Engineering Data User's Guide

Ansys
ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2025 R1
January 2025

Published in the U.S.A.

Table of Contents

Overview

Engineering Data is a resource for material properties used in an analysis system. Engineering Data can be used as a repository for company or department data, such as material data libraries. The Engineering Data workspace is designed to allow you to create, save, and retrieve material models, as well as to create libraries of data that can be saved and used in subsequent projects and by other users.

The following topics cover the basics of the Engineering Data workspace:

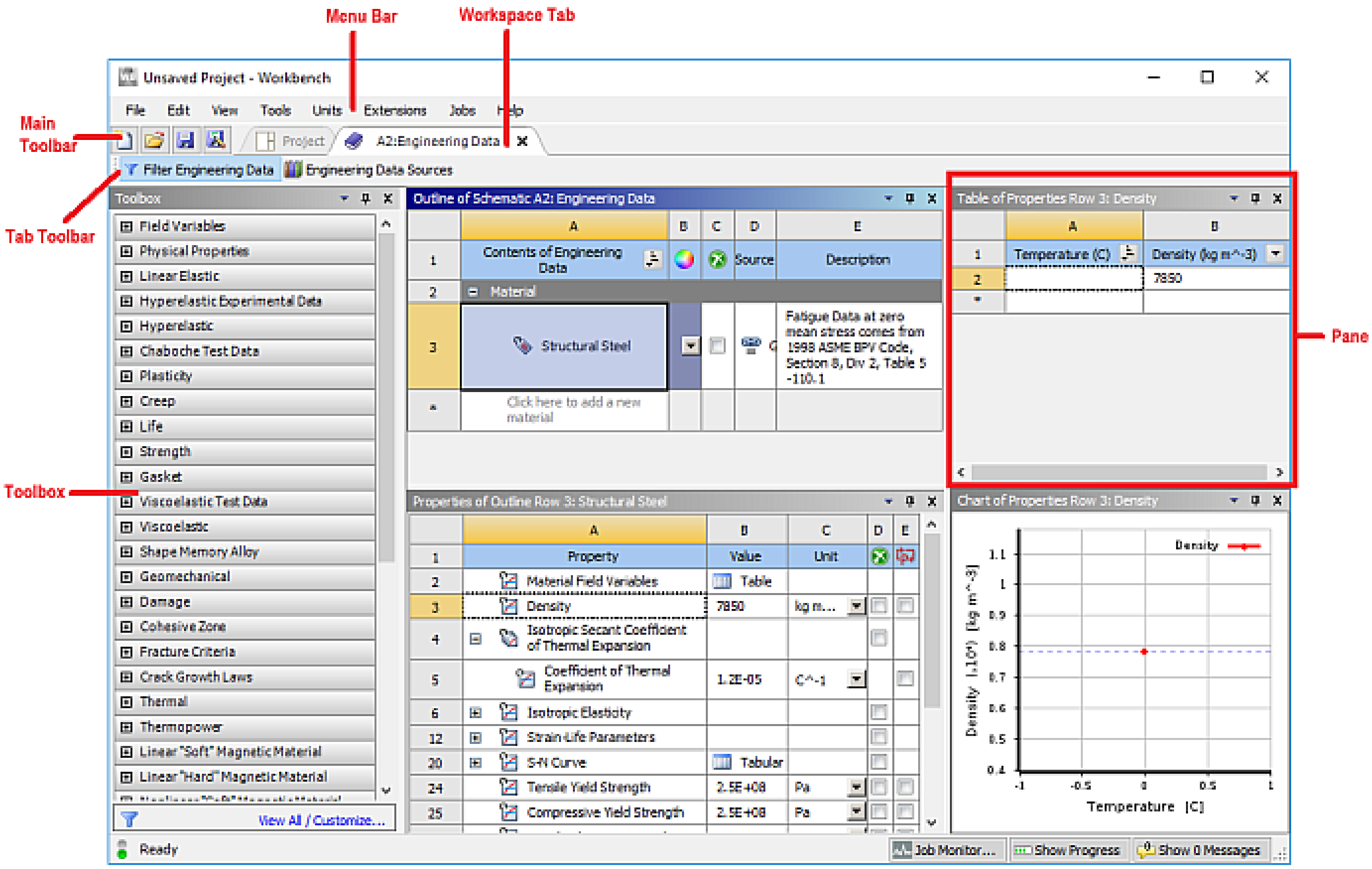
Definitions

The documentation makes use of the following terminology:

Term	Definition
Engineering Data	The cell of a system in the **Project Schematic** that contains engineering data. The default name is **Engineering Data**.
Property	The identifier for the singular information (for example, Density) that together with other properties defines or models the behavior of the material. A property is always defined by at least one table (tabular data). Some properties can contain a collection of tabular data (for example, Isotropic Elasticity).
Property data	The identifier for tabular data (for example, Young's Modulus).

User Interface

The Engineering Data workspace is an integrated into Ansys Workbench. Use the following image and table to understand the default interface layout and elements.



Name	Description
Menu Bar (p. 6)	Allows you to access multiple Workbench menus
Tab Toolbar (p. 7)	Toggles various Engineering Data functions.
Toolbox (p. 7)	Displays data items that can be included in Engineering Data.
Engineering Data Sources Pane (p. 8)	Displays the available data sources and their location, edit state and description.
Outline Pane (p. 8)	Displays the outline of the contents of the selected data source in the **Engineering Data Sources** pane or the contents of Engineering Data.
Properties Pane (p. 12)	Displays the properties of the selected item in the **Outline** pane.
Table Pane (p. 13)	Shows the tabular data for the selected item in the **Properties** pane.
Chart Pane (p. 14)	Displays the chart of the item selected in the **Properties** pane.

Menu Bar

The following items in the menu bar are provided by Engineering Data or affect Engineering Data.

For more information on other menu options, see Menu Bar in the *Workbench User's Guide*.

Menu	Selection(s)	Description
File	**Import Engineering Data**	Imports (p. 20) data into the selected data source.
	Export Engineering Data	Exports (p. 21) the selected data source or selected items to disk.
Edit	**Delete**	Deletes the selected item.
View	*Various Choices*	Adjust the Engineering Data workspace panes. For more details, see View Menu in the *Workbench User's Guide*.
Units	*Various Choices*	Changes the units you see while using Engineering Data. For more details, see Units Menu in the *Workbench User's Guide*.
Extensions	*Various Choices*	Manages extensions and displays extension log files while in Engineering Data. For more details, see Units Menu in the *Workbench User's Guide*.
Help	*Various Choices*	Opens Workbench help. For more details, see Using Help in the *Workbench User's Guide*.

Tab Toolbar

The Engineering Data Tab Toolbar provides the following buttons. Click the button once to turn the function on. Click it again to turn it off.

Button Name	Description	Default
Filter Engineering Data	Filters (p. 15) **Toolbox** content and property data based on the system(s) containing this Engineering Data.	On
Engineering Data Sources	Displays the **Engineering Data Sources** pane (p. 8).	Off

Toolbox

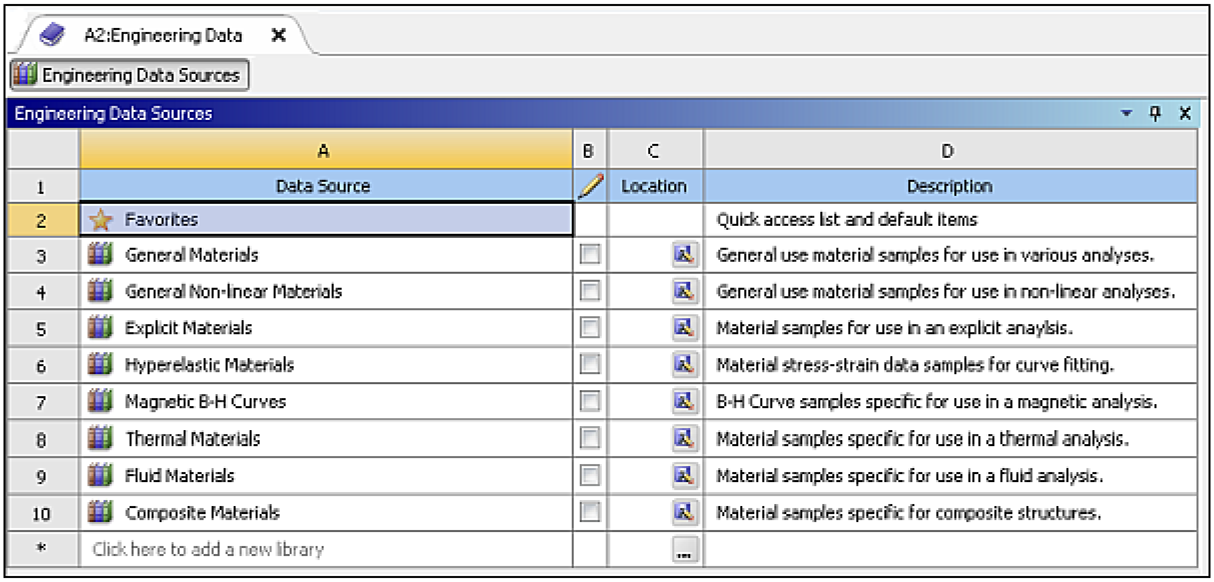
Engineering Data filters the **Toolbox** to those items which are applicable for the current selection (for example, **Structural Steel** as shown in the User Interface (p. 5) image.

The **Toolbox** presents items in the following categories:

- Material properties and models (p. 25)
- Additional tabular data that can be added to a property or model (p. 25)
- Curve fitting of data (p. 48)

Engineering Data Sources Pane

When you select the **Engineering Data Sources** button, as highlighted below, you display the available data sources that are included with the application.



	A	B	C	D
1	Data Source		Location	Description
2	Favorites			Quick access list and default items
3	General Materials			General use material samples for use in various analyses.
4	General Non-linear Materials			General use material samples for use in non-linear analyses.
5	Explicit Materials			Material samples for use in an explicit anaylsis.
6	Hyperelastic Materials			Material stress-strain data samples for curve fitting.
7	Magnetic B-H Curves			B-H Curve samples specific for use in a magnetic analysis.
8	Thermal Materials			Material samples specific for use in a thermal analysis.
9	Fluid Materials			Material samples specific for use in a fluid analysis.
10	Composite Materials			Material samples specific for composite structures.
*	Click here to add a new library		...	

The **Engineering Data Sources** pane allows you to select a data source to display in the **Outline** pane. This pane is used to manage the data sources that are available to you. It displays your libraries ( ) and favorites ( ). You can perform the following actions in this pane:

- Create a new library (p. 19)
- Import (p. 21) and export (p. 21) data sources
- Remove a data source from the list (p. 19)
- Edit a data source (p. 20)

To perform these tasks, select one or more data source check boxes in the **Edit Library** ( ) column. Once checked, the table rows change color and the various fields are editable.

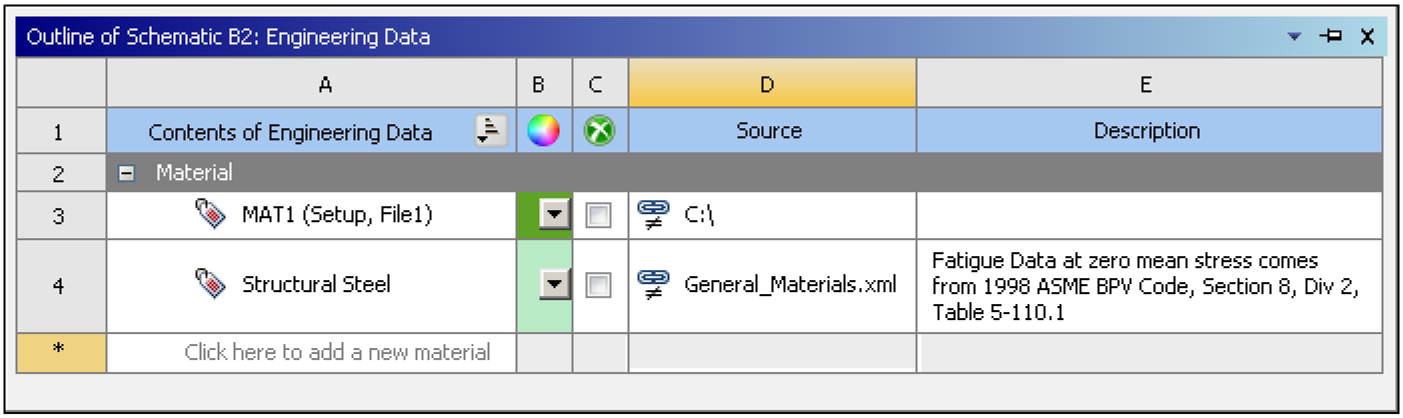
Outline Pane

The **Outline** pane displays an outline of the contents of the selected data source. You can perform the following actions in this pane:

- Create a new material (p. 25)

- Delete a material (p. 26)
- Rename a material
- Suppress a material (p. 47)
- Add a description for a material
- Add a material to the system from an external data source
- Select a default material for the solid and or fluid parts of a model (p. 28)

Use the following image and table to understand the interface elements.

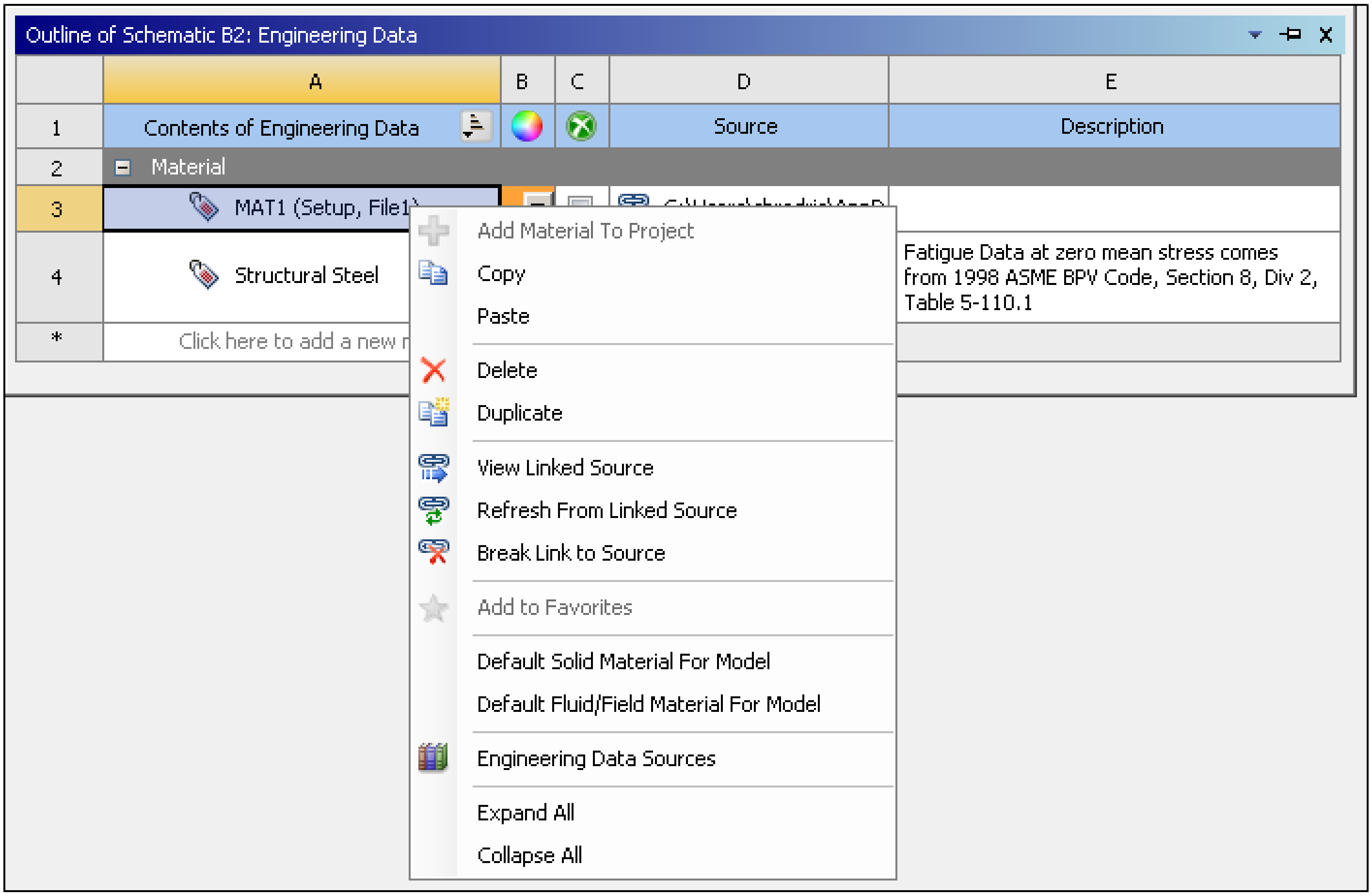


Column Name	Description
Con-tents	Displays the name of the items contained in the selected data source. The type and status of the item is indicated under the **Material** title by an icon to the left of the name. • : The data contained in this material is valid data. • : Some data contained in this material requires attention. For more information, see Validation (p. 47) and Filtering (p. 15).
Color ( )	Allows you to assign a specific color to your materials. This material-based color scheme is transferred to Mechanical. The application assigns random colors by default. You can change the default colors using the palette or you can make manual color value entries.
Suppres-sion ( )	Displays the suppression status of the item and may also be used to switch the status (see Suppression (p. 47)). This column is only displayed when the selected data source is Engineering Data.

Column Name	Description
Source	Displays the location of the material file and whether the data is linked to the file. You can hold the mouse cursor over the link icon to display a tool tip that includes the path to the linked source. Link status is indicated by the following icons. • : The data matches the data in the linked source. • : The data does not match the data in the linked location. • : The data could not be found in the linked source or the linked source is missing.
Descrip-tion	Displays the description for the item contained in the data source. Hold the mouse cursor over the description to display a tool tip with the complete description. If the column is too small, you may still see the contents without resizing.
Add	Adds an item from an external data source to Engineering Data for the system you are editing, and indicates if the item is included in Engineering Data. This column is only displayed when the selected data source is other than Engineering Data. Click **Add** ( ) to add the item to Engineering Data. When an item is included in Engineering Data it is indicated by the **Used in Engineering Data** icon ( ).
Default ( )	Displays items included by default in Engineering Data when it is created in a new system. This column is only displayed when the selected data source is **Favorites**.

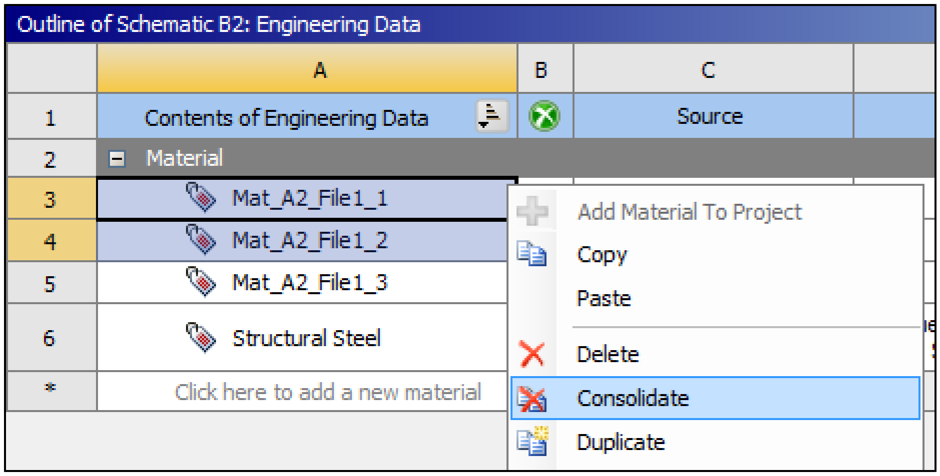
Context Menu Options

Right-clicking a Material file in the **Outline** pane displays a context menu.



In addition to the common menu options, the following selections perform specific actions.

Menu Item	Description
Duplicate	Creates a duplicate of the source file.
View Linked Source	Displays the **Engineering Data Sources** pane.
Refreshed From Linked Source	Refreshes the materials from the source file. This action discards any changes that you have performed.
Break Link to Source	Disconnects the link to your material source file.
Default Solid Material For Model	Automatically assigns the selected material to all solid bodies that do not have a specified material.
Consolidate	Merges all identical selected materials into a single material. Appears in the context menu when you select multiple materials *and* at least two of the selected materials contain the same data and come from the same source.

Menu Item	Description
	

Properties Pane

The **Properties** pane displays the properties for the item selected in the **Outline** pane. You can perform the following actions in this pane:

- Add additional properties, tabular data, or curve fitting (from the **Toolbox**)
- Delete a property (p. 26)
- Modify constant data (p. 26)
- Suppress a property (p. 47)
- Parametrize a property (p. 46)

Column Name	Description
Property	Lists the properties for the item selected in the **Outline** pane. Selecting a property will change the contents of the **Table** pane and **Chart** pane. The type and status of the item is indicated by an icon to the left of the name. • : The material property is described in a single property data (see the Material Definitions (p. 5) topic). • : Some data contained in this material property requires attention (see the Validation (p. 47) and Filtering (p. 15) topics). • : The material property is described in a collection of property data (see the Material Definitions (p. 5) topic). • : Indicates that the collection of property data requires attention (see the Validation (p. 47) and Filtering (p. 15) topics).

Column Name	Description
Value	Changes data for a property or indicates that the data for the property is tabular ( ). If the item selected in the **Outline** pane is not editable, this column is disabled.
Unit	Displays the unit of the data shown in the **Value** column. If the column is editable (see Units Menu), changing the unit converts the value into the selected unit (there is no net change in the data, so the solution is still valid).
Suppression ( )	Dhows the suppression status of the item and may also be used to switch the status (see Suppressing Material Properties (p. 47)).
Parameter ( )	Displays the parametrization status of the item and may also be used to switch the status (see Parametrizing Material Properties (p. 46)).

Table Pane

The **Table** pane displays the tabular data for the item selected in the **Properties** pane. If there are independent variables (for example, Temperature) for the selected item and the item is constant, you can change it to a table by entering a value into the independent variables data cell. If a row is shown with an index of *, you can add additional rows of data. You can sort the data by using the filter menu in the header of the column.

Tabular Data Filter

In Engineering Data, if a material property has more than one independent variable, one of those variables will be chosen by Engineering Data to be a primary independent variable. All independent variables excluding the primary independent variable are shown in the left pane and are used as a filter. The row chosen in the left pane filters the content in the right pane, as shown in the following image.

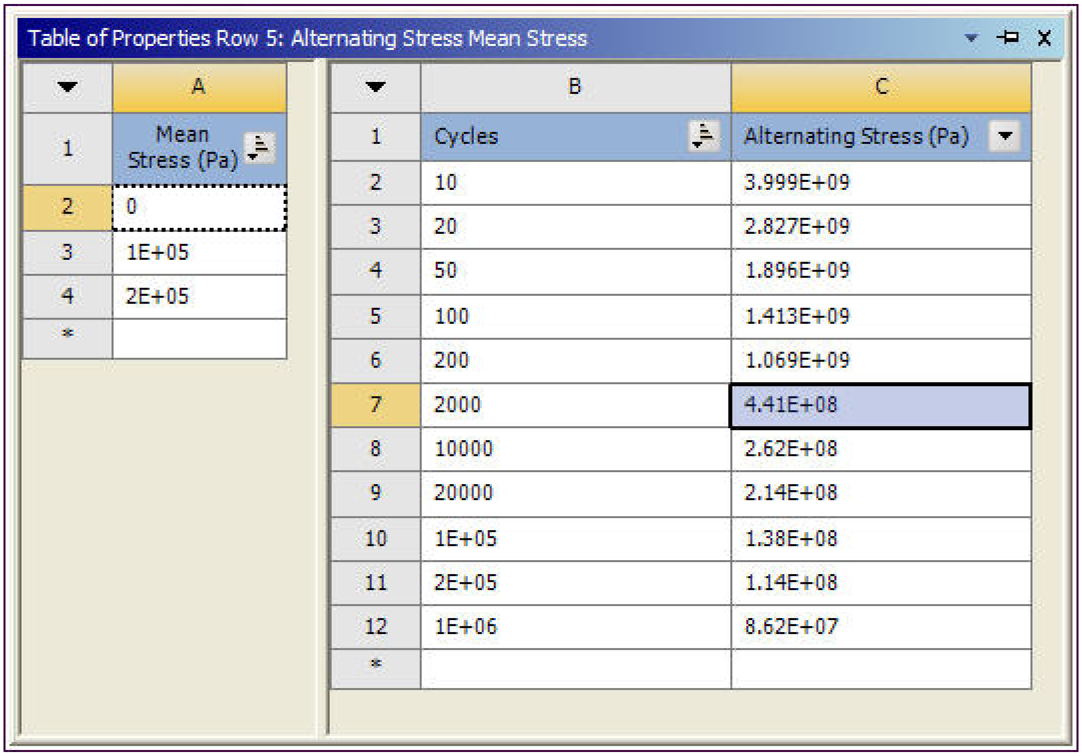


Table of Properties Row 5: Alternating Stress Mean Stress

	A
1	Mean Stress (Pa)
2	0
3	1E+05
4	2E+05
*	

	B	C
1	Cycles	Alternating Stress (Pa)
2	10	3.999E+09
3	20	2.827E+09
4	50	1.896E+09
5	100	1.413E+09
6	200	1.069E+09
7	2000	4.41E+08
8	10000	2.62E+08
9	20000	2.14E+08
10	1E+05	1.38E+08
11	2E+05	1.14E+08
12	1E+06	8.62E+07
*		

The data is separated in this way to make it easier to visualize and modify but the data is maintained in a single table by Engineering Data.

Chart Pane

The **Chart** pane shows the chart of the selected item in the **Properties** pane. If the content is tabular, the pane displays graphs of the data contained in the tables of the selected property.

Selecting a row in the left pane of a table displays a graph for the filtered data only.

Selecting an independent variable column heading displays a graph for each of the independent data. Selecting a dependent variable column heading displays the graph for the chosen data.

Engineering Data

Use the following topics to manage and use Engineering Data.

Accessing Engineering Data

Engineering Data is a resource for material properties used in an analysis system. Engineering Data can be used as a repository for company or department data, such as material data libraries. The Engineering Data workspace is designed to allow you to create, save, and retrieve material models, as well as to create libraries of data that can be saved and used in subsequent projects and by other users.

Engineering Data can be shown as a component system or as a cell in any Mechanical analysis system. As a standalone component system, the workspace accesses all material models and properties by default. When viewed as a cell in a Mechanical analysis system, the workspace shows the material models and properties pertinent to that system's physics.

To access Engineering Data:

1. Insert an Engineering Data component system or a Mechanical system into the **Project Schematic**.
2. Double-click the Engineering Data cell or right-click the cell and select **Edit** from the context menu.
3. The Engineering Data workspace appears. From here, you can navigate through the data for your analysis system, access external data sources, create new data, and store data for future use.

If you share an Engineering Data cell with one or more other analysis systems, be aware that changes in one system will change the data for all systems with which the data is shared.

The data contained in Engineering Data is automatically saved when the project is saved.

Filtering Data

Engineering Data filters the data pertaining to the project system being edited by default. The filtering is based on physics, analysis type, and solver. All data is transferred to the downstream cell regardless of filtering being turned on or off. A selected solver may ignore the data if it is not recognized or supported.

To turn data filtering on and off:

1. Access Engineering Data (p. 15).

2. On the tab toolbar, to turn data filtering off, click **Filter Engineering Data** ( ).

   To turn data filtering back on, click the same button again.

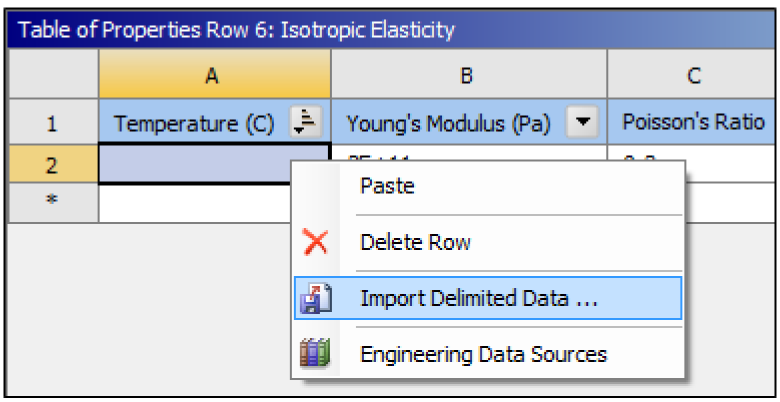
Importing Tabular Data

Use the **Table** pane to populate your tabular data by importing delimited text files.

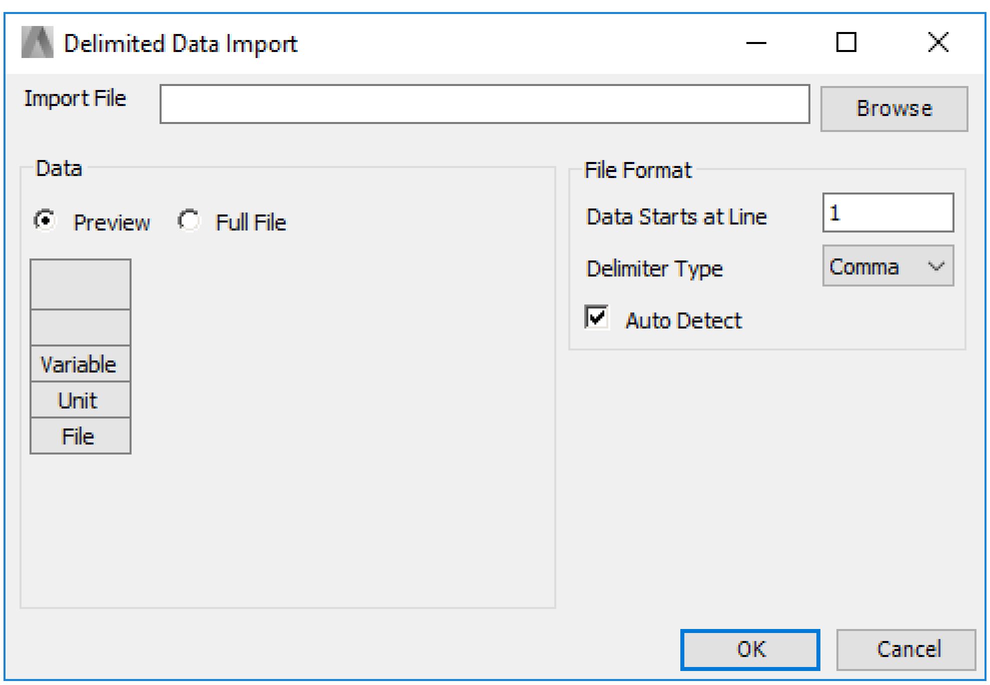
To import material data in tabular form:

1. Access Engineering Data (p. 15).

2. In the **Table** pane, right-click an empty row and select **Import Delimited Data** from the context menu.

   The **Delimited Data Import** dialog box opens, as shown in the following image.

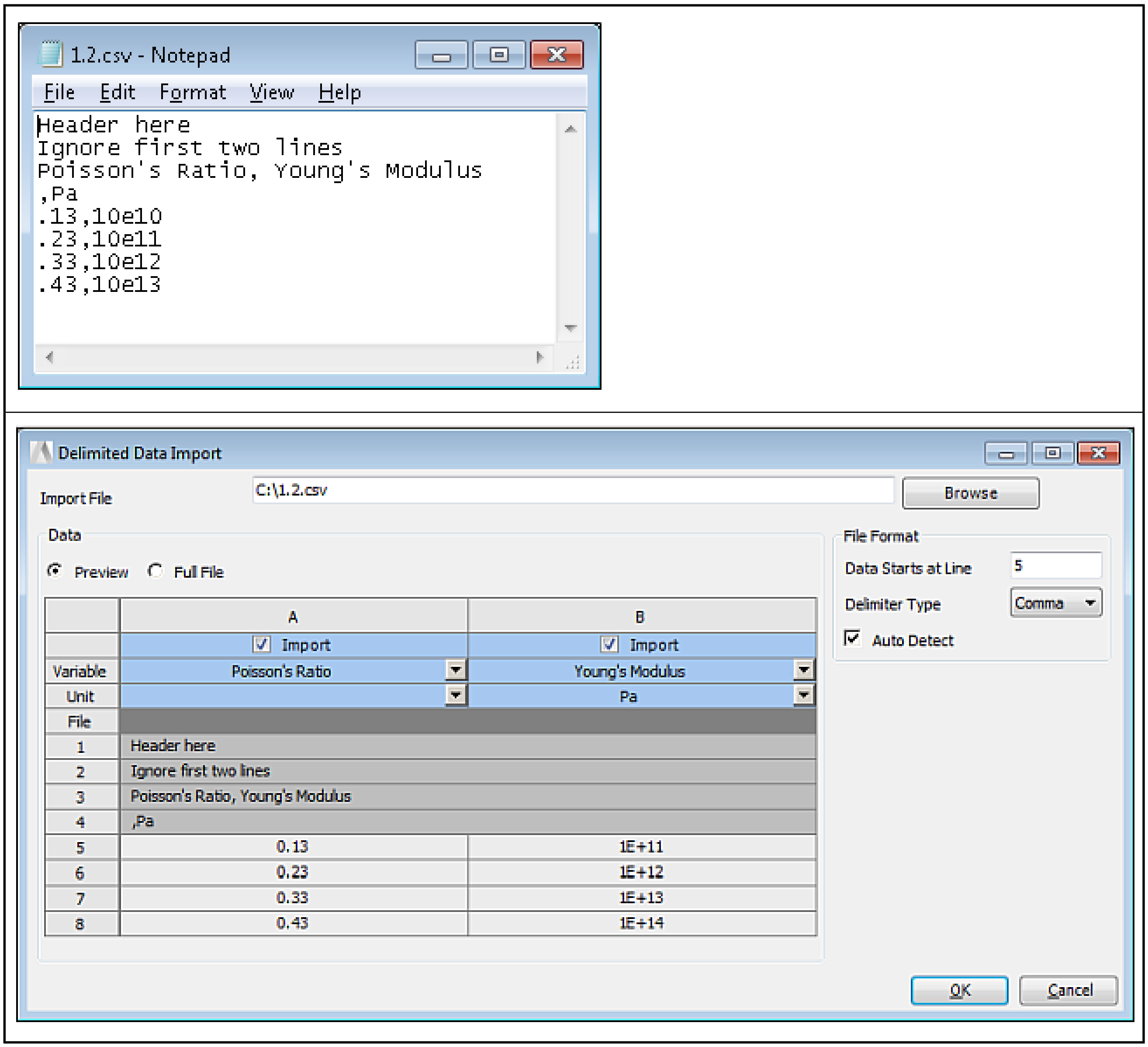
3. The **Auto Detect** feature examines the content of the file to find recognizable delimiters and numeric data and automatically changes the **File Format** settings to match the file formatting. During this process, the feature attempts to convert any string data preceding the numerical data as a variable type and its units.

   To turn this feature off, clear the **Auto Detect** check box.

4. To select a file to import, click **Browse**.

5. Browse to the location of the file you want to import, select it, and click **Open**.

   The data from the file is loaded into the dialog box.



6. To select a specific line in the file to begin the import from, type the number in the **Data Starts at Line** field.

7. Select one of the following options from the **Delimiter Type** drop-down menu:

- **Comma** (default)
- **Semicolon**
- **Space**
- **Tab**

8. To display the header information as well as the first 50 lines of your file, select **Preview**. To display the entire content of the file, select **Full File**.
9. To include or exclude a column of the file from import, select or clear the **Import** check box under the column name.
10. To select or change the information displayed in the **Variable** field, select an option from the drop-down list of available variables provided by the property shown in the **Table** pane.

    These values are read-in by the tool.
11. To select or change the information displayed in the **Unit** field, select an option from the drop-down list of units based on the specified variable.

    For variables that do not require units, this field is left blank.

Note:

- The import operation always appends data in the **Table** pane. Data is never overwritten.
- You can drag and drop your file onto a cell of the **Table** pane to automatically launch the **Delimited Data Import** dialog box.

Working with Data Sources

A data source contains engineering data information. There are three types of data sources used in the Engineering Data workspace: Engineering Data, libraries, and favorites.

Engineering Data

Engineering Data is the source of the material information that is used for the analysis of the system it is contained in. The information in an Engineering Data component system is used if shared to an analysis system. Engineering Data allows you to view, edit, and add data for use in your analysis system.

Library

A library is the term used for a collection of engineering data. You can add items from the library to Engineering Data for use in your analysis system.

Favorites

The favorites (⭐) data source is the location for those items that you use frequently and allows you to mark items as defaults for a new analysis system. You can add items from other data sources to the favorites using the context menu (right-click). You can add items from the favorites to Engineering Data for use in your analysis system. Material defaults for newly created systems may be assigned in the context menu (see Material Defaults (p. 28)).

Opening the Engineering Data Sources Pane

The **Engineering Data Sources** pane displays the available data sources that are included with the application.

To open the **Engineering Data Sources** pane:

1. Access Engineering Data (p. 15).
2. On the tab toolbar, click **Engineering Data Sources**, or right-click the workspace and select **Engineering Data Sources** from the context menu.

Creating a New Library

You can create a new collection of engineering data by adding a new library to the **Engineering Data Sources** pane.

To create a new library:

1. Open the **Engineering Data Sources** pane. (p. 19)
2. In the last row of the **Engineering Data Sources** table, click the empty cell in the **Data Source** column.
3. Type a name for the library and press **Enter**.
4. In the **Save as** dialog box, browse to the location where you want to save the data source.
5. Type a name for the file.
6. Click **Save**.

Removing a Data Source

You can remove a data source from the **Engineering Data Sources** pane.

To remove a data source:

1. Open the **Engineering Data Sources** pane. (p. 19)
2. Right-click a data source and select **Remove from List** from the context menu.

Editing a Data Source

You can change the name of the data source and the description.

To edit a data source:

1. Open the **Engineering Data Sources** pane. (p. 19)
2. Select the **Edit Library** ( ) check box to the right of the data source you want to edit.

   Once checked, the table rows change color and all of the various fields are editable.
3. Edit the text in the **Data Source** and **Description** columns, as required.
4. To save the changes, click **Save** ( ) in the **Location** column.

Importing Data into a Data Source

When you use the **Import Engineering Data** menu item, the data contained in that source is added to the currently selected data source (if edit enabled).

The following types of files are supported for import:

- Engineering Data libraries exported from Workbench 9.0 to 11.0 SP1
- Material(s) file following the MatML 3.1 schema
- Material(s) file generated by AUTODYN

Important:

Exported material data libraries (.xml) are not backwards compatible in previous versions of the application.

To import data into a data source:

1. Open the **Engineering Data Sources** pane. (p. 19)
2. In the **Engineering Data Sources** pane, select a data source from the **Data Source** column.
3. To the right of the data source file, select the **Edit Library** ( ) check box.
4. From the menu bar, select **File** → **Import Engineering Data**.
5. Select a file and click **Open**.

   Note:

   Only recognized data is imported into the data source.

Importing Data as a Data Source

When you use the **Open File** dialog box in the **Engineering Data Sources** pane, the selected data source is added to the list of data sources as a library.

The following types of files are supported for import:

- Engineering Data libraries exported from Workbench 9.0 to 11.0 SP1
- Material(s) file following the MatML 3.1 schema
- Material(s) file generated by AUTODYN
- **Important:**

  Exported material data libraries (.xml) are not backwards compatible in previous versions of the application.

To import data as a data source:

1. Open the **Engineering Data Sources** pane. (p. 19)
2. In the last row of the **Engineering Data Sources** table, click ....
3. Select a file and click **Open**.

   Note:

   Only recognized data is imported.

Exporting a Data Source

You can export selected items in a data source. The MatML 3.1 schema for Material(s) format is supported for export.

To export a data source:

1. Open the **Engineering Data Sources** pane. (p. 19)
2. In the **Engineering Data Sources** pane, select a data source from the **Data Source** column.
3. From the menu bar, select **File** → **Export Engineering Data**.
4. In the **Save as** dialog box, browse to the location where you want to save the data source.
5. Type a name for the file.
6. Click **Save**.

Exporting Individual Data

You can export selected items in a data source. The MatML 3.1 schema for Material(s) format is supported for export.

To export individual data:

1. Open the **Engineering Data Sources** pane. (p. 19)
2. In the **Engineering Data Sources** pane, select a data source from the **Data Source** column.
3. In the **Outline** pane, select one or more items in the **Property** column.
4. From the menu bar, select **File** → **Export Engineering Data**.
5. In the **Save as** dialog box, browse to the location where you want to save the data source.
6. Type a name for the file.
7. Click **Save**.

Combining Data Sources

You can combine multiple data sources into one.

To combine data sources:

Open the **Engineering Data Sources** pane. (p. 19)

1. Import data as a data source (p. 21) for each of the data sources you want to combine.
2. Edit the data source (p. 20) to combine it into an existing data source, or create a library (p. 19) to combine the data sources into a new library.
3. Select a data source.
4. In the **Outline** pane, drag a material into the **Engineering Data Sources** pane and drop the item on the data source you want to combine.
5. To save the changes, click **Save** ( ) in the **Engineering Data Sources** pane **Location** column.

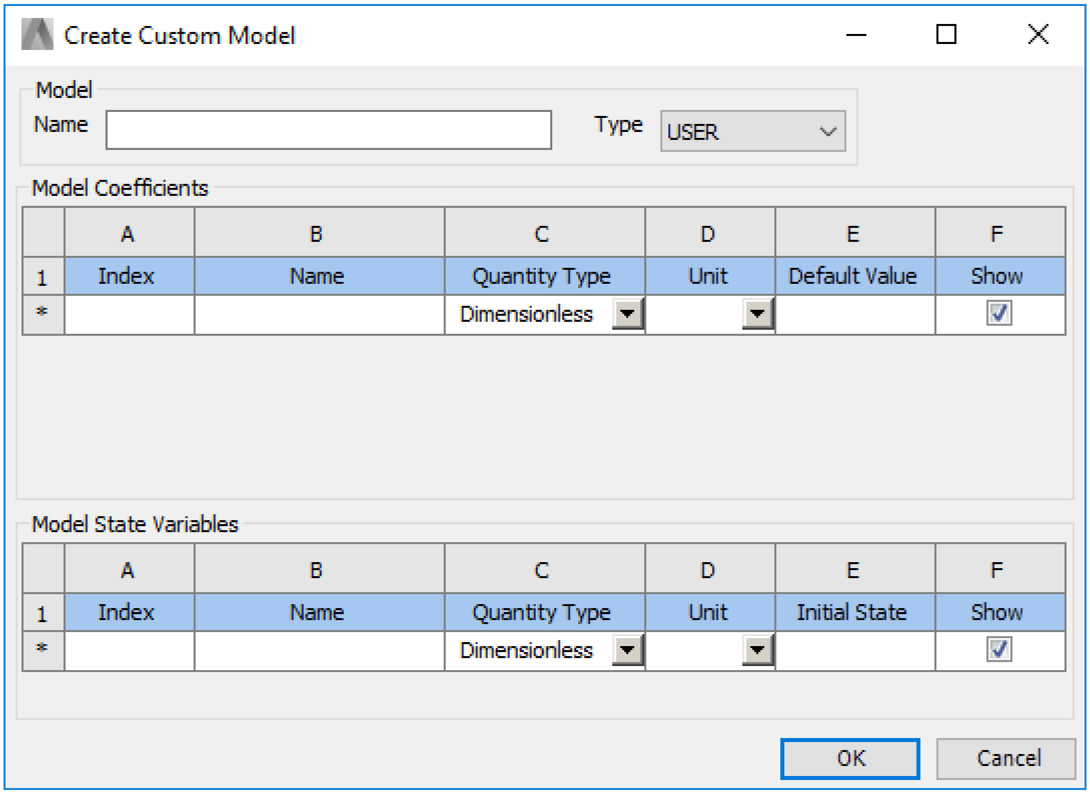
Creating Custom Material Models

The Engineering Data workspace allows you to create custom material models. This advanced option works in conjunction with a Material User Programmable Feature (UPF) of the Mechanical APDL programming interface. You must have the prerequisite expertise in material constitutive modeling and software programming.

To create a custom material model:

1. From the **Custom Material Models** category of the **Toolbox**, double-click **Create Custom Model**.

   The **Create Custom Model** dialog box displays.



2. In the dialog box, enter the following:

- **Name**: User-defined name for the model.
- **Type**: UPF type.
- **Model Coefficients**: User variable input for the UPF subroutine.
- **Model State Variables**: State variable input for the UPF subroutine.

  For each **Model Coefficient** and **State Variable** provide the following:

  – **Index**: Defines the argument index of the variable in the UPF subroutine.

  – **Name**: Defines the name of the variable.

  – **Quantity Type**: Defines the quantity type of the variable.

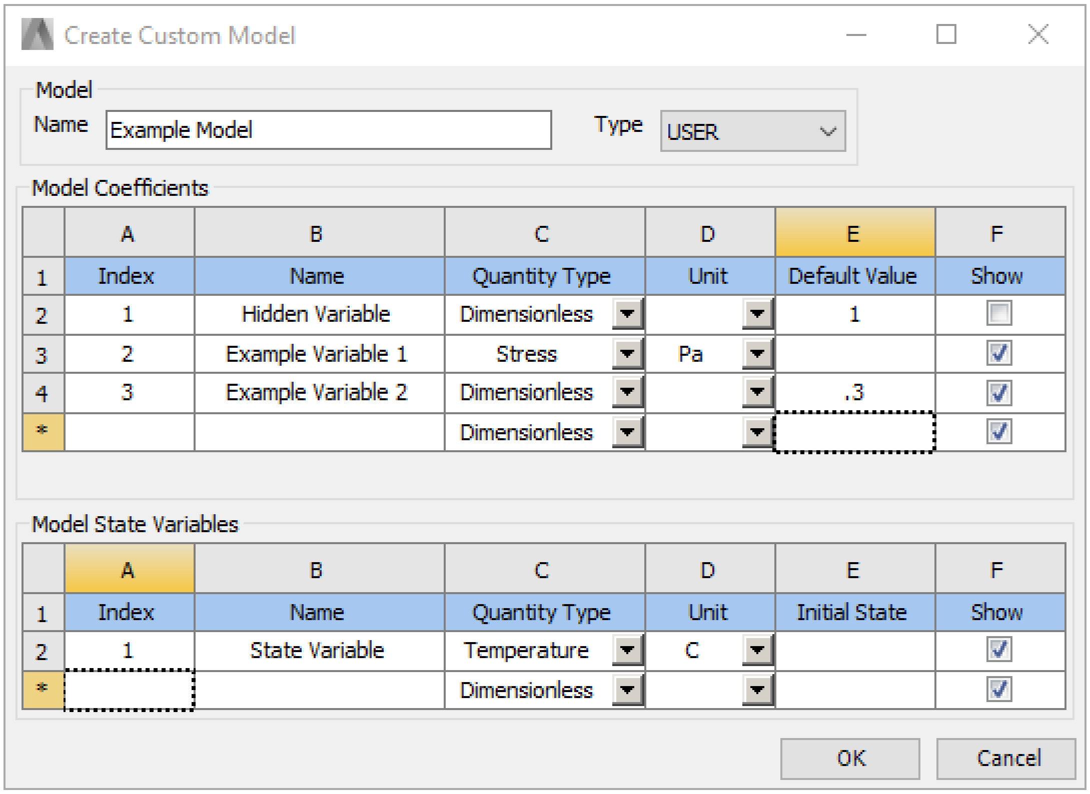
  – **Unit**: Defines the variable's units.

  – **Default Value**: Defines the default value of the variable.

  – **Initial State**: Defines the initial value of the variable.

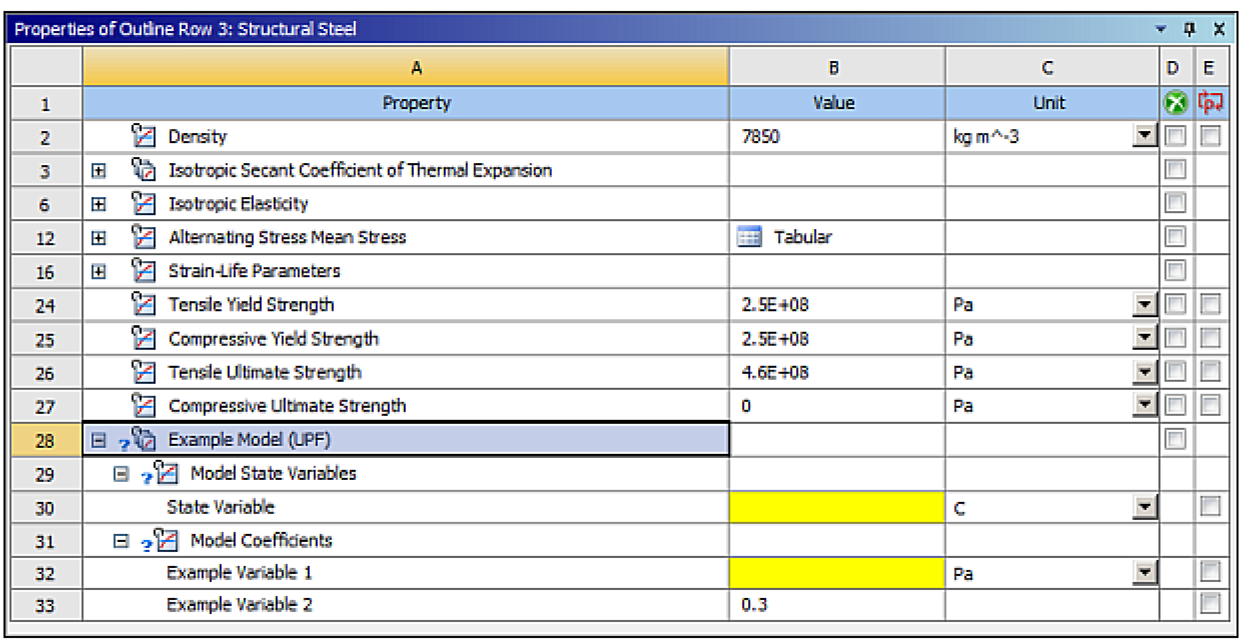
  – **Show**: Determines if the variable is shown in Engineering Data.

The following image displays an example model.



3. Once the model is defined, to add the new model to the **Toolbox** category and the selected material, click **OK**.

   An example of the new model added to Structural Steel is shown in the following image.



Material Data

Use the following topics to understand the function and use of material properties within Engineering Data.

Creating a New Material

You can add a new material to the **Outline** pane.

To create a new material:

1. Access Engineering Data (p. 15).

2. To add a material to libraries or favorites, open the **Engineering Data Sources** pane (p. 19) and mark the data source as editable (p. 20).

3. In the **Outline** pane, click in the cell labeled as **Click here to add a new material**.

4. Type the name of the new material and press **Enter**.

Adding Material Properties

You can add material properties from the **Toolbox** to an existing material.

To add a material property:

1. Access Engineering Data (p. 15).
2. To add a material property to libraries or favorites, open the **Engineering Data Sources** pane (p. 19) and mark the data source as editable (p. 20).
3. In the **Outline** pane, select a material.
4. In the **Toolbox**, double-click the property you want to add.

Deleting Material Properties

You can remove material properties from a material if required.

To delete a material property:

1. Access Engineering Data (p. 15).
2. To remove a material property from libraries or favorites, open the **Engineering Data Sources** pane (p. 19) and mark the data source as editable (p. 20).
3. In the **Outline** pane, select a material.
4. In the **Properties** pane, select the material property to delete.
5. From the menu bar, select **Edit** → **Delete** or right-click the property and select **Delete** from the context menu.

Modifying Material Properties

You can modify both constant and tabular data.

Constant Data

You modify constant data by changing the value and/or unit of that data in the **Properties** pane. The value and unit together constitute one integral piece of information, or *datum*. The value is modified by selecting the cell in the **Value** column and typing in the new value. If available, modifying the unit converts the value to correspond to the new unit (see Units Menu). If the value entered is not in the acceptable range, it is indicated in yellow and causes the state of the Engineering Data cell to change to Attention Required.

Tabular Data

If the data is in a tabular format, it is indicated in the **Value** column ( ). This data is modified in the **Table** pane and each datum is a value and unit as one integral piece. The value is modified by selecting the cell in the variable column you want to change. If the value entered is not in the acceptable range, it is indicated in yellow and causes the state of the Engineering Data cell to change to Attention Required. The unit is shown in the header, and if available, modifying the unit modifies each datum for that variable to have the same unit (see Units Menu).

To modify material properties:

1. Access Engineering Data (p. 15).
2. To modify a material property in libraries or favorites, open the **Engineering Data Sources** pane (p. 19) and mark the data source as editable (p. 20).
3. In the **Outline** pane, select a material that contains the property to modify.
4. Perform one of the following:
   - For constant data, change the value or unit in the **Properties** pane.
   - For tabular data, change the value or unit(s) in the **Table** pane.

Sample Libraries

Engineering Data provides sample material data categorized into several libraries. You must validate that the data is consistent with the material you are using in your analysis. If you are viewing the library in an analysis system, the contents of a material are filtered to that system (see Filtering Data (p. 15)).

The following libraries are included:

GRANTA Materials Data for Simulation,

This is a large library of hundreds of materials records encompassing many materials types and multiple types of constitutive/mathematical models. Use the new Mechanical materials interface to search and browse the available materials, view their properties, and assign the materials directly within your project.

General Materials,

General use materials and consists mostly of metals that can be used in various analyses.

Additive Manufacturing Materials,

Additive manufacturing material samples for use in additive manufacturing analyses.

General Nonlinear Materials,

General use nonlinear materials for performing nonlinear analyses.

Explicit Materials,

Materials containing data, specific for use in an explicit dynamics analysis.

Hyperelastic Materials,

Materials containing stress strain data which can be used to experiment with curve fitting (see the Curve Fitting (p. 48) topic). The data does not correspond to any particular material.

Magnetic B-H Curves,

Materials containing B-H Curve data, specific for use in a magnetic analysis.

Thermal Materials,

Materials containing data, specific for use in a thermal analysis.

Fluid Materials,

Materials containing data, specific for use in a fluid analysis.

Composite Materials,

Materials containing data, specific for use in a composite analysis.

GeoMechanical Materials,

Materials for use with geomechanical models.

Selecting the Default Material Assignment for Model Parts

Engineering Data is configured at installation with the default material assignments of Structural Steel for solid parts and Air for fluid parts. You can change these defaults or remove the default for an individual analysis system or for each newly created analysis system. If the Engineering Data cell is shared and the default is changed, the first Model cell Edit action uses the current default.

To select the default material:

1. In the **Outline** pane, select the material to be used as the default for the **Model** cell.
2. Right-click the material and select either **Default Solid Material for Model** or **Default Fluid/Field Material for Model**.

You can also choose to not have a default material by selecting the current default and removing the selection in the context menu. Including and setting that material as the default for the model (or not setting the default) affects all subsequent systems created in the project.

You can replace a default material assignment in the model by deleting the default material and then refreshing the **Model** cell, which assigns the active default material.

The default material assignment is disabled when an initial connection is made to transfer material data from another system or when the choice is made to import material data with geometry. This behavior is controlled by the **Unset the default material when materials are imported with geometry** Engineering Data user preference. If the option is disabled, the default material assignment occurs if a default is selected.

Supported Properties

The supported material properties are defined by the analysis systems that contain or share the Engineering Data cell. If filtering is active (see Filtering Data (p. 15)), only the supported material properties are displayed. See the list of material properties supported by a given analysis system in the following sections. The minimum material properties required for a given analysis are marked with an asterisk (*). Select the link to read more information about a particular property.

Also note that material property listings in the following sections are appended when field variable support is available. Field variable dependency types include :

- f(T): Temperature
- f(C): Coordinate
- f(F): Frequency
- f(D): Displacement
- f(H): Time History
- f(U): User-defined field variable dependency.
- f(R): Stress ratio

Note that materials can support multiple field variables, for example, Hill Yield, f(T, C, D, H, U).

Note that field variable dependent data cannot be used in the solution process if it is out of scope of the analysis type. For example, if a frequency-dependent density property is defined under Static Structural Analysis, the corresponding Mechanical APDL command is generated by calculating the density at the frequencies default value (defined on the material).

Eigenvalue Buckling

Properties
*Density, f(T,F,C,U) *Isotropic Elasticity, f(T,F,U) Orthotropic Elasticity, f(T,F,U) See Linear Material Properties in the *Material Reference* for a description of the above properties. Anisotropic Elasticity (p. 59) Orthotropic Stress Limits, f(T,F,U) Orthotropic Strain Limits, f(T,F,U) Tsai-Wu Constants, f(T) Puck Constants, f(T,F,U) LaRc03/04 Constants, f(T) Superelasticity, f(T) Shape Memory Alloy, f(T) The above shape memory alloy material model properties are supported only for pre-stress analysis. Gasket Material Model (p. 62) is supported for pre-stress analysis.

Electric

Properties
Electric Anisotropic Relative Permeability *Isotropic Resistivity Orthotropic Resistivity **Piezoelectric** Piezoelectric Matrix

Explicit Dynamics

Properties
Density, f(T,F) Isotropic Elasticity, f(T,F,U) Orthotropic Elasticity, f(T,F,U)

Properties
Viscoelastic
Bilinear Isotropic Hardening, f(T)
Multilinear Isotropic Hardening, f(T)
Bilinear Kinematic Hardening, f(T)
Multilinear Kinematic Hardening, f(T)
Johnson Cook Strength
Cowper Symonds Strength
Steinberg Guinan Strength
Zerilli Armstrong Strength
Neo-Hookean, f(T)
Mooney-Rivlin (2, 3, 5, and 9 Parameter), f(T,F,U)
Polynomial (1st, 2nd, and 3rd Order), f(T)
Yeoh (1st, 2nd, and 3rd Order), f(T)
Ogden (1st, 2nd, and 3rd Order), f(T)
Specific Heat, C_p f(T,U)
Drucker-Prager Strength Linear
Drucker-Prager Strength Stassi
Drucker-Prager Strength Piecewise
Johnson-Holmquist Strength Continuous
Johnson-Holmquist Strength Segmented
RHT Concrete Strength
MO Granular
Bulk Modulus
Shear Modulus
Polynomial EOS
Shock EOS Linear
Shock EOS Bilinear
Crushable Foam
Compaction EOS Linear
Compaction EOS Non-Linear
P-alpha EOS
Plastic Strain Failure
Principal Stress Failure
Principal Strain Failure
Stochastic Failure
Tensile Pressure Failure
Crack Softening Failure

Properties
Johnson Cook Failure Grady Spall Failure Explosive JWL Explosive JWL Miller Ideal Gas EOS Erode on Geometric Strain Erode on Plastic Strain Erode on Failure Erode on Timestep See Material Models Used in Explicit Dynamics Analysis for a description of the above properties. Orthotropic Stress Limits Orthotropic Strain Limits Tsai-Wu Constants Tsai-Wu Constants must be used in conjunction with Orthotropic Stress Limit. Tsai-Wu Constants used in conjunction with Orthotropic Strain Limit are not supported.

Harmonic Acoustics

The following material properties are supported for Harmonic Acoustics analyses.

Properties
*Density, f(T,F) *Speed of Sound, f(T,F) Viscosity, f(T,F) Bulk Viscosity, f(T,F) Isotropic Thermal Conductivity, f(T,F) Orthotropic Thermal Conductivity, f(T,F) Specific Heat, C_p, f(T,F) See the Linear Material Properties section of the Mechanical APDL Material Reference for a description of the above properties. **Perforated Media** The modeling of a perforated media is supported by the use of one of the following models: • Johnson-Champoux-Allard Equivalent Fluid

Properties
• Delany-Bazley Equivalent Fluid • Miki Equivalent Fluid • Complex Impedance and Propagating-Constant Equivalent Fluid • Complex Density and Velocity Equivalent Fluid See the Equivalent Fluid Model of Perforated Material section of the Mechanical APDL Acoustic Analysis Guide for a more detailed description of the above properties.

Harmonic Response

Properties
*Density, f(T,F,C,U) Damping Factor (α) Damping Factor (β) Material Dependent Damping (p. 60) *Isotropic Elasticity, f(T,F,U) Orthotropic Elasticity, f(T,F,U) Anisotropic Elasticity See Linear Material Properties in the *Material Reference* for a description of the above properties. Anisotropic Elasticity (p. 59) Hyperelastic Material Models (p. 62) are supported for pre-stress modal based analysis but not for pure modal based analysis.

Magnetostatic

Properties
*Relative Permeability Coercive Force & Residual Induction B-H Curve Demagnetization B-H Curve Relative Permeability (Orthotropic) *Isotropic Resistivity, f(T) Orthotropic Resistivity See Electromagnetic Material Properties (p. 66) for a description of the above properties.

Modal

Properties
*Density, f(T,F,C,U) Damping Factor (α) Damping Factor (β) Material Dependent Damping (p. 60) Coefficient of Thermal Expansion f(T) (p. 59) *Isotropic Elasticity, f(T,F,U) Orthotropic Elasticity, f(T,F,U) See Linear Material Properties in the *Material Reference* for a description of the above properties. Anisotropic Elasticity (p. 59) Bilinear Isotropic Hardening, f(T) Bilinear Kinematic Hardening, f(T) Multilinear Isotropic Hardening, f(T) Multilinear Kinematic Hardening, f(T) Chaboche Kinematic Hardening, f(T) Anand Viscoplasticity, f(T) Orthotropic Stress Limits, f(T,F,U) Orthotropic Strain Limits, f(T,F,U) Tsai-Wu Constants, f(T) Puck Constants, f(T,F,U) LaRc03/04 Constants, f(T) See Material Strength Limits for a description of the above properties. Superelasticity, f(T) Shape Memory Alloy, f(T) See Shape Memory Alloy (SMA) for a description of the above properties. The above shape memory alloy material model properties are supported for pre-stress modal analysis but not for pure modal analysis. Gasket Material Model (p. 62) is supported for pre-stress modal analysis but not for pure modal analysis. Hyperelastic Material Models (p. 62) are supported for pre-stress modal analysis but not for pure modal analysis.

Modal Acoustics

The following material properties are supported for Modal Acoustics analyses.

Properties
*Density, f(T,F) *Speed of Sound, f(T,F) Viscosity, f(T,F) Bulk Viscosity, f(T,F) Isotropic Thermal Conductivity, f(T,F) Orthotropic Thermal Conductivity, f(T,F) Specific Heat, C_v f(T,F) See the Linear Material Properties section of the Mechanical APDL Material Reference for a description of the above properties.

Modal (Samcef)

The following material properties are supported for Modal analyses using the SAMCEF solver.

Important:

The material properties and models that you use to create SAMCEF solver input must have a temperature dependency. If not, the application does not write the input file. If you have imported materials that do not have an associated temperature dependency, you must add the **Temperature** field in the Engineering Data workspace as a field variable (p. 51) and enter a value (such as `0.0`) to enable the application to write an input file.

Properties
Density, f(T,F) Isotropic Secant Coefficient of Thermal Expansion, f(T, F, C, U) Orthotropic Secant Coefficient of Thermal Expansion, f(T, F, C, U) *Isotropic Elasticity, f(T,F,U) Orthotropic Elasticity, f(T,F,U) Bilinear Isotropic Hardening, f(T) Bilinear Kinematic Hardening, f(T) Multilinear Isotropic Hardening, f(T) Multilinear Kinematic Hardening, f(T) Formulation:

Properties
Cauchy BIOT Kirchof Chaboche Kinematic Hardening, f(T) Nonlinear Elastic Model with Damage Plakin Special Hardening Law, f(T) Plastic Strain Failure Temperature Dependent, f(T)
Refer to the Samcef documentation for a description of the above properties.

Random Vibration & Response Spectrum

Properties
*Density, f(T,F,C,U) Isotropic Secant Coefficient of Thermal Expansion, f(T, F, C, U) Orthotropic Secant Coefficient of Thermal Expansion, f(T, F, C, U) Isotropic Instantaneous Coefficient of Thermal Expansion, f(T) Orthotropic Instantaneous Coefficient of Thermal Expansion, f(T) Damping Factor (α) Damping Factor (β) Material Dependent Damping (p. 60) Coefficient of Thermal Expansion (p. 59) *Isotropic Elasticity, f(T,F,U) Orthotropic Elasticity, f(T,F,U)
See Linear Material Properties in the *Material Reference* for a description of the above properties.
Anisotropic Elasticity (p. 59) Bilinear Isotropic Hardening , f(T) Bilinear Kinematic Hardening, f(T Multilinear Isotropic Hardening, f(T) Multilinear Kinematic Hardening, f(T) Chaboche Kinematic Hardening, f(T) Anand Viscoplasticity f(T) Orthotropic Stress Limits, f(T,F,U) Orthotropic Strain Limits, f(T,F,U) Tsai-Wu Constants, f(T)

Properties
Puck Constants, f(T,F,U) LaRc03/04 Constants, f(T) The above shape memory alloy material model properties are supported for pre-stress modal based analysis but not for pure modal based analysis. Gasket Material Model (p. 62) is supported for pre-stress modal based analysis but not for pure modal based analysis. Hyperelastic Material Models (p. 62) are supported for pre-stress modal based analysis but not for pure modal based analysis.

Rigid Dynamics

Properties
*Density, f(T,F)

Steady-State Thermal

Properties
*Isotropic Thermal Conductivity, f(T,C,U) Orthotropic Thermal Conductivity, f(T,C,U) See Linear Material Properties in the *Material Reference* for a description of the above properties.

Static Structural

Properties
Physical Properties *Density, f(T,F,C,U) Coefficient of Thermal Expansion (p. 59), f(T,F,U) (★ (p. 59)) Orthotropic Secant Coefficient of Thermal Expansion, f(T, F, C, U) Isotropic Secant Coefficient of Thermal Expansion, f(T, F, C, U) Isotropic Instantaneous Coefficient of Thermal Expansion, f(T) Orthotropic Instantaneous Coefficient of Thermal Expansion, f(T) Melting Temperature **Stress-Strain Relationships** *Isotropic Elasticity (Field Variables (p. 51)), f(T,F,U)

Properties
Orthotropic Elasticity (Field Variables (p. 51)), f(T,F,U)
Anisotropic Elasticity (★ (p. 59))
Damping
Damping Factor (α) (Transient Structural only)
Damping Factor (β) (Transient Structural only)
Constant Damping Coefficient (Transient Structural only)
Hyperelasticity
Mooney-Rivlin (2, 3, 5, and 9 Parameter) f(T,F,U) (★ (p. 62))
Neo-Hookean f(T,F,U) (★ (p. 62))
Polynomial (1st, 2nd, 3rd, and Nth Order), f(T) (★ (p. 62))
Yeoh (1st, 2nd, 3rd, and Nth Order), f(T,F,U) (★ (p. 62))
Ogden (1st, 2nd, 3rd, and Nth Order), f(T,F,U) (★ (p. 62))
Arruda-Boyce, f(T,F,U) (★ (p. 62))
Gent, f(T,F,U) (★ (p. 62))
Blatz-Ko f(T,F,U) (★ (p. 62))
Response Function (★ (p. 62))
Ogden Foam (1st, 2nd, 3rd, and Nth Order), f(T,F,U) (★ (p. 62))
Extended Tube, f(T,F,U) (★ (p. 62))
Mullins Effect, f(T,F,U) (★ (p. 62))
General Isotropic Hardening
Bilinear Isotropic Hardening, f(T)
Multilinear Isotropic Hardening, f(T)
Nonlinear Isotropic Hardening, f(T)
Isotropic Hardening Static Recovery, f(T)
General Kinematic Hardening
Bilinear Kinematic Hardening, f(T)
Multilinear Kinematic Hardening, f(T)
Chaboche Kinematic Hardening (p. 61), f(T)
Rate-Dependent Plasticity
Anand Viscoplasticity, f(T)
Exponential Visco-Hardening (EVH) Viscoplasticity, f(T)
Perzyna and Peirce Viscoplasticity, f(T)

Properties
Rate-Independent Plasticity Gurson Model, f(T) (★ **(p. 63)**) Hill Yield, f(T, C, H, D, U). **Fatigue** Strain-Life Parameters S-N Curve Linear S-N Curve Bilinear S-N Curve **Viscoelasticity** Prony Shear Relaxation, f(T) Prony Volumetric Relaxation, f(T) Williams-Landel-Ferry Shift Function (★ (p. 60)) Tool-Narayanaswamy Shift Function (★ (p. 60)) Tool-Narayanaswamy with Fictive Temperature Shift Function (★ (p. 60)) **Stress Tool** Tensile Yield Strength Compressive Yield Strength Tensile Ultimate Strength Compressive Ultimate Strength Gasket Model **Implicit Creep Equations (p. 60)** Strain Hardening, f(T) Time Hardening, f(T) Generalized Exponential, f(T) Generalized Graham, f(T) Generalized Blackburn, f(T) Modified Time Hardening, f(T) Modified Strain Hardening, f(T) Generalized Garofalo, f(T) Exponential form, f(T) Norton, f(T) Combined Time Hardening, f(T)

Properties
Rational polynomial, f(T) Generalized Time Hardening, f(T)
Material Strength Limits Orthotropic Stress Limits, f(TF,U) Orthotropic Strain Limits, f(T,F,U) Tsai-Wu Constants, f(T) Puck Constants, f(T,F,U) LaRc03/04 Constants, f(T) **Note:** Material strength limits can be saved into a material library and they will be written to the Mechanical APDL solver. However, these items cannot be post-processed in Mechanical.
Shape Memory Alloy (SMA) Superelasticity, f(T) Shape Memory Alloy, f(T)
Geomechanical Cam Clay, f(T) (★ (p. 63)) Drucker-Prager, f(T) (★ (p. 64)) Jointed Rock, f(T) (★ (p. 64)) Mohr-Coulomb, f(T) (★ (p. 65)) Porous Elasticity, f(T) (★ (p. 65)) Menetrey-Willam, f(T) (★ (p. 65))
Material Damage Damage Initiation Criteria Damage Evolution Law
Cohesive Zone Exponential for Interface Delamination, f(T) Bilinear for Interface Delamination, f(T) Separation-Distance based Debonding, f(T) Fracture-Energies based Debonding, f(T)
Fracture Criteria

Properties
Linear Fracture Criterion, f(T)
Bilinear Fracture Criterion, f(T)
B-K Fracture Criterion, f(T)
Modified B-K Fracture Criterion, f(T)
Power Law Fracture Criterion, f(T)
Crack Growth Laws
The following material models provide the necessary crack growth laws for fatigue crack growth analyses performed using the SMART Method for Crack-Growth Simulation. Only one material model can be specified for fatigue crack growth in a model.
Paris' Law, f(T) and/or f(R)
Tabular Fatigue Law, f(T) and/or f(R)
The material models listed below cannot be used with crack-closure functions:
Forman Equation, f(T)
Walker Equation, f(T)
NASGRO Equation V3, f(T)
NASGRO Equation V4, f(T)
(★) Refers to the corresponding Engineering Data Help page.

Static Structural (Samcef)

Important:

The material properties and models that you use to create SAMCEF solver input must have a temperature dependency. If not, the application does not write the input file. If you have imported materials that do not have an associated temperature dependency, you must add the **Temperature** field in the Engineering Data workspace as a field variable (p. 51) and enter a value (such as `0.0`) to enable the application to write an input file.

Properties
Density, f(T,F)
Isotropic Secant Coefficient of Thermal Expansion, f(T, F, C, U)
Orthotropic Secant Coefficient of Thermal Expansion, f(T, F, C, U)
*Isotropic Elasticity, f(T,F,U)
Orthotropic Elasticity, f(T,F,U)
Bilinear Isotropic Hardening, f(T)
Bilinear Kinematic Hardening, f(T)
Multilinear Isotropic Hardening, f(T)

Properties
Multilinear Kinematic Hardening, f(T) Formulation: Cauchy BIOT Kirchof Chaboche Kinematic Hardening, f(T) Nonlinear Elastic Model with Damage Plakin Special Hardening Law, f(T) Plastic Strain Failure Temperature Dependent, f(T) Refer to the Samcef documentation for a description of the above properties.

Thermal-Electric

Properties
*Isotropic Thermal Conductivity, f(T,C,U) Orthotropic Thermal Conductivity, f(T,C,U) *Isotropic Resistivity, f(T) Orthotropic Resistivity, f(T) Isotropic Seebeck Coefficient, f(T) Orthotropic Seebeck Coefficient, f(T) See Linear Material Properties in the *Material Reference* for a description of the above properties.

Transient Structural

Properties
Physical Properties *Density, f(T,F,U) Coefficient of Thermal Expansion (p. 59), f(T,F,U) (★ (p. 59)) Orthotropic Secant Coefficient of Thermal Expansion, f(T, F, C, U) Isotropic Secant Coefficient of Thermal Expansion, f(T, F, C, U) Isotropic Instantaneous Coefficient of Thermal Expansion, f(T) Orthotropic Instantaneous Coefficient of Thermal Expansion, f(T)

Properties
Stress-Strain Relationships *Isotropic Elasticity (Field Variables (p. 51)), f(T,F,U) Orthotropic Elasticity (Field Variables (p. 51)), f(T,F,U) Anisotropic Elasticity (★ (p. 59))
Damping Damping Factor (α) Damping Factor (β) Material Dependent Damping (p. 60)
Hyperelasticity Mooney-Rivlin (2, 3, 5, and 9 Parameter) f(T,F,U) (★ (p. 62)) Neo-Hookean f(T,F,U) (★ (p. 62)) Polynomial (1st, 2nd, and 3rd Order), f(T) (★ (p. 62)) Yeoh (1st, 2nd, and 3rd Order), f(T,F,U) (★ (p. 62)) Ogden (1st, 2nd, and 3rd Order), f(T,F,U) (★ (p. 62)) Arruda-Boyce, f(T,F,U) (★ (p. 62)) Gent, f(T,F,U) (★ (p. 62)) Blatz-Ko f(T,F,U) (★ (p. 62)) Response Function (★ (p. 62)) Ogden Foam (1st, 2nd, and 3rd Order), f(T,F,U) (★ (p. 62)) Extended Tube, f(T,F,U) (★ (p. 62)) Mullins Effect, f(T,F,U) (★ (p. 62))
General Isotropic Hardening Bilinear Isotropic Hardening, f(T) Multilinear Isotropic Hardening, f(T)
General Kinematic Hardening Bilinear Kinematic Hardening, f(T) Multilinear Kinematic Hardening, f(T) Chaboche Kinematic Hardening (p. 61), f(T)
Rate-Dependent Plasticity Anand Viscoplasticity, f(T)
Rate-Independent Plasticity

Properties
Gurson Model, f(T) (★ **(p. 63)**) Hill Yield, f(T, C, H, D, U).
Fatigue
Strain-Life Parameters S-N Curve Linear S-N Curve Bilinear S-N Curve
Viscoelasticity
Prony Shear Relaxation, f(T) Prony Volumetric Relaxation, f(T) Williams-Landel-Ferry Shift Function (★ (p. 60)) Tool-Narayanaswamy Shift Function (★ (p. 60)) Tool-Narayanaswamy with Fictive Temperature Shift Function (★ (p. 60))
Stress Tool
Tensile Yield Strength Compressive Yield Strength Tensile Ultimate Strength Compressive Ultimate Strength
Gasket Model
Implicit Creep Equations (p. 60)
Strain Hardening, f(T) Time Hardening, f(T) Generalized Exponential, f(T) Generalized Graham, f(T) Generalized Blackburn, f(T) Modified Time Hardening, f(T) Modified Strain Hardening, f(T) Generalized Garofalo, f(T) Exponential form, f(T) Norton, f(T) Combined Time Hardening, f(T) Rational polynomial, f(T)

Properties
Generalized Time Hardening, f(T)
Material Strength Limits
Orthotropic Stress Limits, f(TF,U)
Orthotropic Strain Limits, f(T,F,U)
Tsai-Wu Constants, f(T)
Puck Constants, f(T,F,U)
LaRc03/04 Constants, f(T)
Shape Memory Alloy (SMA)
Superelasticity, f(T)
Shape Memory Alloy, f(T)
Geomechanical
Cam Clay, f(T) (★ (p. 63))
Drucker-Prager, f(T) (★ (p. 64))
Jointed Rock, f(T) (★ (p. 64))
Mohr-Coulomb, f(T) (★ (p. 65))
Porous Elasticity, f(T) (★ (p. 65))
Menetrey-Willam, f(T) (★ (p. 65))
Material Damage
Damage Initiation Criteria
Damage Evolution Law
Cohesive Zone
Exponential for Interface Delamination, f(T)
Bilinear for Interface Delamination, f(T)
Separation-Distance based Debonding, f(T)
Fracture-Energies based Debonding, f(T)
Fracture Criteria
Linear Fracture Criterion, f(T)
Bilinear Fracture Criterion, f(T)
B-K Fracture Criterion, f(T)
Modified B-K Fracture Criterion, f(T)
Power Law Fracture Criterion, f(T)

Properties
(★) Refers to the corresponding Engineer Data Help page.

Transient Thermal

Properties
*Density, f(T,F) *Isotropic Thermal Conductivity, f(T,C,U) Orthotropic Thermal Conductivity, f(T,C,U) *Specific Heat, C_p f(T,C, U) Enthalpy f(T) Melting Temperature See Linear Material Properties in the *Material Reference* for a description of the above properties.

Parametrizing Material Properties

Data can be parametrized to allow it to be used in parametric studies and design points (see Working with Design Points). To parametrize an item in the **Properties** pane, select the check box in the **Parameterize** ( ). The parameter value can then be changed in the Parameters and Design Points workspace. Use caution when parametrizing data that is dependent on other data to maintain valid data. The parametrized data is always calculated from the original values, so also use caution when modifying data in the parameter workspace to avoid computer precision problems.

Constant Data

When you parametrize constant data, the constant data can be changed in the **Parameters** workspace, but the original datum is not modified. The original datum is restored when the parametrization is removed.

Tabular Data

You can parametrize tabular data by parametrizing the scale and/or offset for all of the tabular data. The scale value varies the curve by multiplying the y-axis value of each point on the curve. The offset is added to or subtracted from a y-axis value for each point on the curve. The equation used for varying each datum value in the tabular data is:

Property = Scale * Nominal Value + Offset

The original tabular data is not modified, but the scale and/or offset datum are restored to the defaults of 1.0 and 0.0 when the parametrization is removed.

To parametrize a material property:

1. Access Engineering Data (p. 15).

2. To parametrize a material property in libraries or favorites, open the **Engineering Data Sources** pane (p. 19) and mark the data source as editable (p. 20).

3. In the **Outline** pane, select a material that contains the property to parametrize.

4. in the **Properties** pane, select the check box in the **Parameterize** ( ) column for the property you want to parametrize.

   For tabular data, use scale and/or offset.

Suppressing Material Properties

Data may be defined but suppressed to prevent it from being sent to a downstream cell in the system. For example, suppressing a material or material property prevents it from being used in the model. Suppressed items are shown by a strike through the name (for example, ~~Structural Steel~~) and the check box being selected in the suppression column.

To suppress a material property:

1. Access Engineering Data (p. 15).

2. To suppress a material property in libraries or favorites, open the **Engineering Data Sources** pane (p. 19) and mark the data source as editable (p. 20).

3. In the **Outline** pane, select a material that contains the property to suppress.

4. In the **Suppression** ( ) column, select the check box for the property you want to suppress.

Suppression of Mutually Exclusive Properties

Some properties are mutually exclusive of each other and require that only one property in the mutually exclusive set be unsuppressed. The addition or removing of the suppression for one of these properties automatically suppresses the other mutually exclusive properties.

For example, defining Bilinear Isotropic Hardening and Multilinear Isotropic Hardening for the same material represents redundant plasticity behavior. Only one behavior can be active for the material. When such a conflict occurs, the property defined last is used and the previously defined, conflicting property is automatically suppressed.

Charting

The axis range can be modified on a chart by selecting **Edit Properties** from the axis context menu (see Setting Chart Properties). This range is used when generating the x-y data for the chart so that you can examine the data beyond the default range.

Validation

The user interface indicates invalid data by displaying a yellow background and changing the state of the material icon (p. 8).

To find the reason for the data being invalid, right-click the invalid item and select **Display Validation Failure** from the context menu.

Curve Fitting

When curve fitting is available for a selected item in Engineering Data a **Curve Fitting** group is accessible in the Workbench **Toolbox**. A curve fitting module can then be added to the selected item.

The data that can be used for the curve fitting module is shown, regardless of whether or not the data is available. If the required data is not available, add it from the **Toolbox**. The data used for curve fitting can be suppressed (in some instances), in the curve fitting module, to prevent its use in the curve fitting calculations.

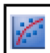
Note:

- Curve fitting is only active for the running session. If you exit the application, you must add the curve fitting module again. Select **Copy Calculated Values to Property** to maintain the calculated information.
- If the active unit system for the project is changed, the current curve fitting modules in the project are recreated using the new unit system. The seed values are reset to their default values.
- If the experimental data is parametrized, the current design point parameter value is used for the computations when selecting **Solve Curve Fit**. It has no effect for other design points when updating a project, because of the interactive nature of performing the fitting.

Curve fitting is specific to a given solver and so the following topics provide specific information:

- Material Model Curve Fitting (p. 48)
- Mechanical Material Curve Fitting (p. 49)

Curve Fitting Icons

	Module for performing curve fitting.
	Indicates that this curve fitting module requires attention.

Material Model Curve Fitting

Curve fitting is specific to a given solver, this topic addresses material models.

To curve fit material models:

1. Access Engineering Data (p. 15).
2. To curve fit material models in libraries or favorites, open the **Engineering Data Sources** pane (p. 19) and mark the data source as editable (p. 20).

3. From the **Curve Fitting** group in the **Toolbox**, select a material model that supports curve fitting.

4. Modify options for the curve fitting calculations.

5. Suppress any experimental data to exclude from the curve fitting calculations.

6. From the **Properties** pane, right-click the curve fitting module and select **Solve Curve Fit** from the context menu.

7. Review the accuracy of the fitted data to the experimental data.

8. From the **Properties** pane, right-click the curve fitting module and select **Copy Calculated Values To Property** from the context menu.

Mechanical Material Curve Fitting

The material curve fitting calculates coefficients of material models that approximate the following experimental data. You can enter the data or copy and paste data from a spreadsheet into the **Table** pane. See the Curve Fitting (p. 48) section for additional specification information.

Hyperelastic Test Data

- Uniaxial Test f(T) (Engineering Strain vs. Engineering Stress)
- Biaxial Test f(T) (Engineering Strain vs. Engineering Stress)
- Shear Test f(T) (Engineering Strain vs. Engineering Stress)
- Volumetric Test f(T)

The following hyperelastic material models support curve fitting of the experimental data. For additional information, see the Material Curve-Fitting chapter in the *Mechanical APDL Material Reference*.

- Mooney-Rivlin
- Ogden
- Neo-Hookean
- Polynomial
- Yeoh
- Arruda-Boyce
- Gent
- Blatz-Ko

Note:

The Volume Ratio equals the ratio of the current volume to that of the original volume and all solution data displayed for postprocessing are true stresses and logarithmic strains.

The experimental data defined for all temperatures is used for curve fitting. Ensure that temperatures are defined consistently for different experimental data. If the experimental data contains temperature mismatch, then the warning message is displayed in the **Messages** pane during **Solve Curve Fit**.

When volumetric data is supplied, a compressible or nearly incompressible model is implied. When no volumetric data is supplied, the model is understood to be incompressible. Supplying zero as a coefficient for the volumetric data also denotes an incompressible model. The curve fitting will calculate the parameters based on an incompressible model when volumetric data is supplied and also when calculating the Stress-Strain points for charting.

Perform curve fitting for the various hyperelastic models to choose the one, based on the range of strain you are interested in, that best matches the experimental data provided.

Chaboche Test Data

Uniaxial Plastic Strain Test Data f(T) (Plastic Strain vs. True Stress)

Note:

To use curve fitting with plasticity, the only experimental data supported is Uniaxial Plastic Strain Test Data. Experimental data for plasticity is path dependent. Multiple Uniaxial Plastic strain test data can be added from **Uniaxial Plastic Strain –Additional Test Data** group which is accessible in the **Toolbox**, when Uniaxial Plastic Strain Test Data is selected.

Chaboche Kinematic Hardening plasticity model supports curve fitting of the Uniaxial Plastic strain test data. For additional information, see the Material Curve-Fitting chapter in the *Mechanical APDL Material Reference*.

Viscoelastic Test Data

- Shear Data- Viscoelastic f(T) (Time vs. Shear Modulus)
- Bulk Data - Viscoelastic f(T) (Time vs. Bulk Modulus)

Note:

Both the above test data are path dependent. Multiple test data can be added from **Viscoelastic - Additional Test Data** group which is accessible in the **Toolbox**, when test data is selected.

Viscoelastic models with curve fitting support:

- Prony Shear Relaxation, f(T)
- Prony Volumetric Relaxation, f(T)

For additional information, see the Material Curve-Fitting chapter in the *Mechanical APDL Material Reference*.

Note:

A change in the **Number of Terms** on the Prony models updates the curve fitting coefficients to allow fitting for the desired number of terms. The calculated values in the curve fitting are normalized, when **Copy Calculated Values to Property** is selected. The normalization uses the following equations, which describe the relationship between the Prony Coefficient (α_i) and the corresponding coefficient generated in curve fitting (A_i). N is the number of terms computed. A_i^K is the square root of K_i and A_i^G is the square root of G_i. $K_{t=\infty}$ and $G_{t=\infty}$ are the shear modulus and bulk modulus at $t=\infty$. This is done to keep all α_i^K and α_i^G values used in the property table positive.

$$K_{t=0}=K_{t=\infty}+\sum_{i=1-N} A_i^K * A_i^K \quad (1)$$

$$\alpha_i^K=\left(A_i^K * A_i^K\right)/K_{t=0} \quad (2)$$

$$G_{t=0}=G_{t=\infty}+\sum_{i=1-N} A_i^G * A_i^G \quad (3)$$

$$\alpha_i^G=\left(A_i^G * A_i^G\right)/G_{t=0} \quad (4)$$

Error Norm for Fit

The error norm can be set to use normalized or absolute error. Normalized error norm considers each experimental datum equally in computing the curve fit. It generally provides better results than the absolute error norm, but in some cases the absolute error norm is a better choice.

Nonlinear Fitting (Ogden, Gent and Chaboche Kinematic Hardening)

For nonlinear curve fitting you can provide seed values for the coefficients or you can fix these seed values. The seed values can be provided for each temperature data. If you do not provide seed values internal defaults is used. It is suggested that you attempt to use seed values based on experience if possible. The nonlinear curve fit most often converges to a local error norm minimum. It may take several attempts (trial seed values) to achieve the desired fit, or copying and pasting the last solution as seed values and solving again.

Curve fitting for viscoelastic models Prony Shear Relaxation and Prony Volumetric Relaxation is nonlinear.

Field Variables

Material data can be dependent on variables such as temperature, shear angle, degradation factor, or user-defined entries. These variables known as *field variables* and are only available for specific material properties and models, as listed below. For more information on this topic, see User-Defined Field Variables in the *Material Reference*.

Engineering Data supports the following field variables:

- **Temperature**
- **Mean Stress**
- **R-Ratio**
- **Frequency**
- **Coordinate X**
- **Coordinate Y**
- **Coordinate Z**
- **Shear Angle**
- **Degradation Factor**
- User-defined variables (as discussed below).

The use of field variables to further define material properties enables you to simulate the effects of environmental conditions and manufacturing processes on material characteristics. This can be of particular importance when you are using the Ansys Composite PrepPost application to model composites. See Variable Material Data in Composite Analyses in the *ACP User's Guide* for additional information.

The material properties and models supported by the field variable feature are based on the selected analysis system. See the Supported Properties (p. 28) section of the documentation for a listing of all of the supported analysis types and the material properties that each analysis supports. Note that certain properties are appended with a function notation to indicate supported fields. This includes:

- Temperature (T)
- Frequency (F)
- Coordinate X (C)
- Coordinate Y (C)
- Coordinate Z (C)
- User-defined (U).

Note:

The coordinate system assigned to the body in the analysis is the coordinate system used for the field variables Coordinate X, Y, and Z. Only Cartesian coordinate systems are supported.

The Shear Angle and Degradation Factor field variables are predefined user-defined variables. Based on this, isotropic elasticity supports all field variables and is denoted as: Isotropic Elasticity f(T,F,U).

However, for Modal Acoustic (p. 35) analysis system, only temperature is supported and so denoted as: **Density f(T)**. For material properties that are defined with unsupported field variables, the application uses the default value for the unsupported field variables to calculate dependent values for the supported field variable(s) values. For example, during a Modal Acoustics analysis, you can enter frequency dependent and/or temperature dependent material properties, however:

- If you define frequency dependent material property data, f(F), your input is calculated using the interpolation of the General Interpolation Library (GIL) based on the default frequency value of the Material Field Variables property.

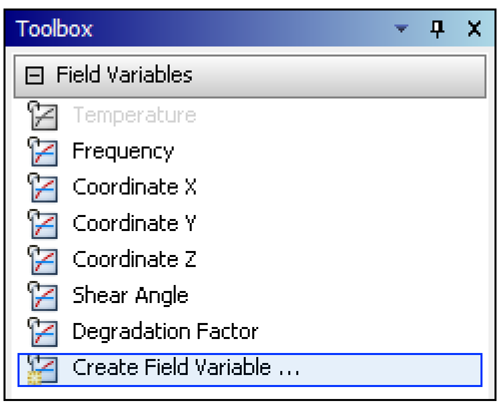
- If you define frequency-dependent and temperature-dependent material property data, solver input is calculated using both the GIL, based on default frequency, and the defined temperature points, for example, **$f(F_{default}, T)$**. Any change to the default frequency value may vary the interpolated values.

Defining System Provided Field Variables

To implement a system provided field variable:

1. Select one of the supported material properties (as listed above).

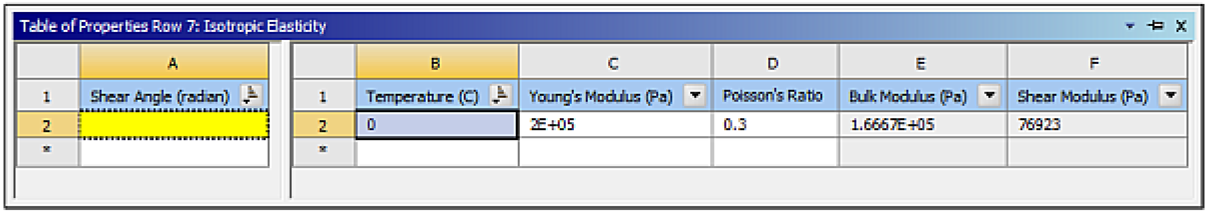
   The application displays the **Field Variables** category in the **Toolbox**.

2. From the **Field Variables** category, select the desired system provided field variable:

   - **Temperature**
   - **Frequency**
   - **Coordinate X**
   - **Coordinate Y**
   - **Coordinate Z**
   - **Shear Angle**
   - **Degradation Factor**

   The new field variable populates the **Table** pane (p. 13). Enter appropriate values.

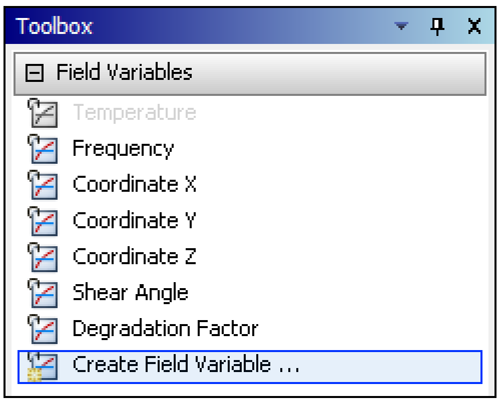


3. Enter appropriate values into the **Table** pane.

Creating User-Defined Field Variables

To implement a user-defined field variable:

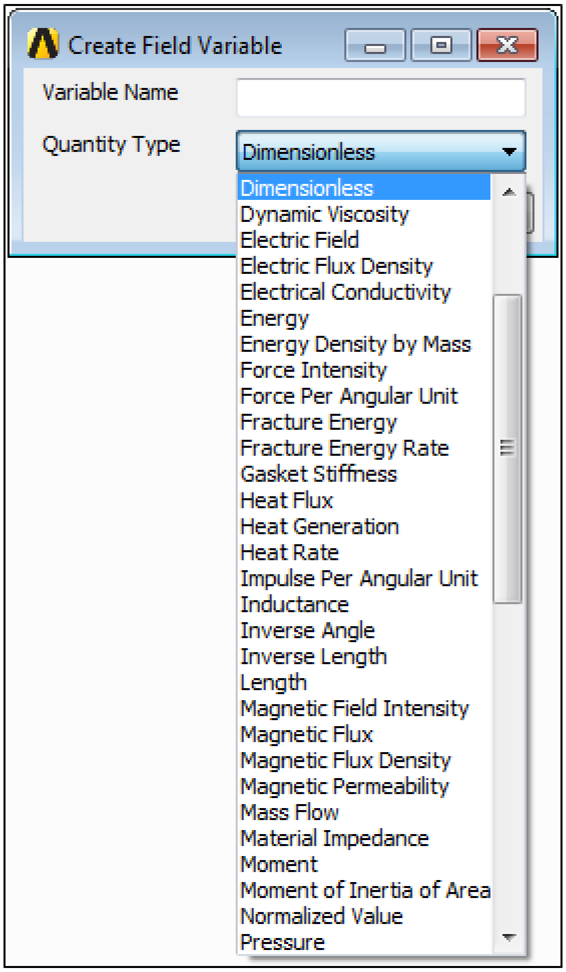
1. In the **Toolbox**, select **Create Field Variable** from the **Field Variables** category.



2. In the **Create Field Variable** dialog box, enter a name for your new field variable.

3. Select an option from the **Quantity Type** drop-down list.

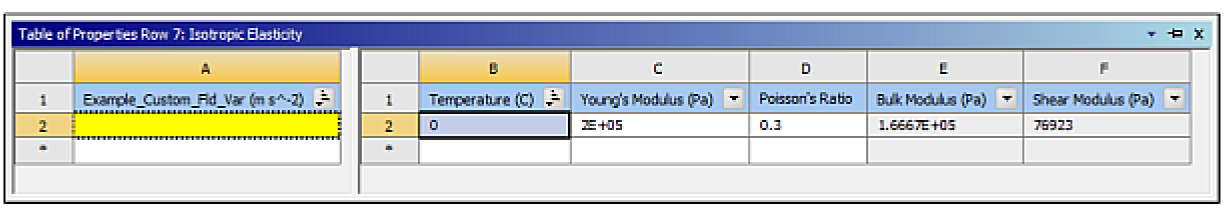
   The default is **Dimensionless**.



4. Once complete, click **OK**.

   The new field variable populates the **Table** pane.

5. Enter appropriate values. Example entries are illustrated below for the user-defined field variable `Example_Custom_Fld_Var`.



6. Select the **Interpolation Options** property to display the following **Table** pane.

   This table defines the algorithm the solver uses to interpolate between the data entered in the **Table** pane from Step 2 to get the values corresponding to the calculated field variable values on a given element. See General Interpolation Library in the *Composites Theory Reference* for more details about the algorithms and options used when creating user-defined field variables.

Table of Properties Row 8: Isotropic Elasticity

	A	B	C
1	Algorithm Type	Normalize	Cache
2	Linear Multivariate	☑	☑

7. Select an option from the **Algorithm Types** drop-down list.

Table of Properties Row 8: Isotropic Elasticity

	A	B	C
1	Algorithm Type	Normalize	Cache
2	Linear Multivariate	☑	☑

Linear Multivariate
Nearest Neighbor
Radial Basis

8. Refine the interpolation algorithm by activating one or more of the following options.

   - **Normalize**: Scale the axes to achieve an unbiased distance metric. The ranges used to scale the axes are defined by the **Lower Limit** and **Upper Limit** values of the **Material Field Variables** table pane.

   - **Cache**: Save previous queries to increase performance.

9. Select the **Material Field Variables** property to display the following table pane.

Table of Properties Row 2: Structural Steel Field Variables

	A	B	C	D	E
1	Variable Name	Unit	Default Data	Lower Limit	Upper Limit
2	Temperature	C	22	Program Controlled	Program Controlled
3	Example_CustomFld_Var	m s^-2	0	Program Controlled	Program Controlled

Field descriptions for the **Material Field Variables** property:

- **Default Data**: Used by the interpolation algorithm when the corresponding field variable is not defined on a given element.

- **Lower Limit**: Defines the lower boundary on the range used when the data is normalized and/or quantized. When you use the **Program Controlled** setting, the value defaults to the minimum value of the data entered for all instances of the corresponding field variable in the selected material.

- **Upper Limit**: Defines the upper boundary on the range used when the data is normalized and/or quantized. When you use the **Program Controlled** setting, the value defaults to the maximum value of the data entered for all instances of the corresponding field variable in the selected material.

Usage Notes

The order in which you add field variables to a material property becomes important if:

- Your material data has multiple field variables.
- The data is structured such that one field variable changes while the others remain fixed.

Whichever field variable varies the most, is the variable that you should insert and define first. Consider the example illustrated below. In this case, the Temperature varies more than the Shear Angle, so it is best to add temperature first and then shear angle. Adding the field variables in this order results in a table structure that, for each shear angle, there is data at multiple temperatures.

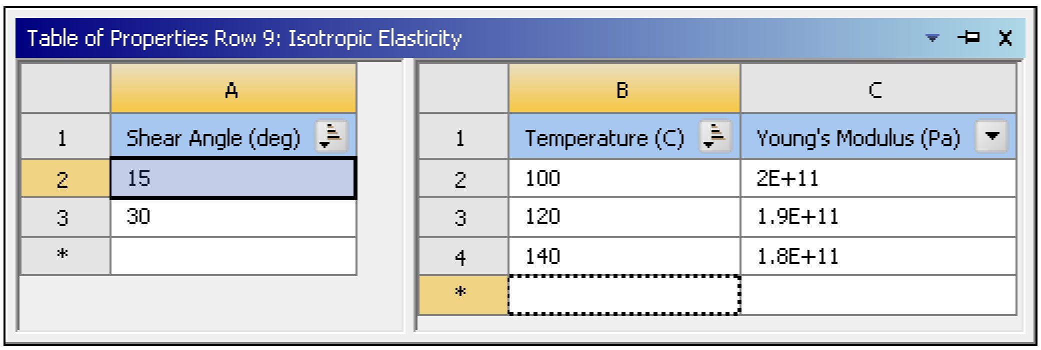


Table of Properties Row 9: Isotropic Elasticity

	A
1	Shear Angle (deg)
2	15
3	30
*	

	B	C
1	Temperature (C)	Young's Modulus (Pa)
2	100	2E+11
3	120	1.9E+11
4	140	1.8E+11
*		

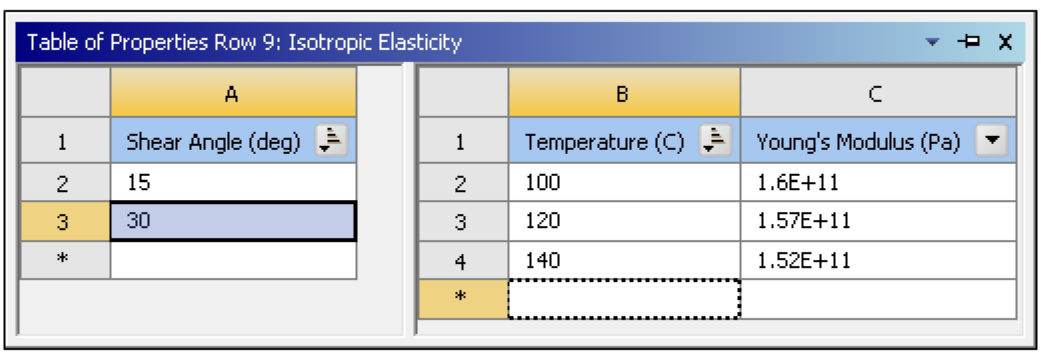
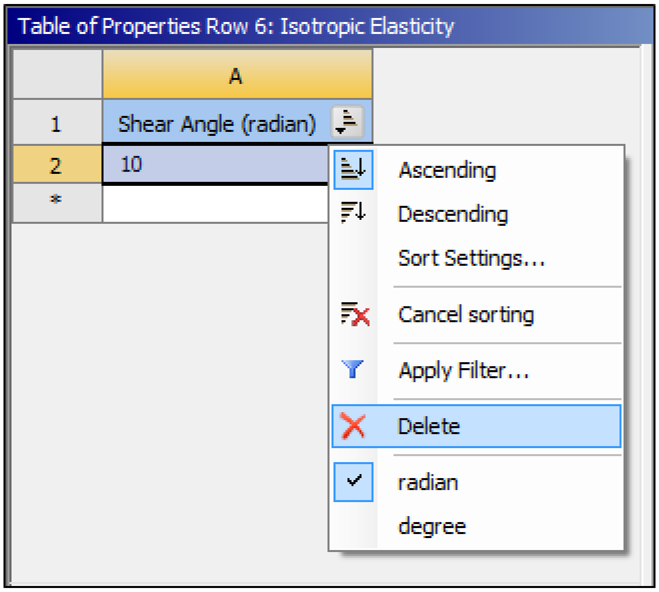


Table of Properties Row 9: Isotropic Elasticity

	A
1	Shear Angle (deg)
2	15
3	30
*	

	B	C
1	Temperature (C)	Young's Modulus (Pa)
2	100	1.6E+11
3	120	1.57E+11
4	140	1.52E+11
*		

The context menu on the **Table** pane for the field variable provides sorting and filtering options.



To remove a field variable from a property, right-click the field variable and select **Delete** from the context menu.

Material Property Support in Mechanical

Select your material properties based on how the material exhibits properties in orthogonal directions (X, Y, and Z), either constant in all directions (isotropic behavior) or different in orthogonal directions (orthotropic behavior). These orthogonal directions in a part, by default, align with the global coordinate system. You may apply a local coordinate system to the part to change the directions. For orthotropic properties, the X, Y, and Z value must be specified for the model to solve (2-D models only use the X and Y values). Those properties which support isotropic or orthotropic behavior is preceded by **Isotropic** or **Orthotropic** (for example, **Isotropic Thermal Conductivity**).

Temperature-dependent properties (the identifier **f(T)** is shown beside the property) are input as tabular data (value vs. temperature). During solution, the material properties are evaluated for the temperature of the integration points of the elements. If the temperature of an integration point falls below or rises above the defined temperature range of tabular data, the solver assumes the defined extreme minimum or maximum value, respectively, for the material property outside the defined range.

The following are supported.

Linear Material Models

Material Models with Nonlinear Behavior (No Unit Conversion)

Material Dependent Damping

Isotropic Hardening

Kinematic Hardening

Chaboche Kinematic Hardening

Hyperelastic Material Models

Gasket Material Model

Gurson Material Model

Linear Material Models

- **Coefficient of Thermal Expansion**

  You can define the coefficient of thermal expansion using the secant or instantaneous method. The secant method includes field variable support for temperature, frequency, coordinate, or user defined field dependency. When you define the coefficient of thermal expansion using the secant method, and you use multiple field variable points, you must define the zero-thermal-strain reference temperature. When conducting a test to measure the thermal expansion, the zero-thermal-strain reference temperature is that temperature at which the test specimen has a zero thermal strain. As the specimen is heated from the zero thermal strain point, the thermal strain is measured at given temperature points. If fields other than Temperature are specified the temperature of the bodies must be the same as the zero-thermal-strain reference temperature value. If only Temperature field is defined then the zero-thermal-strain reference temperature value is specified by the **MPAMOD** command in the Mechanical APDL application, and written as the **MPAMOD** command in the `ds.dat` file. The **MPAMOD** command is written only if the temperature of the bodies using the material differs from the material's zero-thermal-strain reference temperature. The coefficient of thermal expansion values are computed according to the equation documented in Temperature-Dependent Coefficient of Thermal Expansion in the *Theory Reference*.

  For more information on this topic, see Linear Material Properties in the *Material Reference*

- **Anisotropic Elasticity**

  You can define an anisotropic elasticity model by entering the stiffness terms in an elastic coefficient matrix [**D**]. The following graphic show the location of the terms for this symmetric matrix:

D11					
D21	D22				
D31	D32	D33			
D41	D42	D43	D44		
D51	D52	D53	D54	D55	

D61	D62	D63	D64	D65	D66

Each row corresponds to the terms of x, y, z, xy, yz, xz. The stiffness terms must be positive definite (requiring all determinants to be positive). The stiffness terms have units of Force/Area operating on the strain vector.

Note:

If you change the **Format** property from **IEEE** to **MAPDL**, or vise versa, the application automatically clears any data you may have entered and does not perform a conversion.

Material Models with Nonlinear Behavior (No Unit Conversion)

Material models which have nonlinear behavior do not allow for conversion of data from one unit system to another. For these material models, specify the units of the data in the **Reference Units** drop-down list. In Mechanical if an attempt is made to solve the solution, in a unit system which is not compatible with the **Reference Units**, an error is shown and the solution is stopped. The following material models have this restriction:

- Anand Viscoplasticity
- Exponential Visco-Hardening (EVH) Viscoplasticity
- Perzyna and Peirce Viscoplasticity
- Shift Functions

  The Reference Unit setting is applicable for all inputs except the Reference Temperature, where the unit can be set independently. Modification of the Reference Unit setting converts the Reference Temperature to the value corresponding to the temperature unit specified in the Reference Unit.
- Creep

These material models require temperature values to be in an absolute scale. Mechanical writes all temperature inputs for a solver in °C or °F and includes a **TOFFST** command with the value appropriate to the solver unit system setting. This setting allows the computation of these material models to be performed correctly.

Material Dependent Damping

The **Material Dependent Damping** property grouping enables you to define the coefficients **Damping Ratio** and **Constant Structural Damping Coefficient**. If you provide a valid value for the **Damping Ratio** property, the **Constant Structural Damping Coefficient** property is automatically entered as two times that of the **Damping Ratio** value.

For additional information, see Damping in the *Structural Analysis Guide*.

Isotropic Hardening

Engineering Data supports the following general classes of isotropic hardening:

- Bilinear Isotropic Hardening
- Multilinear Isotropic Hardening
- Nonlinear Isotropic Hardening
- Isotropic Hardening Static Recovery

In addition to the above topics, the Rate-Independent Plasticity section of the Mechanical APDL Material Reference includes detailed theoretical and background information on isotropic hardening.

Important:

In release 2022R2 (or prior), the application defined the **Bilinear Isotropic Hardening** and **Bilinear Kinematic Hardening** material models using **Total Strain**. When you open a project from these previous releases, the **Active Table** property for the material models displays the value **Total**. For this scenario, if the temperature is constant, or the same (tabular) temperature values are used for **Isotropic Elasticity** and the hardening model data, the application provides the right-click option **Convert Total to Plastic** for the **Active Table** value cell. See the Bilinear Isotropic Hardening topic in the **Rate-Independent Plasticity** section of the *Mechanical APDL Material Reference* for more information about the conversion.

Kinematic Hardening

Engineering Data supports the following general classes of kinematic hardening:

- Bilinear Kinematic Hardening
- Multilinear Kinematic Hardening
- Chaboche Kinematic Hardening (p. 61)

In addition to the above topics, the Rate-Independent Plasticity section of the Mechanical APDL Material Reference includes detailed theoretical and background information on kinematic hardening.

Chaboche Kinematic Hardening

The **Chaboche Kinematic Hardening** material model option can be defined with respect to temperature and allows up to five kinematic models for each temperature. It is required that all kinematic models be defined for each temperature. You can specify the number of kinematic models in the drop-down list. This adds additional columns for entering the data. Also included as an option is **Chaboche Kinematic Hardening with Recovery**. See the Nonlinear Kinematic Hardening topic in the *Rate-Independent Plasticity* section of the *Mechanical APDL Material Reference* for more information.

Hyperelastic Material Models

Hyperelasticity can be used to analyze rubber-like materials (elastomers) that undergo large strains and displacements, with small volume changes (nearly incompressible materials). Large strain theory is required (in the Mechanical application, set Large Deflection to On).

The hyperelastic material models are isotropic and constant with respect to temperature. The hyperelastic materials are also assumed to be nearly or purely incompressible. Thermal expansion, in the material, is also assumed to be isotropic.

Experimental testing data can be input for a material, and then using the curve fitting module (see Curve Fitting (p. 48)), calculate coefficients for various hyperelastic material models. Another option is to make use of the Response Function which allows the use of experimental testing data and the definition of incompressibility parameters. The allowed experimental testing data are Uniaxial Test Data, Biaxial Test Data, Shear Test Data, Volumetric Test Data, Simple Shear Test Data, Uniaxial Tension Test Data, and Uniaxial Compression Test Data. The definition of the incompressibility parameters is sequential and associated with a given index. To delete values requires that they be deleted from the end of the sequence.

Mullins effect is used for modeling load-induced changes to constitutive response exhibited by some hyperelastic materials. Mullins Effect should be used in conjunction with a hyperelastic material model except for Blatz-Ko and Ogden Foam models.

For additional information on these hyperelastic models see Hyperelasticity in the *Material Reference*

Gasket Material Model

A gasket material model is used to analyze structural components that have a sealing component between them. For additional information on the gasket model, see Gasket in the *Material Reference*. The gasket is defined by a compression load closure curve, unloading data (linear or nonlinear), and optionally transverse shear and the maximum tension stress. The gasket can additionally be defined at different temperature values by adding an additional **Data Set**, found in the **Gasket-Additional Data** category of the **Toolbox** pane for each temperature.

The compression load closure curve defines the pressure in the material at given closure values. To enter this data, select the **Compression** item and enter the data in the **Table** pane.

The unloading data can be entered as linear or nonlinear data. To add unloading data, choose **Linear Unloading** or **Nonlinear Unloading**, found in the **Gasket-Additional Data** category of the **Toolbox** pane. When choosing linear unloading multiple curves are entered into each row of the table in the **Table** pane. If nonlinear unloading is chosen each individual unloading curve is added from the **Toolbox** and then the data is added in the **Table** pane.

To optionally define the maximum tension stress, add **Gasket Parameters**, found in the **Gasket-Additional Data** category of the **Toolbox** pane.

To optionally define the transverse shear or transverse shear and membrane stiffness, add **Transverse Shear** or **Transverse Shear and Membrane Stiffness** form the **Gasket-Additional Data** category of the **Toolbox** pane.

Charts

- Select **Data Set** node in **Properties Pane** to display chart for all the data defined for specific data set or temperature value.
- Select **Gasket Model** property node in **Properties Pane** to display all the gasket data. Each data set is displayed in unique color and associated temperature value can be seen in the chart legend.

Gurson Material Model

You use the Gurson material model to analyze structural components that experience plasticity and damage in porous ductile metals.

See Rate-Independent Plasticity for a more detailed description of the Gurson Material Model.

You define the Gurson model with yield function inputs which allow the growth to occur. You can then optionally combine the yield function with a nucleation model controlled by stress or strain, and/or, a coalescence model.

Once you insert the Gurson Model, the **Gurson - Additional Data** category displays in the **Toolbox** so that you can add the optional additional models:

- **Nucleation Stress Controlled**
- **Nucleation Strain Controlled**
- **Coalescence**

You may specify either stress or strain but not both.

Puck Material Model

The Puck material model is used to analyze the failure of a material in a laminate. In some situations, you may not have experimental data for the Puck parameters but can classify the material as a glass or carbon. You can choose the Puck material classification from a drop-down menu and the appropriate data is filled in. The possible entries are as follows: **Material Specific**, **Glass**, or **Carbon**. **Material Specific** is the default selection.

Cam-Clay

The Cam-Clay material model is contained in the **Geomechanical** Toolbox data item. This property supports Static Structural and Transient Structural analyses. When you added this model, the Porous Elasticity (p. 65) model is also automatically added. For additional information, see Cam-clay in the *Material Reference*.

Cam-Clay includes the following physical properties:

- **Plastic Slope Parameter**
- **Slope of Critical State Line**
- **Initial Size of Yield Surface**

- **Minimum Size of Yield Surface**
- **Dry Part of Yield Surface Modifier**
- **Wetting Part of Yield Surface Modifier**
- **Anisotropic Yield Surface Parameter**

Drucker-Prager

The Drucker-Prager material model is contained in the **Geomechanical** Toolbox data item. This property supports Static Structural and Transient Structural analyses. For additional information, see Drucker-Prager Concrete in the *Material Reference*

Drucker-Prager Base is the primary property and includes the following physical properties:

- **Uniaxial Compressive Strength**
- **Uniaxial Tensile Strength**
- **Biaxial Compressive Strength**

In addition, once you insert the **Drucker-Prager** model, the **Drucker-Prager - Additional Data** data item displays in the **Toolbox** so that you can add one of the following:

- **Softening**
- **Failure Plane Data Set**

Jointed Rock

The Jointed Rock material model is contained in the **Geomechanical** Toolbox data item. This property supports Static Structural and Transient Structural analyses. For additional information, see Jointed Rock in the *Material Reference*.

The **Jointed Rock** property includes the following physical properties:

- **Residual Strength Coupling**
- **Yield Surface**
  - Initial Inner Friction Angle (Degree)
  - Initial Cohesion (Force/length2)
  - Dilatancy Angle (Degree)
  - Residual Inner Friction Angle (Degree)
  - Residual Cohesion (Force/length2)

In addition, once you insert the **Jointed Rock** model, the **Jointed Rock - Additional Data** data item displays in the **Toolbox** so that you can add one of the following:

- **Tension Rankine Yield Surface**
- **Failure Plane Data Set**

Mohr-Coulomb

The Mohr-Coulomb material model is contained in the **Geomechanical** Toolbox data item. This property supports Static Structural and Transient Structural analyses. For additional information, see Mohr-Coulomb in the *Material Reference*.

Yield Surface is the **Mohr-Coulomb** property and includes the following physical properties:

- Initial Inner Friction Angle (Degree)
- Initial Cohesion (Force/length2)
- Dilatancy Angle (Degree)
- Residual Inner Friction Angle (Degree)
- Residual Cohesion (Force/length2)

In addition, once you insert the **Mohr-Coulomb** model, the **Mohr-Coulomb - Additional Data** data item displays in the Toolbox so that you can add the optional additional model **Tension Rankine Yield Surface**.

Porous Elasticity

The Porous Elasticity material model is contained in the **Geomechanical** Toolbox data item. This property supports Static Structural and Transient Structural analyses. Porous Elasticity is automatically added when the Cam Clay (p. 63) model is added. For additional information, see Porous Elasticity in the *Material Reference*.

Porous Elasticity includes the following physical properties:

- **Swell Index**
- **Elastic Limit of Tensile Strength**
- **Poisson's Ratio**
- **Initial Void Ratio**

Menetrey-Willam

The Menetrey-Willam material model is contained in the **Geomechanical** Toolbox data item. This property supports Static Structural and Transient Structural analyses. It is useful for modeling geo-mechanic materials such as concrete. For additional information, see Menetrey-Willam in the *Material Reference*.

Menetrey-Willam Base and **Dilatancy Angle** are the primary properties and includes the following physical properties:

- **Menetrey-Willam Base**
  - Uniaxial Compressive Strength
  - Uniaxial Tensile Strength
  - Biaxial Compressive Strength
- **Dilatancy Angle**

In addition, once you insert the **Menetrey-Willam** model, the **Menetrey-Willam - Additional Data** data item displays in the **Toolbox** so that you can add **Softening** (Linear or Exponential).

Three Network Model

The **Three Network Model** is a special hyperlasticity material model intended specifically for visco-plastic materials.

You insert this model using the **Three Network Model** option from the **Toolbox**.

For the theoretical background of this material model, the steps to use it, as well as a description of the generated output, see the Three Network Model topic in the Special Hyperelasticity section of the *Mechanical APDL Material Reference*.

Bergstrom-Boyce Model

Bergstrom-Boyce is a special hyperviscoelasticity material model intended specifically for elastomer materials.

To add this material model to your analysis, right-click the **Bergstrom-Boyce** option contained in the **Hyperviscoelasticity** category of the **Toolbox** and select **Include**. You can also add the material by double-clicking the option.

> **Note:**
>
> Note that this material model does not support the Engineering Data option **Display Values in Project Units**. As a result, the material properties are not affected when you change the units in the Workbench **Unit** drop-down menu. You must manually change the units for the material using the **Stress Units** and **Time Units** properties of the material.

For the theoretical background of this material model, the steps to use it, as well as a description of the generated output, see the Bergstrom-Boyce Material topic in the Special Hyperelasticity section of the *Mechanical APDL Material Reference*.

Electromagnetic Material Properties

Linear "Soft" Magnetic Material

This category characterizes magnetic material assuming a constant permeability, that is, no saturation effects. Permeability is simply defined as the ratio of B to H: $\mu = B/H$. Permeability is more easily expressed in terms of relative and free-space values: $\mu = \mu_o \mu_r$. Free-space permeability, μ_o, is equal to 4πe-07 H/m. Relative permeability, μ_r is a multiplier of free-space permeability. Free-space permeability is defined internally within the program. You are required to supply a relative permeability value. This category is applicable to nonmagnetic material such as air, copper, aluminum. It can also be used as an approximation to magnetic materials when a B-H curve is not available. If the material exhibits constant properties in all directions (isotropic behavior) then select **Relative Permeability** and enter the appropriate value. If the material exhibits different permeability in different orthogonal directions (orthotropic), then select **Relative Permeability (Orthotropic)** and enter values for three orthogonal directions (X, Y, Z). By default, the global coordinate system is used when the material is applied to a part in the Mechanical application. If desired, you can apply a local coordinate system to the part. The material orthogonal properties align with the coordinate system assigned to the part. For orthotropic material properties, all property values must be entered for the model to properly solve.

Linear "Hard" Magnetic Material

This category characterizes hard magnetic materials such as permanent magnets. The demagnetization curve of the permanent magnet is assumed to have a constant slope. The demagnetization curve intersects the H axis at a value corresponding to the coercive force, H_c. The curve also intersects the B-axis at a value corresponding to the residual induction, B_r. You must enter the **Coercive Force** and **Residual Induction** values. (Use a positive value for the **Coercive Force**). A permanent magnet is polarized along an axis of the part. By default, the global coordinate system is used when the material is applied to a part in the Mechanical application. If desired, you can apply a local coordinate system to the part. Align the X-axis of the coordinate system in the direction of the North pole of the magnet. The coordinate system may be Cartesian or cylindrical. A cylindrical system may be used for radially oriented permanent magnets.

Nonlinear "Soft" Magnetic Material

This category characterizes soft materials that exhibit nonlinear behavior between B and H. Select **B-H Curve** to enter nonlinear B-H data. The nonlinear behavior is described by a single B-H curve. You may create a curve by entering B and H data points in Engineering Data, or you may choose from a library of B-H curves for typical properties (p. 27). For material exhibiting orthotropic behavior, you may also select **Relative Permeability (Orthotropic)**. You may elect to apply the B-H curve in any one or all three orthotropic directions, and specify a constant relative permeability in the other directions. If you use the orthotropic option, you can apply a local coordinate system to the part in the Mechanical application instead of using the default, global coordinate system. When creating B-H curves, observe the following guidelines:

a. The curve should be smooth and continuous.

b. Extend the curve well beyond the operating location to accurately capture local high saturation levels. The slope of the curve should asymptotically approach that of free-space permeability. The program extrapolates beyond the end of the curve at a slope equal to free-space permeability if required during the simulation.

c. Group data points around the knee of the curve for better curve-fitting.

d. For best convergence of the simulation, the curve should approach the (0,0) point asymptotically. A new point in the curve near the curve origin may cause convergence problems.

Nonlinear "Hard" Magnetic Material

This category characterizes hard magnetic materials such as permanent magnets. The demagnetization curve of the permanent magnet is described by a series of B-H data points located in the second quadrant. Select **Demagnetization B-H Curve** to enter this data. The first data entry point should be at B = 0, H = -H_c. A permanent magnet is polarized along an axis of the part. In the Mechanical application, you can apply a local coordinate system to the part, instead of using the default, global coordinate system. Align the X-axis of the coordinate system in the direction of the North pole of the magnet. The coordinate system may be Cartesian, cylindrical. A cylindrical system may be used for radially oriented permanent magnets. When creating B-H curves, observe the following guidelines:

a. The curve should be smooth and continuous.

b. The curve may extend into the first quadrant.

c. Group data points around the knee of the curve for better curve-fitting.

Electric

This category defines the electrical material models, including:

- **Anisotropic Relative Permeability**
- **Anisotropic Electric Loss Tangent**
- **Isotropic Resistivity**
- **Orthotropic Resistivity**

Choose your material properties based on how the material exhibits properties in orthogonal directions, either constant in all directions (isotropic behavior) or different in orthogonal directions (orthotropic). By default, the global coordinate system is used when you apply these properties to a part in the Mechanical application. If desired, you can apply a local coordinate system to the part. The material orthogonal properties align with the coordinate system assigned to the part. For orthotropic material properties, all property values must be entered for the model to properly solve.

Piezoelectric

This category contains the following material models.

- **Piezoelectric Matrix**: This model is based on a matrix for tabular data. The model has the following properties:
- **Anisotropic Elastic Loss Tangent**
- **Anisotropic Viscosity**

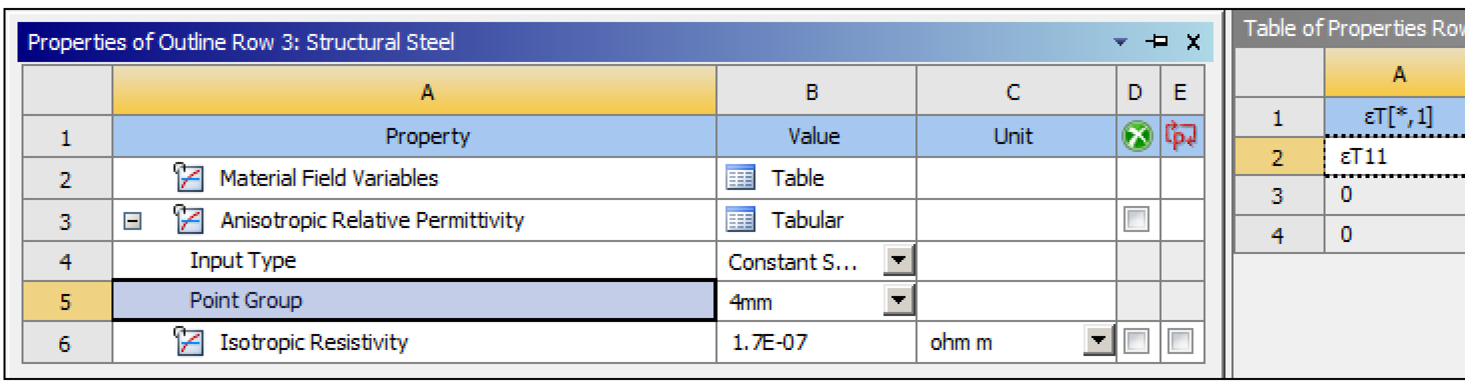
> **Note:**
>
> If you change the **Format** property from **IEEE** to **MAPDL**, or vise versa, the application automatically clears any data you may have entered and does not perform a conversion.

Crystallographic Point Group

Point Group is available for the Anisotropic Elasticity (p. 59), Anisotropic Relative Permeability (p. 66), and Piezoelectric Matrix (p. 66) material models. Using **Point Group**, you can select from the following symmetry operations:

- Unspecified (default)
- 32
- 3m
- 4mm
- 6mm
- 23
- 43mm

Based on your selection, a corresponding table/matrix populates automatically with the appropriate data, as shown here for Anisotropic Relative Permeability using the 4mm option.



Mutually Exclusive Properties

The properties that are mutually exclusive are grouped in the following table.

• Isotropic Elasticity • Orthotropic Elasticity • Mooney-Rivlin • Neo-Hookean • Polynomial • Yeoh • Ogden

• Arruda-Boyce • Gent • Blatz-Ko • Ogden Foam • Extended Tube
• Bilinear Isotropic Hardening • Multilinear Isotropic Hardening • Bilinear Kinematic Hardening • Multilinear Kinematic Hardening
• Relative Permeability • Relative Permeability (Orthotropic) • Coercive Force & Residual Induction • Demagnetization B-H Curve
• Thermal Conductivity Isotropic • Thermal Conductivity Orthotropic
• Isotropic Resistivity • Orthotropic Resistivity
• Orthotropic Seebeck Coefficient • Seebeck Coefficient
• Relative Permeability • B-H Curve • Coercive Force & Residual Induction • Demagnetization B-H Curve
• Mooney-Rivlin • Neo-Hookean • Polynomial • Yeoh

• Ogden • Arruda-Boyce • Gent • Blatz-Ko • Ogden Foam • Extended Tube • Anand Viscoplasticity
• Williams-Landel-Ferry Shift Function • Tool-Narayanaswamy Shift Function • Tool-Narayanaswamy with Fictive Temperature Shift Function
• Superelasticity • Shape Memory Effect
• Isotropic Elasticity • Orthotropic Elasticity • Anisotropic Elasticity • Blatz-Ko • Ogden Foam • Mullins Effect
• Exponential for Interface Delamination • Bilinear for Interface Delamination • Separation-Distance based Debonding • Fracture-Energies based Debonding
• Linear Fracture Criterion • Bilinear Fracture Criterion • B-K Fracture Criterion • Modified B-K Fracture Criterion • Power Law Fracture Criterion

CAD Materials

The materials assigned to parts in a CAD package can be used in Mechanical by selecting the **Material Properties** check box in the Geometry cell Properties pane. The materials assigned to parts in the CAD package is assigned to corresponding parts in the Mechanical application.

CADMaterials Engineering Data Source: When a model is refreshed after importing CAD geometry, a temporary MatML 3.1 file is created, which contains the materials from the CAD package. This file is added as a Data Source in Engineering Data as `CADMaterials` and the file description identifies the system it belongs to. This file is overwritten when a CAD update operation is performed, so the file should not be edited directly.

See the following workflows related to the use of materials from CAD packages.

A. Using CAD Materials

1. Select the **Material Properties** check box in the Geometry cell and import geometry.
2. Refresh the model. The materials assigned to parts in the CAD package is assigned to corresponding parts in the Mechanical application.

> **Note:**
>
> If the part in the CAD package does not have a material assigned, the default material is assigned in Mechanical (see Selecting the Default Material Assignment for Model Parts (p. 28)). This only happens the first time the geometry is attached. If you subsequently update your geometry from the source application, Mechanical, by default, does not assign the default material to new bodies. Review the description for the **Assign Default Material** property in the Geometry object reference section of the *Mechanical User Guide* for additional information.

B. Modifying Material Properties

1. Follow workflow A. Using CAD Materials (p. 72).
2. Add the material from CADMaterials Data Source to Engineering Data and modify it.
3. Refresh the model.

   Mechanical uses the material defined in Engineering Data.

C. Deleting Material in Engineering Data

1. Follow workflow B. Modifying Material Properties (p. 72).
2. Edit the Engineering Data cell and delete the material.
3. Refresh the model.

The default material is assigned to corresponding parts in Mechanical (see Selecting the Default Material Assignment for Model Parts (p. 28)).

D. Updating Material from CAD

1. Follow workflow A. Using CAD Materials (p. 72).
2. Open geometry in the CAD application.
3. Modify material properties in the CAD application and save.
4. Update geometry from CAD.
5. Refresh the model.

   Mechanical uses the modified material from CAD.

E. Materials with Same Name or Synonym

1. Define some materials in Engineering Data.
2. Select the **Material Properties** check box in the Geometry cell and import geometry.
3. Refresh the model.

   If the material in Engineering Data has the same name or synonym (seen in the tooltip) as the material defined in the CAD package, then the Mechanical application uses the material defined in Engineering Data and not from CAD package.

Appendix

This section includes the following:

Appendix A: Material Library File Format

The Engineering Data module follows the MatML 3.1 schema for saving material data to external libraries on disk. For more information, see MatML: A Data Interchange Markup Language. To view an example of the Ansys MatML 3.1 format, follow the directions in Exporting Individual Data (p. 22) to export material data to a file. Then, open the exported file with a text or XML editor.

Appendix B: Custom Material Models

Introduction

Your installation of Workbench provides templates that enable you to customize the Engineering Data Workspace to provide inputs for a user-defined material model.

This appendix describes the files that you will use as templates to create your own property categories and the corresponding property data for your own user-defined material model(s).

Assumptions and Prerequisites

The process of creating user-defined materials requires you to edit installation files. Therefore, it is assumed that you have the necessary skills and experience to perform these tasks. Expertise in material constitutive modeling and software programming is necessary and experience writing XML. Furthermore, creating user-defined material models also requires validation and testing. Ansys, Inc. strongly recommends that you test both the single elements and the multiple elements with various loading conditions to ensure correct results. This feature supports shared memory and distributed parallel processing. However, you are responsible for ensuring that your code can use parallel processing.

In addition, in order to use user-defined material model(s) in Engineering Data, you must first follow the steps to create the corresponding Subroutine in Mechanical APDL. See the Custom Material Models section of the Mechanical APDL Material Reference before continuing.

The files to be customized contain instructional comments to assist you with the modification process. The following sections further elaborate upon and explain the steps for customization.

Material Properties File

Metadata Element

Create Material Models

Customization Files for Engineering Data Workspace The following files are included within your production installation. They enable you to create interface elements within the Engineering Data Workspace.

- **MAPDL_UserMat_Properties.xml**: Defines a user-defined material model's constants to be sent to Mechanical. These can be displayed in the **Property** pane or hidden. Installation location:

  ```
  Drive:\Program Files\ANSYS Inc\version\Addins\EngineeringData\MetaData
  ```

- **MAPDL_UserMat_DefinitionMethods.xml**: Defines the category and names to display for the user-defined material model(s) in the **Toolbox** pane. Installation location:

  ```
  Drive:\Program Files\ANSYS Inc\version\Addins\EngineeringData\MetaData
  ```

- **MAPDL_UserMat_Relationships.xml**: Defines dependencies and mutual exclusions for the custom material model(s). Installation location:

  ```
  Drive:\Program Files\ANSYS Inc\version\Addins\EngineeringData\MetaData
  ```

- **EngineeringData.config**: Defines the files to be used by the product and if your edits are invalid, you can disable the product. Installation location:

  ```
  Drive:\Program Files\ANSYS Inc\version\Addins\EngineeringData\bin\<machine-type>
  ```

These files include instructional comments that guide you during the modification process. Each file is described in subsequent sections.

Important:

ANSYS recommends that you use a text editor that allows you to expand and collapse sections in the XML files for ease of editing.

Procedure

1. Open the following installation directory:

   ```
   Drive:\Program Files\ANSYS Inc\version\Addins\EngineeringData\MetaData
   ```

2. Highlight and copy each of the following files to a new folder. This helps to ensure that the integrity of the original files is maintained.

   - **MAPDL_UserMat_Properties.xml**
   - **MAPDL_UserMat_DefinitionMethods.xml**
   - **MAPDL_UserMat_Relationships.xml**

3. Rename the new files by replacing “MAPDL” with your company’s name. For example:
   - **MyCompany**_UserMat_Properties.xml
   - **MyCompany**_UserMat_DefinitionMethods.xml
   - **MyCompany**_UserMat_Relationships.xml

Once copied to a new folder and renamed, you can begin editing the files.

> **Important:**
>
> We recommend that you use a text editor that allows you to expand and collapse sections in the XML files for ease of editing.

Once you feel that your edits are complete and accurate, you need to modify the **EngineeringData.config** file. This is a critical program file.

> **Warning:**
>
> If you introduce an error into EngineeringData.config file, you may disable the product. Therefore, it is critical that you have a backup copy of the file before performing edits to it.

1. Open the following installation directory:

```
Drive:\Program Files\ANSYS Inc\version\Addins\EngineeringData\bin\<machine-type>
```

2. Make a copy of **EngineeringData.config** before editing.
3. Open the file in any text editor and search for "UserMat" and duplicate the lines related to "UserMat" below the existing lines.
4. Replace the filename with your filename and then change enabled to "true" as highlighted below.
5. Make sure that you have copied your files to the installation folder.

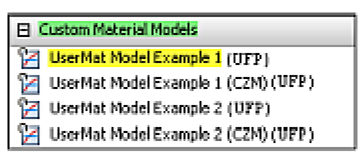
```
  <MetaFile enabled="true" filename="MyComany_UserMat_Properties.xml">
    Metadata file containing properties, propertyDatas, and variables known to be used for MAPDL UserMat.
  </MetaFile>
</MetaData>
<DefinitionMethods>
  <DefinitionMethod enabled="true" filename="ANSYS_StructuralMaterialDefinitionMethods.xml" />
  <DefinitionMethod enabled="true" filename="ANSYS_ThermalMaterialDefinitionMethods.xml" />
  <DefinitionMethod enabled="true" filename="ANSYS_ElectricMaterialDefinitionMethods.xml" />
  <DefinitionMethod enabled="true" filename="ANSYS_MagneticMaterialDefinitionMethods.xml" />
  <DefinitionMethod enabled="true" filename="AUTODYN_MaterialDefinitionMethods.xml" />
  <DefinitionMethod enabled="true" filename="SAMCEF_MaterialDefinitionMethods.xml" />
  <DefinitionMethod enabled="true" filename="ABAQUS_MaterialDefinitionMethods.xml" />
  <DefinitionMethod enabled="true" filename="FLUID_MaterialDefinitionMethods.xml" />
  <DefinitionMethod enabled="true" filename="ACP_MaterialDefinitionMethods.xml" />
  <DefinitionMethod enabled="false" filename="nCode_MaterialDefinitionMethods.xml" />
  <DefinitionMethod enabled="true" filename="MyComany_UserMat_DefinitionMethods.xml" />
</DefinitionMethods>
<Relationships>
  <Relationship enabled="true" filename="ANSYS_Relationships.xml" />
  <Relationship enabled="true" filename="AUTODYN_Relationships.xml" />
  <Relationship enabled="true" filename="LSDYNA_Relationships.xml" />
  <Relationship enabled="true" filename="SAMCEF_Relationships.xml" />
  <Relationship enabled="true" filename="ABAQUS_Relationships.xml" />
  <Relationship enabled="true" filename="NASTRAN_Relationships.xml" />
  <Relationship enabled="true" filename="FLUID_Relationships.xml" />
  <Relationship enabled="true" filename="ACP_Relationships.xml" />
  <Relationship enabled="false" filename="nCode_Relationships.xml" />
  <Relationship enabled="true" filename="MyComany_UserMat_Relationships.xml" />
</Relationships>
```

6. Open the Engineering Data Workspace and select the **Filter Engineering Data** button. The **Toolbox** should display your material model(s).

Material Definition Methods File

The content of the User Material Definition Methods file, *MyCompany_DefinitionMethods.xml*, is illustrated below. This file defines the category and property names that are displayed in the Engineering Data Toolbox.

```
<Notes>
  This file is used to identify and group the definition methods for material properties. The user
  will be presented with this as a searchable tree view.
</Notes>
<MaterialPropertyDefinitionCategories target="ANSYS">
  <!--  The physicsFilter can be changed, see other files for applicable string -->
  <PhysicsCategory physicsFilter="Structural" name="Structural">
    <!--  The name can be customized-->
    <PropertyCategory name="Custom Material Models">
      <Property name="UserMat Model Example 1"/>
      <Property name="UserMat Model Example 1 (CZM)"/>
      <Property name="UserMat Model Example 2"/>
      <Property name="UserMat Model Example 2 (CZM)"/>
    </PropertyCategory>
  </PhysicsCategory>
</MaterialPropertyDefinitionCategories>
```



As shown above, the **<PropertyCategory>** element defines the Engineering Data Toolbox category and the **<Property>** element defines a property contained in the category.

As illustrated, you can have more than one property. In addition, this is where you also create additional property categories. You can copy and paste these lines to help you create additional properties and property categories.

Note:

- The string for the name attribute must be the same as that used in the *MyCompany_UserMat_Properties.xml* file for the name of the user-defined material model to be valid. An example of this entry is shown below.

```
<ParameterValue parameter="groupPropertyData"  format="string">
  <Data>UserMat Model Example 1 </Data>
</ParameterValue>
```

- The text "(UPF)" is appended to the string by Engineering Data to clearly distinguish user-defined material models from those provided by ANSYS.
- Make sure that your tags are properly formatted with beginning ("<") and ending ("/>") characters.

Material Relationships File

The content of the User Material Relationships file, *MyCompany_UserMat_Relationships.xml*, is illustrated below. Your material model may require specific material data, such as Density in order for it to be computed. This file defines those types of dependencies and also defines the mutual exclusions for your material model(s).

```
<Notes>
  This information defines the material models that a particular solver target can
  utilize.  It also defineds the rules for mutually exclusive logic.
</Notes>
<PropertyDependencies>
  <!--  The UserMat requires that Density be specified with it.-->
  <Property name="UserMat Model Example 1">
    <Property name="Density"/>
  </Property>
  <Property name="UserMat Model Example 1 (CZM)">
    <Property name="Density"/>
  </Property>
  <Property name="UserMat Model Example 2">
    <Property name="Density"/>
  </Property>
  <Property name="UserMat Model Example 2 (CZM)">
    <Property name="Density"/>
  </Property>
</PropertyDependencies>
<MutualExclusions>
  <MutuallyExclusive>
    <ModelData>Elasticity</ModelData>
    <ModelData>Mooney-Rivlin</ModelData>
    <ModelData>Neo-Hookean</ModelData>
    <ModelData>Arruda-Boyce</ModelData>
    <ModelData>Gent</ModelData>
    <ModelData>Blatz-Ko</ModelData>
    <ModelData>Polynomial</ModelData>
    <ModelData>Yeoh</ModelData>
    <ModelData>Ogden</ModelData>
    <ModelData>Response Function</ModelData>
    <ModelData>Ogden Foam</ModelData>
    <ModelData>Extended Tube</ModelData>
    <ModelData>Exponential Strain Energy Potential</ModelData>
    <ModelData>UserMat Model Example 1</ModelData>
  </MutuallyExclusive>
</MutualExclusions>
```

The **PropertyDependencies** element provides all material property dependencies. A given property is specified which then contains those properties which it is dependent upon.

Note:

- The string for the name attribute must be the same as that used in the *MyCompany_UserMat_Properties.xml* file for the name of the user-defined material model to be valid. An example of this entry is shown below.

```
<ParameterValue parameter="groupPropertyData"  format="string">
  <Data>UserMat Model Example 1 </Data>
</ParameterValue>
```

- Make sure that your tags are properly formatted with beginning ("<") and ending ("/>") characters.

Mutual Exclusions

A user-defined material model may not be able to be used with other material data (such as isotropic elasticity) because the user-defined material model provides the needed computation (such as elasticity). The relationships file enables you to specify this information so that when a user would add the user-defined material model to the material the other material properties or models will be suppressed automatically if in the material.

Any mutual exclusion to other material models is defined within the **MutualExclusions** element. You use the subelements (**ModelData**) to specify those models which cannot be used together. There can be more than one **MutuallyExclusive** element specified. Add the **ModelData** line multiple times with a string as a property name which cannot be used with the user-defined material model.

Material Properties File

The *MyCompany_UserMat_Properties.xml* file contains the specific data that is sent to Mechanical. Within the file, the Bulk Details element houses the data elements.

The illustration shown below displays the **BulkDetails** element in a collapsed view. The additional elements within **BulkDetails** are the elements that you will make changes to.

```
<BulkDetails>
  <!--  The contents of the Name element cannot be modified. -->
  <Name>ANSYS Engineering Data Property Definition</Name>
  <!--
  EXAMPLE - Model with State Variables hidden
  -->
  <!--The following PropertyData will create a property with the name from the Metadata (UserMat Model Example 1)
      and define three variables.   The UserMat Qualifier must be defined for this PropertyData to be
      written as TB,USER. -->
  <PropertyData property="prUSER" minOccurrences="1">
  <PropertyData property="prSTATE" minOccurrences="1">
  <PropertyData property="prUSER" minOccurrences="1">
  <!--
  EXAMPLE - Temperature Dependent Model with State Variables mixed display
  -->
  <!--The following PropertyData will create a property with the name from the Metadata (UserMat Model Example 2)
  <PropertyData property="prUSER" minOccurrences="1">
  <!--The following PropertyData will create a property with the name from the Metadata (UserMat Model Example 2)
  <PropertyData property="prSTATE" minOccurrences="1">
  <!--The following PropertyData will create a property with the name from the Metadata (UserMat Model Example 2 (CZM))
  <PropertyData property="prUSER" minOccurrences="1">
</BulkDetails>
```

Within the **BulkDetails** element, the **PropertyData** element defines the input data for each user-defined material model.

```
<!--
EXAMPLE - Model with State Variables hidden
-->
<!--The following PropertyData will create a property with the name from the Metadata (UserMat Model Example 1)
    and define three variables.   The UserMat Qualifier must be defined for this PropertyData to be
    written as TB,USER. -->
<PropertyData property="prUSER" delimiter=";" minOccurrences="1">
  <Data format="string">-</Data>
  <Qualifier name="UserMat">USER</Qualifier>
  <!-- The following ParameterValue allows for temperature dependence for the model.  To enable change
       the maxEntries to floor(1000/NPTS). -->
  <ParameterValue parameter="pa1" format="float" minEntries="0" maxEntries="2">
    <Data>7.888609052210118054117285652827 9e-31</Data>
    <Qualifier name="Variable Type">Independent</Qualifier>
  </ParameterValue>
  <!--  The following ParameterValue specifies the value to be used as the switch for multiple
        models defined in a UserMat subroutine in this case it is model 1.  The user doesn't need
        to see this so it is not displayed (Display).  Also specified is that this is the first constant
        "C1" for the TBDATA command (UserMat Constant).-->
  <ParameterValue parameter="paC1" format="float" lowerLimit=">0.0">
    <Data>1</Data>
    <Qualifier name="Display">False</Qualifier>
    <Qualifier name="UserMat Constant">1</Qualifier>
    <Qualifier name="Variable Type">Dependent</Qualifier>
  </ParameterValue>
  <!--  The following ParameterValue specifies that this is the second constant "C2"
        for the TBDATA command (UserMat Constant).-->
  <ParameterValue parameter="paC2" format="float" lowerLimit=">0.0">
  <!--  The following ParameterValue specifies that this is the third constant "C3"
        for the TBDATA command (UserMat Constant).-->
  <ParameterValue parameter="paC3" format="float" lowerLimit=">0.0">
  <ParameterValue parameter="groupPropertyData" format="string">
    <Data>UserMat Model Example 1</Data>
  </ParameterValue>
</PropertyData>
```

The **PropertyData** element requires the property attribute (property="prUSER") which is a string data type that corresponds to an id attribute in the **Metadata** element within the properties files. You may use any unique string as the identification for a given **Property**. The **PropertyData** element also requires the **minOccurrences** attribute (minOccurrences="1") attribute. You use the **ParameterValue** element within **PropertyData** to define the data for a user-defined material model.

<Data format>=string</Data>

You must include this line as shown for compatibility requirements.

<Qualifier name="UserMat">string</Qualifier>

Identifies this **PropertyData** as being defined in UserMat. The string should be specified as "USER" to define inputs for this user-defined material model (**TB**, USER) or "STATE" to define state variables for this user-defined material model (**TB**, STATE) or a material model (e.g. CZM) which supports USER as a valid TBOPT field (such as **TB**,CZM,,,,USER).

<Qualifier name="Display">False</Qualifier>

This Boolean allows the **PropertyData** to be hidden (False) in the GUI but the data is sent to Mechanical.

The individual field data and behaviors for a user-defined material model are defined within the **ParameterValue** element. The following attributes are applicable for this element.

Attribute	Value	Description
parameter	"string"	The string corresponds to the id attribute in a **ParametersDetails** element of the **Metadata** element. You may use any unique string as the identification for a given Parameter (user-defined material model field).
format	"string"	This attribute is required for compatibility requirements and the string value must equal "float".
lowerLimit	"string"	This string is the lower limit of valid data. It is possible to use ">0.0" for a positive non-zero value. The default is negative the maximum double numerical representation.
upperLimit	"string"	The string is the upper limit of valid data. It is possible to use "<0.0" for a negative non-zero value. The default is the maximum double numerical representation.

```
<!--
EXAMPLE - Model with State Variables hidden
-->
<!--The following PropertyData will create a property with the name from the Metadata (UserMat Model Example 1)
    and define three variables.   The UserMat Qualifier must be defined for this PropertyData to be
    written as TB,USER. -->
<PropertyData property='prUSER' delimiter=";" minOccurrences='1'>
  <Data format='string'>-</Data>
  <Qualifier name="UserMat">USER</Qualifier>
  <!-- The following ParameterValue allows for temperature dependence for the model.  To enable change
       the maxEntries to floor(1000/NPTS). -->
  <ParameterValue parameter='pa1' format="float" minEntries="0" maxEntries="2">
    <Data>7.8886090522101180541172856528279e-31</Data>
    <Qualifier name='Variable Type">Independent</Qualifier>
  </ParameterValue>
  <!--  The following ParameterValue specifies the value to be used as the switch for multiple
        models defined in a UserMat subroutine in this case it is model 1.  The user doesn't need
        to see this so it is not displayed (Display).  Also specified is that this is the first constant
        "C1" for the TBDATA command (UserMat Constant).-->
  <ParameterValue parameter='paC1' format="float' lowerLimit=">0.0">
    <Data>1</Data>
    <Qualifier name='Display'>False</Qualifier>
    <Qualifier name='UserMat Constant">1</Qualifier>
    <Qualifier name='Variable Type">Dependent</Qualifier>
  </ParameterValue>
  <!--  The following ParameterValue specifies that this is the second constant "C2"
        for the TBDATA command (UserMat Constant).-->
  <ParameterValue parameter='paC2' format="float' lowerLimit=">0.0">
  <!--  The following ParameterValue specifies that this is the third constant "C3"
        for the TBDATA command (UserMat Constant).-->
  <ParameterValue parameter='paC3' format="float' lowerLimit=">0.0">
  <ParameterValue parameter='groupPropertyData" format='string">
    <Data>UserMat Model Example 1</Data>
  </ParameterValue>
</PropertyData>
```

Further defining **ParameterValue** content includes the following:

<Data=string</Data>

The string is the default value for the parameter to send to Mechanical. If the desire is to have a blank field in the GUI to indicate a value should be entered (shown in yellow), use the value included in the example, the result of pow(2,-100).

<Qualifier name="UserMat Constant">string</Qualifier>

The string is an integer value which specifies this parameter value's position in **TBDATA** command. For example, the user input value of parameter "paC3" will be placed at the fifth position in **TB-DATA** command.

```
<ParameterValue parameter="paC3" format="float" lowerLimit=">0.0">
  <Data>7.88860905221011805411728565282790e-31</Data>
  <Qualifier name="UserMat Constant">5</Qualifier>
  <Qualifier name="Variable Type">Dependent</Qualifier>
</ParameterValue>
```

<Qualifier name="Variable Type">string</Qualifier>

This string should be specified as **Dependent** for a dependent parameter and as **Independent** for an independent parameter (such as Temperature).

Temperature Dependent Parameter Value

When the data can vary with temperature then you need to include a parameter (in the example, **ParameterValue** parameter="pa1") as the first element of the **PropertyData** element. You need to also include the following two attributes to define the number of temperature points for which data will be provided.

minEntries

minEntries="string": The string is an integer value. A value of **"0"** allows the user to enter data without any temperature points.

maxEntries

maxEntries="string": The string is an integer value. For example, a value of **"2"** allows up to a maximum of two temperature data points to be specified. The maximum entries allowed by Mechanical APDL is limited by the following calculation:

maxEntries <= 1000 / (number of user-defined material model inputs)

```
<ParameterValue parameter="pa1" format="float" minEntries="0" maxEntries="2">
  <Data>7.88860905221011805411728565282790e-31</Data>
  <Qualifier name="Variable Type">Independent</Qualifier>
</ParameterValue>
```

For the example content, the parameter "pa1" is defined in the **Metadata** table as **Temperature**. You can use an alternate string if desired (such as "temp").

STATE Dependent Parameter Value

If your material model also needs **STATE** data to complete the inputs, include the **groupPropertyData** parameter as the last element in the **PropertyData** element. In the example illustrated below, placing this element in more than one **PropertyData** element will group these **PropertyData** under a single property "UserMat Model Example 1" in the GUI.

```
<ParameterValue parameter="groupPropertyData" format="string">
  <Data>UserMat Model Example 1</Data>
</ParameterValue>
```

The parameter **groupPropertyData** is defined in the **Metadata** table with the name Material Property which facilitates this behavior. You can use an alternate string if desired (such as "group").

Metadata Element

The details for the attributes used on **PropertyData** (property="") and **ParameterValue** (parameter="") are contained within **Metadata** element. **Metadata** content is illustrated below.

```
<Metadata>
  <!--
  ### Parameter Details
  -->
  <ParameterDetails id="pa1">
    <!-- If a parameter is not Unitless, it means the user can use the unit for the value.
    <Name>Temperature</Name>
    <Units name="Temperature">
      <Unit>
        <Name>C</Name>
      </Unit>
    </Units>
  </ParameterDetails>
  <ParameterDetails id="paC1">
  <ParameterDetails id="paC2">
  <ParameterDetails id="paC3">
  <ParameterDetails id="paC4">
  <!--
  <ParameterDetails id="paSV1">
  <ParameterDetails id="paSV2">
  <!--  Key to group PropertyData into a single Property -->
  <ParameterDetails id="groupPropertyData">
  <!--
  <PropertyDetails id="prUSER">
    <Name>Model Coefficients</Name>
    <Unitless/>
  </PropertyDetails>
  <PropertyDetails id="prSTATE">
</Metadata>
```

Parameter Details

The string used by **ParameterDetails** is a unique identification for the parameter to be used in conjunction with a **ParmeterValue**'s attribute parameter.

Name

<Name>"string"</Name>: This string to be displayed in the GUI as the name of the parameter.

```
<ParameterDetails id="pa1">
  <!-- If a parameter is not Unitless, it means
  the user can use the unit for the value.-->
  <Name>Temperature</Name>
  <Units name="Temperature">
    <Unit>
      <Name>C</Name>
    </Unit>
  </Units>
</ParameterDetails>
```

Units

The Units element defines the name of the physical quantity for which defined units are available. For example, units of a parameter which has Stress units (Pa) can be defined as shown below. Use **Unitless** for parameters without units as shown above.

```
<Units name="Stress">
  <Unit>
    <Name>Pa</Name>
  </Unit>
</Units>
```

Caution:

Use extreme caution if you are choosing the unit of measure. This value is converted to the Unit System specified for the solution and if you do not enter a supported unit, you could produce incorrect data for the solution.

Property Details

The only **PropertyDetails** attribute is id="string". The string is a unique identification for the property (prUSER) to be used in conjunction with a **PropertyData**'s attribute property. Following elements are used to further **specify PropertyDetails**.

Name

<Name>"string"</Name>: The string to be displayed in the GUI as the name of the user-defined material model.

Units

This is required for compatibility and should not be modified.

```
<PropertyDetails id="prUSER">
  <Name>Model Coefficients</Name>
  <Unitless/>
</PropertyDetails>
```

Index

S

T

U

W