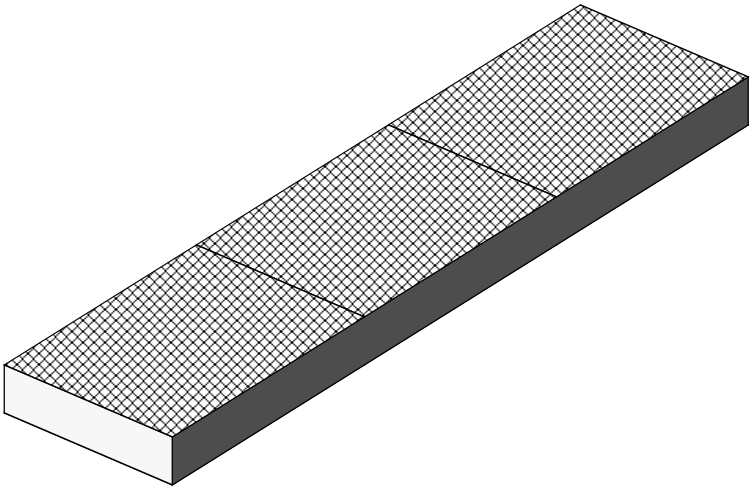
WORKSHOP 12

Material Property Definition

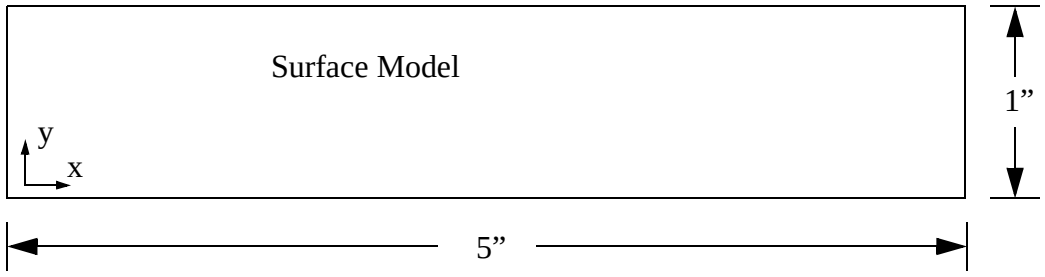


Objective:

- 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...

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:	**Surface**
Method:	**XYZ**
Vector Coordinate List	**<5, 1, 0>**

Apply

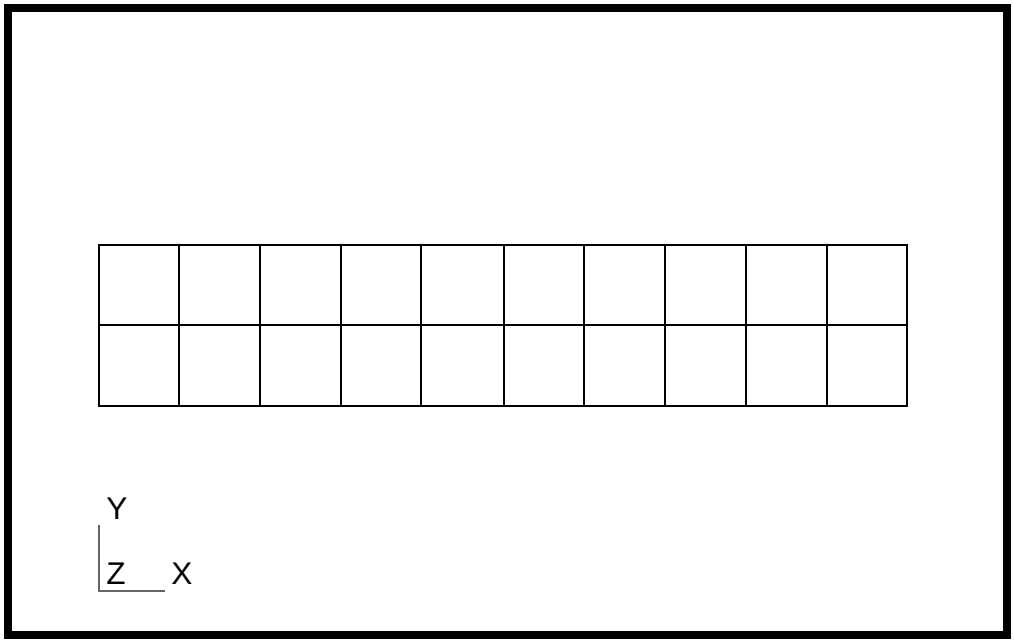
To create the finite element model, click on the **Finite Elements** radio button in the *Main Form*.

Finite Elements

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length	**0.5**
Element Topology	**Quad4**
Surface List	**Surface 1**

Apply

Your model should look like the one shown below.



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**

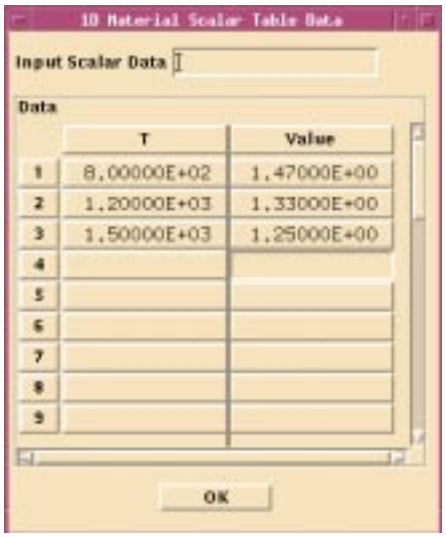
Using the data listed in Table 1 of this exercise, define the field for E11.

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.



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:	**Create**
Object:	**2D Orthotropic**
Method:	**Manual Input**
Material Name	**mat_orth2d**

Input Properties...

Constitutive Model	**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	**E11**
Elastic Modulus 22	**E22**
Poisson's Ratio	**0.32**
Shear Modulus 12	**G12**
Shear Modulus 23	**G23**
Shear Modulus 13	**G13**
Thermal Expan. Coeff 11	**ALPHA11**
Thermal Expan. Coeff 22	**ALPHA22**

Apply

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

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...