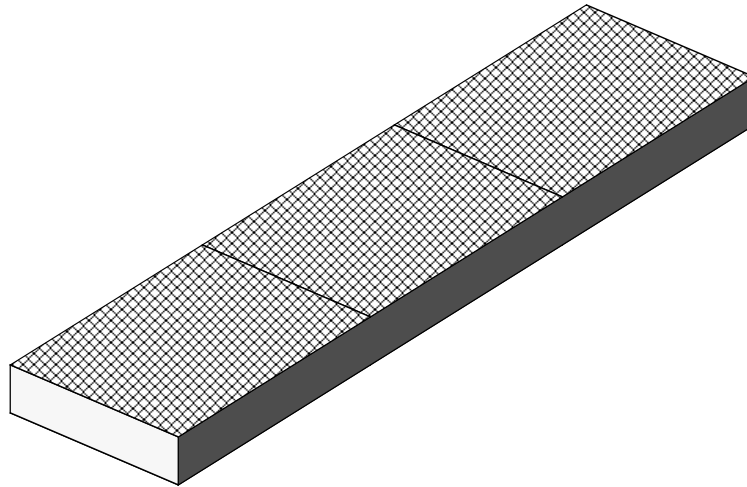


---

## WORKSHOP 11

# *Material Property Definition*



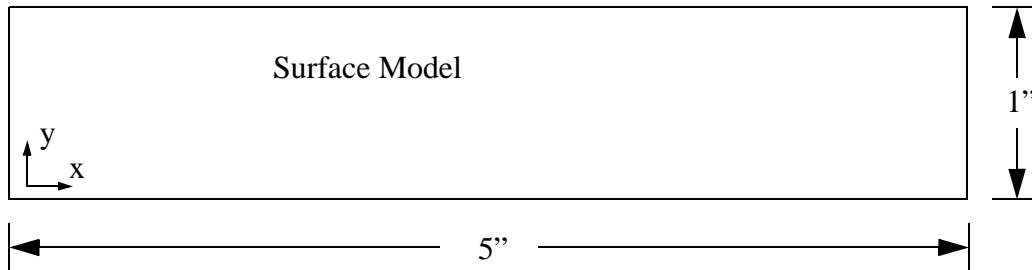
### **Objective:**

- Create a material that has temperature dependent properties.



**Model Description:**

In this exercise you will create several fields that represent the variation of material properties with respect to temperature. You will use fields to define a composite material. This exercise has been designed to contain the required steps that are necessary to create almost any material definition in MSC.Patran.



Finite Element Mesh:  
Global Edge Length= 0.5 in  
QUAD4 elements

Analysis Code: MSC/NASTRAN

Properties:  
Thickness = 0.020 in  
2d Orthotropic material name: mat\_orth2d

**Figure 11-1**

**Table 1: Temperature Dependent Material Properties**

T (°F)	E <sub>11</sub> (Msi)	E <sub>22</sub> (Msi)	v <sub>12</sub>	G <sub>12</sub> (Msi)	G <sub>23</sub> (Msi)	G <sub>13</sub> (Msi)	α <sub>11</sub> (Mils/ in/°F)	α <sub>22</sub> (Mils/ in/°F)
800	1.47	0.364	0.320	0.119	0.227	0.335	0.50	58.90
1200	1.33	0.183	0.320	0.060	0.196	0.303	0.00	71.10
1500	1.25	0.161	0.320	0.053	0.199	0.300	-0.25	15.60

---

## Suggested Exercise Steps:

- Create a new database named **material.db**.
- Change the *Tolerance* to **Default** and the *Analysis Code* to MSC/NASTRAN.
- Create the geometry and the finite element mesh using the information shown in Figure 11-1.
- Create an individual field for each material property listed in Table 1 above that varies with respect to temperature. Use E11, E22, G12, G13, G23, ALPHA11, and ALPHA22 for the field names.
- Create a 2D Orthotropic material named, **mat\_orth2d**, that incorporates the material property fields.
- Define a shell element property named **Prop\_1**. Use the **mat\_orth2d** material to complete its definition and apply it to all the finite elements of your model.

## Exercise Procedure:

1. Create a new database and name it **material.db**. Select the **Default** *Tolerance* and **MSC/NASTRAN** *Analysis Code* in the *New Model Preferences* form.

### File/New...

*New Database Name*

**material**

**OK**

### New Model Preference

*Tolerance*

◆ **Default**

*Analysis Code:*

**MSC/NASTRAN**

**OK**

2. Create the geometry and the finite element mesh using the information shown in Figure 12-1.

### ◆ Geometry

*Action:*

**Create**

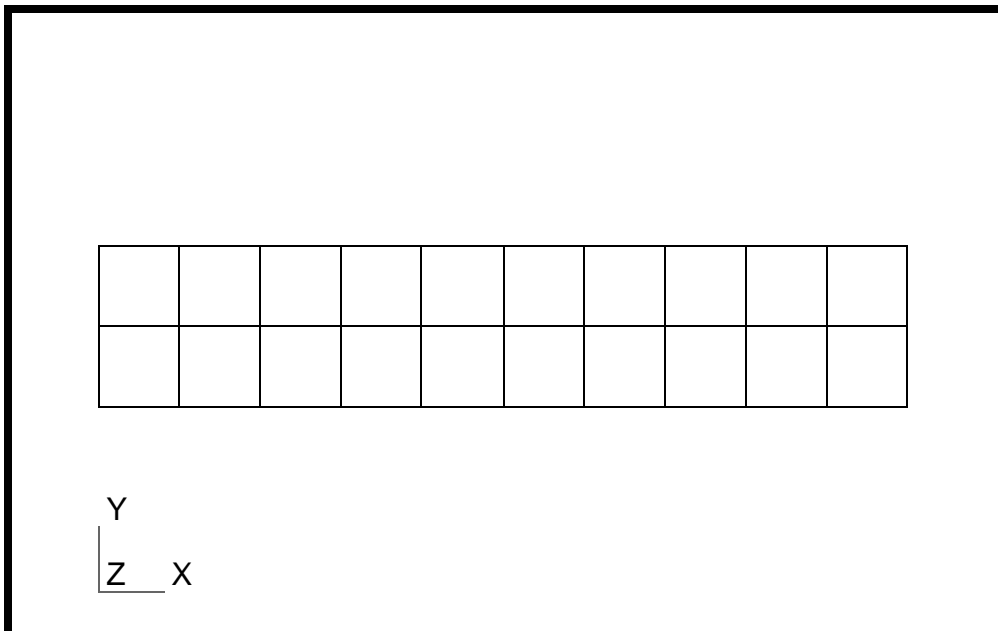
*Object:*   
*Method:*   
*Vector Coordinate List*

To create the finite element model, click on the **Finite Elements** radio button in the *Main Form*.

◆ **Finite Elements**

*Action:*   
*Object:*   
*Type:*   
*Global Edge Length*   
*Element Topology*   
*Surface List*

Your model should look like the one shown below.



## Temperature Dependent Properties

3. Create an individual field for each material property listed in Table 1 above that varies with respect to temperature. Use E11, E22, G12, G13, G23, ALPHA11, and ALPHA22 for the field names.

To define the 2D Orthotropic material, you must create the fields which will define the variation of each material property with respect to temperature.

### ◆ Fields

Action:

Create

Object:

Material Property

Method:

Tabular Input

Using the data listed in Table 1 of this exercise, define the field for E11.

Field Name

E11

Active Independent Variable

Temperature

**Input Data...**

This will open the *1D Material Scalar Table Data* form. Click in the value cells and enter the values shown in Table 1 for E11. Your form should look like this.

	T	Value
1	8.00000E+02	1.47000E+00
2	1.20000E+03	1.33000E+00
3	1.50000E+03	1.25000E+00
4		
5		
6		
7		
8		
9		

OK

Apply

Repeat these steps to create the remaining fields for the other temperature dependent material properties. Use the following names for these fields: (See table 1 on page 11-3)

E22, G12, G13, G23, ALPHA11, ALPHA22.

4. Create a 2D orthotropic material named **mat\_orth2d** that incorporates the material property fields.

**2D  
Orthotropic  
Material**

◆ **Materials**

Action:	<input type="text" value="Create"/>
Object:	<input type="text" value="2D Orthotropic"/>
Method:	<input type="text" value="Manual Input"/>
Material Name	<input type="text" value="mat_orth2d"/>
<input type="button" value="Input Properties..."/>	
Constitutive Model	<input type="text" value="Linear Elastic"/>

Specify each material property by clicking in the Value databoxes on the *Input Options* form, and picking the appropriate field name from the *Temperature Dependent Fields* listbox that will appear at the bottom of the form. Since the Poisson's Ratio listed in Table 1 is constant at all temperatures, enter its value manually.

Elastic Modulus 11	<input type="text" value="E11"/>
Elastic Modulus 22	<input type="text" value="E22"/>
Poisson's Ratio	<input type="text" value="0.32"/>
Shear Modulus 12	<input type="text" value="G12"/>
Shear Modulus 23	<input type="text" value="G23"/>
Shear Modulus 13	<input type="text" value="G13"/>
Thermal Expan. Coeff 11	<input type="text" value="ALPHA11"/>
Thermal Expan. Coeff 22	<input type="text" value="ALPHA22"/>
<input type="button" value="Apply"/>	
<input type="button" value="Cancel"/>	

**Apply  
Material to  
Model**

5. Define a 2D shell element property named **prop\_1**. Use the **mat\_orth2d** material to complete its definition and apply it to all the finite elements of your model.

**◆ Properties**

<i>Action:</i>	<b>Create</b>
<i>Dimension:</i>	<b>2D</b>
<i>Type:</i>	<b>Shell</b>
<i>Property Set Name</i>	<b>prop_1</b>
<i>Options:</i>	<b>Homogeneous</b> <input type="checkbox"/>
	<b>Standard Formulation</b> <input type="checkbox"/>

**Input Properties...**

<i>Material Name</i>	<b>m:mat_orth2d</b>
<i>Thickness</i>	<b>0.020</b>

**OK**

*Select Members*

Select All Finite Elements

You may have to click on the **2d Element** icon.



**Add**

**Apply**

**File/Quit...**