

At this stage you should play around a bit with the shape generator to familiarize yourself with the user interface.

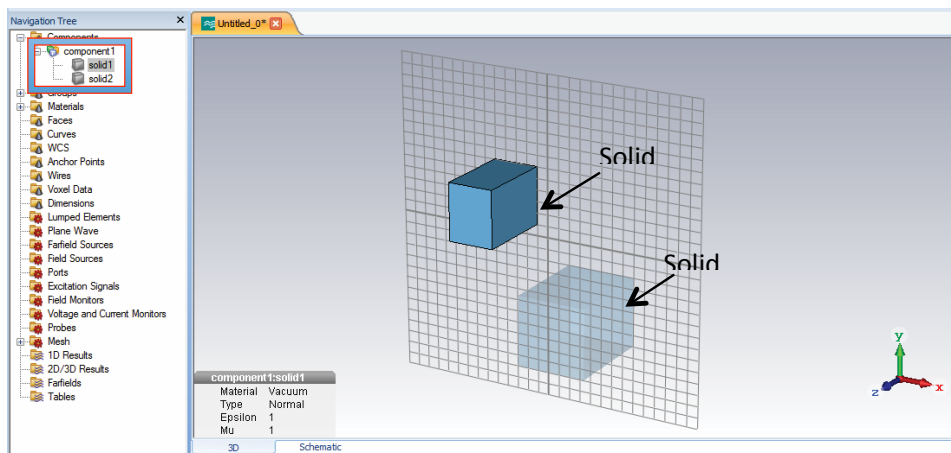
## Getting Started

### Select Shapes

After a shape has been defined, it is automatically cataloged in the navigation tree. You can find all shapes in the *Components* folder. If you open this folder, you will find a subfolder called *component1*, which contains all defined shapes. The name for each primitive is assigned in the final shape dialog box when the shape is created. The default names start with “solid” followed by an increasing number: solid1, solid2, etc.

You may select a shape by clicking on the corresponding item in the navigation tree. Note that after you select a shape, it will be displayed opaquely while all others will be drawn transparently (see the picture below). This is how the modeler visualizes shape selection. A shape can also be selected by double-clicking on it in the main window. In this case, the corresponding item in the navigation tree will also be selected. Holding down the *Ctrl* key, while double-clicking a shape in the main view, allows you to select multiple shapes. You may also select ranges of shapes in the navigation tree by holding down the *Shift* key while clicking on the shapes’ names.

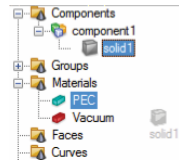
Another powerful way to select multiple shapes is the *Rectangle Selection* feature. Choose *View: Selection Rectangle Selection* and define a rectangular area in the main view by clicking and dragging with the mouse. All shapes within this rectangle are selected. Take a few seconds to familiarize yourself with the shape selection mechanism.



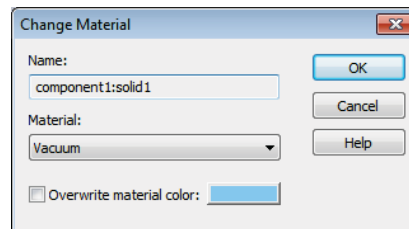
You may change the name of a shape by selecting it and choosing *Modeling: Edit Properties Rename (F2)*. You can then change the name of the shape by editing the item text in the navigation tree.

color.

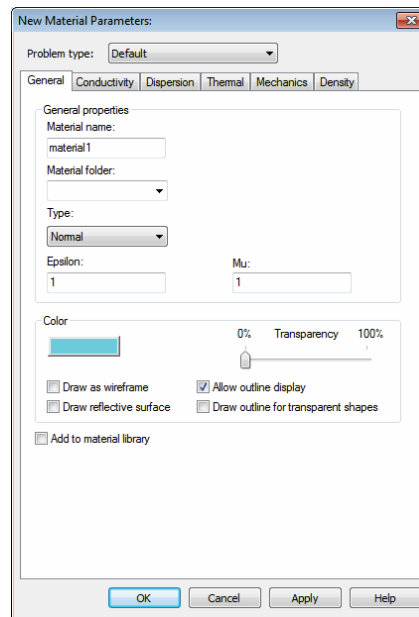
To change the material properties or the color of an individual shape you can assign it to another material. This can be done by dragging the solid in the navigation tree to the target material or vice versa:



Another method is to select the shape and choose *Modeling: Materials New/Edit Change Material* (this option can also be found in the context menu when a shape is selected). The following dialog box will open:

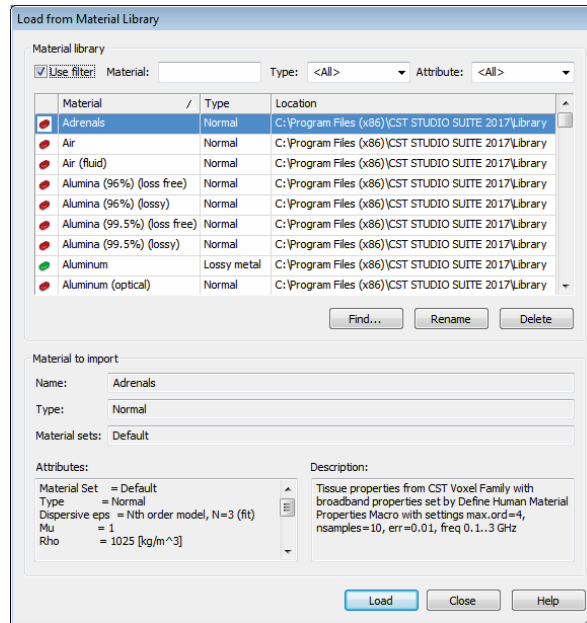


In this dialog box you may select an existing material from the list or define a new one by selecting the item *[New Material...]* from the list. In the latter case, another dialog box will open:



In this dialog box you have to specify the *Material name* and the *Material type* (e.g. perfect electric conductor (PEC), normal dielectric (Normal), etc.). Note that the available material types as well as the corresponding options depend on the currently used module. You can also change the color of the material by clicking the *Color* button. Use the *Material folder* field to arrange the materials in different sub folders. After clicking the *OK* button, the new material is stored and appears in the *Materials* folder in the navigation tree. Selecting a particular material

in the navigation tree also highlights all shapes that belong to this material. All other shapes will then be drawn transparently.

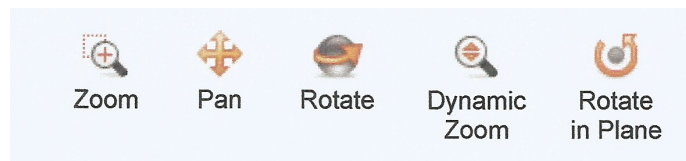


In order to simplify the definition of frequently used materials, a material database is available. Before a material definition from the database can be used, you have to add it to the current project by selecting *Modeling: Materials Material Library Load from Library*. This operation will open the following dialog box displaying the contents of the database:

You may select an existing material from the list and click the *Load* button to add the material definition to the *Materials* folder in the navigation tree. Once the material is available in this folder, it can be used in the current project. You can also add a material that has been defined in the current project to the database by selecting the material in the navigation tree and then choosing *Modeling: Materials Material Library Add to Library*.

## Change the View

So far we have created and viewed the shapes by using the default view. You can change the view at any time (even during shape generation) using some simple commands as explained below. The view will change whenever you drag the mouse while holding down the left button, according to the selected mode. You can select the mode by choosing *View: Mouse Control Zoom / Pan / Rotate / Dynamic Zoom / Rotate in Plane* or by selecting the appropriate item from the status bar:



The mode setting affects the behavior as follows:

- **Zoom:** In this mode a zoom window can be defined by dragging the mouse. After you release the left mouse button, the zoom factor and the view location will be updated so that the rectangle fills up the main window.
- **Pan:** The structure will be translated in the screen plane following the mouse cursor movement.

- **Rotate:** The structure will be rotated around the two screen axes. The center of the rotation will be the point on the structure where the mouse button was pressed, indicated by a red mark. If the selected location is outside the structure, the bounding box center point will be used as rotation center.
- **Dynamic Zoom:** Moving the mouse upward will decrease the zoom factor while moving the mouse downward will increase the zoom factor.
- **Rotate in Plane:** The structure will be rotated in the screen's plane.

The dynamic view-adjusting mode ends when you release the left mouse button. You can reset the zoom factor by choosing *View: Change View Reset View (Space)* or

## Apply Geometric Transformations

So far you have seen how to model simple shapes and how to change the view of your model. This section focuses on applying geometric transformations to your model.

We assume that you have already selected the shape (or multiple shapes) to which a transformation will be applied (e.g. by double-clicking on a shape in the main view).

You can then open the transformation dialog box by choosing *Modeling: Tools Transform* or by choosing the item *Transform* from the context menu. In the dialog box, you are asked to select one of the following transformations:

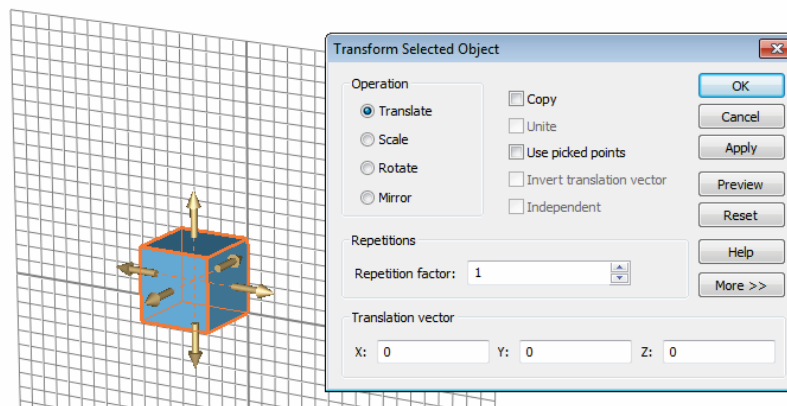
**Translate:** This transformation applies a vector translation to the selected shape.

**Scale:** By choosing this transformation, you can scale the shape along the coordinate axes. By unchecking *Scale uniform* you may specify different scaling factors for each coordinate direction.

**Rotate:** This transformation applies a rotation of the shape around a coordinate axis by a fixed angle. You may additionally specify the rotation center in the *Origin* field (click on *More* if the option is not immediately available). The center may be the center of the shape (calculated automatically) or any specified point. Specify the rotation angle and axis settings by entering the corresponding angle in the entry field for the corresponding axis (e.g. entering 45 in the y field while leaving all other fields set to zero performs a rotation around the y-axis of 45 degrees).

**Mirror:** This transformation allows one to mirror the shape at a specified plane. A point in the mirror plane is specified in the *Mirror plane origin* field, and the plane's normal vector is given in the *Mirror plane normal* input field.

A final example will demonstrate the usage of the transformation feature. Assume a brick has been defined and selected as depicted below. Open the transform dialog box by choosing the appropriate item from the context menu or *Modeling: Tools Transform*.



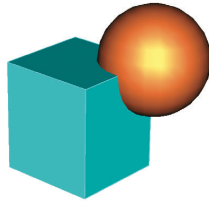
Started 33 Now the screen should look as follows:

## Combine Shapes Using Boolean Operations

Probably the most powerful operation to create complex shapes is to combine simple shapes using Boolean operations. These operations allow you to add shapes together, to subtract one or more shapes from another, to insert shapes into each other, and to intersect two or more shapes.

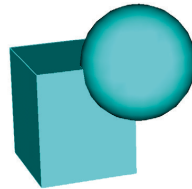
Let us consider two shapes – a sphere and a brick – on which to perform Boolean operations.

This list names all available Boolean operations and shows the resulting body for each combination:



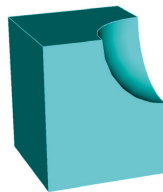
### **Add brick to sphere**

Add both shapes together to obtain a single shape. The resulting shape will assume the component and material settings of the first shape.



### **Subtract sphere from brick**

Subtract the second shape from the first to obtain a single shape. The resulting shape will assume the component and material settings of the first shape.



### **Intersect brick and sphere**

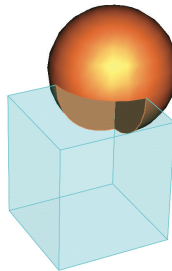
Intersect two shapes to form a single shape. The resulting shape will assume the component and material settings from the first shape of this operation.



### Trim sphere

= Insert brick into sphere

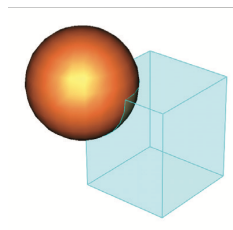
The first shape will be trimmed by the boundary of the second shape. Both shapes will be kept. The resulting shapes will have no intersecting volume.



### Insert sphere into brick

= Trim brick

The second shape will be inserted into the first one. Again both shapes will be kept. The resulting shapes will have no intersecting volume.



Note that not all of the Boolean operations above are directly accessible. As you can see, some of the operations are redundant (e.g. a trimming operation can be replaced by an insertion operation when the order of the shapes is reversed).

You can access the following Boolean operations by choosing the corresponding items: *Modeling: Tools Boolean Add / Subtract / Intersect / Insert*. Operations are accessible only when a shape has been selected (in the following referred to as “first” shape). After the Boolean operation has been activated, you will be prompted to select the “second” shape. Pressing the *Return* key performs the Boolean combination. The result depends on the type of Boolean operation:

**Add (+):** Add the second shape to the first one – keeps the component and material settings of the first shape.

**Subtract (-):** Subtract the second shape from the first one – keeps the component and material settings of the first shape.

**Intersect (\*)**: Intersect the first with the second shape – keeps the component and material settings of the first shape.

**Insert (/)**: Insert the second shape into the first one – keeps both shapes while changing the first shape only.

The trim operations are only available in a special “Shape intersection” dialog box which appears when a shape is created that intersects or touches areas with existing shapes. This dialog box will be explained later.

When multiple shapes are selected, you can access the Boolean add operation to unite all selected shapes. You can also select more than one shape when you are prompted to specify the *second* shape for Boolean subtract, intersect or insert operations.

## Pick Points, Edges, or Faces from within the Model

Many construction steps require the selection of points, edges, or faces from the model. The following section explains how to select these elementary entities interactively. For each of the “pick operations”, you must first select the appropriate pick tool e.g. *Modeling: Picks* *Picks*. Pick points, Pick edge Show point Clear picks edges or center pick list

faces

After you activate a pick tool, the mouse cursor will change indicating that a pick operation is in progress. In addition, all pickable elements (points, edges, or faces) will be highlighted in the model. Now you can double-click on an appropriate item. Alternatively, you can cancel the pick mode by pressing the *Esc* key.

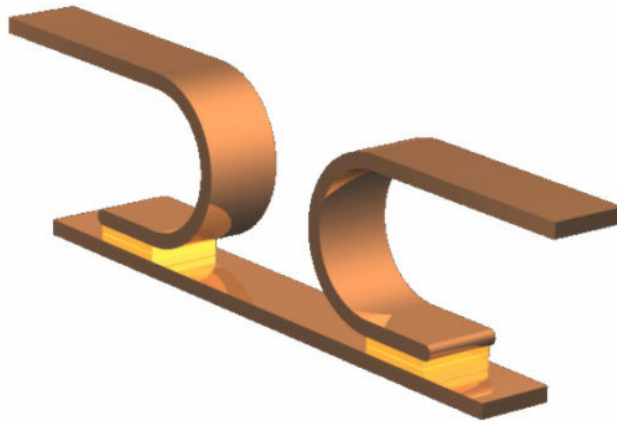
**Note**: You cannot pick edges or faces of a shape when another shape is currently selected. In this case you should either select the proper shape or deselect all shapes.

As soon as you double-click in the main view, the pick mode will be terminated and the selected item will be highlighted. Note that if the *Modeling: Picks* *Picks* *Keep Pick Mode* option is activated, the pick operation will not terminate after double-clicking. In this case you have to cancel the pick mode by pressing the *Esc* key. This mode is useful when multiple items have to be selected and it would be cumbersome to re-enter the pick mode several times.

The following list gives an overview of the available pick modes. Whenever the main structure view is active, keyboard shortcuts (listed in parentheses) can be used to activate a particular pick mode. The main structure view can be **activated by left-clicking once on the main drawing window.**

- **Pick Points, Edges, or Faces (S)** : Double-click close to an edge, an end point of an edge, or a face. The corresponding item will be selected.
- **Pick End Point (P)**: Double-click close to the end point of an edge. The corresponding point will be selected.
- **Pick Edge Center (M)** : Double-click on an edge. The mid-point of this edge will be selected.
- **Pick Circle Center (C)** : Double-click on a circular edge. The center point of this edge will be selected. The edge need not necessarily belong to a complete circle.
- **Pick Point on Circle (R)** : Double-click on a circular edge. Afterward an arbitrary point on the circle will be selected. This operation is useful when matching radii in the interactive shape creation modes.
- **Pick Face Center (A)** : Double-click on a planar face of the model. The center point of this face will be selected.
- **Pick Point on Face (O)**: Double-click on a point on the model to select it.
- **Pick Edge (E)** : Double-click on an edge of the model to select it.
- **Pick Face (F)** : Double-click on a face of the model to select it.
- **Pick Edge Chain (Shift+E)**: Double-click on an edge of the model. If the selected

# Stationary bar



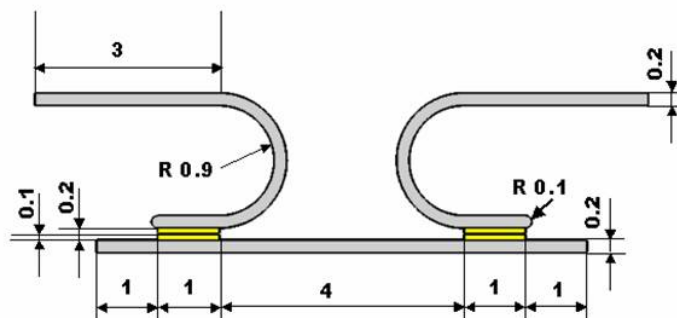
## Geometric Construction and Solver Settings

### Introduction and Model Dimensions

In this tutorial, a circuit breaker consisting of two contact springs connected by a bridge is analyzed. The contact springs are constructed using the line and arc curve tools. The resulting curve can then be traced to form a solid. On either side of the contact area, a gold pad is attached that can be constructed with the extrusion tool. For the construction of the bridge, a local coordinate system is employed to make creation of the brick easier.

One matter of concern is the current flow from one contact over the bridge to the other contact. Therefore, two current ports are defined for the stationary current solver. If the current amplitude increases too much (e.g. caused by a short circuit), the bridge is pulled away from the springs - which is an effect caused by the current's magnetic field. The current field can be calculated by the stationary current solver and will be used as a source field for a magnetostatic calculation.

We strongly recommend that you carefully read through the CST EM STUDIO *Workflow & Solver Overview* manual before starting this tutorial.



All dimensions in the detail drawing are given in cm.