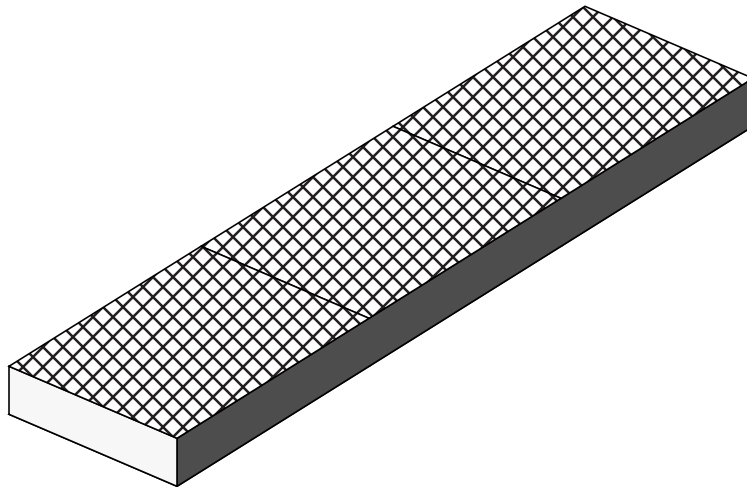

LESSON 12

Material Property Definition



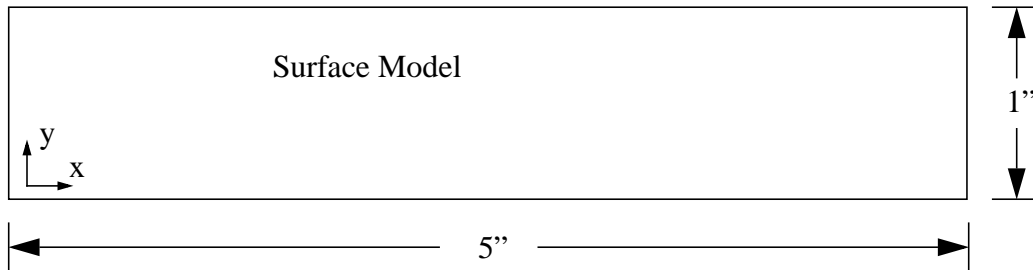
Objective:

- 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 12-1

Table 1: Temperature Dependent Material Properties

T (°F)	E ₁₁ (Msi)	E ₂₂ (Msi)	ν ₁₂	G ₁₂ (Msi)	G ₂₃ (Msi)	G ₁₃ (Msi)	α ₁₁ (Mils/ in/°F)	α ₂₂ (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 12-1.
- 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 Database...

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

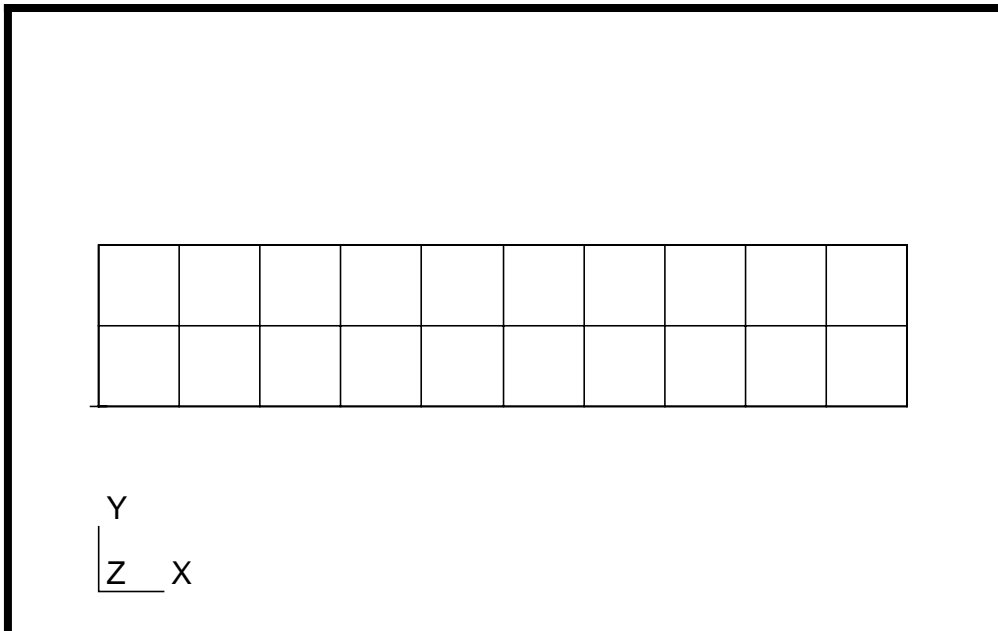
<i>Object:</i>	Surface
<i>Method:</i>	XYZ
<i>Vector Coordinate List</i>	<5, 1, 0>
Apply	

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

◆ **Finite Elements**

<i>Action:</i>	Create
<i>Object:</i>	Mesh
<i>Type:</i>	Surface
<i>Global Edge Length</i>	0.5
<i>Element Topology</i>	QUAD 4
<i>Surface List</i>	Surface 1
Apply	

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 12-3)

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:

Object:

Method:

Material Name

Constitutive Model

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

Elastic Modulus 22

Poisson's Ratio

Shear Modulus 12

Shear Modulus 23

Shear Modulus 13

Thermal Expan. Coeff 11

Thermal Expan. Coeff 22

**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**

Action:

Create

Dimension:

2D

Type:

Shell

Property Set Name

prop_1

Options:

Homogeneous

Standard Formulation

Input Properties...

Material Name

m:mat_orth2d

Thickness

0.020

OK

Select Members

Select All Finite Elements

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



Add

Apply

File/Quit