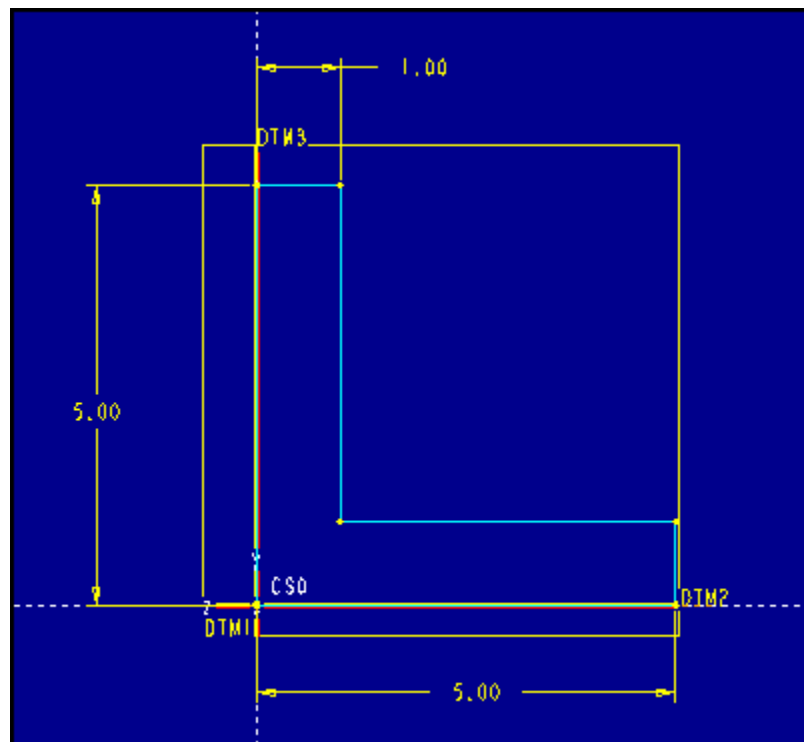


Pro/Mechanica Tutorial

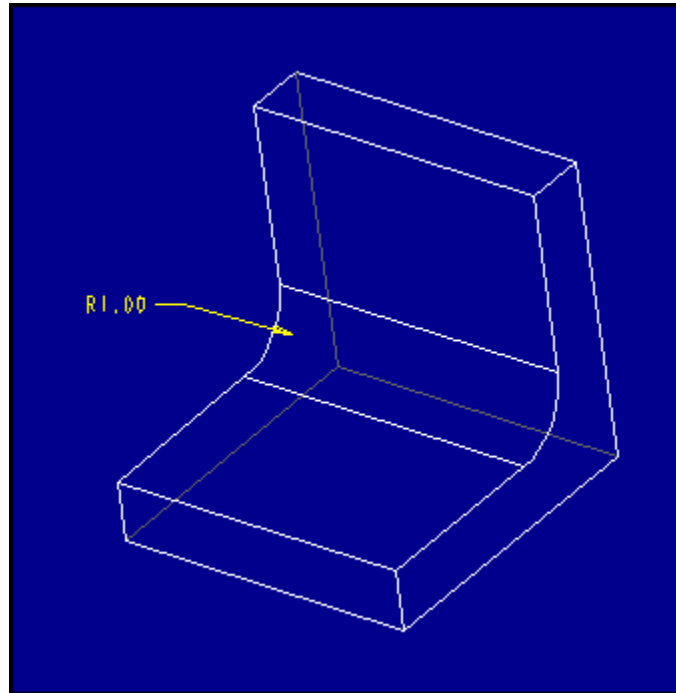
Getting Started

The first thing we need is to create our initial design concept. To keep it simple, we will start out with an L bracket.

- Create a part in Pro/E and name it **BRACKET**.
- Create default datums and default coordinate system.
- Create an L-shaped protrusion with a depth of 5.00 as shown below. Select DTM1 as your sketching plane and DTM2 as your top reference. (Take note of the sketch below in reference to the datum planes).



- Create a 1.00 inch fillet round as a separate feature as pictured below.



This will represent our initial design concept. At this point we would like to learn a few things about this shape:

1. Will it carry our load in an efficient manner ?
 2. Where is it the weakest ?
 3. Can we better utilize our material and how ?
- Access Pro/MECHANICA from the part menu by selecting **MECHANICA**.
 - Since we will be doing structural analysis, select **STRUCTURE**
 - Then lets start to define our **MODEL**.

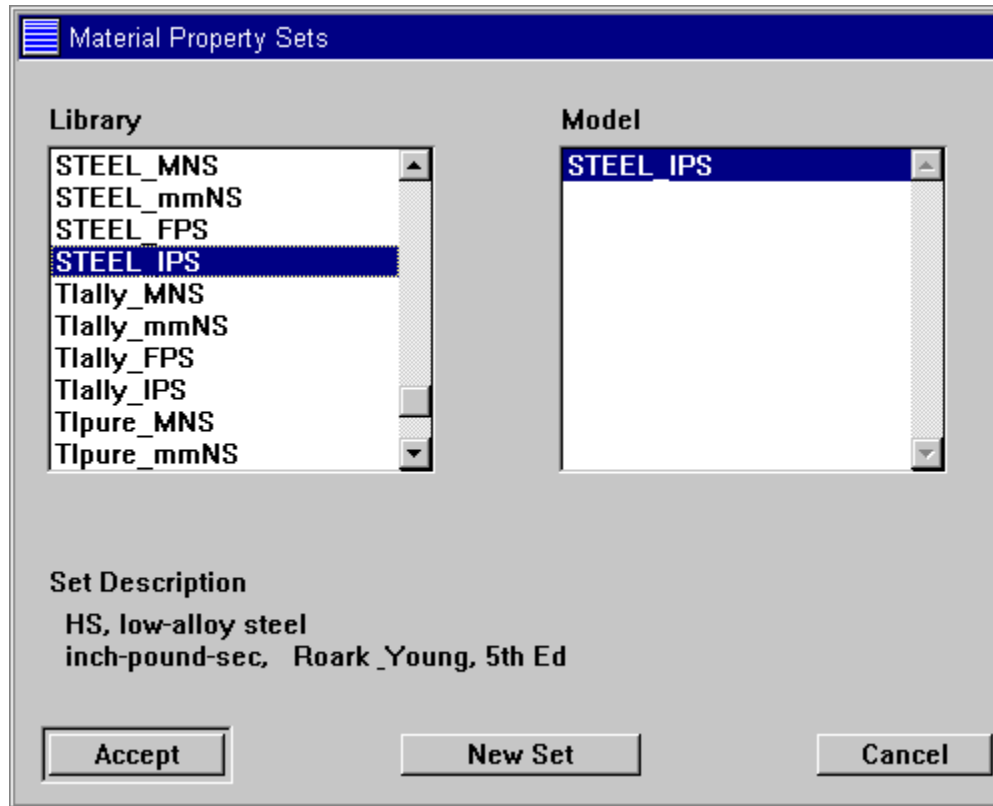
Operating Environment

The process of assigning material properties, constraints and loads to our model is analogous to setting up our operating environment. The information we provide will indicate how our model is fastened or restricted, what external forces are acting on it, and what are the material characteristics of our model.

Material Properties

Material properties can be assigned from the existing library or created as needed. For our bracket we will use Steel, which exists in the material library.

- Select **MATERIAL** then **ASSIGN** it to the **PART**.
- For this exercise we will use **STEEL_IPS** as shown below.



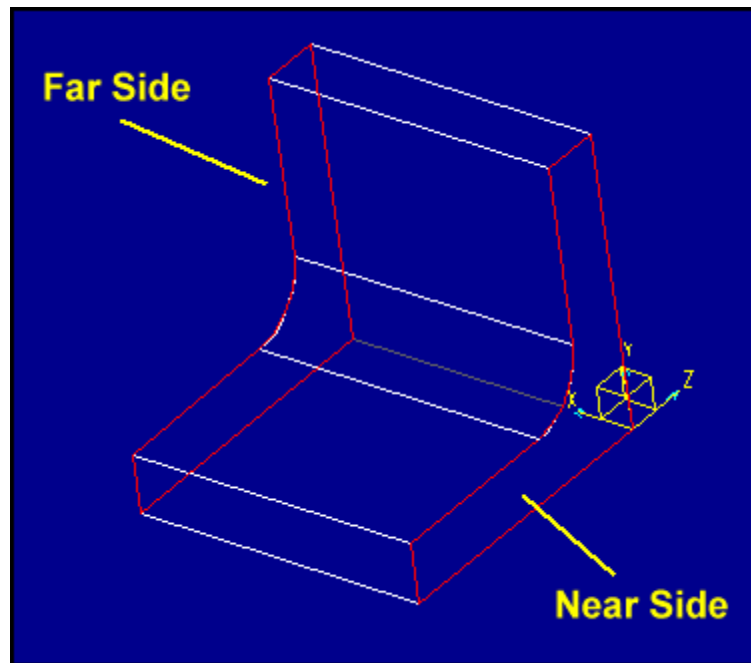
Idealizations



Idealizations allow you to apply Shells, Beams, Masses, Springs and Spot Welds to your model. We can simplify this model (to speed up computations) by representing it as a plate. Mechanica will create a mid-plane between the two surfaces we've selected. This mid-plane will then be used to analyze the model.

Select **SHELLS** followed by **PAIRS** and then **DEFINE PAIR**. Select the far and near side surfaces as shown below. By doing this you are manually creating the pairing for Mechanica to use.

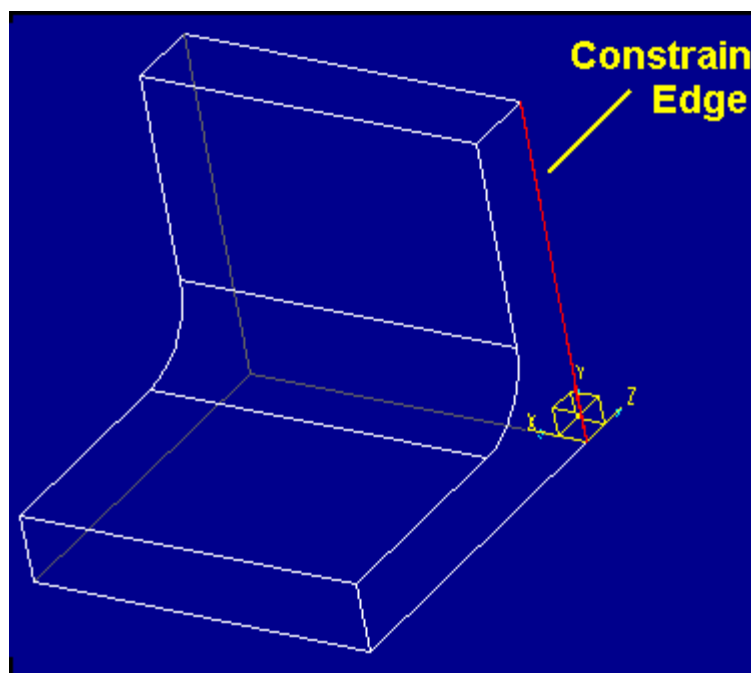
- Show the pair by selecting **COMPRESS** from the shell menu. The yellow surface in the center of our bracket represents the mid-plane.



Constraints

We will apply one constraint by fixing the back of our bracket, simulating a weld or several fasteners. By applying this constraint the back surface of the bracket will not be able to move or rotate in any direction.

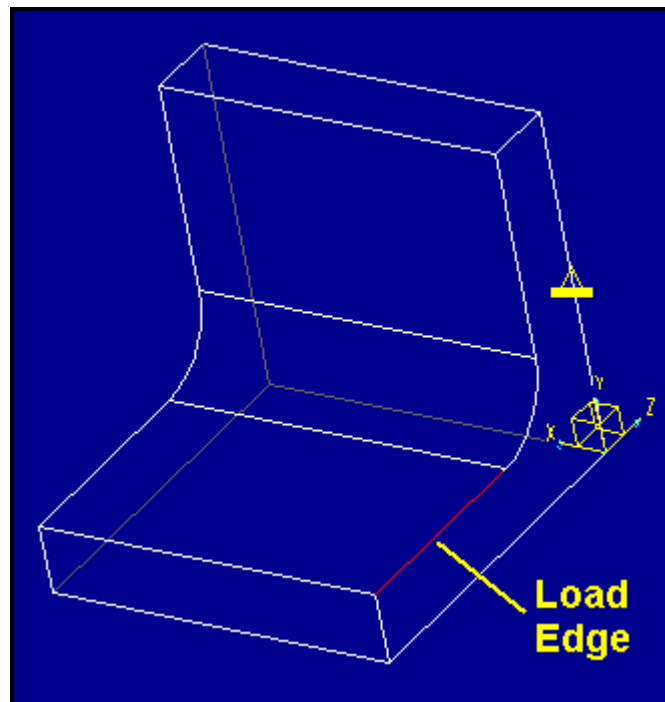
- Select **CONSTRAINTS** and select the rear edge as shown below.
- **FIX** the Displacement and Rotation in all directions.



Loads

We will apply a load on the foot of our L bracket.

- Select **LOADS** and **EDGE/CURVE** and select the edge as shown below.
- Apply a load in the Y direction of -100.



Analysis

We've completed defining our environment for our model. The next step will be to create and run an analysis.

- Select **ANALYSIS** and name it **bracket_1**.
- Set the Convergence to **10%** and Max Polynomial Order to **9**.
- Confirm that the boxes for Stresses, Rotations, and Reactions are checked.

Remember, the user determines the accuracy of the solution by specifying the convergence percentage.

Analysis Definition

Analysis Name: Type:

Constraint Set:

Load Sets:

Convergence Method:

Convergence: % on

Polynomial Order (1-9) Min: Max:

Plotting Grid (2-10):

Calculate: Stresses Rotations Reactions

Now we have completed our analysis form and are ready to run it.

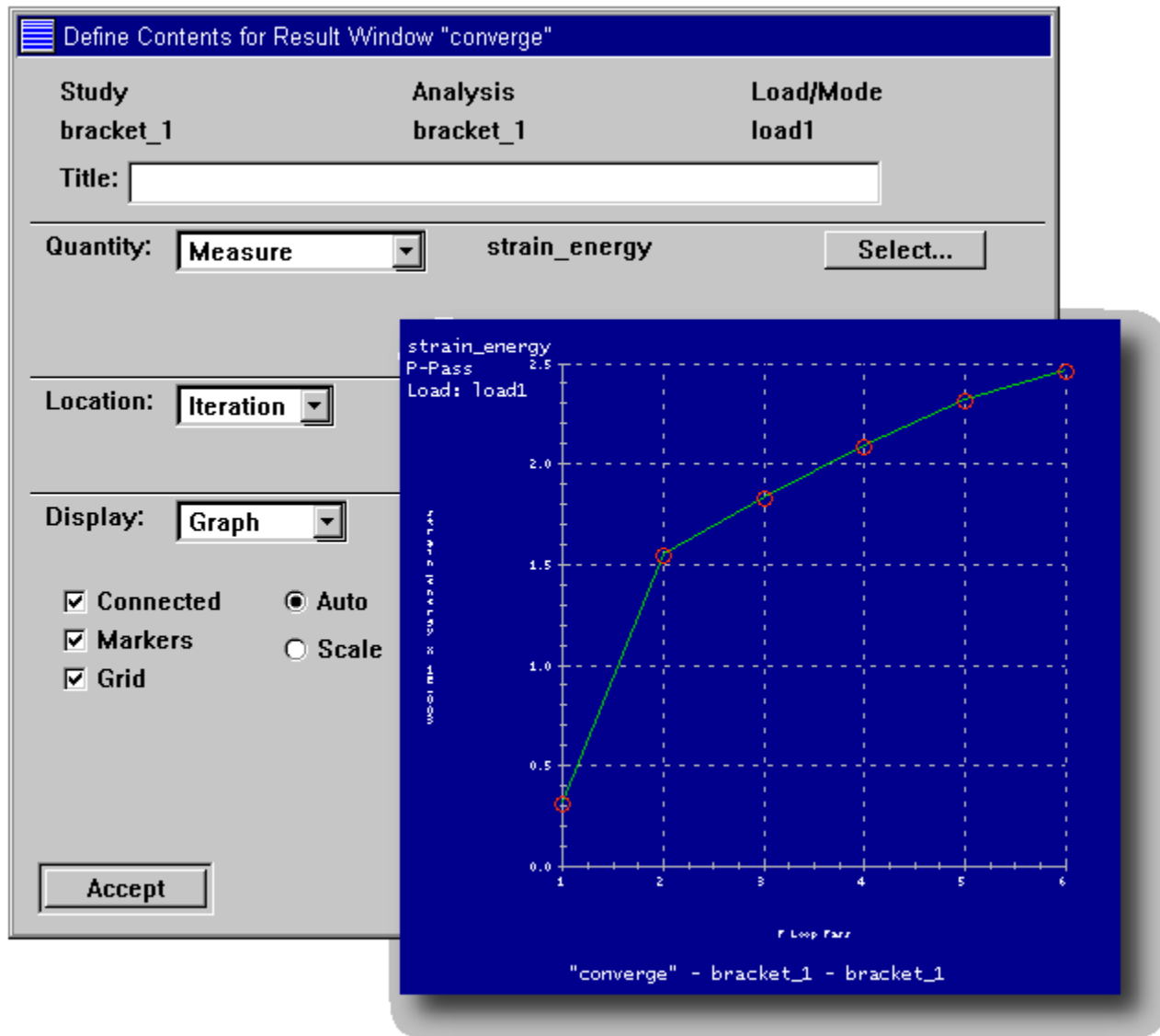
- Select **RUN** from the Mech Struct menu.
- Confirm **brack_1** is highlighted and select **START**.
- As the problem is solving you can examine the status of the solution by selecting **SUMMARY**

Results

To view results in Mechanica you first need to create a results window in which you select what you want to view and how you want to view it.

The first window we create will be a convergence graph. This graph will reveal how Mechanica converged on a solution. The solution process continued until two consecutive passes were within our specified convergence of 10%.

- **CREATE** a results window and name it **converge**.
- Select **bracket_1** as the analysis we would like to create results from.
- The standard results form appears, for Quantity select **MEASURE** and then pick **SELECT** and select **STRAIN_ENERGY** from the list.
- To view the results window hi-lite **converge** in the show column and select **SHOW**. The converge graph should be similar to the graph below. Notice how Mechanica continues to refine the solution until the convergence percentage is achieved.



Our second results window will be a fringe (color contour) plot of the Von-Mises stress.

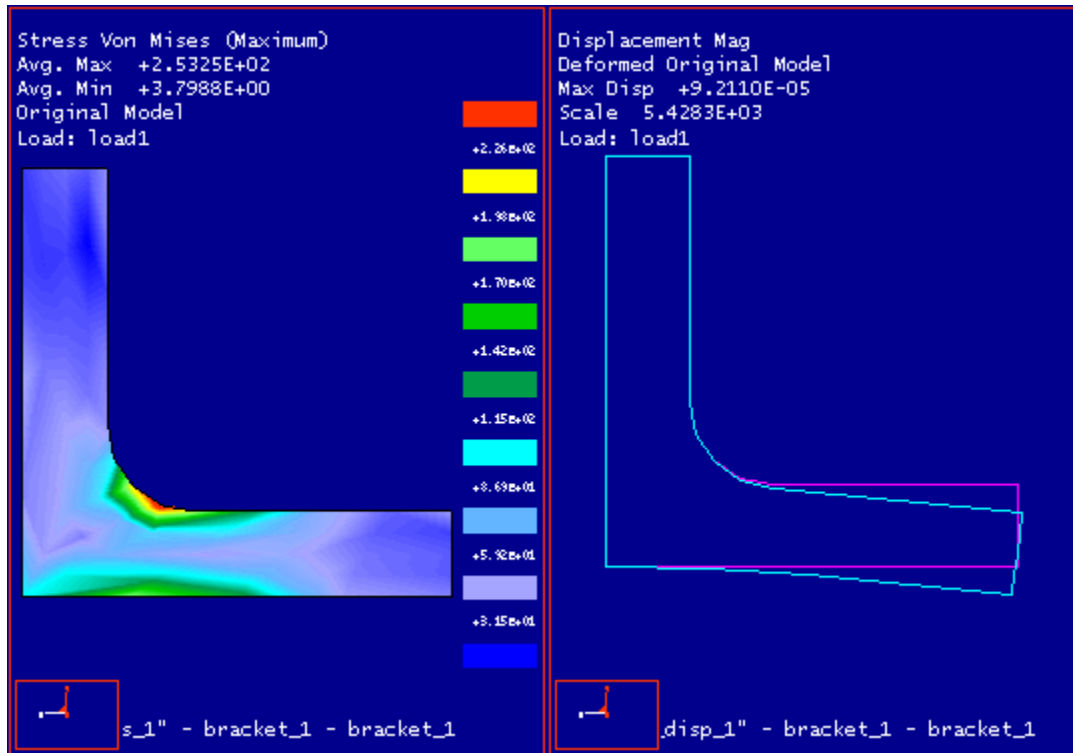
- **CREATE** a results window and name it **von_mises_1**.
- **ACCEPT** **bracket_1** as the analysis we wish to create a results window for (bracket_1 should be already highlighted since we previously used it).
- In the standard results window, select **STRESS** for the Quantity and then **VON-MISES**.

The third results window we will create will be an animated displacement.

- **CREATE** the third window and name it **anim_disp_1**.

- Again, select **bracket_1** as the analysis.
- For Quantity select **DISPLACEMENT** and for Display select **ANIMATION**. Also, change the Frames to **16** and Accept this form.

Show the Von-Mises and Displacement Animation plots simultaneously. This is accomplished by highlighting **von_mises_1** and **anim_disp_1** from the Show column and then selecting **SHOW**. The windows should be similar to those below.

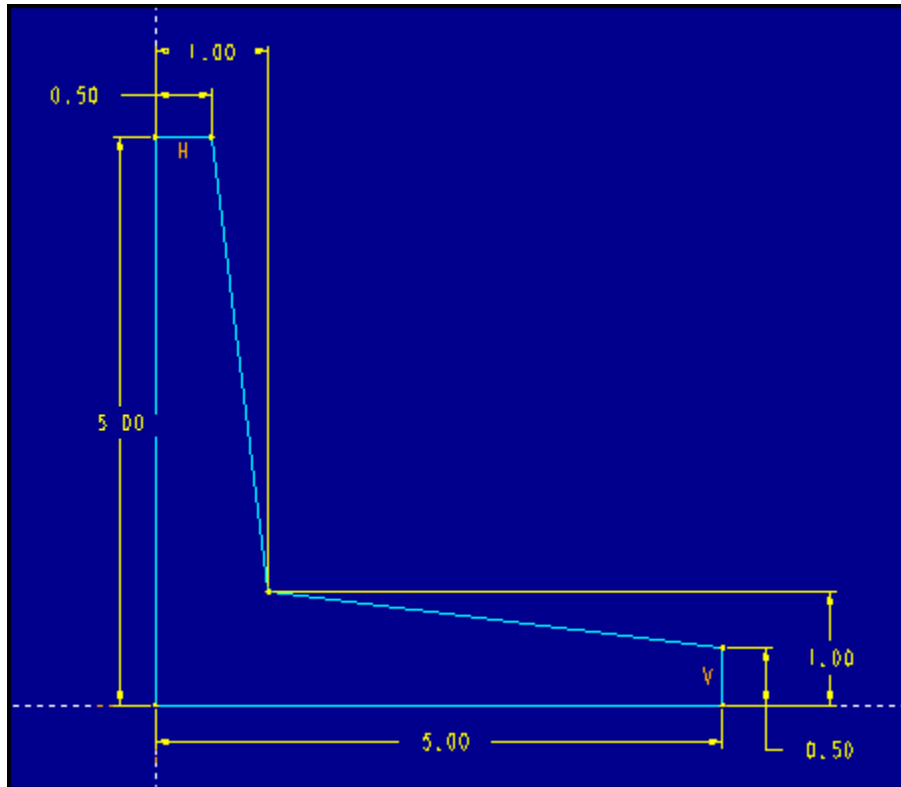


To start the animation of the displacement window select **CONTROLS**, select the result window you want to control, which is the Displacement Mag window in this case, and then select **START**.

Alternative Design Concept

The previous result windows reveal some interesting things, one being the Von-Mises Stress plot indicating that much of the material in the foot is unnecessary. Based on these results we are going to make some modifications to our initial design concept and see if we can add some improvements. We are uncertain whether these changes will result in an improved design, however, Mechanica will allow us to quickly evaluate these modifications.

- **REDEFINE** the sketch of base feature as shown below. Use **GEOM TOOLS / MOVE** to redefine this sketch and dimension as shown.



Now we have made modifications to our old design and have created an un-tested design. Lets run an analysis and see how this new design compares to our original design.

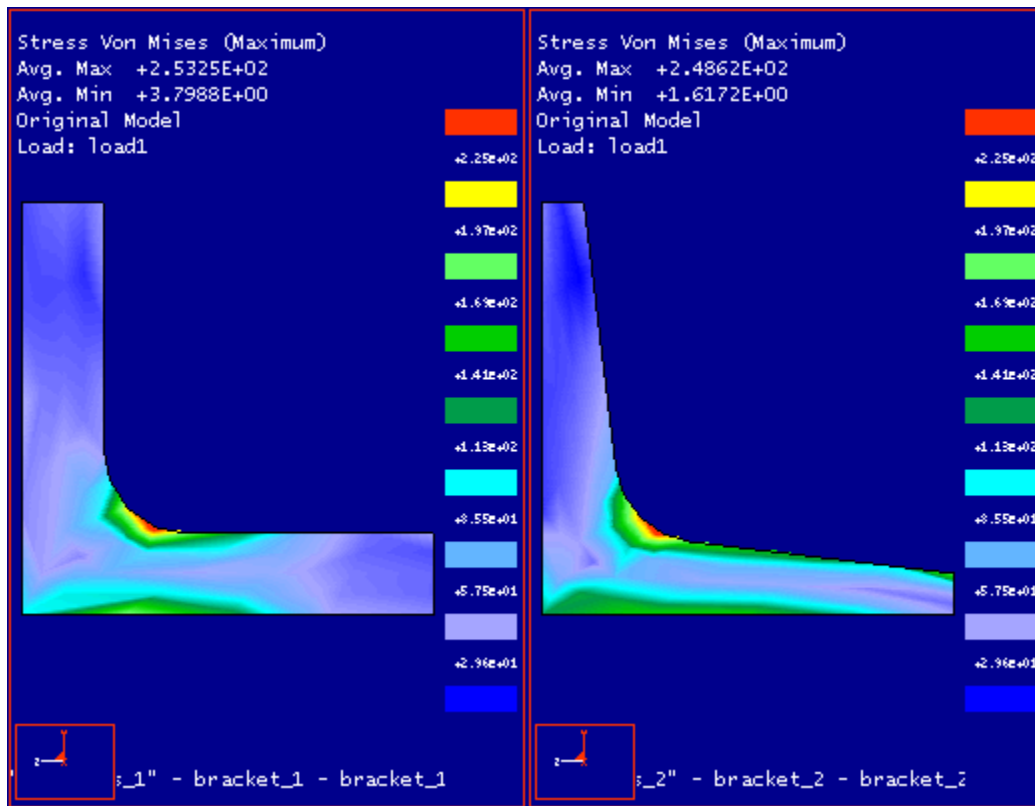
- Create a new **ANALYSIS** by performing a **COPY** of **bracket_1** to **bracket_2** (Select **bracket_1** in the edit column and select copy).
- **RUN** the new analysis **bracket_2**.
- Create a **RESULT** window for the Von Mises stress called **von_mises_2** of our **bracket_2** analysis.

We can compare the two designs by showing their result windows simultaneously.

- Select **bracket_1** and **bracket_2** from the show column of Results and de-select the other result windows which may be selected. **SHOW** these results windows.

The two result windows are shown simultaneously, however, the fringe plots are not at the same scale. We can tie the two scales together such that red in one window is equal to red in the other and so on.

- Select **CONTROLS** and pick one of the result windows and then select **TIE** and pick the other result window. Notice the color of the legend in both windows are now equal.



These results indicate that our modifications to the bracket utilize the material better (the stress is distributed more evenly than it was in the original design).

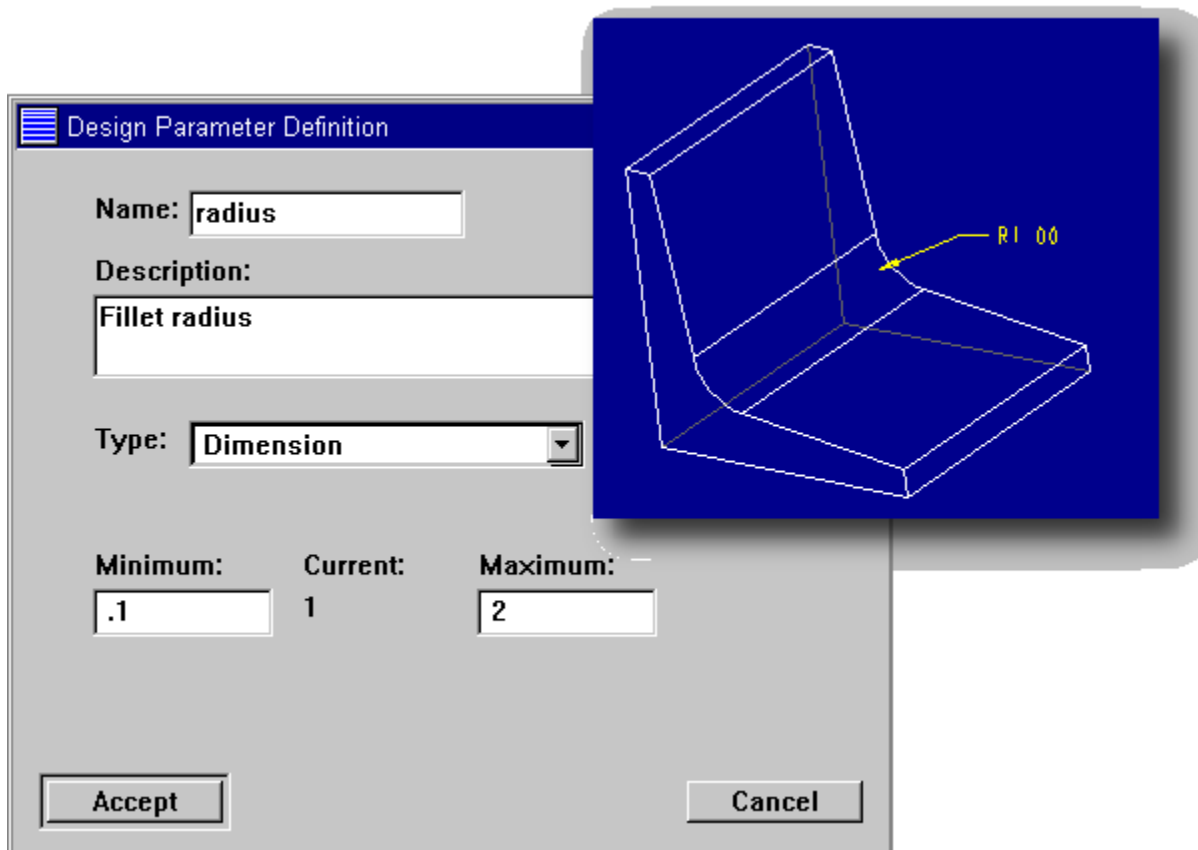
Design Parameters

At this point we have created an initial design concept and tested it. Based on our results we made some modifications and arrived at an alternative design. After completing tests on the alternative design and comparing the results to the initial design we have concluded the alternative is a better design. Can we make more improvements to our design? And if so, how?

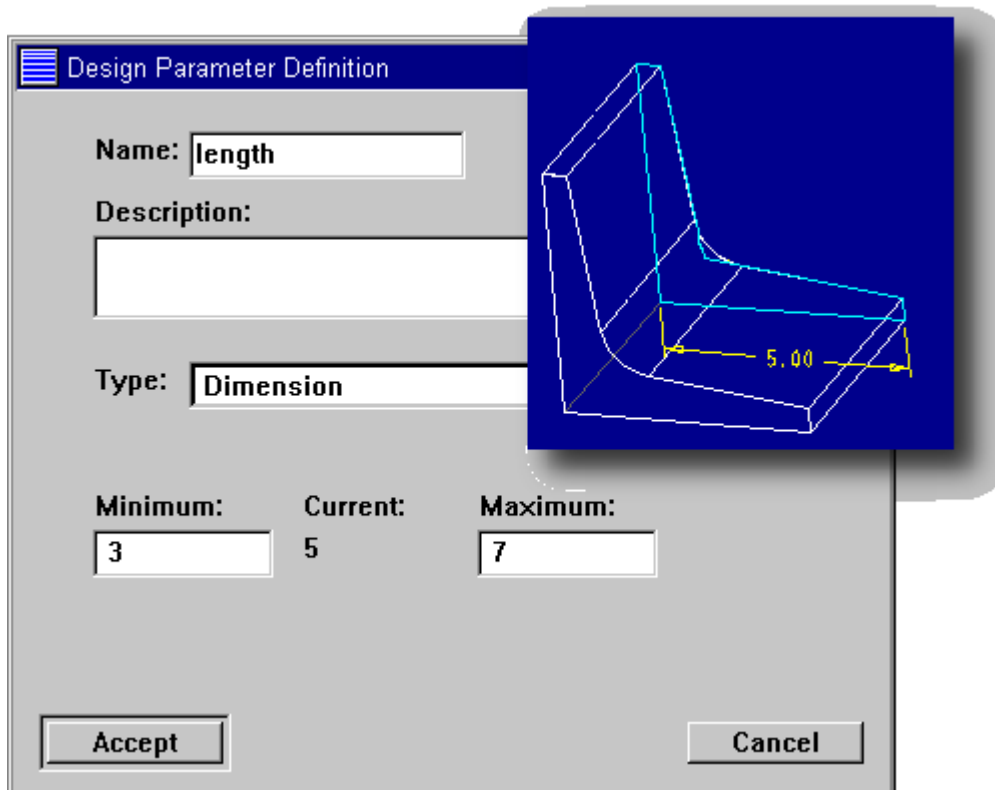
Design Parameters allow us to assign a parameter as not having a fixed value but rather having a range of acceptable values. We want to understand how modifying the parameter in this range affects the quality of our design; does it increase or decrease the stress? How does it affect the displacement?

We will create four design parameters and see how they impact our overall design. Also, we can compare design parameters to one another to see which ones affect the design more or less than others.

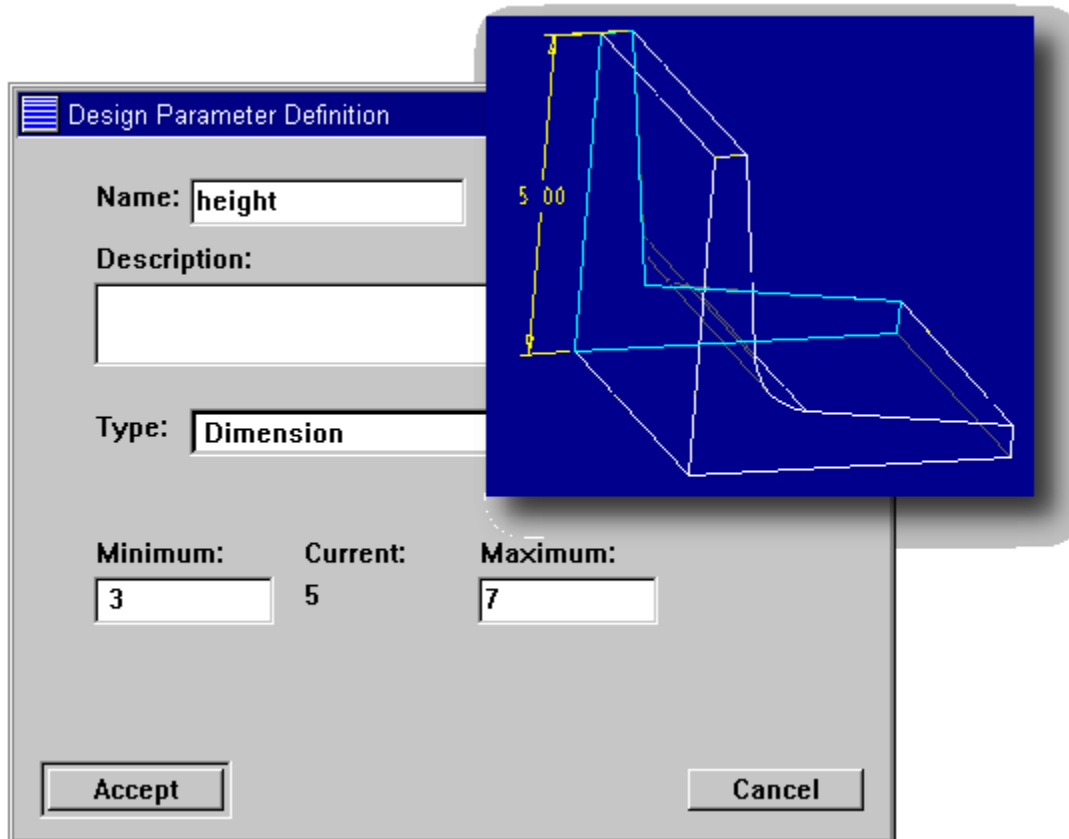
- Create a design parameter by selecting **DSGN CONTROLS** then **DESIGN PARAM** and name it **radius**. Select the fillet radius from the model, as shown below, as the design parameter. Fill in the form as below.



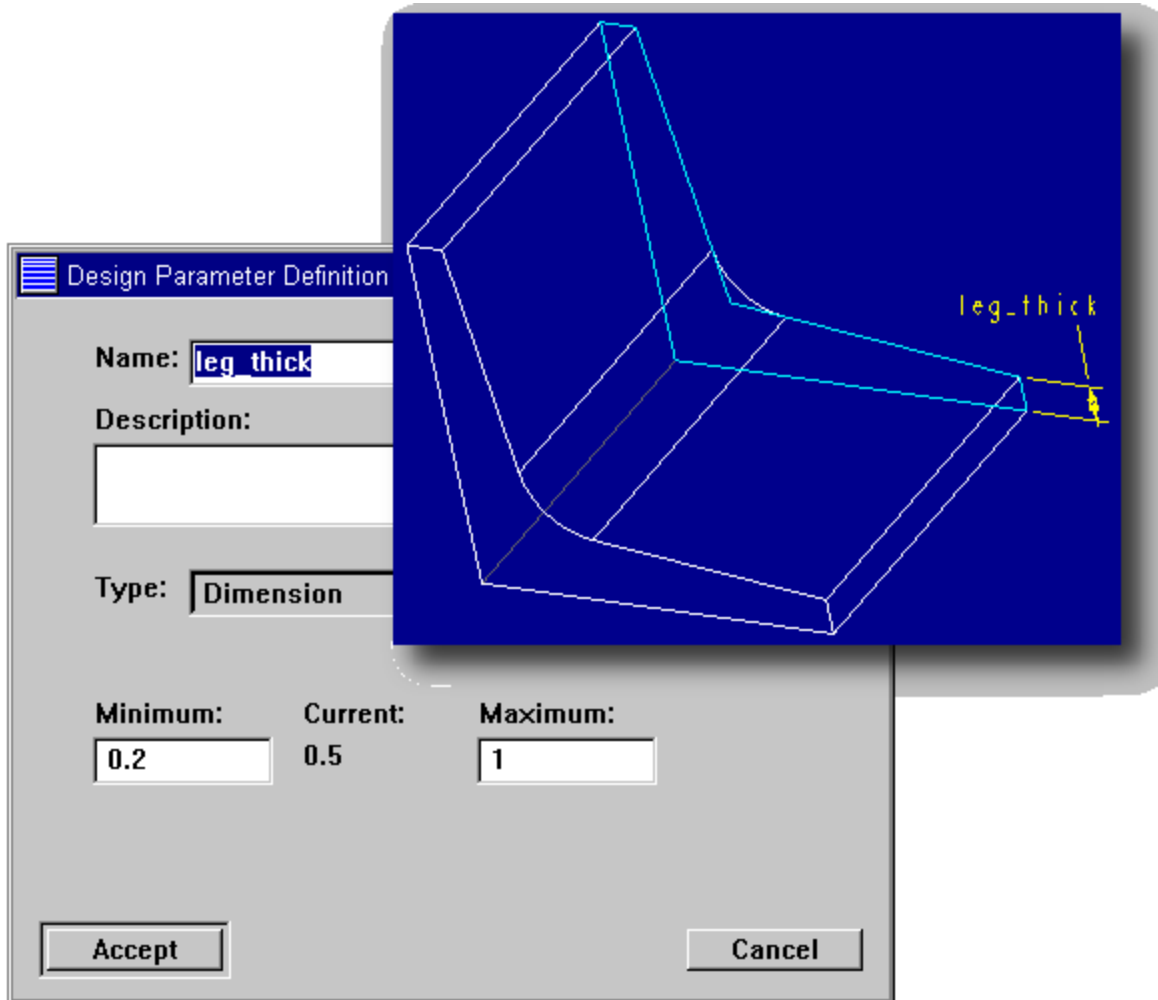
- Create a second design parameter called **length** as shown below.



- Create a third design parameter called **height** as shown below.



- Create the fourth and final design parameter called **leg_thick** as shown below.



Now that we have created the four design variables it is a good idea to animate the model. The animation steps the model through the possible extremes of the design parameters. This serves two purposes:

1. Ensures that the parameter changes the model in the way you anticipated.
2. Verifies that the range available to the design parameter does not violate the geometry.

- Select **SHAPE ANIMATE**
- Select all four Design Parameters
- Reduce the number of intervals to 4
- Select the **ANIMATE** button

Design Studies

Sensitivity Studies will allow us to perform what-if scenarios. What if we increase or decrease the fillet radius? What effect will that have on my stresses, displacements, modal frequency, etc.?

We will create Design Studies to see how our design parameters influence our design.

- Select **DESIGN STUDY** from the menu and fill in the form as shown below.

Design Study Definition

Study Name: Type:

Description:

Analyses:

- bracket 1 (Static)
- bracket 2 (Static)

Parameters: Start: End:

Parameters:	Start:	End:
<input checked="" type="checkbox"/> radius	<input type="text" value="Minimum"/>	<input type="text" value="Maximum"/>
<input type="checkbox"/> length		
<input type="checkbox"/> height		
<input type="checkbox"/> leg_thick		

Number of Intervals:

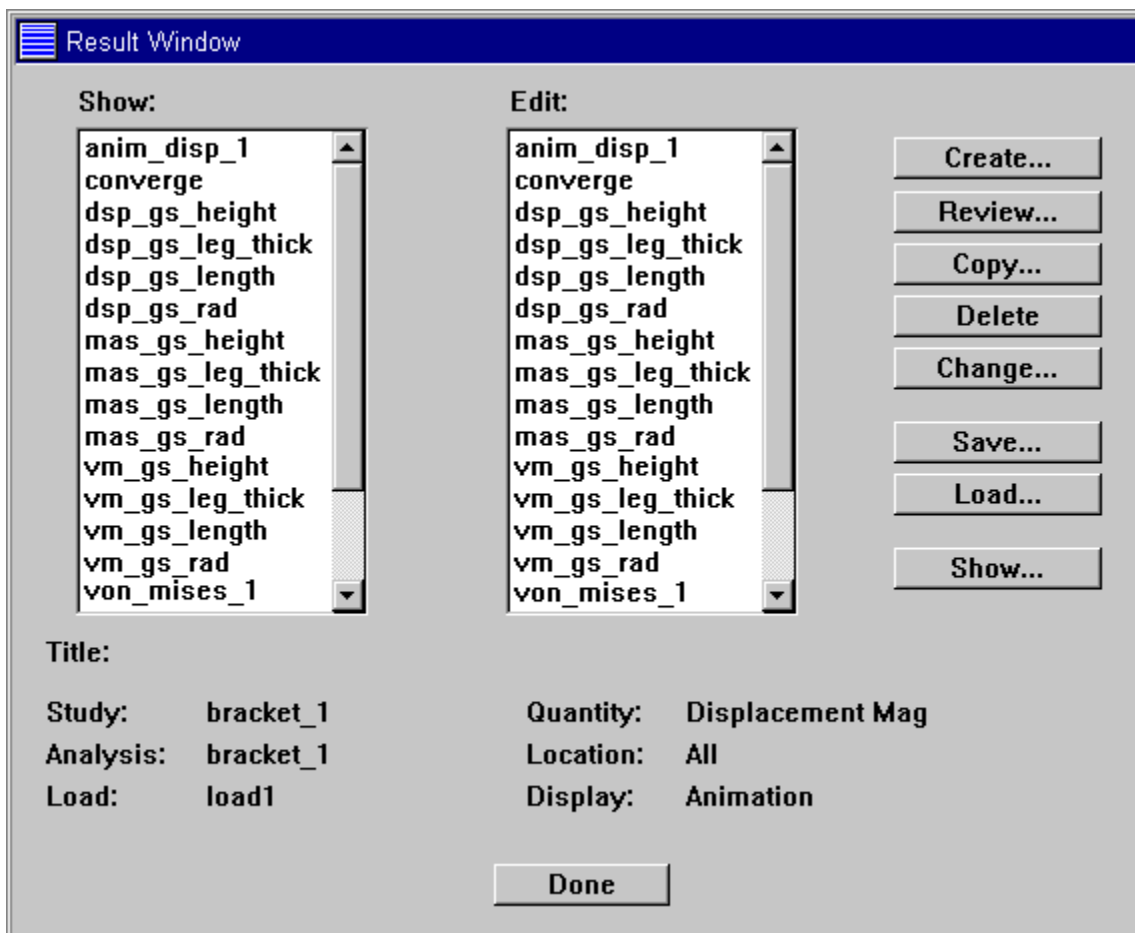
Repeat P-Loop Convergence

- **RUN** the design study in the same manner as bracket_1 and bracket_2 were run.
- Examine the **SUMMARY** as Mechanica runs the sensitivity study. Notice all the meshing, solving and incrementing of the design parameter is accomplished by Mechanica and requires no intervention from the user.
- Create a **RESULTS** window of the Von Mises stress called **vm_gs_rad** and view the resultant graph.
- Create additional result windows for the mass, **mas_gs_rad** and for the displacement, **dsp_gs_rad** and view the impact of changing the fillet radius has on these values.
- Create three additional design studies as follows:
 - For the **length** design parameter with design study named **gs_length**
 - For the **height** design parameter with design study named **gs_height**
 - For the **leg_thick** design parameter with design study named **gs_leg_thick**
- Run all three design studies.
- Create results windows for Von Mises stress, mass, and displacement for each sensitivity study and

name them as shown below.

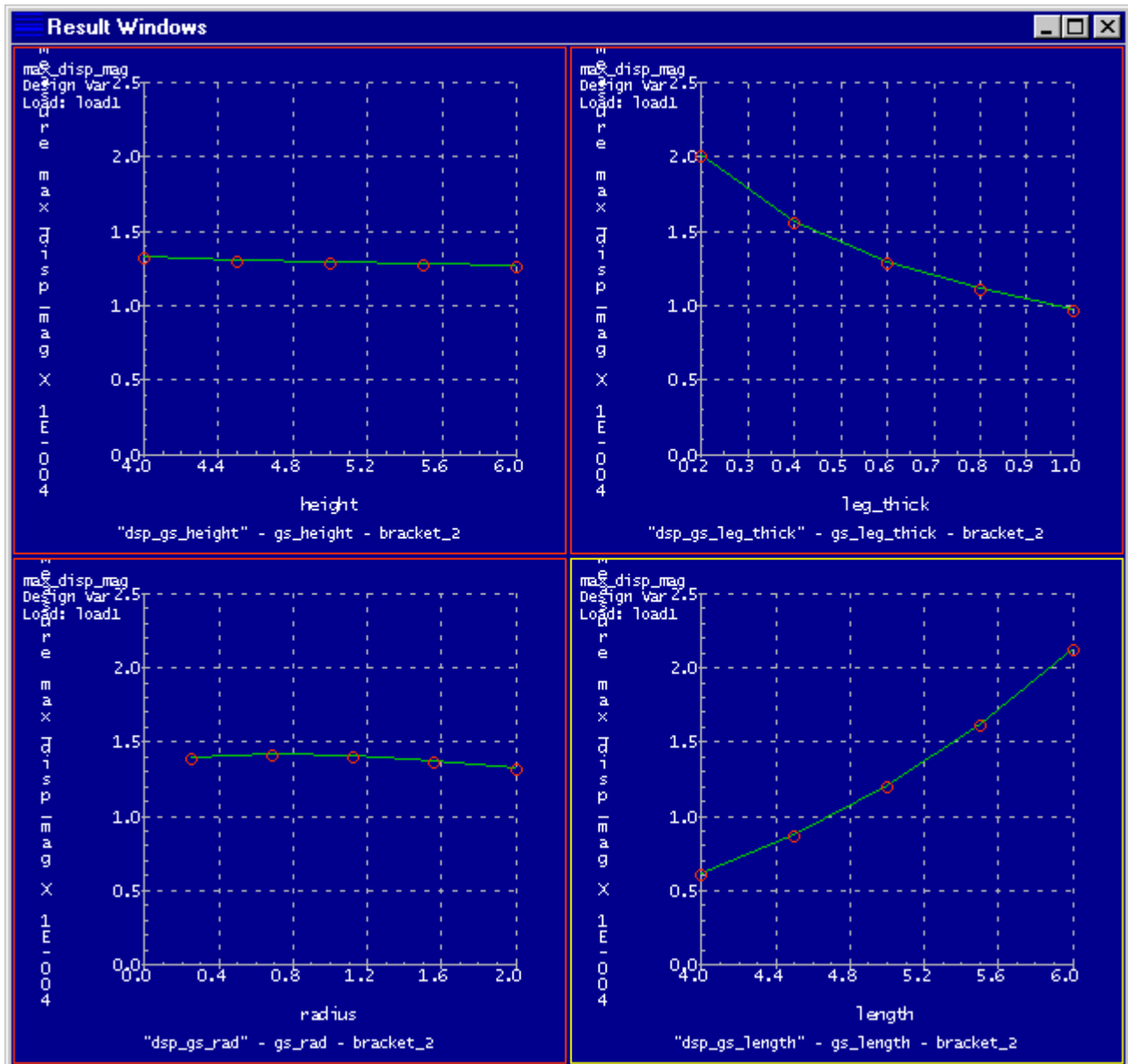
Sensitivity Study Name	Quantity	Result Window Name
gs_length	Mass	mas_gs_length
-	Von Mises stress	vm_gs_length
-	Displacement	dsp_gs_length
gs_height	Mass	mas_gs_height
-	Von Mises stress	vm_gs_height
-	Displacement	dsp_gs_height
gs_leg_thick	Mass	mas_gs_leg_thick
-	Von Mises stress	vm_gs_leg_thick
-	Displacement	dsp_gs_leg_thick

The following image reveals all the result windows you should have created.



Now we have completed running the sensitivity studies and the creation of all the results windows. Lets compare the result windows to one another to see what parameters have a greater influence on the model. We will first compare what effect the parameters have on the model's displacement.

- **Show** the four result windows for displacement, **dsp_gs_rad**, **dsp_gs_height**, **dsp_gs_length**, and **dsp_gs_leg_thick**.
- The four graphs are shown, however, the scales are not the same. Select **Controls** and then select a graph to use as the control, in this case select the length parameter graph since the y scale is the largest of all the graphs.
- Select **Tie Quantity** and pick one of the other graphs. Notice the y axis scale of this graph is now the same scale as the controlling graph.
- Select **Tie Quantity** again followed by selecting another graph.
- Continue the previous step for the third and final graph. Now all four graphs should have the same y scale as show below.



Optimization

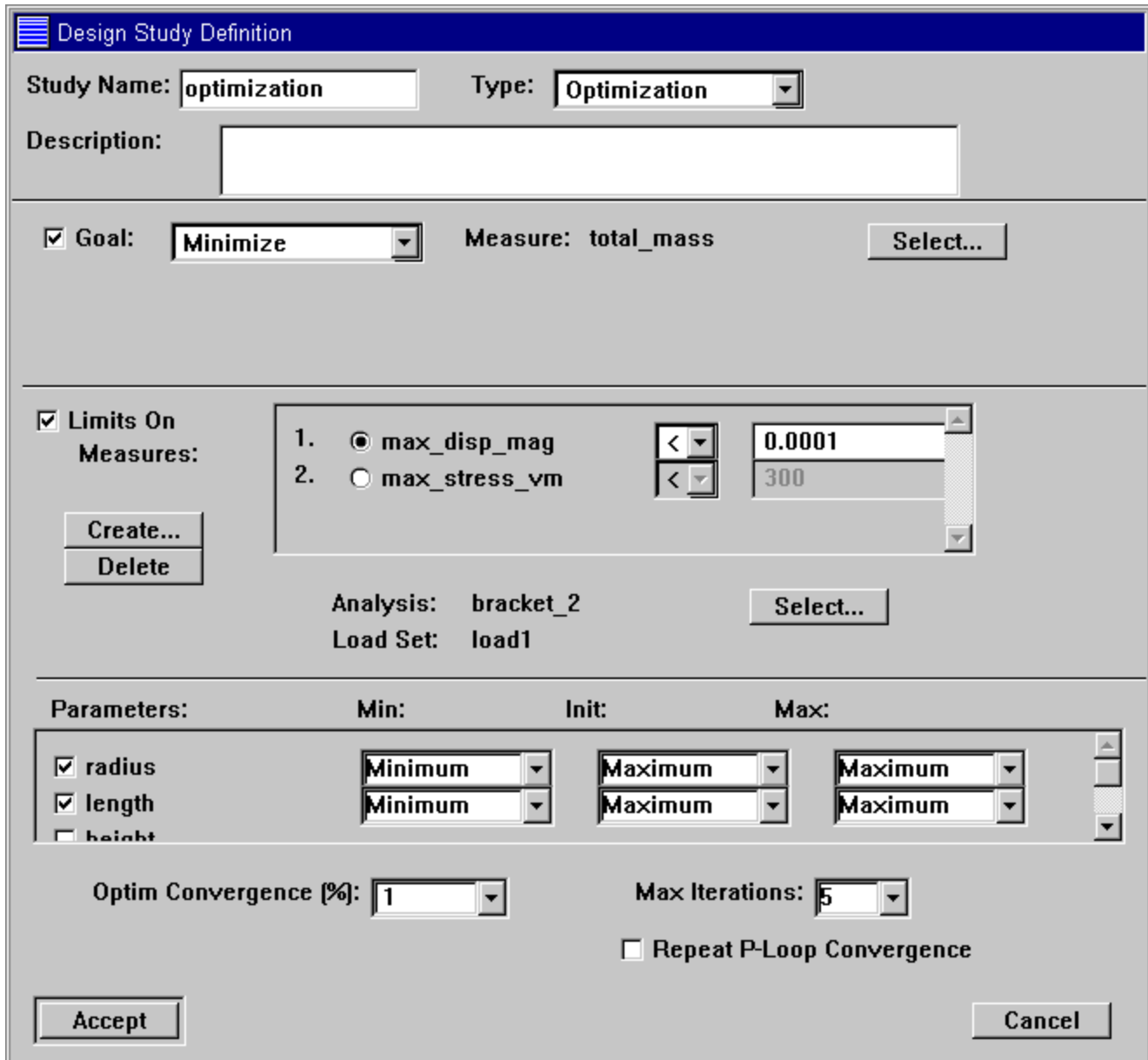
The finale is now to use the information we have gathered thus far and optimize our design. The design parameters reveal how much a parameter can effect the design. For our bracket we created four design parameters and based on our results, the **height** design parameter has little effect on our design. Therefore, when we optimize our model we can eliminate the height parameter as one of the design parameters to find an optimal value for.

Optimizations require three essential ingredients:

1. What is the goal (minimize mass, minimize displacement, maximize natural frequency, etc.)?
2. What are the design restraints (stresses, displacements, temperatures, frequencies, etc.)?
3. What can be changed and within what range can it be changed (design parameters)?

Create an optimization form as shown below.

- The goal is to **Minimize** the total mass of the design.
- The limits on our design are that the maximum displacement is not greater than **0.0001** and the maximum Von Mises stress is not greater than **300**.
- Based on our inspection of the sensitivity study results, the height design parameter has little effect on our model. Therefore, we will not include this parameter in our optimization.
- Confirm the **radius**, **length**, and **leg_thick** design parameters are checked and their intial values are set to the Maximum.
- Set the Max Iterations to **5**.
- After creating the optimzation form, start it and examine the summary as Mechanica optimizes our design.



The image shows a 'Design Study Definition' dialog box. At the top, the title bar reads 'Design Study Definition'. Below it, the 'Study Name' is 'optimization' and the 'Type' is 'Optimization'. The 'Description' field is empty. The 'Goal' is checked and set to 'Minimize', with the 'Measure' set to 'total_mass'. There is a 'Select...' button next to the measure. Under 'Limits On Measures', two items are listed: '1. max_disp_mag' (selected with a radio button) and '2. max_stress_vm'. The first item has a value of '0.0001' and the second has '300'. There are 'Create...' and 'Delete' buttons. The 'Analysis' is 'bracket_2' and the 'Load Set' is 'load1', both with 'Select...' buttons. The 'Parameters' section has a table with columns for 'Parameters', 'Min:', 'Init:', and 'Max:'. The parameters are 'radius', 'length', and 'height'. 'radius' and 'length' are checked, and 'height' is not. All 'Min:' and 'Max:' values are set to 'Maximum'. Below this, 'Optim Convergence (%)' is set to '1' and 'Max Iterations' is set to '5'. There is an unchecked checkbox for 'Repeat P-Loop Convergence'. At the bottom are 'Accept' and 'Cancel' buttons.

Design Study Definition

Study Name: optimization Type: Optimization

Description:

Goal: Minimize Measure: total_mass Select...

Limits On Measures:

1. max_disp_mag 0.0001

2. max_stress_vm 300

Create... Delete

Analysis: bracket_2 Select...

Load Set: load1

Parameters:	Min:	Init:	Max:
<input checked="" type="checkbox"/> radius	Minimum	Maximum	Maximum
<input checked="" type="checkbox"/> length	Minimum	Maximum	Maximum
<input type="checkbox"/> height			

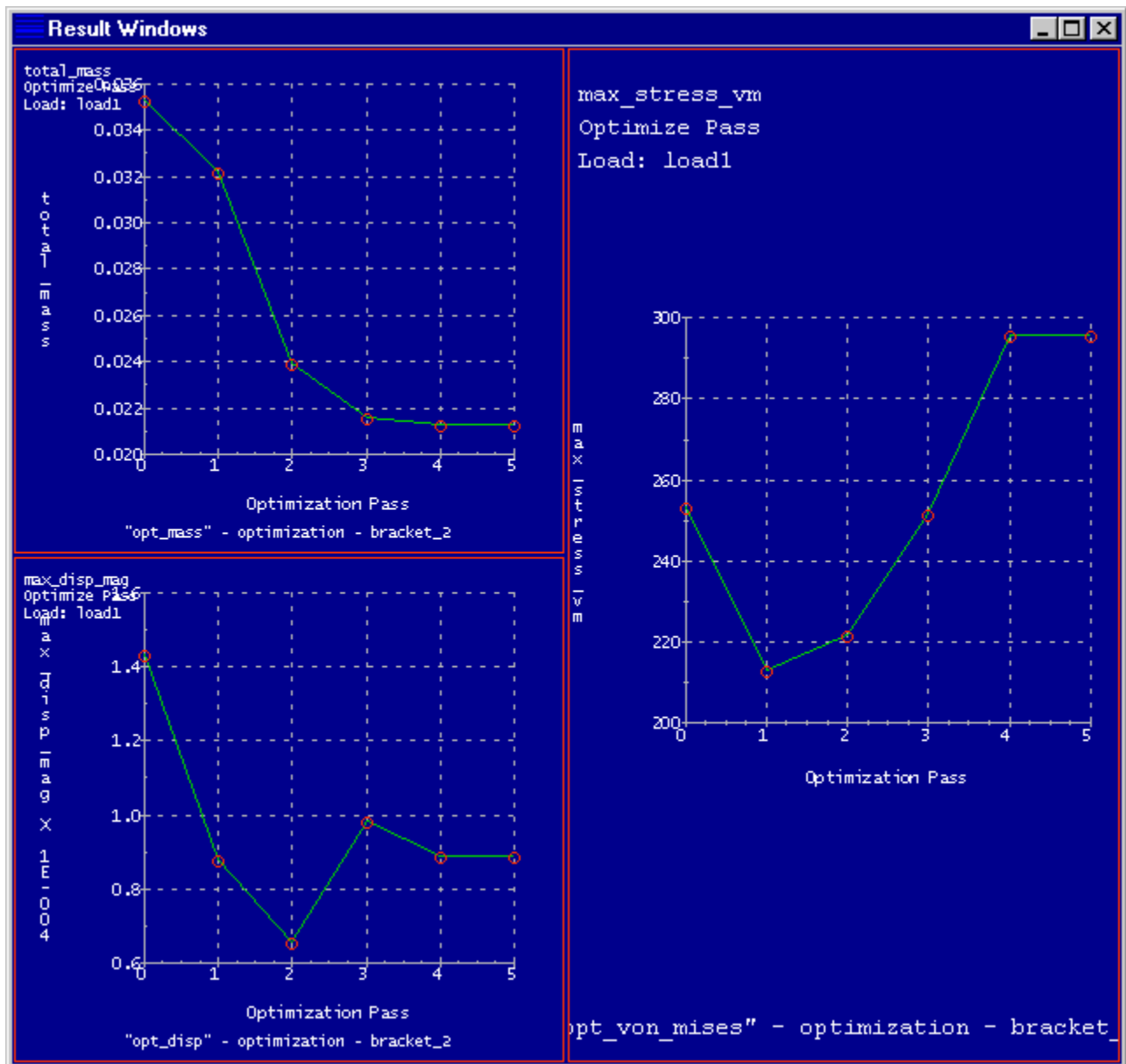
Optim Convergence (%): 1 Max Iterations: 5

Repeat P-Loop Convergence

Accept Cancel

After the completion of the optimization we can examine a number of things.

- Results windows can be created showing the iterations of the optimization as shown below.



These are result windows for the total mass, maximum Von Mises stress and maximum displacement.

The optimization history can be read such that you are stepped through the optimization steps. The final step is the optimized model where you can choose to accept these new values.

- Select **Dsgn Controls** and then **Optimize Hist**. Then select **Search Study** and select our optimization.
- Pro/E will then prompt you through each iteration and when the final iteration is reached, which is the optimized model, you can choose to accept or deny the new values.

Below is a side by side comparison of the Von Mises stress plots of the original design vs. the optimized design. Note that the horizontal leg of the optimized model utilizes the material to it's fullest, while the original model contains a lot of blue, which indicates under stressed material. Further inspection into the optimized model also shows the vertical leg contains under stressed material. Future design improvements could be made to reduce the thickness of the vertical leg.

