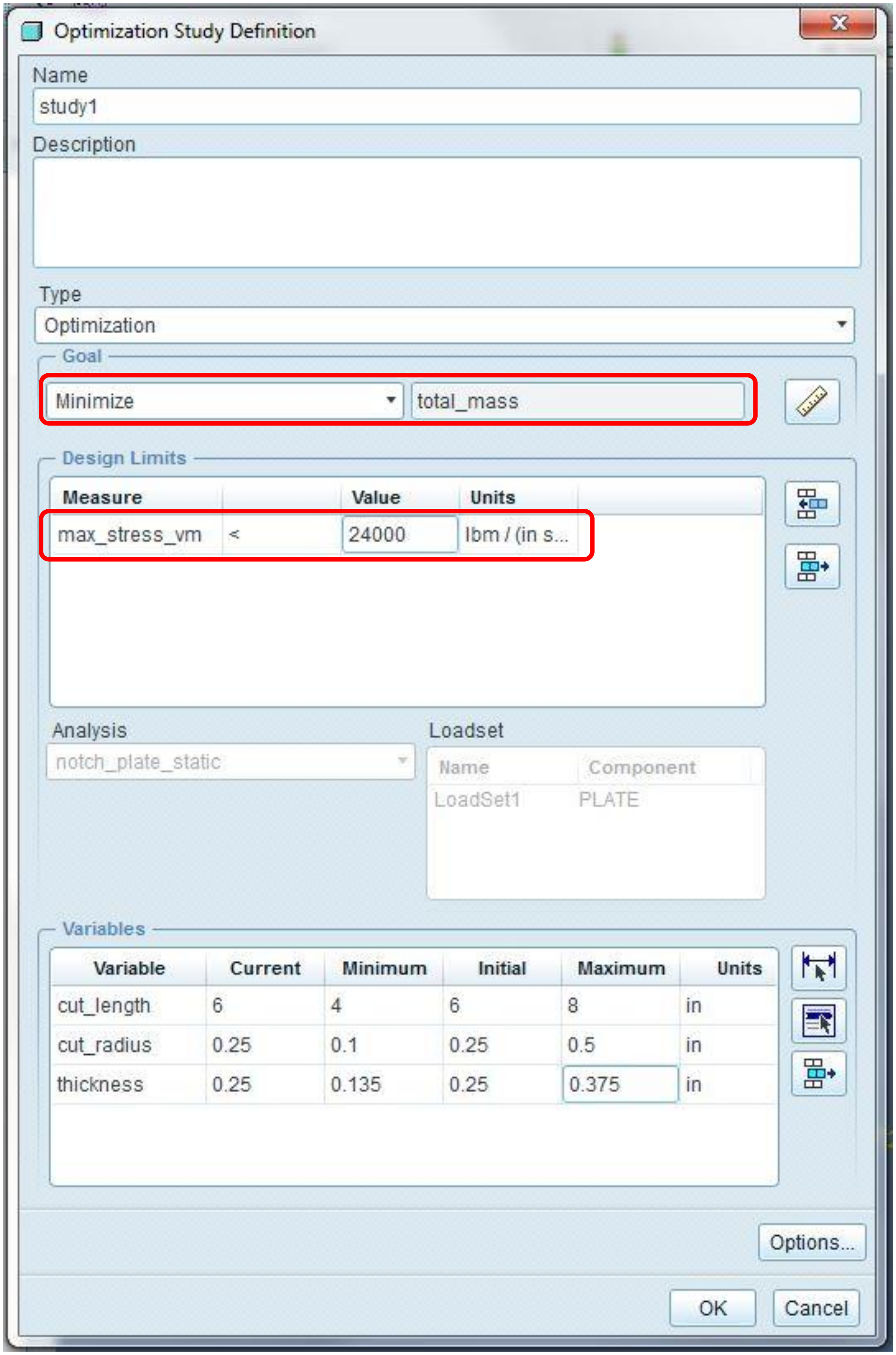
-Maximum Thickness
-Maximum Cut length
-Minimum Cut Radius

6.4 The Optimization Study on Total Mass

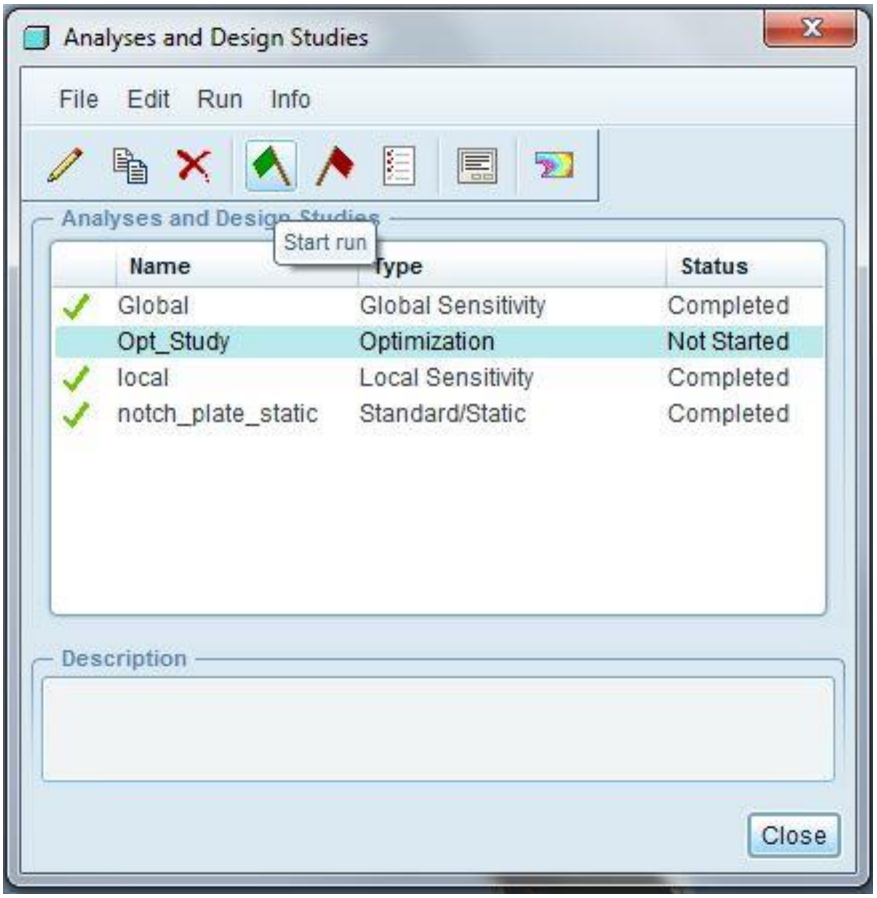
File>New Optimization Design Study



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)

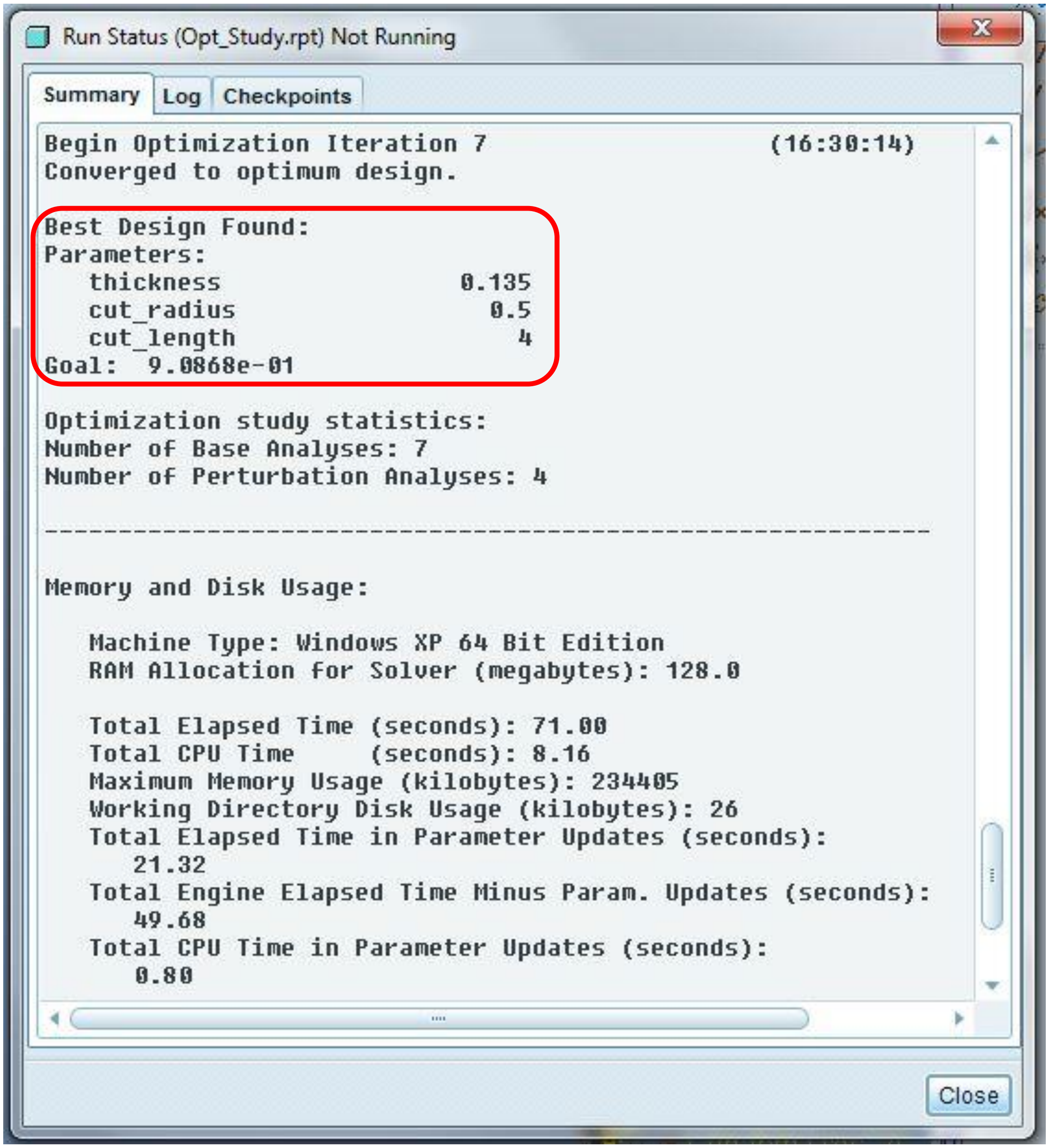


Run the design study, it will take several minutes.



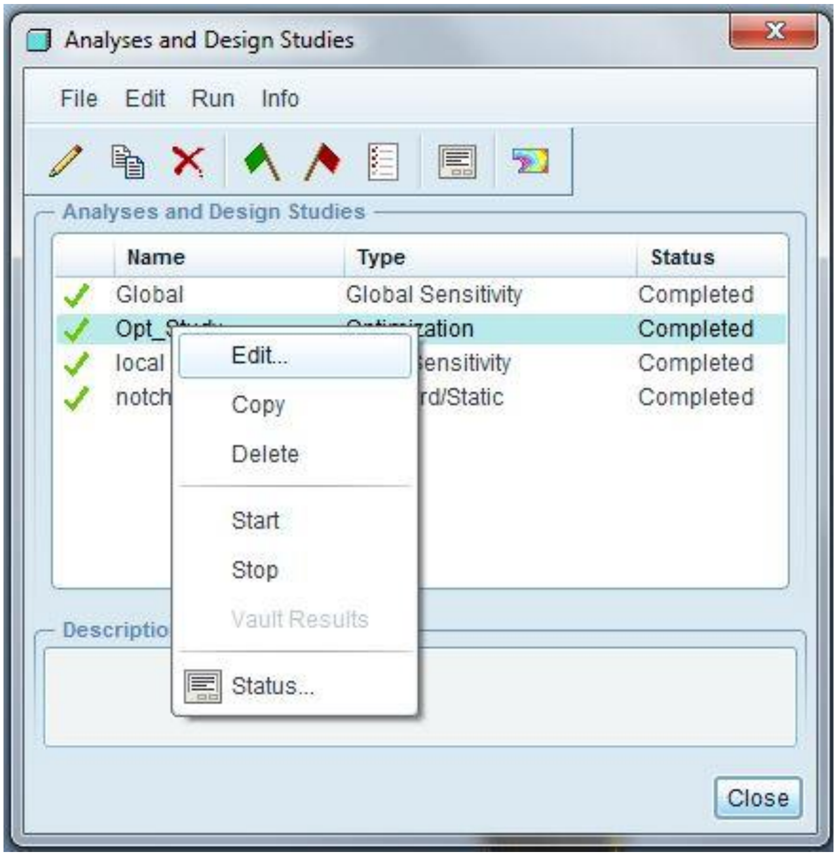
Check the status file and compare it to the shown below.

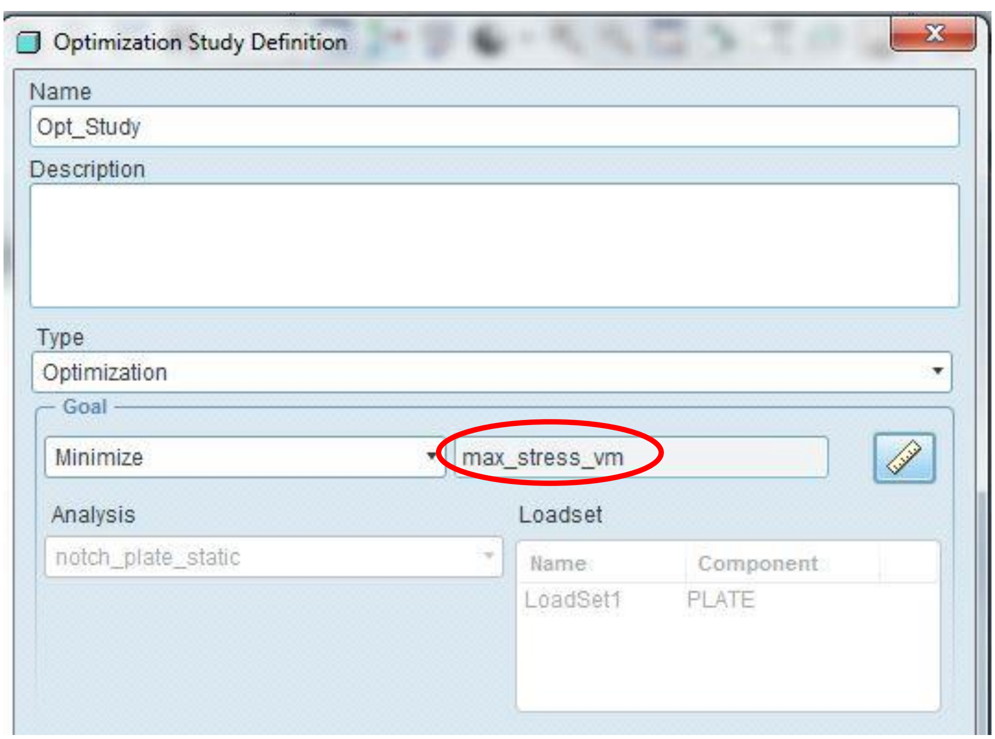
The final values:



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)





Click OK and run the study

The final values to get the Minimum Von Mises Stress