# **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:

- 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.
- 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)





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	Von Mises Stresses	Name	Description	Initial Value (in)		
weight of the				value(in)		
part		Cut_length	Dimension	6		
			from left			
			edge to cut			
		Cut_radius	Notch radius	0.25		
		Thickness	Plate	0.25		
			Thickness			

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

sis Info	Applications Tools Window Help
	Standard Sheetmetal Welding Legacy Mechanica
	Plastic Advisor Mold/Casting Mechanica Simulation application

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.

Mechanica Model Setup
Capability Mode
Mechanica Lite
- Model Type
Structure
Mode
FEM Mode Advanced <<
- TypeExpan
@ 3D
2D Plane Stress (Thin Plate)     2D Plane Strein (Infinitely Thick)
2D Axisymmetric
- Coordinate System
No Items
- Geometry
Surfaces, Edges, Curves
No Items
- Default Interface
Bonded
OK Cancel

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

<u>Idealizations</u>:- 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

<u>Materials</u>:- 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.

Materials			X
File Edit Show			
🗅 🗊 🥒 🖣 🗙	රිග්		
- Materials in Library -		ſ	– Materials in Model –
- D - M	aterial Directory 🗸 😽 Search		
📲 Organize 🗸 🏢 Vie	ws 🎢 Tools 🗸 😡		STEEL
Common Folders	al2014.mtl		
🛃 Desktop	al6061.mtl		
Av Documents	B brass.mtl		
field019	bronze.mtl		
C* Working Directory	Lg cu.mti		
	Lē epoxy.mti		
Material Directory	Ligife20.mtl		
Tavorites	Lighte 30.mt	<b>DDD</b>	
		444	
	Te ma mt		
	Tanvion.mti		
	B pvc.mt		
	ass.mtl		
	b steel.mtl		
	🕞 tially.mti		
	Bipure.mtl		
	atungsten.mtl		
• Folder Tree			
			OK Cancel

# Select Properties/Material Assignment

🔄 Materials	🔊 🔫 🔽
🔄 Material Assignment	
Material Or	
Create a	a Material Assignment
A	
De Snell Properties	
j,L Shell Properties L × Beam Sections	
jua Shell Properties ăx Beam Sections ⊻x Beam Orientations	
jua Shell Properties ăx Beam Sections ∠x Beam Orientations ダ Beam Releases	9.
は、Shell Properties ▲ Beam Sections と Beam Orientations の Beam Releases ま Spring Properties	

Material Assignment	nt 🔀				
- Name	1				
MaterialAssign1					
- References					
Components	•				
- Properties Material					
STEEL •	More				
Material Orientation					
CONTRACTOR AND	More				
(None) *	10010				

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



<u>AutoGEM</u>:- 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

File Info			
AutoGEM Refe	rences ——		
All with Proper	ties	•	Create
k			Delete

This confirmation of the elements box should come up.

# Select Close

🗐 AutoGEI	M Sumn	nary	8	Diagnostics : A	utoGEI	M Mesh	×
Entities C	reated:			File Edit Vie	ew Ir	nfo	
Beam: Tri: Quad: Tetra: Wedge: Brick:	0 2 9 0 0	Edge: Face: Face-Face Link: Edge-Face Link:	28 11 0 0	Simulation Diag	nostic: elem	<mark>Source</mark> s ∉AutoGEM	Igno
Criteria Sa Angles (D Min Edge Max Aspec	atisfied: egrees Angle: ct Ratio: Time: 0	): 12.52 Max Edge Angle : 9.61 - 00 min - CPU Time: - (	: 125.99	4	•	4 (	
Lipour		Close					
					Cl	ose	

Click on the setup simulation display button on the toolbar

Settings	Modeling Entities	Loads / Constraints	Set Visibilities	Me
- Mesh				
	Entities	-		
✓ M	esh Points	✓ Beams		
So 🗹 So	lids	🗹 Links		
St	ells	🛃 Bonding E	lements	
Displ	splay Shells with Ze ay Quality ie idium	ero Thickness		

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

 AutoGEM

 Image: Dispute the mesh ?

 Image: Dispute the mesh ?

Click Yes to save the mesh

#### 5. 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.

### 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

- Name	)					- 12	
end_l	bad						2
Mem	ber of Set –						_
Loads	3et1				•	New	<i>I</i>
Refer	ences						
Edges	s/Curves						1
Edges	Curves : 🔘	Individu	ola	Inter	it		
- Prop CSYS : J. W	erties World CS	) Sel	ected		Adva	nced >	~
- Prop CSYS : پلز W - Force	erties World CS	) Sel	ected		Adva	nced >	×
- Prop CSYS : J. W - Force Comp	erties @ World CS 	) Sel	ected	) ( imen npor	Adva t —	nced >	~
- Prop CSYS : J, W - Force Comp	erties World CS ionents 500	) Sel	ected Cor X	men	Adva it	nced >	~
- Prop CSYS : J. W - Force Comp K	erties World CS ionents 500 -500	) Sel	ected Cor X Y	men mpor	Adva t — nents	nced >	~
- Prop CSYS : J, W - Force Comp K Y Z	erties World CS onents 500 -500 0	O Sel	ected Cor X Y Z	mpor 0 0	Adva t	nced >	~

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

File Edit View	Insert Properties AutoGEM /	Analysis Info Appli
C C C C C C C C C C C C C C C C C C C	<ul> <li>Force/Moment Load</li> <li>Pressure Load</li> <li>Bearing Load</li> <li>Gravity Load</li> <li>Centrifugal Load</li> <li>Centrifugal Load</li> <li>Displacement Constraint</li> <li>Planar Constraint</li> <li>Displa</li> </ul>	Cement Constraint
TOP     FRONT     FRONT     FRONT     Sketch 1     From Particular     Sketch 2	<ul> <li>Ball Constraint</li> <li>Create a Dis</li> <li>Symmetry Constraint</li> <li>Shell</li> <li>Beam</li> </ul>	placement Constraint
<ul> <li>              € Arrude 2          </li> <li>             Insert Here         </li> <li>             Materials         </li> </ul>	Spring	-
Material As     Material As     Material As	Connection	
To A Sec	Simulation Measure	
	Model Datum Surface Region Volume Region	Z
	🔠 Mechanism Load	

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

Constraint	
Name	
end_Constraints	
- Member of Set	
ConstraintSet1 • New	
- References	
Edges/Curves 🔹	
Edges/Curves :  Individual Intent	
Edge	
Coordinate System     World      Selected	
<b>st</b> ,wcs	
- Translation	PRT CSYS DEFA_1
X • 🗿 👫 👫	ZZYXX
Y • 🗿 👫 👬	
Z • 🗿 👫	
N/A	A 2
- Rotation	
x	
Y + MAX	
7 +	(107.1 [i
N/A	
N/O	
OK Cancel	

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

New Modal-		2	
New Buckling			
New Fatigue			Status
New Prestress	8		
New Dynamic	•		
New Standard Design Study			
New Sensitivity Design Study			
New Optimization Design Study			
Close			
escription ————			

s e Constraint Sets Component Set1 PLATE	Loads Sum Load Name LoadSet1	d Sets Component PLATE
s e Constraint Sets Component Set1 PLATE	Loads Sum Load Name LoadSet1	d Sets Component PLATE
s e Constraint Sets Component Set1 PLATE	Loads Sum Loa Name LoadSet1	d Sets Component PLATE
ar	Inertia R	elief
emperature Converger	nce Output Ex	cluded
Stribution	Ele	ements
:k		
al Order 3 3		
	emperature istribution Converge % al Order 3 3	emperature istribution Convergence Output Exe k al Order 3 3

Select OK

Click in <b>Start</b> , answer yes to activate	ate the error d	ection
--	-----------------	--------

	Start	
Name	Batch	Status
	Settings	
Description -		

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

Edg Checkpoints		
max_stress_xx:	1.941327e+04	
<pre>max_stress_xy:</pre>	3.026795e+03	
<pre>max_stress_xz:</pre>	0.000000e+00	
<pre>max_stress_yy:</pre>	4.513252e+03	
<pre>max_stress_yz:</pre>	0.00000e+00	
<pre>max_stress_zz:</pre>	0.00000e+00	
min_stress_prin:	-1.632201e+04	
strain_energy:	1.195698e-02	
Analysis "quick_check"	'Completed (16:50:03)	
Total Elapsed Time	(seconds): 3.55 (seconds): 0.58	
Maximum Memory Usag Working Directory D Results Directory S	e (kilobýtes): 177450 Disk Usage (kilobytes): 0 Size (kilobytes):	
Maximum Memory Usag Working Directory D Results Directory S 451 .\quick_check Run Completed	e (kilobýtes): 177450 Disk Usage (kilobytes): 0 Size (kilobytes):	
Maximum Memory Usag Working Directory D Results Directory S 451 .\quick_check Run Completed Fri Jul 26, 2013 16:	e (kilobýtes): 177450 Disk Usage (kilobytes): 0 Size (kilobytes): S50:03	
Maximum Memory Usag Working Directory D Results Directory S 451 .\quick_check Run Completed Fri Jul 26, 2013 16:	e (kilobýtes): 177450 visk Usage (kilobytes): 0 size (kilobytes): .50:03	

# 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

nah	vses and Design	Studies	
	Name	Туре	Status
	quick_check	Edit Copy Delete Start Stop	Completed
esc	ription ———	Status	

otch_plate_static			
escription:			
Constraints         Combine Constraint Sets         Name       Component         ConstraintSet1       PLATE	Loads - Sum I Name LoadSe	Load Sets	omponent ATE
Nonlinear	Inerti	a Relief	•
Itervals Distribution	Carbon	Elements	
Multi-Pass Adaptive			
Polynomial Order Minimum Maximum Converge on Local Displacement, Local Strain Local Displacement and Local St	nits — rcent Converg Energy and G rain Energy	gence 1	D Stress
) Measures			

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

```
X
Run Status (notch_plate_static.rpt) Not Running
 Summary Log Checkpoints
        max_stress_xx: 1.941327e+04
max_stress_xy: 3.026795e+03
max_stress_xz: 0.000000e+00
max_stress_yy: 4.513252e+03
max_stress_yz: 0.000000e+00
max_stress_zz: 0.000000e+00
max_stress_zz: 0.000000e+00
                                                     16.0%
                                                                               .
                                                    20.4%
                                                     0.0%
                                                    2.0%
                                                    0.0%
                                                     0.0%
                               -1.632201e+04
        min_stress_prin:
                                                      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

nalwaas and Dasi			
Name	Type	view results of a Design Study or Status	Finite Elemer
notch_plate_:	static Standard/Static	Completed	

Fill out the result window as follows:

lame	Title
Window1	VM Stress Results
<ul> <li>Study Selec</li> <li>Design Study</li> </ul>	tion
🗃 notch	_plate_static notch_plate_static
- Display type	)
Fringe	×
Thinge	<u>.</u>
Quantity Dis	play Location Display Options
Quantity Dis	play Location Display Options
Quantity Dis	play Location Display Options
Quantity Dis Stress Component	play Location Display Options           Ibm / (in sec^2)
Quantity Dis Stress Component von Mises	play Location Display Options           Ibm / (in sec^2)
Quantity Dis Stress Component von Mises	play Location Display Options           •         Ibm / (in sec^2)         •           •         ontributions from shells         •
Quantity Dis Stress Component von Mises Include co Membra	play Location       Display Options         ▼       Ibm / (in sec^2)         ▼       Ibm / (in sec^
Quantity Dis Stress Component von Mises Include co Membra At Maximu	play Location       Display Options         ▼       Ibm / (in sec^2)         ▼       Ibm / (in sec^

# Click on Display Options Tab

Check out the Deformed, animate and continuous Tone boxes

Vame	Title	
Window1	VM Stress Resu	Ilts
Window1 - Study Select Design Study Design Study The select Design Study Design Study Display type Fringe Quantity Display type Continuou Legend Leve Contour Label Cor IsoSurfac	VM Stress Resu ion plate_static noto play Location us Tone Is Is s ntours es	Its Its Isis Interplate_static Isis Interplate_static Isis Iay Options I Constraints Show Element Edges I Show Constraints Show Bonding Elements Isis Show Bonding Element Isis Sho
		Animate Auto Start Frames 8

## 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  $1.82 \times 10^4$  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.

#### 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.

<u>Design Controls</u>:- 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

New Static New Modal		2	
New Buckling New Fatigue			Status
New Prestress New Dynamic	+		Completed
New Standard Design Study New Sensitivity Design Study			
New Optimization Design Study Close	-		
escription			

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

Sensitivity Study	Definition			X	B	$\sim$
Name					Ē	
local					Trat.	1
Description						1
						$\sim$
					0	××
						× <del>X</del> ×
Туре						5
Local Sensitivity				•		ø
- Analyses	tic (Static)				U.	•
regenerate (Mod	el Regeneration Only	0				
					57	
					(PH)	
					T	
					AXR .	
- Variables					WW	
Variable	Current	Setting	Units			
	Guitone	octang	Units		m	
				Select	limension f	from model
					凤	
					₿.	
					ت) ا	
					6	
					13	
					1 50	
					5 <b>0</b>	
				F	0	
					B,	•
			OK	Cancel		

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

J Sensitivity Study I	Definition			24
Name				
local				
Description				
242				
Туре				
Local Sensitivity				•
- Analyses				
notch_plate_stati	c (Static)			
regenerate (Mode	el Regeneration Only	y)		
				21
Variables				
Variables	Current	Setting	Units	
Variables	Current 0.25	Setting 0.25	Units in	
Variables	Current 0.25	Setting 0.25	Units in	
Variables Variable cut_radius	Current 0.25	Setting 0.25	Units in	
Variables	Current 0.25	Setting 0.25	Units in	
Variables	Current 0.25	Setting 0.25	Units in	
Variables Variable cut_radius	Current 0.25	Setting 0.25	Units in	
Variables Variable Cut_radius	Current 0.25	Setting 0.25	Units in	
Variables	Current 0.25	Setting 0.25	Units in	
Variables Variable cut_radius	Current 0.25	Setting 0.25	Units	
Variables Variable cut_radius	Current 0.25	Setting 0.25	Units in	
Variables Variable Cut_radius	Current 0.25	Setting 0.25	Units in	
Variables Variable cut_radius	Current 0.25	Setting 0.25	Units in	
Variables Variable cut_radius	Current 0.25	Setting 0.25	Units	
Variables Variable cut_radius	Current 0.25	Setting 0.25	Units in	
Variables Variable cut_radius	Current 0.25	Setting 0.25	Units in	

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

Name Iocal Description Type Local Sensitivity Analyses notch_plate_static (Static) regenerate (Model Regene Variables Variables Cu cut_radius 0.21 cut_length 6 thickness 0.21				-
Iocal Description Type Local Sensitivity Analyses notch_plate_static (Static) regenerate (Model Regene Variable Variable Cu cut_radius 0.24 Cu thickness 0.24				
Description         Type         Local Sensitivity         Analyses         notch_plate_static (Static)         regenerate (Model Regene         Variables         Variables         cut_radius       0.21         cut_length       6         thickness       0.21				
Type Local Sensitivity Analyses notch_plate_static (Static) regenerate (Model Regene Variables Variable Cu cut_radius 0.2 cut_length 6 thickness 0.2				
Variables       Variable     Cu       cut_radius     0.24       cut_length     6       thickness     0.24	ration Only)			-
cut_radius0.24cut_length6thickness0.24	rent	Setting	Units	
cut_length 6 thickness 0.2	;	0.25	in	
thickness 0.2		6	in	
	;	0.25	in	₩,
				0

Click on **OK** 

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

nalyses and Desig		4
Name	Туре	Status
local	Local Sensitivity	Not Started

Click on Yes to activate the error detection

	] Ana	lyses and Design Stud	lies		Diagnostics : Analysis	local	x
	File	Edit Run Info			File Edit View In	fo	
	0			c		Source	Igno
ſ	- Ana	lyses and Design Stu Name	dies	Status	A total of 0 eleme	Solver	ĘÂ
	•	local notch_plate_static	Local Sensitivity Standard/Static	Completed Completed	Begin base analy     Begin sensitivity a     Begin sensitivity a     Begin sensitivity a     Run completed	Solver Solver Solver Solver	
ſ	- Des	cription		Close	Cia	ise	

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

An	alyses and Design Stud	ies		🔲 Run Status (local.rpt) Not Running	x
File	e Edit Run Info			Summary Log Checkpoints	
	alyses and Design Stud	dies Display stu	dy status	inertia_yy: 4.885867e+02 inertia_yz: 3.735963e+00 inertia_zz: 5.186080e+02	*
	Name	Туре	Status	Load Set: LoadSet1	
1	local	Local Sensitivity	Completed		
	notch_plate_static scription	Standard/Static	Completed	<pre>max_beam_bending: -0.0000000e+00 max_beam_tensile: -0.0000000e+00 max_beam_total: -0.0000000e+00 max_disp_mag: -2.002241e-04 max_disp_x: -3.871234e-05 max_disp_y: 1.964461e-04 max_disp_z: -2.325949e-18 max_disp_z: -2.325949e-18 max_rot_mag: -2.442807e-05 max_rot_ag: -2.442807e-05 max_rot_x: -2.046169e-20 max_rot_y: 2.701477e-19 max_rot_z: 2.442807e-05 max_stress_prin: -8.097903e+04 max_stress_vm: -7.867077e+04 max_stress_xx: -8.088861e+04 max_stress_xx: -1.261165e+04 max_stress_yy: -1.261165e+04 max_stress_yy: -1.880522e+04 max_stress_yy: -2.906716e-11 max_stress_yy: -2.906716e-12 max_stress_prin: 6.800838e+04 strain_energy: -4.982076e-02</pre>	
					ose

Click Close

Click on the review result button

Name	Type	eview results of a Design Study or I	Finite Element a
local	Local Sensitivity	Completed	

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

Result Window	w Definition
Name	Title
Window1	VM_thickness
- Study Selection	n -
Design Study	Analysis
local	notch_plate_static *
Quantity Displ Graph Ordina	ay Location Display Options te (Vertical) Axis
Measure	*
Jndet	ined
- Graph Locati	on
Design Var	•
thickness:PLA	TE
	OK OK and Show Cancel

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

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	•	

Result Window	v Definition	×
Name	Title	
Window1	VM_thickness	
- Study Selection	n	
Design Study	Analysis	
local	notch_plate_static	•
Graph Quantity Displ Graph Ordina	ay Location Display Options te (Vertical) Axis	<b></b>
Measure		•
max_s	itress_vm	/ (in sec^2) 🔹
- Graph Locati	on	
Design Var		
thickness:PLA	TE	•
-	OK OK and Show	Cancel

#### Click on OK and Show



Click on the insert a new definition button

ame	little			
Vindow2	VM_C	Cut_length		
- Study Selec	tion —			
Design Study	1	Analysis		
local		notch_plate_static		
- Display type Graph	e ———			
- Display type Graph Quantity Dis - Graph Ordi	e play Loca nate (Vert	tion Display Options		
- Display type Graph Quantity Dis - Graph Ordi Measure	e ————————————————————————————————————	tion Display Options		
- Display type Graph Quantity Dis - Graph Ordi Measure	e	tion Display Options	Ibm / (in sec^2)	
- Display type Graph Quantity Dis - Graph Ordi Measure Measure	e play Loca nate (Vert	tion Display Options ical) Axis — vm	[bm / (in sec^2)	
- Display type Graph Quantity Dis - Graph Ordi Measure Measure - Graph Loca	e nate (Vert <_stress_t	tion Display Options ical) Axis	[bm / (in sec^2)	
- Display type Graph Quantity Dis - Graph Ordi Measure I I I I I I I I I I I I I I I I I I I	e nate (Vert c_stress_1	tion Display Options ical) Axis	[bm / (in sec^2)	

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

New Static New Modal		2	
New Buckling			Status
New Paugue	3	ty	Completed
New Dynamic	F	с	Completed
New Standard Design Study			
New Sensitivity Design Study			
New Optimization Design Study			
Close	_	ļ	
escription ————————————————————————————————————			

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

Sensitivity Stud	y Definition				23
ame					
Global					
escription					
ype Global Sensitivit - Analyses notch_plate_st regenerate (Mc	/ atic (Static) Idel Regeneratio	m Only)			-
- Variables	Current	Start	End	Units	
cut_radius	0.25	0.1	0.5	in	
cut_length	6	4	8	in	
thickness	0.25	0.135	0.375	in	₩+
4		2000.		<b></b> →	

Click on **OK** 

Run the global sensitivity study



## Click on Review result button

Fill out the dialog boxes with following data

Vame Ti	tle	
Window1	M_thickness_gss	
- Study Selection -		
Design Study	Analysis	
Global	notch_plate_static	
- Display type Graph		*
Oisplay type     Graph     Quantity     Display L     Graph Ordinate (	ocation Display Options Vertical) Axis	•
Oisplay type Graph	ocation Display Options Vertical) Axis	•
Display type     Graph  Quantity Display L Graph Ordinate ( Measure      Measure      max_stre	ocation Display Options Vertical) Axis ss_vm	•
- Display type	ocation Display Options Vertical) Axis ss_vm	•
Display type     Graph  Quantity Display L      Graph Ordinate (      Measure      Measure      Graph Location -	ocation Display Options Vertical) Axis ss_vm	•
Display type     Graph  Quantity Display L      Graph Ordinate (      Measure      Measure      Graph Location -      Design Var	ocation Display Options Vertical) Axis ss_vm	•

## 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)

escription ype Optimization Goal Minimize Design Limits Measure max_stress_vm	5	▼ tot       Value       24000	al_mass Units Ibm / (in s	5		· ·
ype Optimization - Goal — Minimize - Design Limits — Measure max_stress_vm	4	• tot Value 24000	al_mass	5		
- Design Limits	×	Value 24000	Units	5		
Measure max_stress_vm	4	Value 24000	Units Ibm / (in s	5		
max_stress_vm	<	24000	lbm / (in s	5		in l
		1	_			-
			LoadSet1	PLATE		
- Variables	Current	Minimum	Inifial	Maximum	Units	
cut length 6	current	4	6	8	ín	
cut_radius 0	.25	0.1	0.25	0.5	in	
thickness 0	.25	0.135	0.25	0.375	in	<b>*</b>

Run the design study, it will take several minutes.

	Name	run Type	Status
1	Global	Global Sensitivity	Completed
	Opt_Study	Optimization	Not Started
1	local	Local Sensitivity	Completed
14			
_			

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

The final values:

Run Status (Opt_Study.rpt) Not Running	X
Summary Log Checkpoints	
Begin Optimization Iteration 7 (16:30:14) Converged to optimum design.	*
Best Design Found: Parameters: thickness 0.135 cut_radius 0.5 cut_length 4 Goal: 9.0868e-01	
Optimization study statistics: Number of Base Analyses: 7 Number of Perturbation Analyses: 4	
Memory and Disk Usage: Machine Type: Windows XP 64 Bit Edition RAM Allocation for Solver (megabytes): 128.0	
Total Elapsed Time (seconds): 71.00 Total CPU Time (seconds): 8.16 Maximum Memory Usage (kilobytes): 234405 Working Directory Disk Usage (kilobytes): 26 Total Elapsed Time in Parameter Updates (seconds): 21.32	
Total Engine Elapsed Time Minus Param. Updates (seconds): 49.68 Total CPU Time in Parameter Updates (seconds): 0.80	•
•	*
	Close

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)

Analyses	and Design :	Studies —		
Na	me	Туре	¢.	Status
🥒 Glo	bal	Globa	al Sensitivity	Completed
🧹 Opt	-	<u></u>	zation	Completed
🧹 loca	Edit		lensitivity	Completed
🧹 note	ch Copy		rd/Static	Completed
	Delet	е		
	Start			
	Stop			
Descripti	io Vault	Results	-	
	E Status	3		

ame				
Dpt_Study				
escription				
ype				
Optimization				33
- Goal				
Minimize	• max_	_stress_vm		
Minimize Analysis	• (max_	_stress_vm Loadset		
Minimize Analysis notch_plate_static	•(max_	stress_vm Loadset Name	Component	
Minimize Analysis notch_plate_static	•(max_	stress_vm	Component	
Minimize Analysis notch_plate_static	• (max_	_stress_vm Loadset Name LoadSet1	Component PLATE	

#### Click OK and run the study

The final values to get the Minimum Von Mises Stress

23 Run Status (Opt\_Study.rpt) Not Running Summary Log Checkpoints ۸ Result of Optimization Iteration 6 Parameters: 0.375 thickness 0.108579 cut\_radius cut\_length 4 Goal: 7.6672e+03 Status of Optimization Limits: 1. max\_stress\_vm 7.6672e+03 < 2.4000e+04 (satisfied withir (16:40:44) Resource Check Elapsed Time (sec): 65.75 CPU Time (sec): 9.27 (kb): Memory Usage 236517 Wrk Dir Dsk Usage (kb): 26 Begin Optimization Iteration 7 (16:40:44) Converged to optimum design. Best Design Found: Parameters: thickness 0.375 cut radius 0.108579 cut\_length 4 Goal: 7.6672e+03 Optimization study statistics: Number of Base Analyses: 7 Number of Perturbation Analyses: 22 \_\_\_\_\_ w Þ 4 🤇 ..... Close