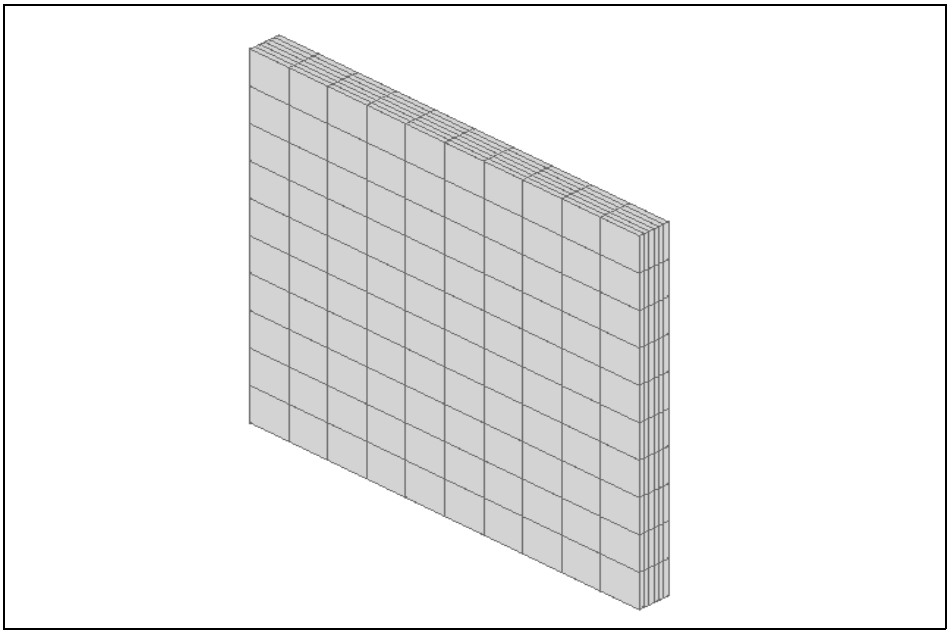
## WORKSHOP 8

# *Thermal Stress Analysis of a Bi-Metallic Plate*



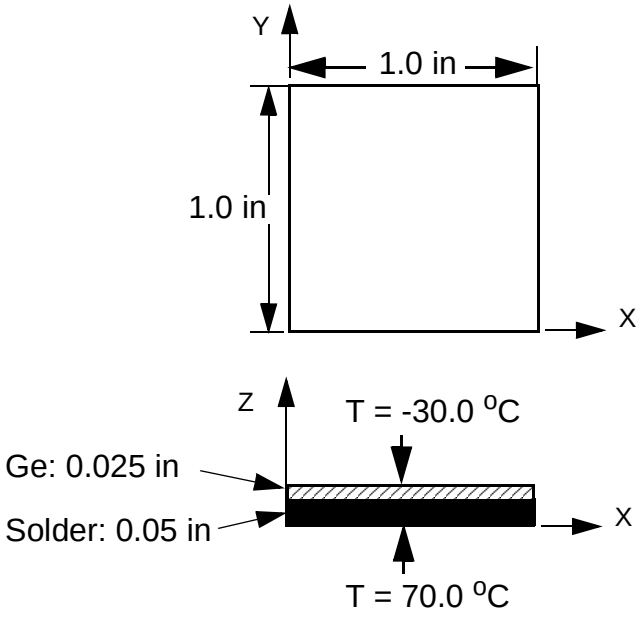
## Objectives:

- Create the geometry of the Bi-Metallic Plate.
- Apply the thermal loading conditions.
- Run the steady-state thermal analysis.
- Run static analysis from the thermal result.

# Model Description:

In this example we will perform the thermal stress analysis of a bi-metallic strip. We will build the entire model from geometric construction so that we can apply loads directly on the geometry. The dimension of the bi-metallic strip is one inch by one inch. The thickness for the solder type material is 0.05 inch, and the thickness of the Ge material is 0.025 inch. Thus the assembly thickness is 0.075 inch.

The top surface temperature boundary condition is -30 degrees C, and the bottom surface temperature boundary condition is 70 degrees C. We will determine the temperature distribution by running a steady-state thermal analysis.




$K_{Ge}$ = 1.524 W/in-$^{o}$C

$K_{solder}$ = 1.27 W/in-$^{o}$C

$E_{Ge}$ = 1.885E7 lb/in$^2$
$G_{Ge}$ = 0.933E7 lb/in$^2$
$\alpha_{Ge}$ = 5.8E-6 in/in-$^{o}$C

$E_{Solder}$ = 1.3E7 lb/in$^2$
$\nu_{Solder}$ = 0.4
$\alpha_{Solder}$ = 2.47E-5 in/in-$^{o}$C

$T_{ref}$ = -30 $^{o}$C

# Exercise Procedure:

1. Start up MSC.Nastran for Windows 4.0.2 and begin to create a new model.

Double click on the icon labeled MSC.Nastran for Windows V4.0.2.

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

*Open Model File:* **New Model**

2. Create the geometry for the plate.

**Tools/Advanced Geometry...** ● **Standard**

**OK**

**Geometry/Surface/Plane...**

**OK**

**OK**

**Cancel**

**View/Rotate... <F8>**

**Isometric**

**OK**

**Geometry/Volume/Extrude...**

**Select All**

**OK**

| | *X:* | *Y:* | *Z:* |
|---|---|---|---|
| *Base:* | **0** | **0** | **0** |
| *Tip:* | **0** | **0** | **0.05** |

**OK**

*(Select the front surface of the solid (**Surface 2**).)*

**OK**

| | | | |
|---|---|---|---|
| *Base:* | **0** | **0** | **0** |
| *Tip:* | **0** | **0** | **0.025** |

**OK**

**Cancel**

Use Autoscale and to better view the model.

**View/Autoscale... <Ctrl+A>**

**Isometric**

**OK**

3. Create a material called **Ge** and **solder** and specify thermal material properties.

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

**Model/Material...**

| | |
|---|---|
| *Title:* | **Ge** |
| *Youngs Modulus, E:* | **1.885e7** |
| *Shear Modulus, G:* | **0.993e7** |
| *Expansion Coeff, a:* | **5.8e-6** |
| *Conductivity, k:* | **1.524** |
| *Reference Temp:* | **-30** |

**OK**

| | |
|---|---|
| *Title:* | **solder** |
| *Youngs Modulus, E:* | **1.3e7** |
| *Poisson's Ratio, nu:* | **0.4** |

*Expansion Coeff, a:* **2.47e-5**

*Conductivity, k:* **1.27**

*Reference Temp:* **-30**

**OK**

**Cancel**

4. Create a property called **GE** and **solder** to apply to the plate itself.

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

**Model/Property...**

*Title:* **Ge**

*Material:* **1..Ge**

**Elem/Property Type...**

*Volume Elements:* **Solid**

**OK**

**OK**

*Title:* **solder**

*Material:* **2..solder**

**OK**

**Cancel**

5. Define the mesh size on the palte.

**Mesh/Mesh Control/Size Along Curve...**

*(Select the 4 corners edges of solid 1 that are in the z direction (**Edge 9 to 12**).)*

**OK**

● **Number of Element** **4**

**OK**

*(Select the 4 corner edges of Solid 2 (**Edge 17 to 20**).)*

**OK**

● **Number of Element** **2**

**OK**

**Cancel**

**Mesh/Mesh Control/Default Size...**

*Size:* **0.1**

**OK**

**Mesh/Geometry/Volume...**

*(Select the back solid (**solid 1**).)*

**OK**

*Property:* **2..solder**

**OK**

**Mesh/Geometry/Volume...**

**OK**

*(Select the front solid (**solid 2**).)*

**OK**

*Property:* **1..Ge**

**OK**

6. Check for coincident nodes and merge them.

**Tools/Check/Coincident Nodes...**

**Select All**

**OK**

When asked if you wish to specify an additional range of nodes to merge, select **No**.

**No**

☒ **Merge Coincident Entities**

**OK**

7. Apply temperature loads and boundary conditions.

**Model/Load/Set...**

*Title:* **thermal**

**OK**

**Model/Load/On Surface...**

*(Select the back surface (**surface 1**).)*

**OK**

*(highlight)* **Temperature**

*Value:* **70**

**OK**

*(Select the front surface (**surface 7**).)*

**OK**

*Temperature/Value:* **-30**

**OK**

**Cancel**

**Model/Load/Body...**

*(next to Thermal options)* ☒ **Active**

*Default Temperature T:* **20**

**OK**

8. Perform the thermal analysis.

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

*Type:* **20..Steady-State Heat Transfer**

**OK**

*File Name:* **BiM-thermal**

**Write**

☒ **Run Analysis**

**OK**

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

**Yes**

*File Name:* **BiM-plate**

**Save**

When the MSC.Nastran manager is through running, MSC.Nastran for Windows 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 smoothly, we will not bother with the details this time.

**Continue**

9. Post-Process the thermal result.

**View/Select... <F5>**

*Contour Style:* ● **Contour**

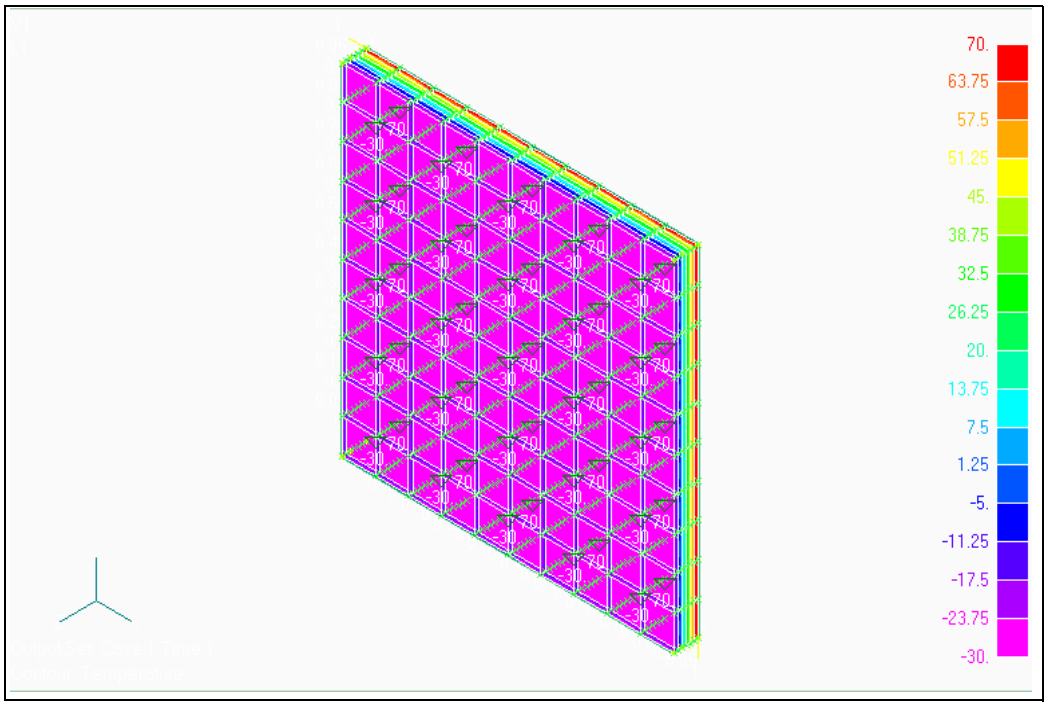
**Deformed and Contour Data...**

*Contour:* **31..Temperature**

**OK**

**OK**

Figure 8-1: Thermal Result.




10. Now apply the loading condition for static analysis.

**View/Select... <F5>**

*Contour Style:* ● **Non-Model Only**

**OK**

**Model/Laod/Set...**

*ID:* **2**

*Title:* **static**

**OK**

**Model/Load/From Output...**

*Nodal Loads:* ● **Temperature**

**OK**

*Output Set:* **1..Case 1 Time 1**

*X Vector:* **31..Temperature**

**OK**

**Quick Options <Ctrl+Q>**

**Labels Off**

**Done**

**Model/Constraint/Set...**

*Title:* **constraint**

**OK**

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

**Method^**

**On Point**

*(Select the two left corners of the front surface (**Point 15 and 16**).)*

**OK**

*DOF:* ☒ **TX**

**OK**

**Method^**

**On Point**

*(Select the top right corner of the front surface (**Point 14**).)*

**OK**

*DOF:* ☒ **TY**

**OK**

**Method^**

**On Point**

*(Select the four corners of the front surface (**Point 13 to 16**).)*

**OK**

*DOF:* ☒ **TZ**

**OK**

When asked “OK to Overwrite (No = combine)?”, respond No.

**No**

**Cancel**

11. Static Analysis

**Filel/Export/Analysis Model...**

*Type:* **1..static**

**OK**

*File Name:* **BiM-static**

**Write**

☒ **Run Analysis**

**OK**

**Yes**

**Continue**

12. Post-Process the thermal stress result.

**Quick Options <Ctrl+Q>**

**All Entities Off**

☒ **Element**

**Done**

**View/Select... <F5>**

*Contour Style:* ● **Contour**

**Deformed and Contour Data...**

*Output Set:* **2..MSC/NASTRAN Case 1**

There will be a warning, click OK.

**OK**

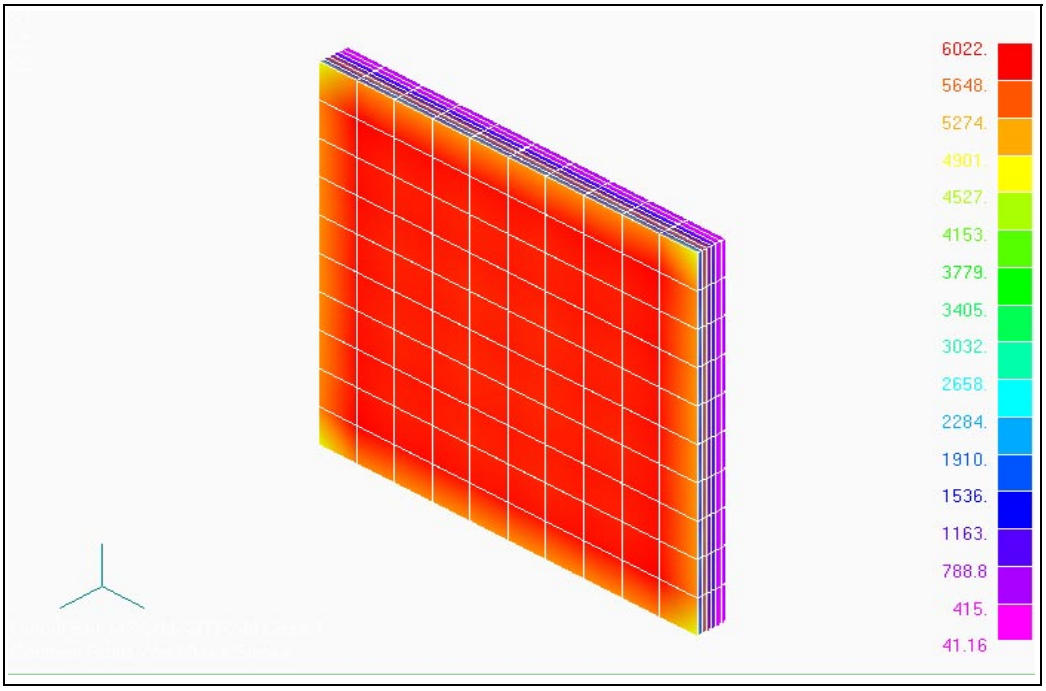
*Contour:* **60031..Solid Von Mises Stress**

**OK**

*Model Style:* ☒ **Render**

**OK**

Figure 8-2: Von Mises Stress Result.




**View/Select... <F5>**

*Deformed Style:* ● **Deform**

**Deformed and Contour Data...**

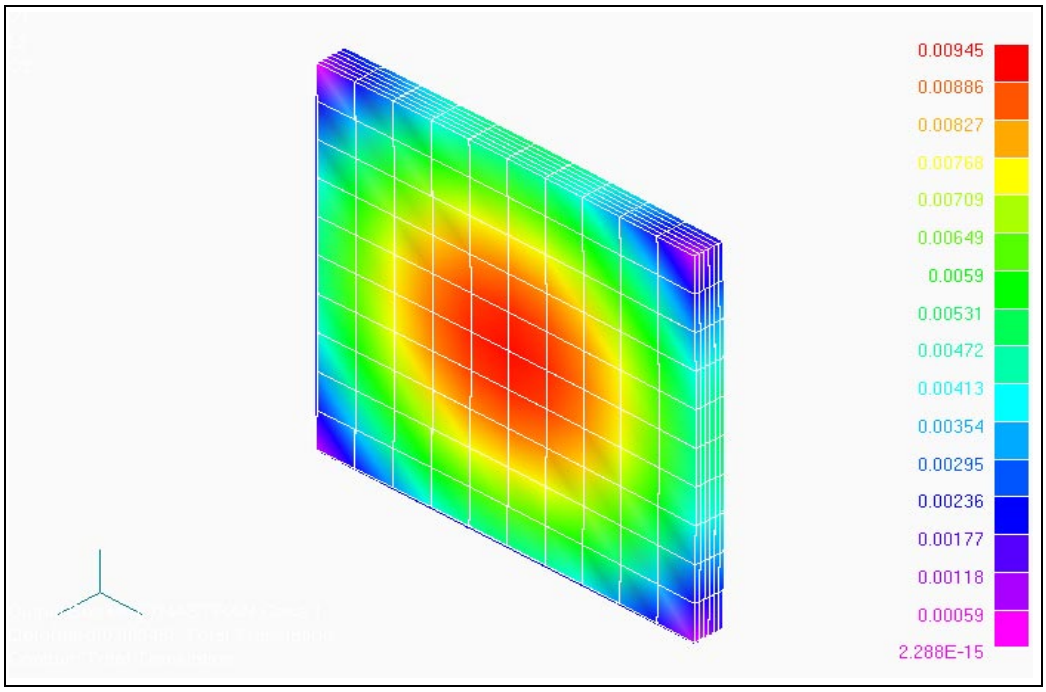
*Output Vectors /Deformation:* **1..Total Translation**

*Output Vectors/Contour:* **1..Total Translation**

**OK**

**OK**

Figure 8-3: Displacment Result.




When done, exit MSC.Nastran for Windows.

**File/Save**

**File/Exit**