Step-by-Step Instructions

Integrating Algor and Pro/E
On the desktop click on the Engineering folder and open Algor.
Once in Algor click on file and open
In the open dialog box go to the file type and select the 3d model type. For this application select Pro/Engineer part file (*.prt).
Algor will open up the file and provide the create surface mesh dialog box. Click on Mesh.
The completed mesh should look like this.
After the mesh is completed click on Transfer to Superdraw in the Tools menu.
The 3d model should look like this in Superdraw
Click on the XZfront button in the View Utilities toolbar.
Go to FEA Add, Stress and Vibration Analysis, Boundary Conditions
Select the left side of the flat plate
The resulting fixed boundary condition should look like this.
Click done on the Boundary Conditions tool bar.
Go to FEA Add, Stress and Vibration Analysis, Apply Forces.
Click X Direction, then click done.
Click Magnitude.
Type in 3000 in the Magnitude dialog box at the bottom of the screen.
Click Box Apply.
Select the right side of the flat plate.
The result should look like this.
Click Done.
Open up Model Data Control by clicking on the Modal Data button.
Click on Data, use defaults, click OK.
Click on Material, select Steel 4130, click OK.
Click on Global Data, under the Multipliers tab add 1 under Pressure.
Under the Output Tab check Displacement/eigenvector data, and Stress data.
Under the Solution tab, change Solution Options to Banded, then click OK.
On Model Data Control click check, and the result should look like this. Close or minimize this window when done.
On Model Data Control click Analysis, click on Analyze once, step back and wait don’t touch the machine.
Click done when finished.
On Model Data Control click Results, and the result should look like this.
Go to the Results menu and click on Displacement Data, click on Displacement Model.
The results should look like this with a scale of 100%
Go to Results and click on Von Mises Stress.
The resulting Von Mises Stress should look like this.
Once Von Mises Stress are shown open the View Utilities tool bar and rotate the model.