

MATERIALS

Material Types

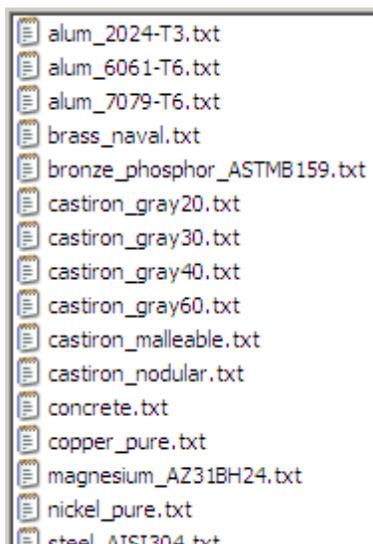
There are two basic material types in FEMdesigner AD, **mechanical** and **thermal**, which are used in mechanical and thermal loadcases, respectively. As the names suggest, the mechanical material type contains the mechanical properties of a material, including the elastic or Young's modulus, Poisson's ratio, density, and others. The thermal material type contains the thermal properties of a material, which include conductivity, heat capacity, and others. For mechanical loadcases, only mechanical materials can be assigned, and for thermal loadcases, only thermal materials are allowed.

Sample Materials

FEMdesigner AD is installed with some sample mechanical and thermal materials for use in analyses. These materials are for demonstration purposes only, and are not intended to accurately represent your engineering materials.

IMPORTANT: SAMPLE MATERIALS SUPPLIED WITH FEMDESIGNER ARE FOR REFERENCE ONLY AND CONTAIN GENERALIZED VALUES THAT ARE NOT INTENDED FOR DESIGN VALIDATION. THE USER IS RESPONSIBLE TO INSURE THAT ALL MATERIAL PROPERTY VALUES BEING USED FOR ANALYSIS ARE ACCURATE FOR THEIR PURPOSES.

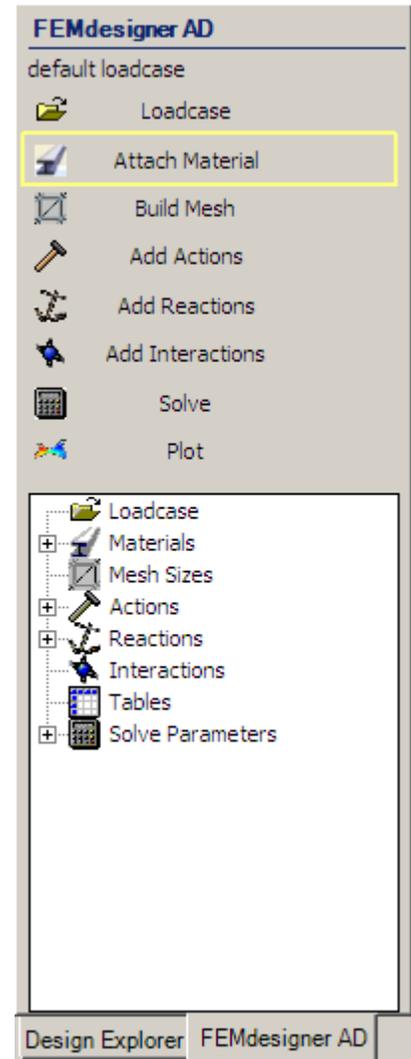
The user should always check material property values and edit the sample materials or create their own materials to accurately represent the materials that will be used to fabricate the parts and assemblies that they are analyzing.



Material Files

Material properties are stored in plain text files (*.txt), one file for each material, with the material property information organized inside the file in a simple format. In order for a material to be displayed in the selectable list of standard materials inside FEMdesigner, the material file must be stored in the default location according to its material type. For mechanical materials, this location is C:\Program Files\FEMdesigner\FemDes\Materials\Mechanical, and correspondingly, FEMdesigner AD thermal materials are stored in C:\...FemDes\Materials\Thermal. A Windows explorer listing of some sample mechanical materials that are installed with FEMdesigner AD is shown at left. Material files can also be stored in other directories of the user's choice and be selected for use in an analysis by navigating to that directory and selecting the desired material file name, as explained in the **Materials Dialog Box** section below.

FEMdesigner AD Explorer



Inside the Material File

In the material text file, material properties are listed in a specific order, using a specific format, as shown here in the examples. The user can edit the material files installed with FEMdesigner AD, or create custom materials by following this format.

The first line of each material file must specify length units, and all properties in that file must be expressed with respect to those length units. To illustrate matching length and material property units,

groups of matching units are color coded in the material file schematic above.

Mechanical Material File Schematic

material units [m, mm, in]	;<Description> (required for display on model)
E [Pa, MPa, psi]	;Young's modulus for isotropic materials, or for direction 1 in orthotropic materials
nu [no units]	;Poisson's ratio
CTE [1/°C, 1/°C, 1/°F]	;Coefficient of thermal expansion
rho [kg/m ³ , kg/mm ³ , lbf/in ³]	;Density
T [m, mm, in]	;Plate thickness (for future use)
E2 [Pa, MPa, psi]	;Young's modulus in direction 2 for orthotropic
G2 [Pa, MPa, psi]	;Shear modulus between directions 1 and 2
phi [degrees]	;Angle between direction 1 and global x-axis
CTE2 [1/°C, 1/°C, 1/°F]	;CTE in direction 2
Y [Pa, MPa, psi]	;Yield stress, tension
UTS [Pa, MPa, psi]	;Ultimate tensile strength
L [Pa, MPa, psi]	;Endurance limit (for future use)
C [Pa, MPa, psi]	;Yield stress, compression

Mechanical Materials

In the material file schematic shown above right, the length units specified [in brackets] on line 1, determine which of the respective material property units listed [in brackets] below are to be used.

For example, if length units of meters (m) are specified, the elastic (or Young's) modulus must be specified in Pascals (Newtons per square meter). Similarly, with a length unit of inches (in), this modulus must be expressed in pounds force per square inch (psi).

Example: Aluminum alloy 7075, T6 temper

in	;AL 7075-T6 (required Description)
10400000	;Young's modulus (psi)
0.33	;Poisson's ratio
0.000013	;Coefficient of thermal expansion (in/in°F)
0.102	;Density (lbf/in ³)
0	;Plate thickness (for future use)
0	;E2 (Young's modulus in direction 2, psi)
0	;G2 (shear modulus in direction 2, psi)
0	;phi (angle, direction 1 to x-axis, degrees)
0	;CTE2 (CTE in direction 2, in/in°F)
73000	;Yield stress (psi, tension)
83000	;Ultimate tensile strength (psi)
0	;L (endurance limit, psi – for future use)
0	;Yield stress (psi, compression)

At left, we have the text from a material file that represents the 7075 alloy of aluminum, with a T6 temper. The first line specifies a length unit of inches (in), and all following material properties are expressed in appropriate units.

TIP: Scientific Notation

Scientific notation is allowed when specifying material properties, e.g. the Young's modulus in the example at left could also be expressed as 1.04e7, and the CTE as 1.3e-5.

On any line of the material file, text after a semicolon (;) is considered comment text and is ignored by FEMdesigner AD, except for line 1, where it is required as the material Description,

and will be the visible material label when attached to a part. Comment text is not a required part of the file, but comments can be very informative for explaining the numerical values in the file, so we highly recommend that all user-generated material files contain these descriptive comments.

IMPORTANT: the zero values shown in the material files are required. If you do not know the value of a material property, it is necessary to put a zero on that line of the text file. If not, FEMdesigner AD will use the next numerical value in the file as the value of the unknown material property. Also, even though for a linear static analysis only the Young's modulus and Poisson's ratio are required, it is strongly recommended to include as many accurate material property values as possible in the material file to avoid erroneous analysis results due to insufficient material data.

The properties E2, G2, Phi, and CTE2 shown in the material file are used to specify orthotropic material properties. Please see the diagram below in the **Non-linear Materials** section for a graphical representation of orthotropic material properties and their orientation.

Thermal Materials

For thermal materials, the format is very similar to mechanical materials, with different material properties being specified. The schematic representation of a thermal material file at right shows the different material properties and the units used to represent them, just as in the mechanical material file schematic. In the example given of a naval brass alloy, our length units are specified as millimeters ("mm" in line 1), so all of our material properties that contain length units are expressed in millimeters also.

Thermal Material File Schematic

material units [m, mm, in]	;<Description> (required for display)
kx [W/m·°C, W/mm·°C, BTU/s-in·°F]	;Isotropic conductivity, or in global X direction if ky and/or kz are non-zero
ky [W/m·°C, W/mm·°C, BTU/s-in·°F]	;Conductivity in global Y direction
kz [W/m·°C, W/mm·°C, BTU/s-in·°F]	;Conductivity in global Z direction
rho [kg/m ³ , kg/mm ³ , lbf/in ³]	;Density
c [J/kg·°C, J/kg·°C, BTU/lb·°F]	;Specific heat capacity

User-defined Materials

Using the examples and material file schematics above, the user can create custom mechanical and thermal materials. Simply open an existing material text file and edit the values inside the file, or create a plain text file (*.txt) in the format provided, making sure that material property values are unit-consistent (as specified in the schematics), and save the file in the appropriate directory (mechanical or thermal, depending on material type), with an appropriate name. This enables the user to create and use their own materials, including non-standard and proprietary materials, in any analysis. As mentioned above, these custom material files can also be saved in locations other than the default locations, and can be accessed for analysis through the navigation mechanism provided in the Materials dialog box.

Example: Naval brass

mm	;Brass, naval (required Description)
116e-3	;kx, isotropic conductivity (W/mm·°C)
0	;ky (zero value...not used)
0	;kz (zero value...not used)
8407e-9	;rho, density (kg/mm ³)
377	;c, specific heat capacity (J/kg·°C)

Another method for creating a user-defined material is to enter the material properties directly into the Materials dialog box (see **"Materials Dialog Box"** below). However, after being assigned to a part, the material created will be saved only with the currently active loadcase, making it a custom material for that loadcase only. It will not create a material text file for future use.

TIP: Naming Material Files

When creating a custom material file, we recommend giving the file a name that identifies the specific material in the file, because when the user selects materials to attach to parts in FEMdesigner AD, the file name is displayed in the list of selectable materials.

Non-linear Materials

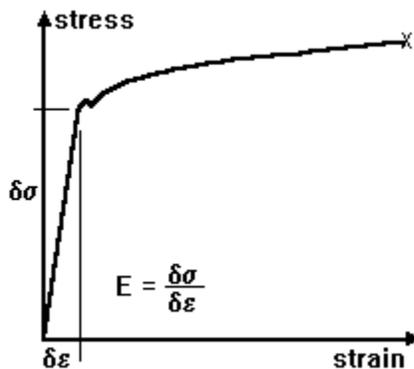
Non-linear materials are represented in several different ways in FEMdesigner AD:

- **Bi-linear** elastic-perfectly plastic materials based on the elastic (Young's) modulus and yield stress
- **Orthotropic mechanical** materials with different material properties in orthogonal directions, at a specified angle to the global X-axis
- **Orthotropic thermal** materials with different conductivities in the global X, Y and Z directions

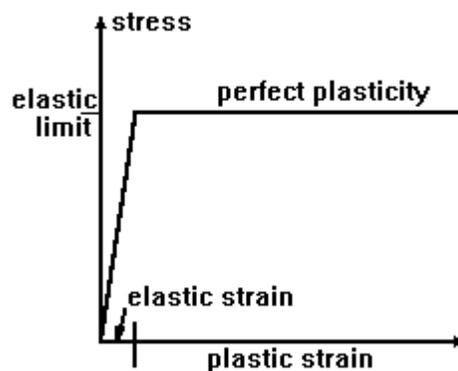
Bi-linear Materials

FEMdesigner AD models plastic deformation with bi-linear **elastic-perfectly plastic materials**, which if represented on a stress-strain curve, would be two straight line segments joined together, one segment representing the elastic portion of the strain, and the other representing perfectly plastic behavior beginning at yield stress. Through independent testing, we have found that even though this method does not explicitly account for strain hardening effects and other non-linearities that occur during actual plastic deformation, FEMdesigner's calculations of plastic deformation behavior in metals is quite accurate when an appropriate yield stress is used, such as a 0.2% offset yield. We can compare the FEMdesigner plastic deformation model to a real world stress-strain curve to see how they differ:

Typical "real-world" Stress-Strain Curve



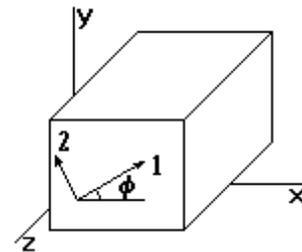
FEMdesigner Idealized Stress-Strain Curve



Orthotropic Materials

FEMdesigner AD can also model **orthotropic mechanical materials** having different properties in two perpendicular directions, as might typically be found in composite materials. In the material file, we specify the material properties in these two orthogonal directions (direction 1 and direction 2), and also the angle ϕ between direction 1 and the global X-axis. Schematically, the diagram at right shows the orientation of the material properties to the global coordinate system. Oftentimes the part being analyzed is oriented to align with the global axes and the angle ϕ is zero, but we provide a method to specify these properties if they are not aligned with the coordinate axes. This does

require, however, that to properly calculate results for parts with orthotropic materials, the user must orient the part properly: FEMdesigner AD assumes that the material property directions 1 and 2 are perpendicular, and the plane described by those two axes is parallel to the global XY plane.



For modeling **orthotropic thermal materials**, different conductivities in the global X, Y, and Z directions can be specified in the thermal material file, as shown above. In this case, the different conductivities must be aligned with the global axes for accurate results – there is no adjustment mechanism as there is for orthotropic mechanical materials. Referring to the “Thermal Material File Schematic” above, if **ky** = 0 and **kz** = 0, then **kx** is used as the isotropic conductivity throughout the material. If a non-zero value is present for **ky** or **kz**, that value will be used as the thermal conductivity in that direction. In other words, **kx** will be used as the thermal conductivity in any direction (Y or Z) that has a zero value.

Material/Model Unit Conversion

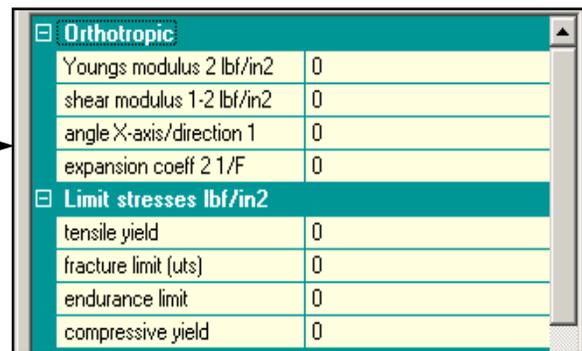
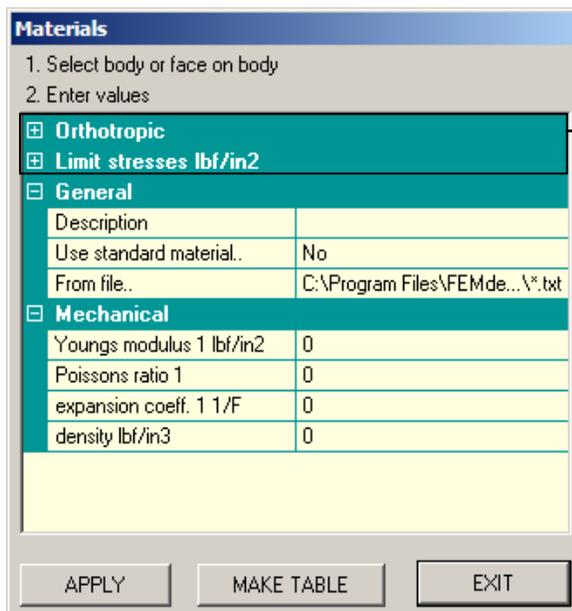
If the Alibre model units are different than the length units specified in the FEMdesigner material file, FEMdesigner will automatically convert the material properties into units that match the model. When a material is selected, the material properties are converted and displayed in current model units in the Materials dialog box (see below) *before* being applied, so that an accurate conversion can be confirmed. This means that for any material, the material properties need only be specified in one file, in units that are consistent within that material file, and those properties will be automatically converted for any models with different units. Multiple material files with different units for the same material are not necessary. This is true for both mechanical and thermal materials.

Materials Dialog Box

When the “Attach Material” menu is selected (red box at right), the Materials dialog box is displayed (below left). Depending on the type of loadcase that is currently active, different material property fields will be displayed. For a mechanical loadcase, mechanical material properties will be displayed, and for a thermal loadcase, thermal material properties will be displayed.



Materials Dialog Box (mechanical)



Expanded View (mechanical) – Orthotropic, Limit Stresses

The dialog box is divided into several expandable and contractible data areas (using the +/- signs) that display material property data. The information displayed in these areas corresponds to the information in material property files, and can be entered either directly via keyboard input, or can be auto-filled by FEMdesigner AD when the user selects a material. The material properties, when

attached to a part, are saved with the active loadcase, so multiple loadcases with different materials can be created and saved for a single part. Mechanical and thermal materials will display different data areas:

Mechanical

- Orthotropic
- Limit Stresses
- General
- Mechanical

Thermal

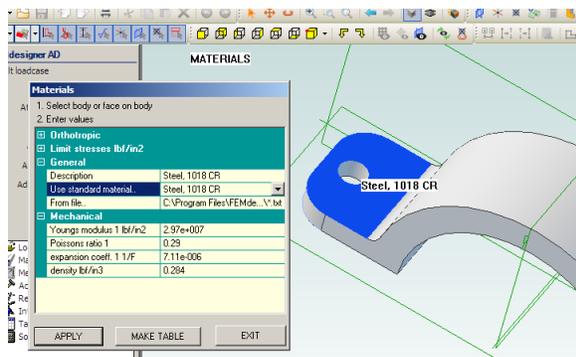
- General
- Thermal

Applying Materials

To apply a material to a part, the data areas must be filled in with the values of the material properties. This can be done by manually entering values into those data fields, or by selecting a material file, which will automatically fill in the data fields. The user can also select a material file as a starting point and edit the values that were imported from the file.

After the data fields are filled in, the material can be applied to a part by selecting a face on the part and then selecting the “APPLY” button in the dialog box.

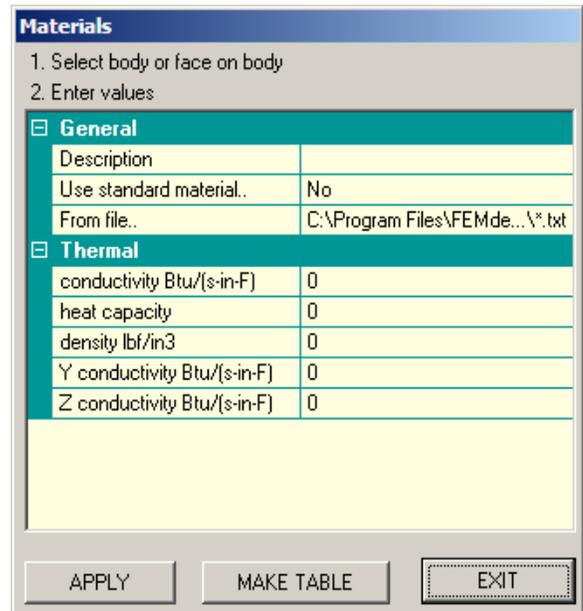
When a material is applied to a part, the “Description” data field will be displayed at the center of the face selected:



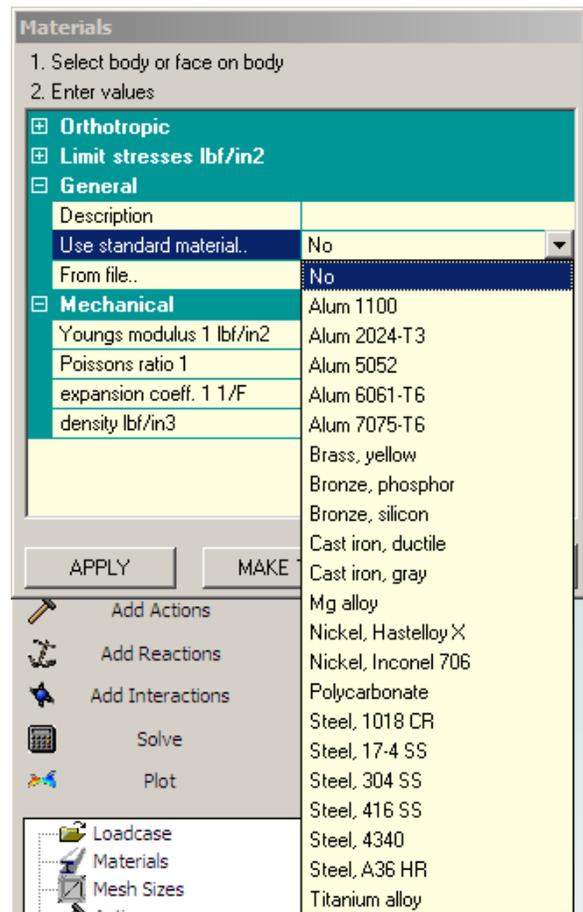
Selecting Materials

The simplest way to apply a material is to select a standard material from the “Use standard material..” drop-down list, shown at right. The list of materials displayed is created from the file names of the material files (*.txt) stored in the default material file directory (see the **Material Files** section above). User-defined custom material files stored in this default directory will also appear in this list, so it is important that custom material files are named to indicate the material they represent.

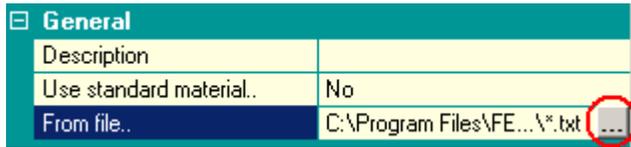
Materials Dialog Box (thermal)



Material Selection Drop-down List



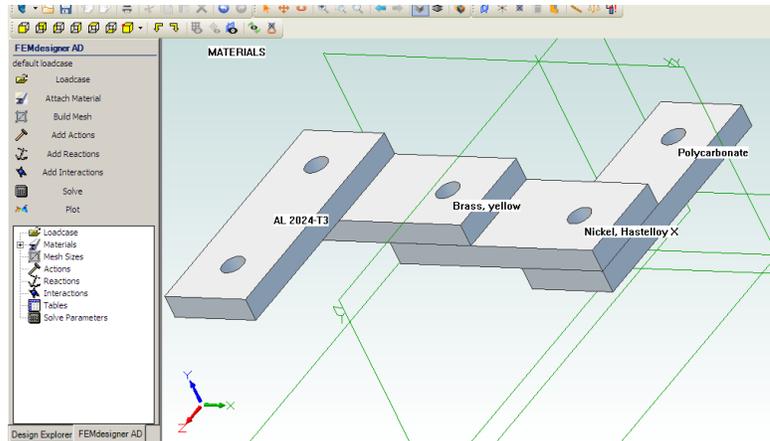
To select custom material files that are not stored in the default directories, those materials can be selected by searching the file system via the “From file..” search box – click the field to the right of “From file..” and then select the ellipsis (three dots), shown here circled in red. This will open a standard Windows file system search box to find the desired material file.



Displaying Assigned Materials

To see what materials have been applied to parts in the current loadcase, simply click the “Attach Material” menu to launch the Materials dialog box, as described at the beginning of this section. If any materials have been applied to parts, the material Description will be displayed directly on the part, at the center of the face to which it was originally applied. As you can see in the example, every part in an assembly can have a different material applied to it.

Loadcase Materials Displayed on Screen



Editing Loadcase Materials

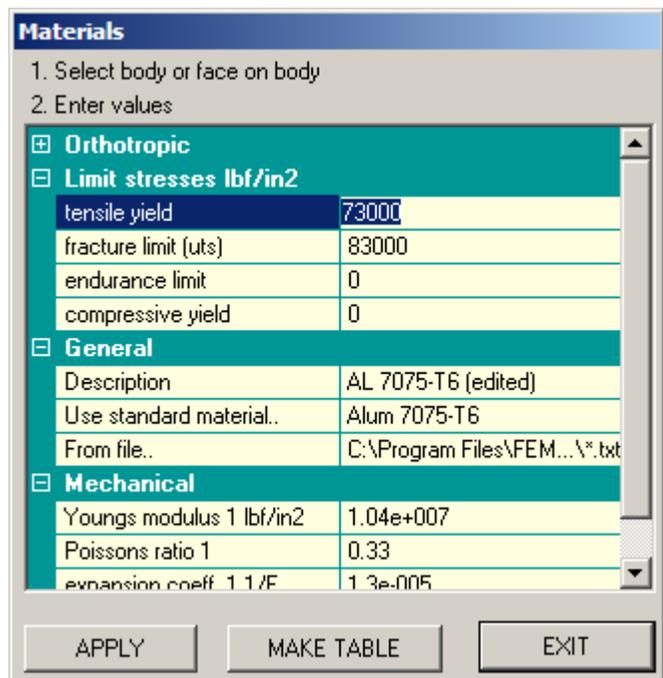
As briefly mentioned above, materials used in a loadcase can be edited. There are two ways to do this:

- 1) Before being applied to part, using the Materials dialog box, and
- 2) After being applied to a part, using the tree view in the lower section of the FEMdesigner AD explorer.

Editing In Materials Dialog

To edit a material before it is applied to a part, select a material as outlined above so that the material properties are displayed in the Materials dialog box. To change a material property, simply click in the desired material property data field and enter a new value, as shown at right for “tensile yield” at the top of the dialog box. After the desired properties have been edited, apply the new material to a part, which saves it with the active loadcase. Remember that this new material will only exist within the currently active loadcase, and will not be available for other analyses.

Edit Properties – Materials dialog box



TIP: Material Description

When editing material data that was imported from a material file, whether in the Materials dialog box or in the tree view, we strongly suggest that the “Description” field be altered to indicate this is an edited standard material, e.g. “AL-7075 T6 – edited.”

NOTE: When filling in or editing data fields manually, always pay special attention to the units that are specified for each field in the dialog box. These required units match the design units of the current part or assembly, and for accurate calculations all properties must be entered in the appropriate units. In the example above, our limit stresses are expressed in lbf/in² or psi, because the model design units are inches.

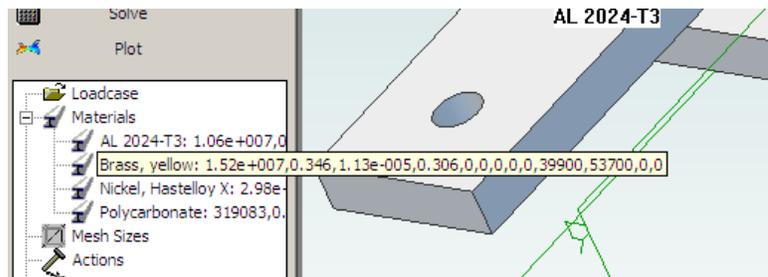
Editing in the Tree View

After a material is applied to a part, it is saved with the loadcase, and any further editing of this loadcase material

must now be done using the lower tree view of the FEMdesigner AD explorer. In this tree view section, the “Materials” entry can be expanded to display all materials assigned in the current loadcase, and each individual material displayed can be edited with a right-click.

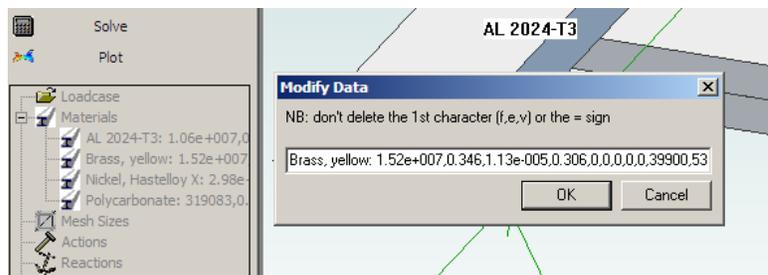
In the example shown, four different materials have been used in an assembly. Clicking the plus sign next to “Materials” in the lower tree view displays the list of materials that have been applied in this loadcase, and hovering the mouse over “Brass, yellow” displays the full list of its properties, in the same order that they appear in a material text file.

Hover Mouse to Display Properties



Right-clicking on a material will allow the user to either modify or delete the material. Selecting “Modify” displays the “Modify Data” dialog box shown here, where the material properties are again shown in the same order as contained in a material text file. When editing these values, it would be advisable to have a listing of the material file properties in their proper order for reference.

Right-click to Modify Properties



IMPORTANT: loadcase materials are different from material files. Material files are external, independent text files that contain material property information that can be used by any loadcase, where loadcase materials are specific to and saved only with the current active loadcase. A material file can be imported into a loadcase by selecting and applying the material to a part as outlined above, but once applied to the part, it is a loadcase material and is completely independent of the material file.