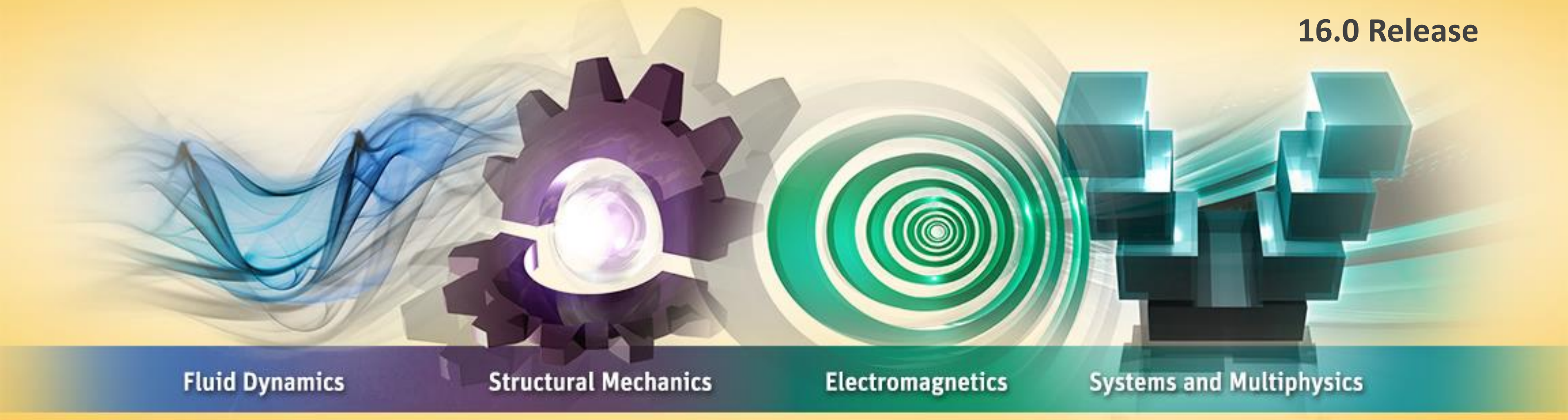
Workshop 12.1
Parameter Management

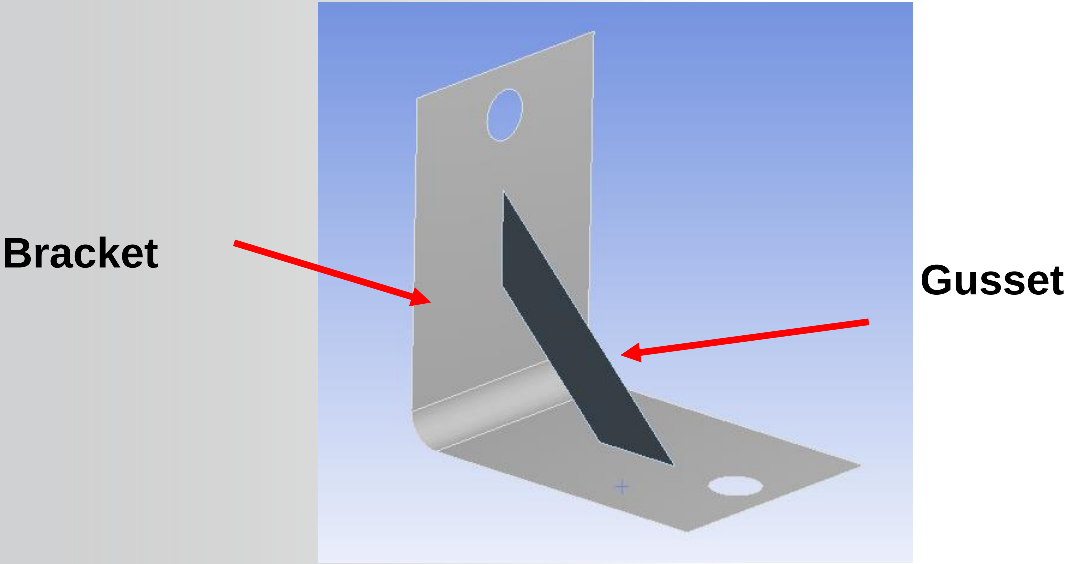
16.0 Release



Introduction to ANSYS Mechanical

Goals

- Use the Workbench Parameter Workspace to setup multiple scenarios to explore structural responses in the bracket shown.
- Various combinations of gusset and bracket thicknesses will be evaluated.

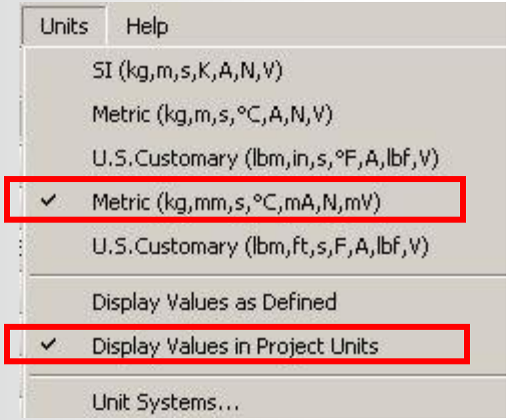


Project Schematic

Open the Project page.

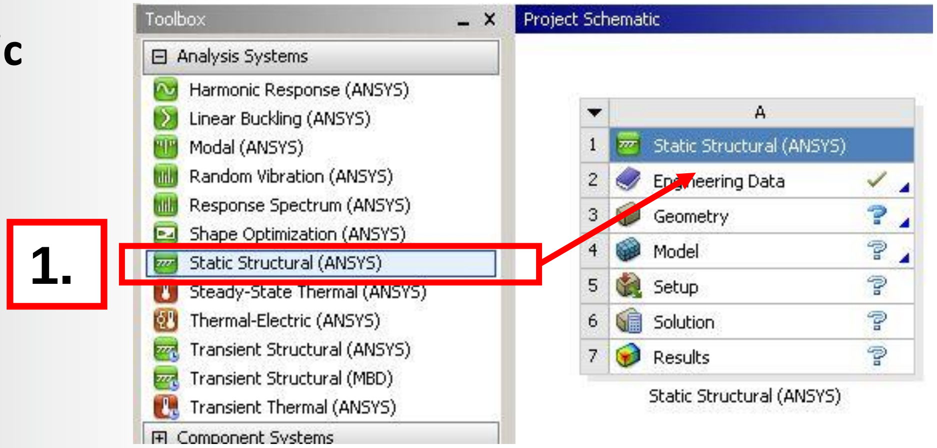
From the Units menu verify:

- **Project units are set to “Metric (kg, mm, s, C, mA, mV).**
- **“Display Values in Project Units” is checked.**

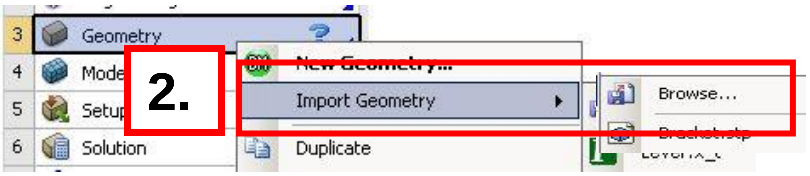


. . . Project Schematic

1. **From the Toolbox double click "Static Structural" to create a new system.**



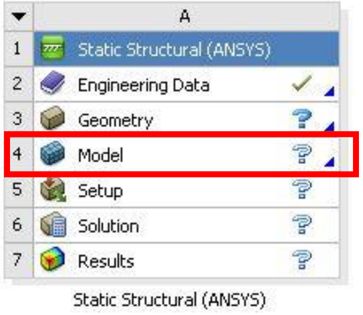
2. **RMB the geometry cell and "Import Geometry" and browse to "Bracket.stp".**



Preprocessing

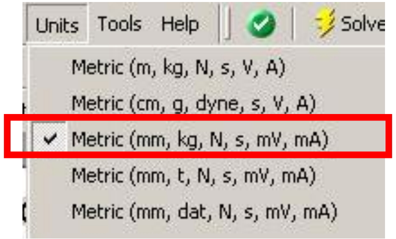
3. Double click the “Model” cell to open the Mechanical application.

3.



4. Set/verify the working unit system:
   – “Units > Metric (mm, kg, N, s, mV, mA)”.

4.

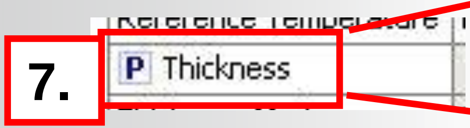


. . . Preprocessing

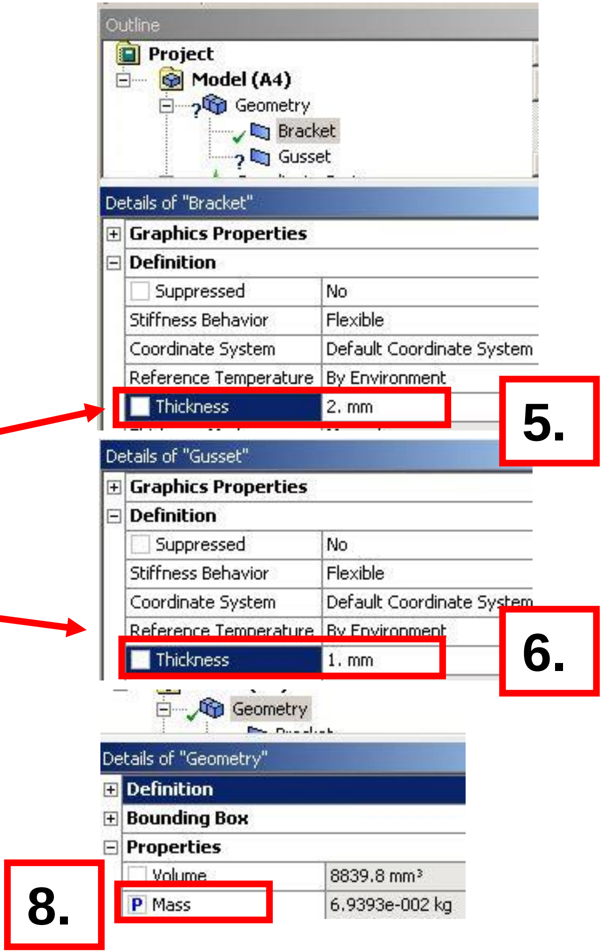
5. Highlight the part “Bracket” and enter a thickness = 2mm in the details.

6. Highlight the part “Gusset” and enter a thickness = 1mm in the details.

7. Make both thicknesses parametric by toggling the check box.



8. Highlight the Geometry branch and, in the Property details, toggle the Mass parameter on.

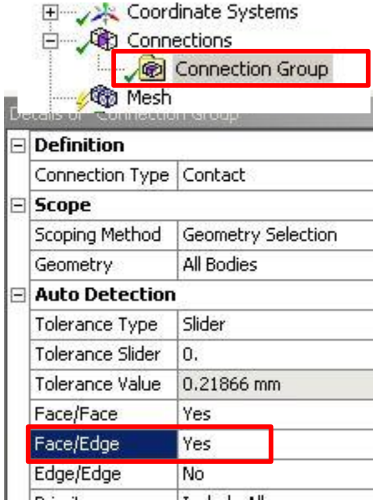


. . . Preprocessing

9. Highlight the Connections branch, RMB > Insert > Connections Group.

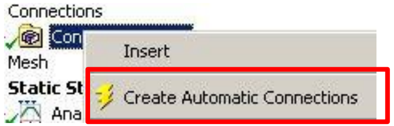
9.

10. In the details for the connections group change the Auto Detections for Face/Edge to “Yes”.



10.

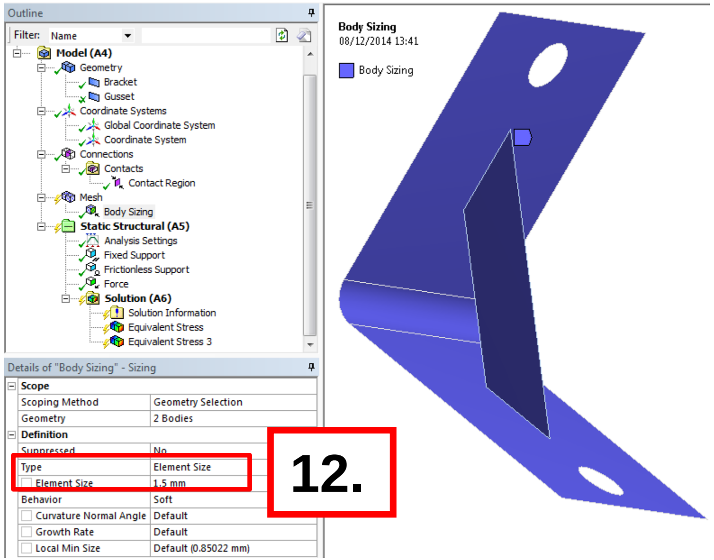
11. Highlight the connections group “RMB > Create Automatic Connections”.



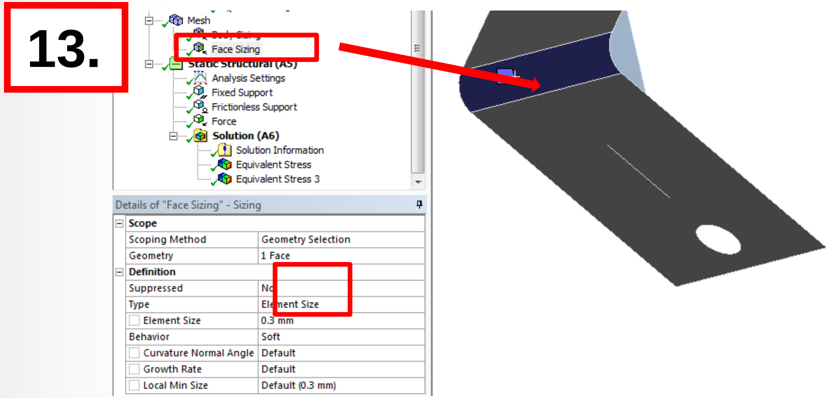
11.

Mesh control

12. Insert a body sizing for both bodies and set the element size to 1.5 mm.



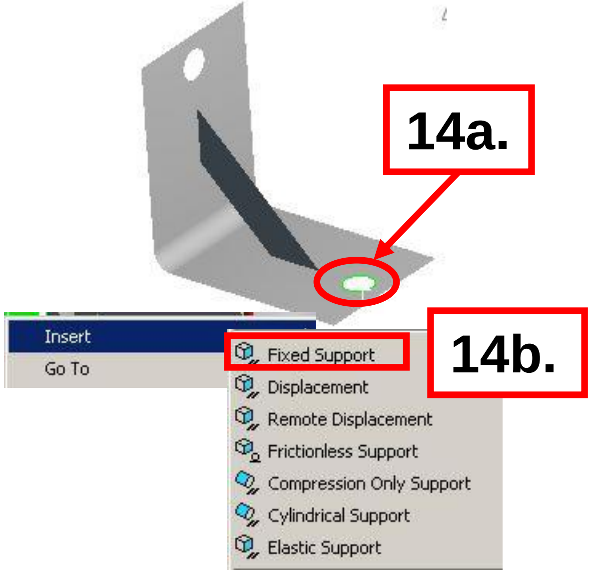
13. Insert a face sizing of 0.3 mm.

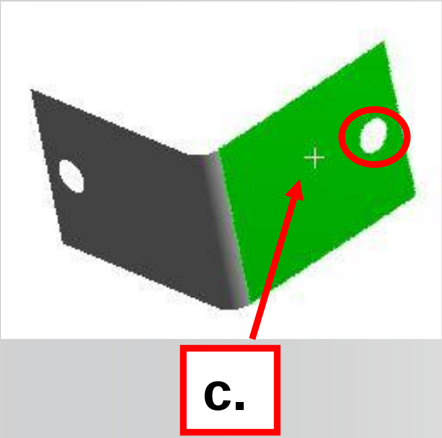


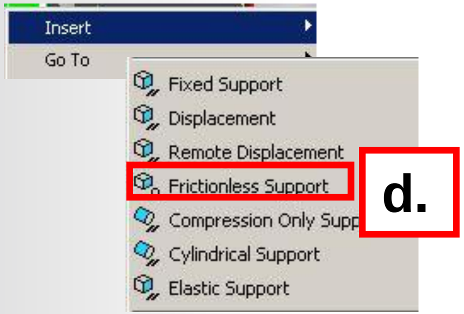
Environment

14. Apply constraints to the model (highlight “Static Structural” branch (A5):
    a. Select the edge of one hole.
    b. RMB > Insert > Fixed Support.

    c. Highlight the face surrounding the fixed hole.
    d. RMB > Insert > Frictionless Support.







. . . Environment

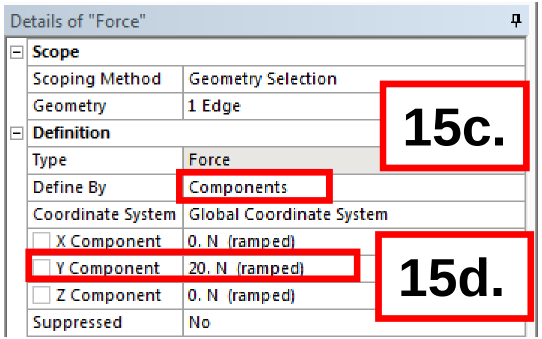
15. Apply Loads to the model:

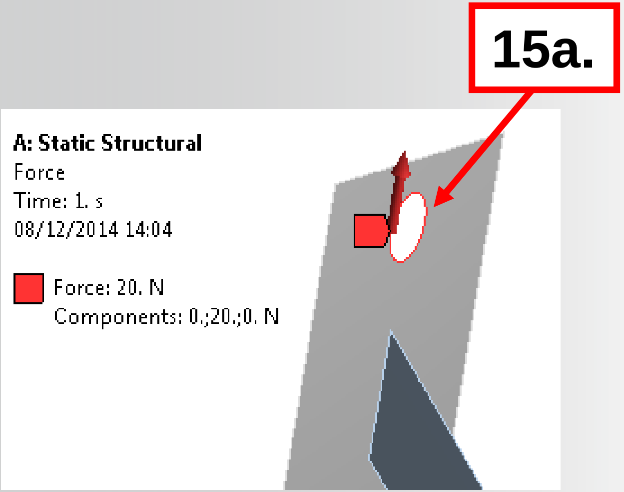
a. Select the edge of the hole shown below.

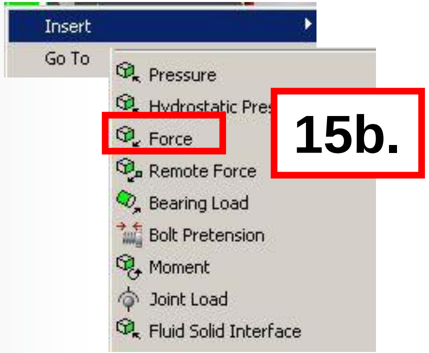
b. RMB > Insert > Force.

c. In the Details switch to the component method.

d. Enter a magnitude of 20 N in the Y direction.



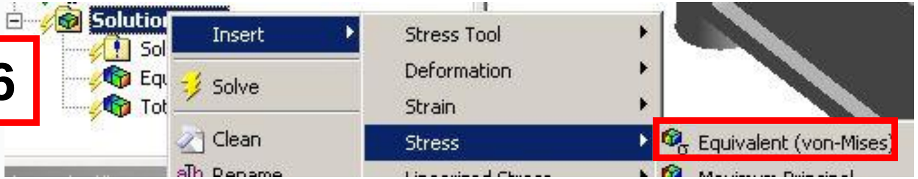




Solution Setup

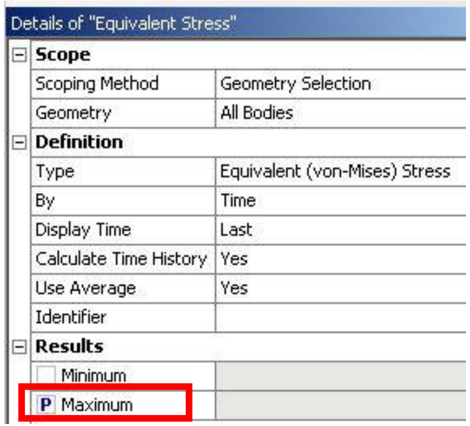
16. Insert Results (highlight Solution branch (A6):
    a. RMB > Insert > Stress > Equivalent (von Mises).

16
.



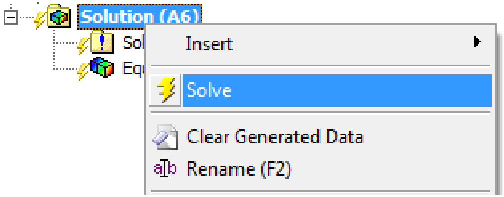
17. In the result detail, toggle the “Maximum” result as a parameter.

17.

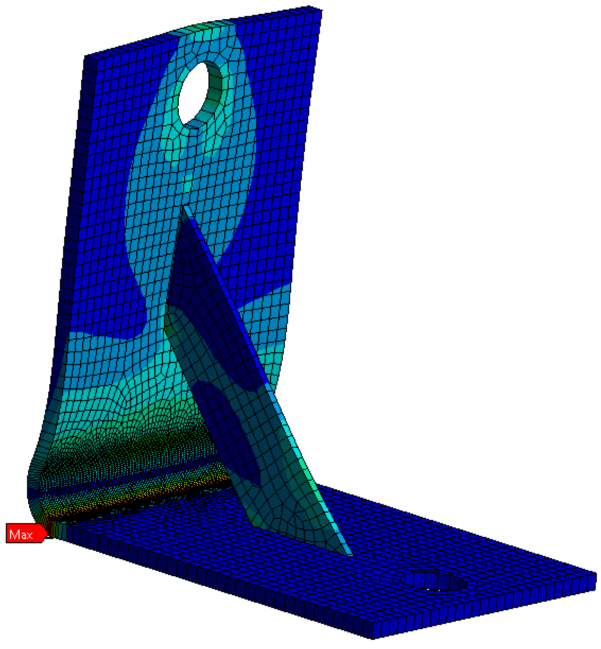


Solve and postprocessing

18. Launch a Solve : RMB on solution > solve



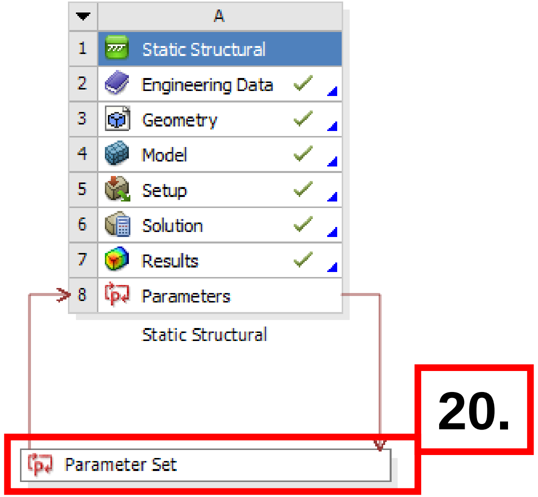
19. Postprocess equivalent stresses



Parameter Management

20. **Access the Parameter Set:**
    a. **Return to WorkBench project schematic**
    b. **From the schematic double click “Parameter Set”.**

When the parameter workspace opens make sure the 2 thicknesses, the mass and the stress are all shown in the parameter list.



Outline of All Parameters

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	Static Structural (A1)			
4	P3	Bracket Thickness	2	mm
5	P4	Gusset Thickness	1	mm
*	New input parameter	New name	New expression	
7	Output Parameters			
8	Static Structural (A1)			
9	P1	Geometry Mass	0.069393	kg
10	P2	Equivalent Stress Maximum	2.3111	MPa
*	New output parameter		New expression	
12	Charts			

Table of Design Points

	A	B	C	D	E	F	G	H
1	Name	P3 - Bracket Thickness	P4 - Gusset Thickness	P1 - Geometry Mass	P2 - Equivalent Stress Maximum	Retain	Retained Data	Note
2	Units	mm	mm	kg	MPa			
3	DP 0 (Current)	2	1	0.069393	2.3111	☑	✓	
*						☐		

. . . Parameter Management

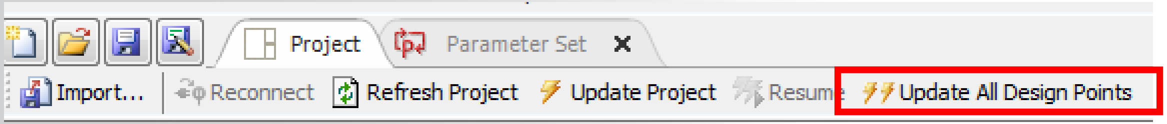
21. Enter thickness values as shown below.

21.

Table of Design Points

	A	B	C	D	E
1	Name	P3 - Bracket Thickness	P4 - Gusset Thickness	P1 - Geometry Mass	P2 - Equivalent Stress Maximum
2	Units	mm	mm	kg	MPa
3	DP 0 (Current)	2	1	0.069393	2.3111
4	DP 1	2	2		
5	DP 2	3	1		
6	DP 3	3	2		
7	DP 4	4	1		
*					

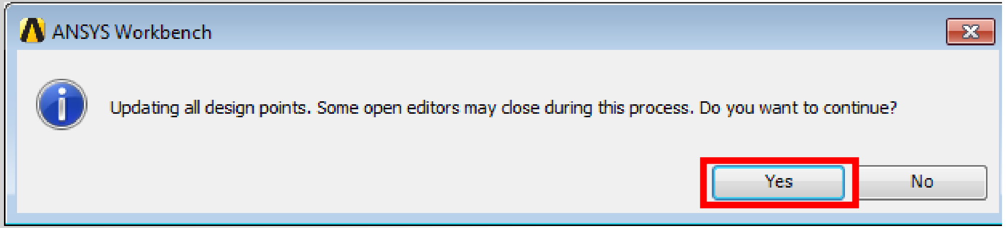
22. "Update All Design Points" will instruct Mechanical to execute a solve for each scenario in the Design Point table.



22.

. . . Parameter Management

Once the update process begins a message will appear as shown here. In fact the Mechanical application window will close during the update process. This is normal.



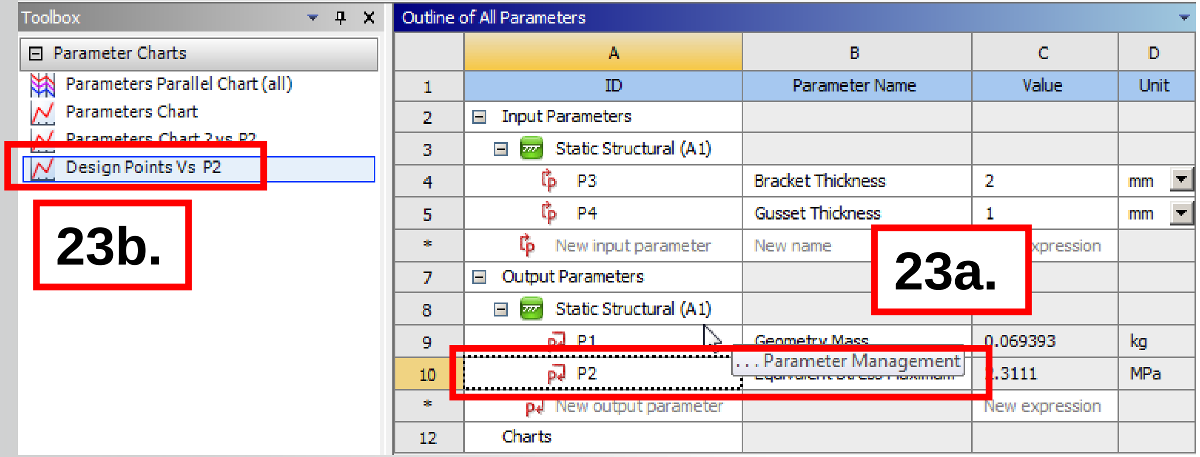
When the updates are complete the table will show calculated values for both output parameters.

Table of Design Points

	A	B	C	D	E	F	G	H
1	Name	P3 - Bracket Thickness	P4 - Gusset Thickness	P1 - Geometry Mass	P2 - Equivalent Stress Maximum	Retain	Retained Data	Note
2	Units	mm	mm	kg	MPa			
3	DP 0 (Current)	2	1	0.069393	2.3111	☑	✓	
4	DP 1	2	2	0.072631	2.2086	☐		
5	DP 2	3	1	0.10247	1.0972	☐		
6	DP 3	3	2	0.10571	1.0491	☐		
7	DP 4	4	1	0.13555	0.64505	☐		
*						☐		

. . . Parameter Management

23. There are several ways we can present the design point information. In this case we'll see how output quantities vary with each design point:
    a. Highlight the output parameter "Equivalent Stress Maximum" (P2 here).
    b. Double click the "Design Points Vs P2" choice in the Toolbox (again, parameter numbers will vary depending on the order of their definition).

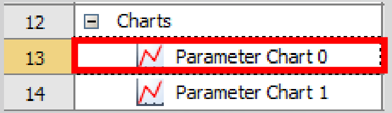


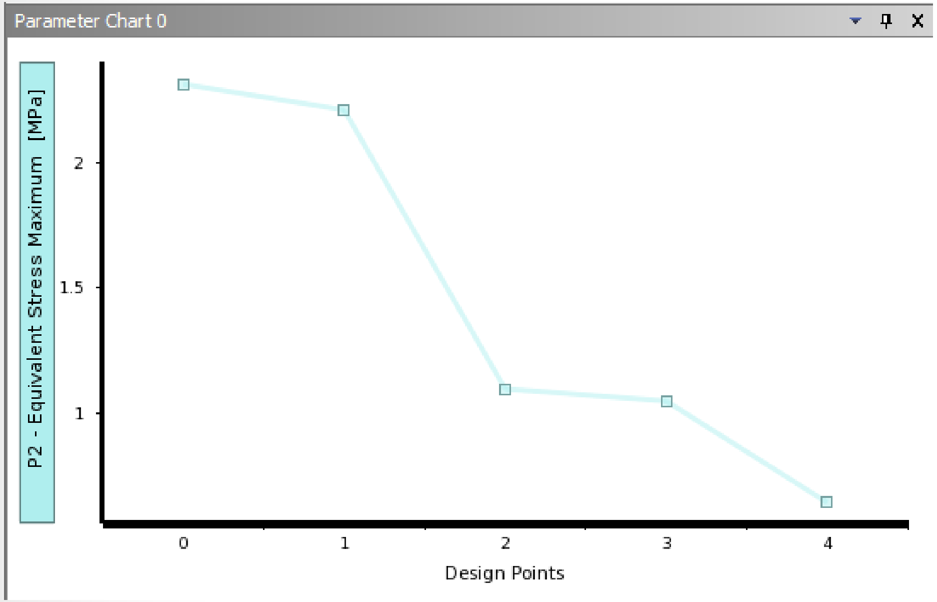
- Repeat the above steps with the "Geometry Mass" parameter (P1 in this case).

. . . Parameter Management

24. Highlight the Parameter Chart 0 chart to display:

24.

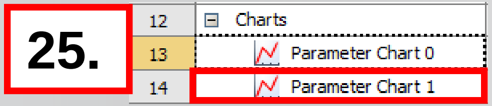


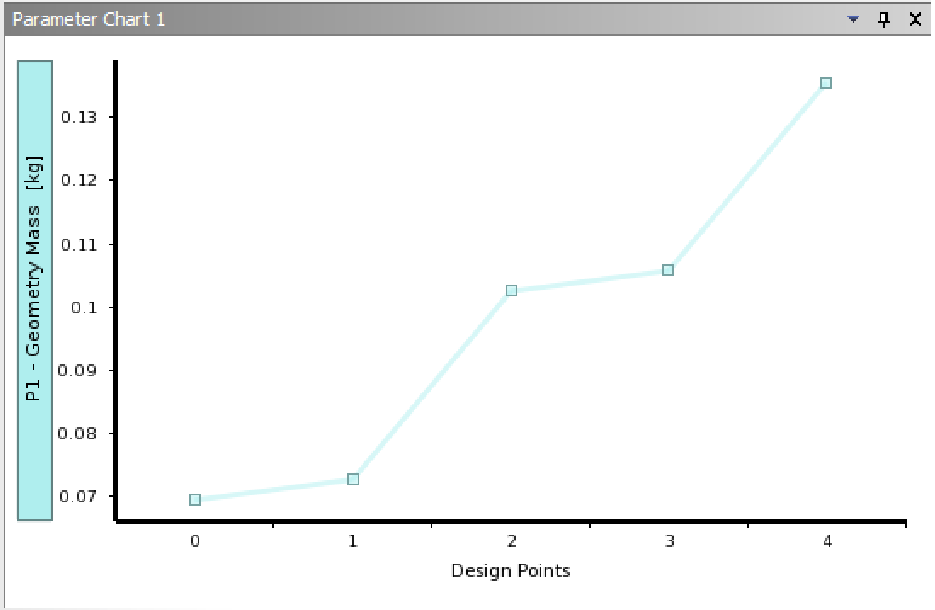


Stress per Design Point

. . . Parameter Management

25. Highlight the Parameter Chart 1 to display.



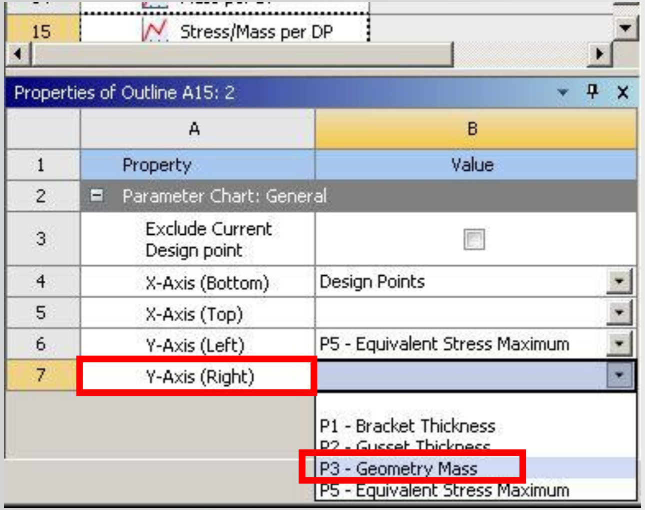


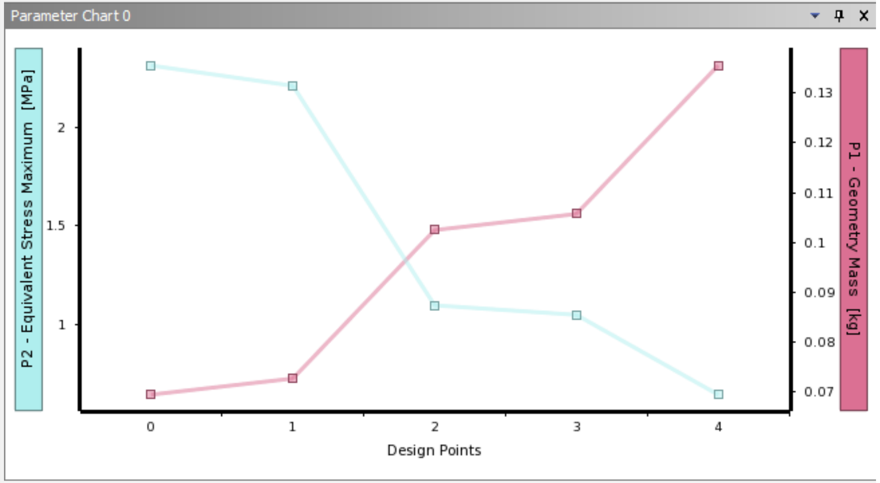
Mass per Design Point

. . . Parameter Management

Repeat step 24 and create a stress vs DP plot.

In the properties window choose to display “Geometry Mass” on the right side Y axis as shown below.





Plots like this one allow us to visualize the trade off that often accompanies these kinds of choices.

Go further!

Specifications for this study impose to a maximum Von Mises stress below 15MPa, a mass below 0.1 Kg and that the frequency of the first mode below than 2000Hz.

Please find the good candidate which respect the three conditions detailed before.