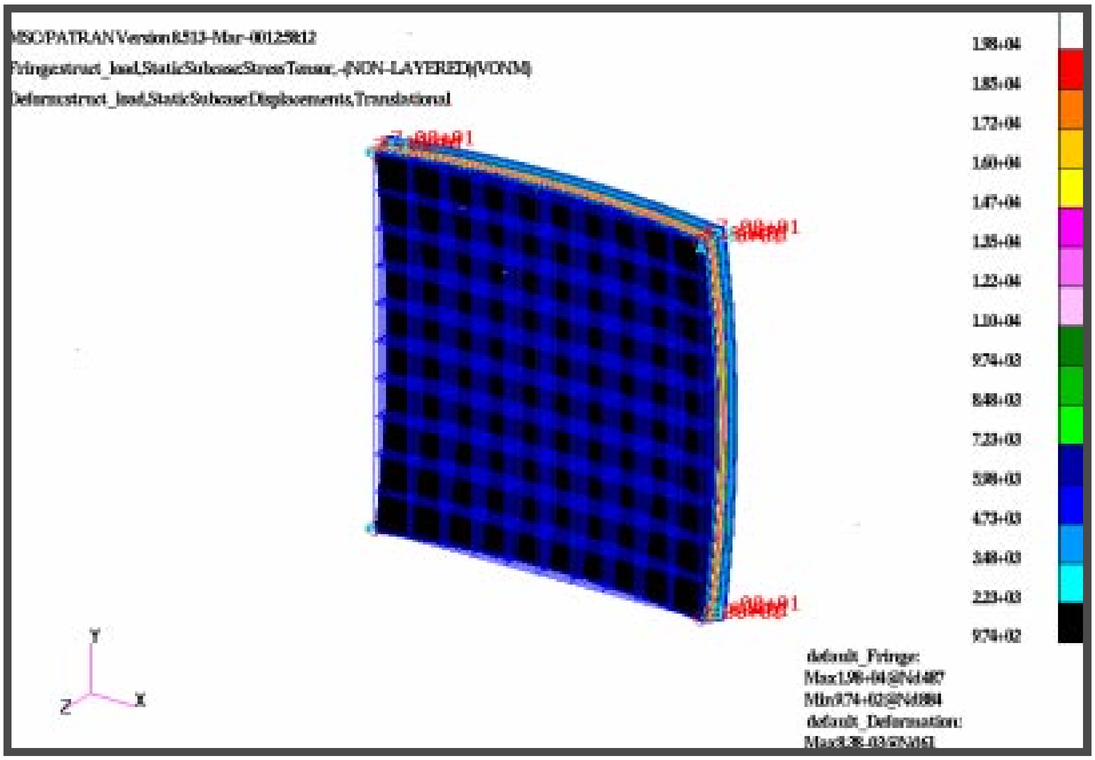
WORKSHOP 10

Thermal Stress Analysis of a Bi-Metallic Plate

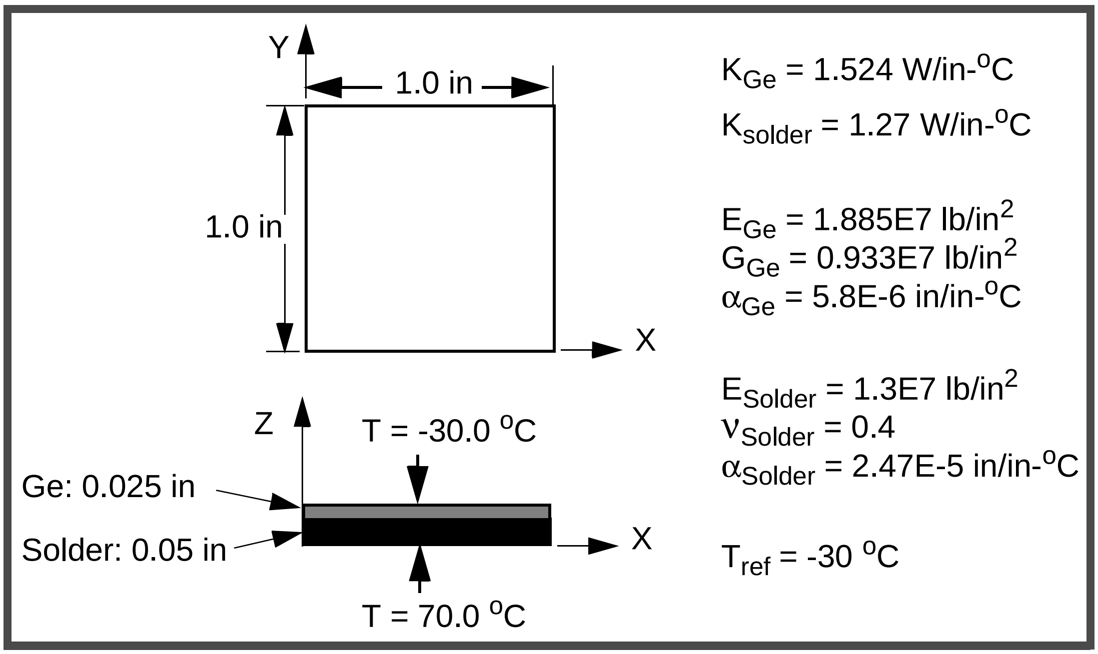


Model Description:

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

Figure 10.1



Prior to the development of the MSC.Patran MSC.Nastran Heat Transfer interface, one would request:

TEMP(PUNCH)=all

in the MSC.Nastran Case Control section of the thermal run. The temperature load is then created and saved inside the punch file. In the subsequent thermal stress analysis one can access this file by defining

TEMP(LOAD)=1

in the Case Control section of the ensuing stress analysis run.

However, using MSC.Patran you can use the **Create-Spatial-FEM** command after you have postprocessed the thermal result in the viewport. We will use this technique to apply a thermal load for the stress analysis. Also, we will analyze the thermal stress analysis for the free-free expansion by enforcing a minimum number of constraints to fix-rigid body motion.

Suggested Exercise Steps:

- Create a new database called **ex10.db**
- Create a geometry representing a bi-metallic strip.
- Mesh the solid using Uniform Mesh Seed for solid 1 and Mesh using HEX8 for both solids.
- Merge all coincident nodes using Equivalence action in the Finite Elements menu
- Specify thermal material properties.
- Define properties using 3D solid for each individual parts.
- Apply temperature boundary conditions to the solid.
- Analyze, perform, and read the results.
- Define a spatial FEM Field based on the temperature Profile.
- Define the new material properties using structural analysis.
- Apply different loads and boundary conditions for the solid
- Perform the structural analysis and read the results.

Exercise Procedure:

1. Open a new database. Name it **ex10.db**

File/New...

New Database Name:	**ex10**

OK

The viewport (PATRAN's graphics window) will appear along with a *New Model Preference* form. The *New Model Preference* sets all the code specific forms and options inside MSC.PATRAN.

In the *New Model Preference* form set the *Analysis Code* to **MSC.Nastran**

Tolerance:	◆ **Based on Model**
Analysis Code:	**MSC/NASTRAN**
Analysis Type:	**Thermal**

OK

2. Create the Model.

◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**XYZ**
Vector Coordinates List:	**<1 1 0>**
Origin Coordinates List:	**[0 0 0]**

Apply

◆ **Geometry**

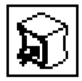
Action:	**Create**
Object:	**Solid**
Method:	**Extrude**
Translation Vector:	**<0 0 0.05>**

Auto Execute

Surface List:	**Surface 1**

Apply

Click on the **Solid Face** icon.

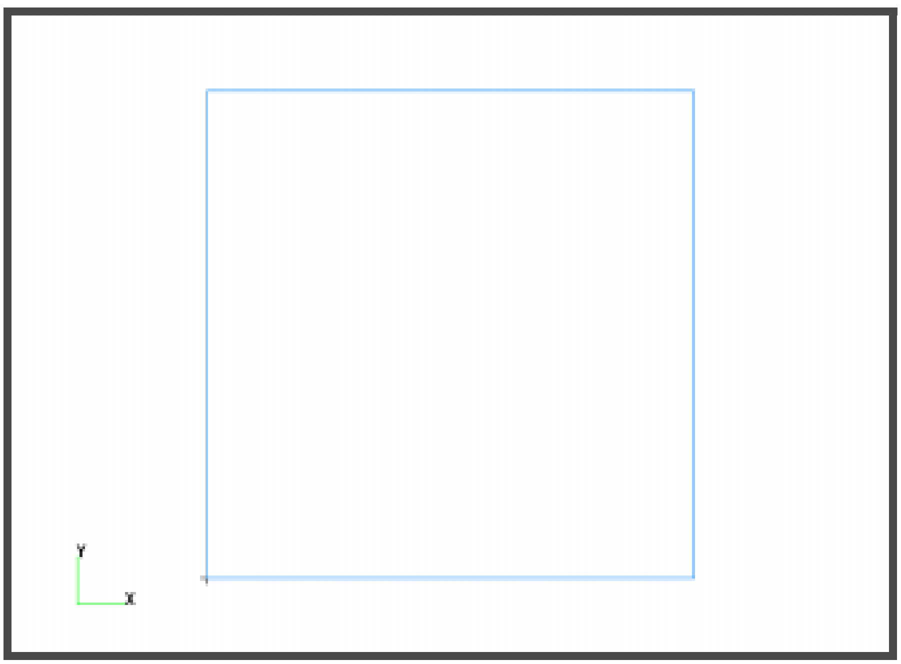


Solid Face

Translation Vector:	**<0 0 0.025>**
Surface List:	**Solid 1.6**

Apply

Your model should look like the following figure.



3. Mesh the Solids.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**

Number:	**4**

Click on the four corners of Solid 1. Hold shift key down while you click.

Curve List:	**Solid 1.1.1 1.2.1 1.2.3 1.1.3**

Apply

Number:	**2**

Click on the four corners of Solid 2. Hold shift key down while you click.

Curve List:	**Solid 2.1.1 2.2.1 2.2.3 2.1.3**

Apply

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Solid**
Global Edge Length:	**0.1**
Element Topology:	**Hex8**
Solid List:	**Solid 1 2**

Apply

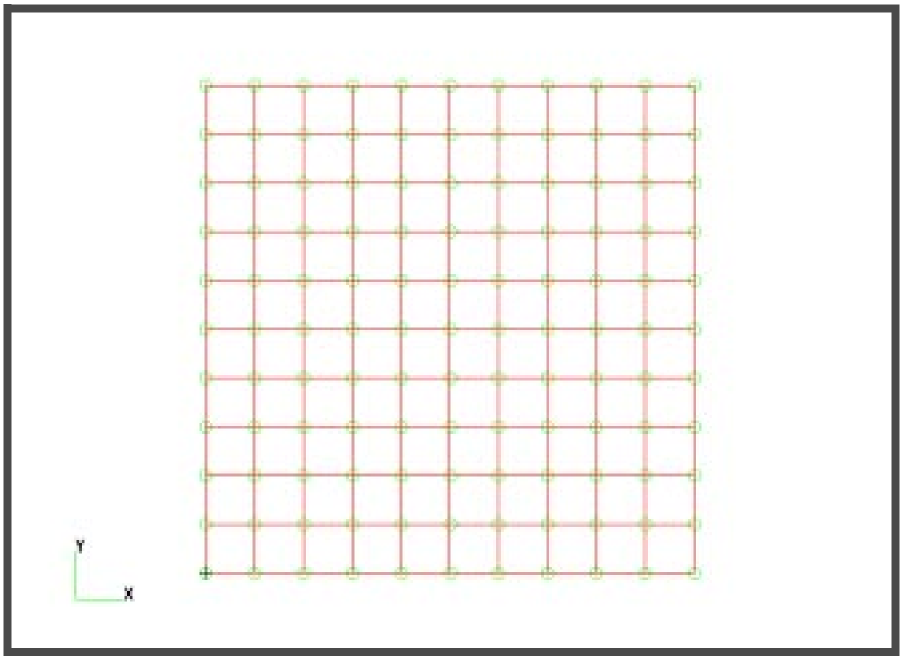
4. Remove Coincident Nodes.

◆ **Finite Elements**

Action:	**Equivalence**
Object:	**All**
Type:	**Tolerance Cube**
Equivalencing Tolerance:	**0.005**

Apply

Your model should look like the following figure.



5. Specify Thermal Material Properties.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**Ge**
Input Properties...	
Constitutive Model:	**Solid properties**
Thermal Conductivity:	**1.524**
Apply	
Cancel	
Material Name:	**Solder**
Input Properties...	
Constitutive Model:	**Solid properties**
Thermal Conductivity:	**1.27**
Apply	

Cancel

6. Define Element Properties.

◆ **Properties**

Action: **Create**

Object: **3D**

Type: **Solid**

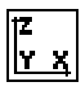
Property Set Name: **Ge**

Input Properties...

Material Name: **m:Ge**

OK

Click on the **Bottom View** icon.



Bottom View

Select Members: **Solid 2**

Add

Apply

Property Set Name: **Solder**

Input Properties...

Material Name: **m:Solder**

OK

Select Members: **Solid 1**

Add

Apply

7. Apply temperature boundary conditions.

◆ **Load/BCs**

Action: **Create**

Object: **Temp(Thermal)**

Type: **Nodal**

Analysis Type: **Thermal**

New Set Name: **temp_bottom**

Input Data...

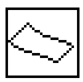
Boundary Temperature: **70**

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

Click on the **Surface or Face** icon.



Surface or Face

Select Geometry Entities: **Surface 1**

Add

OK

Apply

New Set Name: **temp_top**

Input Data...

Boundary Temperature: **-30**

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

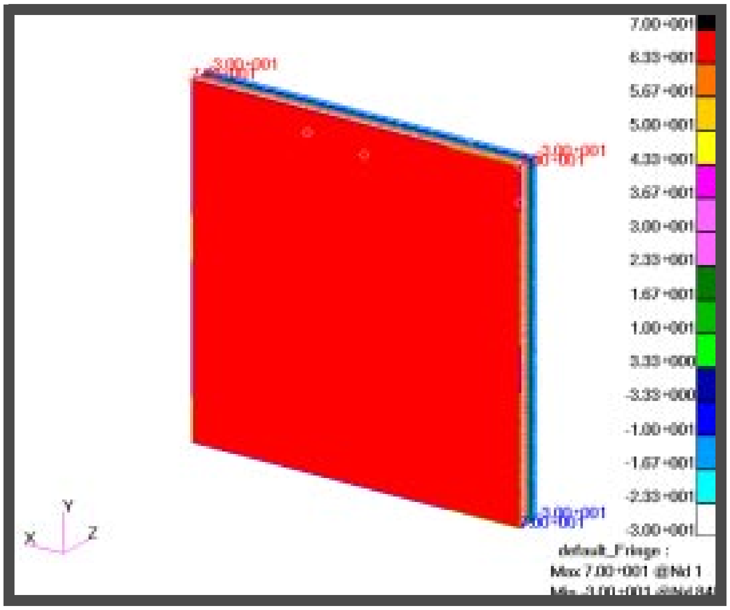
Select Geometry Entities: **Solid 2.6**

Add

OK

Apply

Your model should look like the following figure.



8. Perform the Thermal Analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Job Name:	**ex10**

Apply

An MSC.Nastran input file called **ex10.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green.

Submitting the Input File for Analysis:

9. Submit the input file to MSC.Nastran for analysis.

   9a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran ex10.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

   9b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran ex10 scr=yes**. Monitor the run using the UNIX **ps** command.

10. When the run is completed, edit the **ex10.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

11. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

12. Proceed with the Reverse Translation process, that is, attaching the **ex10.xdb** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

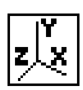
Action:	**Attach XDB**
Object:	**Result Entities**
Method:	**Local**
Select Results File	
Select Results File	**ex10.xdb**
OK	
Apply	

13. Display the Results.

◆ **Results**

Form Type:	**Default, PW Linear: 100. % of Load**
Select Results Cases:	**Temperatures**

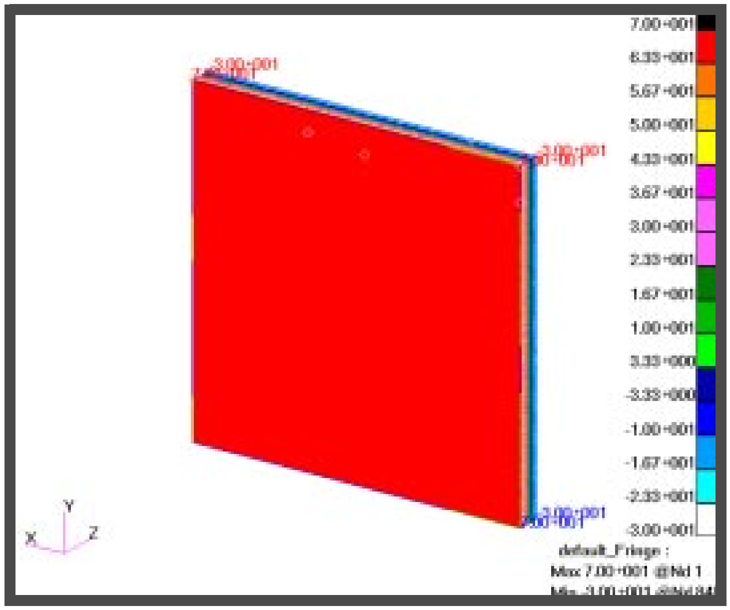
Click on the **Iso 1 View** icon to change the view.



Iso 1 View

Apply

Your model should look like the following figure.



14. Define a Spatial FEM Field based on the Temperature Profile.

◆ **Fields**

Action: **Create**

Object: **Spatial**

Method: **FEM**

Field Name: **t_load**

FEM Field Definition: ◆ **Continuous**

Field Type: ◆ **Scalar**

Mesh/Results Group Filter: ◆ **Current Viewport**

Select Group: **default_group**

Apply

15. Change the Analysis type to Structural.

Preferences/Analysis...

Analysis Type: **Structural**

OK

16. Specify Structural Material Properties.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**Solder_st**
Input Properties...	
Constitutive Model:	**Linear Elastic**
Elastic Modulus:	**1.3e7**
Poisson Ratio:	**0.4**
Thermal Expan. Coeff:	**2.47e-5**
Reference Temperature:	**-30.0**
Apply	
Cancel	
Material Name:	**Ge_st**
Input Properties...	
Constitutive Model:	**Linear Elastic**
Elastic Modulus:	**1.885e7**
Shear Modulus:	**0.933e7**
Thermal Expan. Coeff:	**5.8e-6**
Reference Temperature:	**-30.0**
Apply	
Cancel	

17. Assign Element Properties.

◆ **Properties**

Action:	**Create**
Object:	**3D**

Type: **Solid**

Property Set Name: **Ge_st**

Options: **Standard Formulation**

Input Properties...

Material Name: **m:Ge_st**

OK

Select Members: **Solid 2**

Add

Apply

When asked, "Solid 2 already has been associated to an element property region. Overwrite the association?", answer Yes.

Yes

Property Set Name: **Solder_st**

Options: **Standard Formulation**

Input Properties...

Material Name: **m:Solder_st**

OK

Select Members: **Solid 1**

Add

Apply

When asked, "Solid 1 already has been associated to an element property region. Overwrite the association?", answer Yes.

Yes

18. Create a New Load Case.

◆ **Load Cases**

Action: **Create**

Load Case Name: **struct_load**

Load Case Type: **Static**

Apply

19. Define a Temperature Load.

◆ **Load/BCs**

Action: **Create**

Object: **Temperature**

Type: **Nodal**

Analysis Type: **Structural**

Current Load Case: **struct_load**

New Set Name: **temp_load**

Input Data...

Load/BC Set Scale Factor: **1.0**

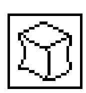
Temperature: **f:t_load**

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

Click on the **Solid** icon.



Solid

Select Geometry Entities: **Solid 1 2**

Add

OK

Apply

20. Apply constraints on the four corner points of the top surface.

◆ **Load/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
Analysis Type:	**Structural**
New Set Name:	**fix_x**
Input Data...	
Load/BC Set Scale Factor:	**1.0**
Translations <T1 T2 T3>	**<0., , >**
OK	
Select Application Region...	
Geometry Filter:	◆ **Geometry**

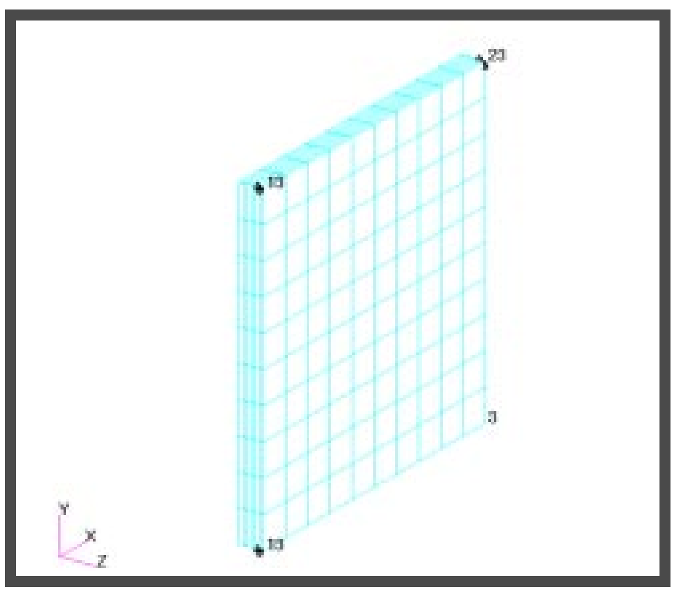
Click on the **Point** icon.

Point

Select Geometry Entities:	**Point 9 10**
Add	
OK	
Apply	
New Set Name:	**fix_y**
Input Data...	
Load/BC Set Scale Factor:	**1.0**
Translations <T1 T2 T3>	**< , 0., >**
OK	
Select Application Region...	
Geometry Filter:	◆ **Geometry**
Select Geometry Entities:	**Point 11**
Add	
OK	

Apply	
New Set Name:	**fix_z**
Input Data...	
Load/BC Set Scale Factor:	**1.0**
Translations <T1 T2 T3>	**< , , 0.>**
OK	
Select Application Region...	
Geometry Filter:	◆ **Geometry**
Select Geometry Entities:	**Point 9:12**
Add	
OK	
Apply	

Your model should look like the following figure.



21. Perform the Structural Analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Job Name:	**ex10_st**

Subcase Select...

Subcases For Solution Sequence:101	**struct_load**
Subcases Selected:	**default**

OK

Apply

Submitting the Input File for Analysis:

22. Submit the input file to MSC.NASTRAN for analysis.

    To submit the MSC.PATRAN **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran ex10_st.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

23. When the run is completed, edit the **ex10_st.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing **WARNING** messages indicate modeling errors.

24. Read in the Analysis Results.

◆ **Analysis**

Action: **Read Output2**

Object: **Result Entities**

Method: **Translate**

Job Name: **ex10_st**

Select Results File...

ex10_st.op2

OK

Apply

25. Display the Results.

◆ **Results**

Select Results Cases: **struct_load, Static Subcase**

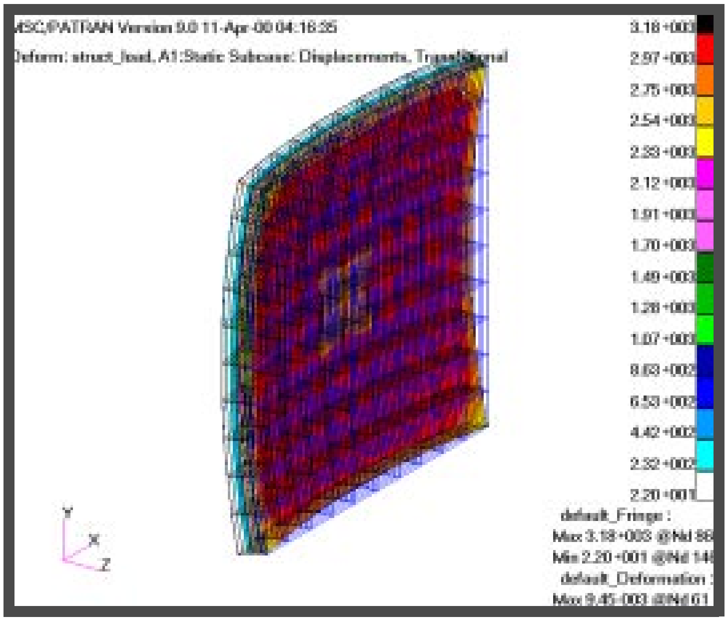
Select Fringe Result: **Stress Tensor**

Result Quantity: **von Mises**

Select Deformation Result: **Displacements, Translational**

Apply

Your model should look like the following figure.



The reference or zero stress state for the assembly is initialized at -30 ^{o}C. The thermal coefficient of expansion for the solder is approximately four times that of Ge. When the temperature gradient associated with the temperature boundary conditions is applied, the solder layer wants to grow significantly more than the Ge layer due not only to the higher coefficient of thermal expansion, but also because of the higher temperature relative to TREF. The Ge layer ends up with a more complex stress pattern due to its four corner points being constrained, the distribution of temperature through the layer, and the growth enforced by the solder layer. The free surface of the solder layer exhibits the low stress levels.

Quit MSC.Patran when you have completed this exercise