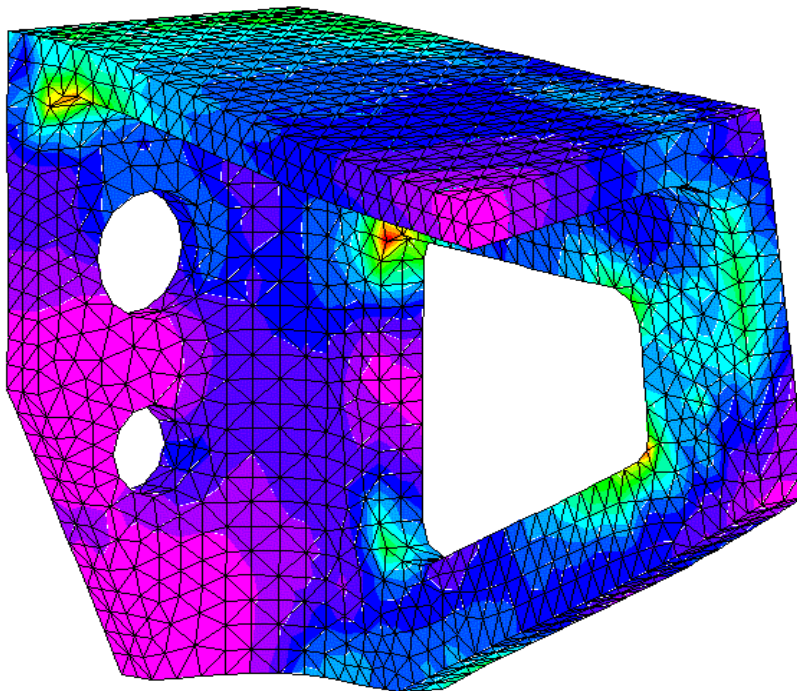


---

## WORKSHOP 2

# *Step File Import (V4.0)*



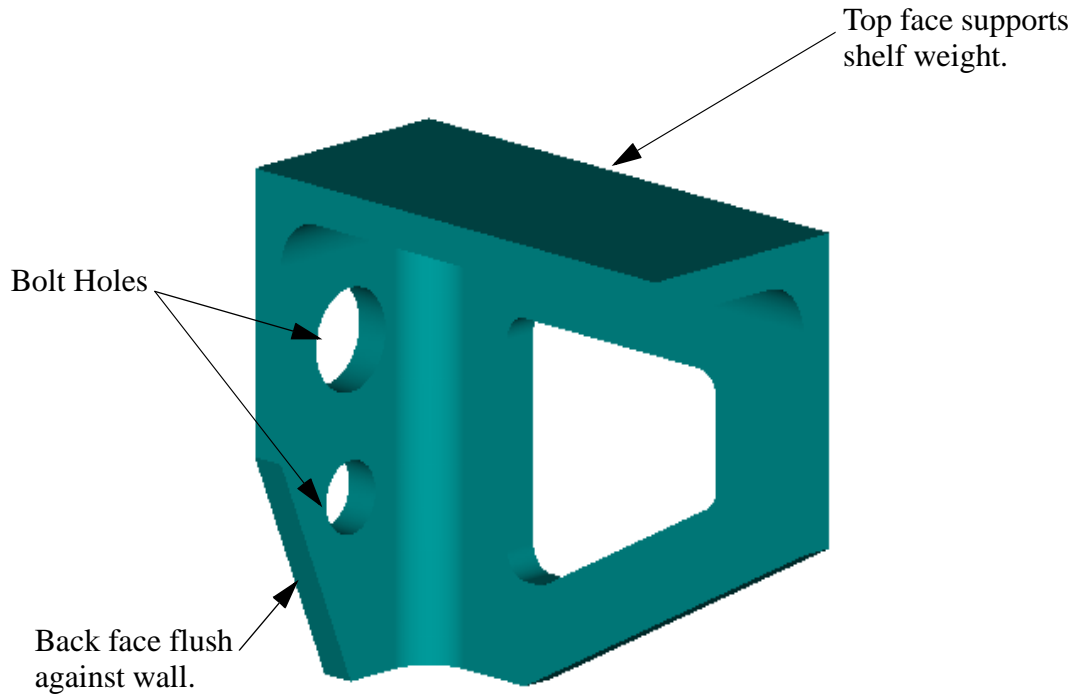
### **Objectives:**

- Import a STEP file.
- Apply loads to the model.
- Run an MSC.Nastran linear static analysis.
- Create an animation that shows the effects of the loading.



## Model Description:

Below is a model of a shelf bracket. This bracket is expected to support up to 72 lbs of load. Given the surface area of the top (and the assumption that the load is evenly distributed) this works out to a 5 psi loading on the top face. The bracket is bolted to a wall along its back face.



Run an analysis on the bracket and perform various postprocessing techniques in order to determine the stresses on the bracket.

---

## Exercise Procedure:

1. Start up MSC.Nastran for Windows V4.0 and begin to create a new model.

Double click on the icon labeled **MSC.Nastran for Windows V4.0**.

On the *Open Model File* form, select **New Model**.

*Open Model File:*

**New Model**

2. Import the model file.

From the pulldown menu, select **File/Import/Geometry**.

**File/Import/Geometry...**

*File Name:*

**bracket.STP**

**Open**

**OK**

Rotate the model to an isometric view.

**View/Rotate...**

<F8>

**Isometric**

**OK**

3. Create a material called **steel**.

From the pulldown menu, select **Model/Material**.

**Model/Material...**

*Title:*

**steel**

*Young's Modulus:*

**30e6**

*Poisson's Ratio:*

**.3**

**OK**

**Cancel**

4. Create the solid element properties for the model.

From the pulldown menu, select **Model/Property**.

**Model/Property...**

*Title:*

**solid**

*Material:*

**1..steel**

**Elem/Property Type...**

**Solid**

**OK**

**OK**

**Cancel**

5. Create the mesh for the model.

**Mesh/Geometry/Solids...**

**Tet Meshing**

*Element Size:*

**0.19**

**OK**

*Property:*

**1..solid**

**Midside Nodes**

**OK**

Turn off mesh before creating constraints on the model.

**View/Options...**

**<F6>**

**Quick Options...**

**Node**

**Element**

**Done**

**OK**

- 
6. Create the constraints on the model.

**Model/Constraint/Nodal...**

*Title:*

**constraint**

**OK**

**Methods^**

**On Surface**

On the bracket, select the back face with holes (surface 1).

**OK**

*DOF:*

**TZ**

**OK**

**Methods^**

(select **On Surface**)

On the bracket, select the top and bottom faces of the holes (surfaces 16, 17, 22, 23).

**OK**

*DOF:*

**TX**    **TY**

**OK**

When asked what to do with the existing set:

**No**

(combine load sets)

**Cancel**

7. Create the loading on the model.

**Model/Load/Elemental...**

*Title:*

**Pressure**

**OK**

**Methods^**

**On Surface**

Select the top face of the bracket (surface 3).

**OK**

Select **Pressure** as the load type.

*Value:*

**5.0**

**OK**

*Method:*

**Near Surface**

*Surface:*

(select the top face - 3)

**OK**

**Cancel**

- Turn on all analysis entities and turn off all the Load/Constraint markers.

**View/Options...**

<F6>

**Quick Options...**

**Analysis Entities On**

**Load/Constraint Off**

**Done**

**OK**

Turn off the workplane.

**Tools/Workplane...**

<F2>

**Draw Workplane**

**Done**

Regenerate drawing.

**View/Regenerate...**

<CTRL+G>

(or double click left mouse button)

- 
9. Run the analysis.

**File/Export/Analysis Model...**

*Analysis Format/Type:*

**1..Static**

**OK**

Change the directory to **C:\Temp**.

*File Name:*

**bracket**

**Write**

*Additional Info:*

**Run Analysis**

**OK**

When asked if you wish to save the model, respond **Yes**.

**Yes**

*File Name:*

**bracket**

**Save**

When the MSC.Nastran manager is through running, MSC.Nastran will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran successfully, we will not bother with the details this time.

**Continue**

10. Clean up the display..

**View/Options...**

<F6>

**Quick Options...**

**All Entities Off**

**Element**

**Done**

**OK**

<b>View/Rotate...</b>	<b>&lt;F8&gt;</b>
X:	<input type="text" value="15"/>
Y:	<input type="text" value="45"/>
Z:	<input type="text" value="0"/>
<input type="button" value="OK"/>	

11. Post a quick results plot, including a contour plot of VonMises stress on the deformed model.

<b>View/Select...</b>	<b>&lt;F5&gt;</b>
<i>Model Style:</i>	<input checked="" type="radio"/> <b>Full Hidden Line</b>
<i>Deformed Style:</i>	<input checked="" type="radio"/> <b>Deform</b>
<i>Contour Style:</i>	<input checked="" type="radio"/> <b>Contour</b>
<input type="button" value="Deformed and Contour Data..."/>	
<i>Output Set:</i>	<input type="text" value="1..MSC.Nastran Case 1"/>
<i>Deformation:</i>	<input type="text" value="1..Total Translation"/>
<i>Contour:</i>	<input type="text" value="60031 Solid VonMises Stress"/>
<input type="button" value="OK"/>	
<input type="button" value="OK"/>	

When done, exit MSC.Nastran for Windows

This concludes the exercise.

**File/Save**

**File/Exit**

