

TurboCAD Online Help

TurboCAD is a CAD and 3D modeling application the features of which include the 2D and 3D tools, surface and solid modeling, handle-based editing, advanced facet editing, parametric objects. TurboCAD supports the most of industry standard file formats (DXF, DWG, DWF, HPGL, DGN etc.) and is completely programmable. The following are two excerpts from the online help of TurboCAD version 6:

- ⇒ [DXF \(DWG\) Tolerance Support](#)
- ⇒ [3D Object Boolean Operations](#)

DXF (DWG) Tolerance Support

In the world of science and technology, a tolerance is a leeway for variation from a kind of standard. This is a permissible deviation from a specified value of a structural dimension.

We use dimensions to add measurements to a drawing, and we use tolerances to specify by how much a dimension can vary. *Geometric tolerancing* is a way to show maximum allowable deviations of form, profile, orientation, location, and *runout* of a feature.

To define the tolerance information for a single dimension, *feature control frames* are used. A feature control frame consists of two or more compartments.

The first of two mandatory compartments contains a geometric characteristic symbol that represents the geometric characteristic to which a tolerance is being applied, for example: a form, orientation, or runout. Form tolerances control straightness, flatness, circularity, cylindricity, and profiles of a line and surface.

The second of two mandatory compartments contains the tolerance value. Where applicable, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol.

Note: Material conditions apply to features that can vary in size. At maximum material condition (**M**, also known as **MMC**), a feature contains the maximum amount of material stated in the limits. At **MMC**, a hole has the minimum diameter, whereas a shaft has the maximum diameter. At least material condition (**L**, also known as **LMC**), a feature contains the minimum amount of material stated in the limits. At **LMC**, a hole has the maximum diameter, whereas a shaft has the minimum diameter. Regardless of Feature Size (**S**, also known as **RFS**) means a feature can be any size within the stated limits.

For a tolerance feature control frame with two tolerance values (Tolerance 1 and Tolerance 2), the second of two mandatory compartments contains the value of Tolerance 1 and is followed by a third, identical compartment that holds the value of Tolerance 2.

Other (optional) compartments commonly contain a pair of symbols each. These are a datum reference letter and a material condition symbol.

Note: A datum is a theoretically exact point, axis, or plane from which you make measurements and verify dimensions. Usually, two or three mutually perpendicular planes perform this task best. These are jointly called the datum reference frame.

As a rule, we deal with three datum references—primary, secondary, and tertiary—marked by letters A, B, and C, respectively.

To specify the symbols and values for a feature control frame, use the Tolerance dialog.

To open the Tolerance dialog:

1. Select the desired feature control frame in your drawing.
2. Open the Properties dialog by doing one of the following:
 - Double-click on the feature control frame.
 - Choose Format | Properties.
 - Click Properties on the Inspector bar.
 - Right-click and select Properties.
3. Click the Others tab and select Tolerance from the Additional Pages list.
4. Click Go To Page. This will open the Tolerance dialog

The Tolerance dialog provides a number of controls for customizing the settings of the feature control frame. Each compartment of the tolerance feature control frame has a specific set of controls. These are:

Sym—Clicking this compartment brings up the Symbol dialog, where you can select one of the available geometric characteristic symbols.

Tolerance (#)—Use the options of this compartment to specify three parameters—the tolerance value and two symbols, diameter and material condition, preceding the value and succeeding it, respectively. The tolerance value, editable via the data entry field, indicates the amount by which the geometric characteristic can deviate from a perfect form. The diameter symbol can be inserted or discarded by toggling the box preceding the tolerance value field. Checking the box after the tolerance value field displays the Material Condition dialog where you can select a symbol that will be inserted into the MC box for the given tolerance value.

Datum (#)—Use the options of this compartment to specify one of three datum references—primary, secondary, and tertiary. The datum reference can consist of a value and a modifying symbol. A datum is a theoretically exact geometric reference used to establish the tolerance zone for a feature. Use the data entry field to enter the datum reference value. Checking the box succeeding the datum reference value field displays the Material Condition dialog, where you can select a symbol that will be inserted into the MC box for the datum reference value. These symbols act as modifiers to the datum reference.

Protected Tolerance Zone—Use the options in this compartment to specify the projected tolerance zone value and symbol in the feature control frame. A projected tolerance zone controls the variation in height of the extended portion of a fixed perpendicular part and refines the

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tolerance to that specified by positional tolerances. Enter a value in the data entry field. A projected tolerance zone symbol (**P**) can be inserted or discarded by toggling the box succeeding the height entry field.

Datum Identifier—Use the data entry field to specify a datum identifying string. A datum is a theoretically exact geometric reference from which you can establish the location and tolerance zones of other features. A point, line, plane, cylinder, or other geometry can serve as a datum.

For customizing the parameters of text within the tolerance feature control frame, use the options (**Text Height** and **Font**) in the Frame Parameters control group. **Text Height** helps to define the text height. **Font** lets you choose a preferable font; otherwise, check the “Use the document font” box.

3D Object Boolean Operations

TurboCAD Boolean tools create a new 3D object by combining two previously existing 3D objects. The two objects must be selected in sequence. The first object selected is called the primary object. The second object selected is called the secondary object. For the Boolean tools to work, the two objects must intersect in Model Space.

Note: For creating and laying out your drawing, TurboCAD provides you with two drawing environments—Model Space and Paper Space. Model Space is the environment in which you create your drawing, usually called a model. In this area you do drafting and design work, creating two-dimensional drawings or three-dimensional models. Paper Space is the environment in which you create the final layout of your drawing for printing or plotting it on paper. In Paper Space you usually arrange the drawing's elements on a sheet of paper. Switching between Paper Space and Model Space can be done by using the Menu: Workspace|Model Space/Paper Space options, or from the Model Space and Paper Space buttons in the Standard toolbar. For this purpose, you also can use the Model Space/Paper Space button.

Four Boolean operations are defined for 3D objects in TurboCAD. These are *addition*, *subtraction*, *intersection*, and *3D slicing*.

Related Topics

[Add](#)

[Subtract](#)

[Intersect](#)

[3D Slice](#)

Add (Boolean Operation)

Remember that TurboCAD lets you create solid objects directly (for example, using the Box and Sphere options of the Insert|3D Object menu) or indirectly (for example, first, using the Prism option of the Insert|3D Object menu and then making the prism solid with the help of the Properties dialog).

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To add two solid objects, do as follows:

1. If necessary, enable the Model Space mode (the WorkSpace | Model Space menu).
2. Activate the Add tool (Modify | Boolean Operations).
3. Select the first solid object.
4. Select the second solid object.

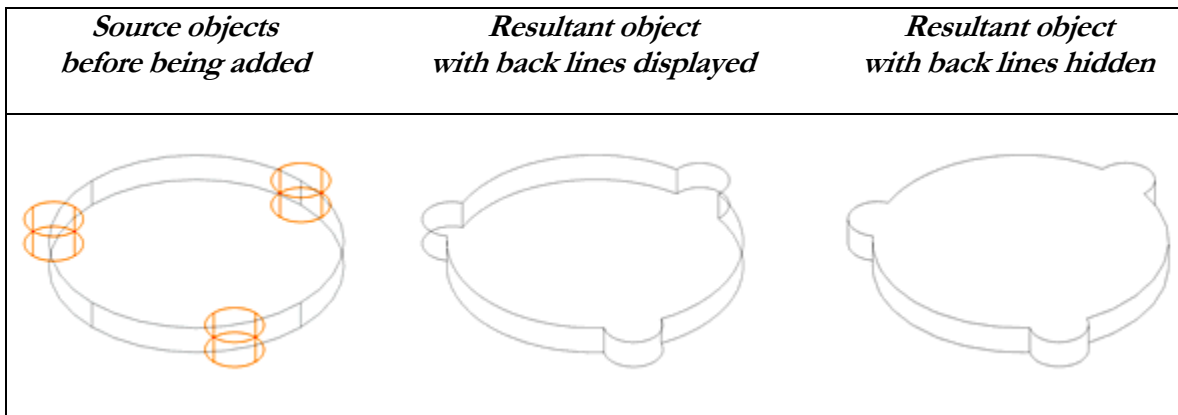
TurboCAD creates a 3D object, which is a result of the Boolean addition of two selected objects.

It is noteworthy that the local menu provides a number of options:

- Turn on a local snap (Vertex, Grid, Nearest On Graphic, etc.).
- Open the Properties dialog to customize the properties of the most recently created object.
- Cancel the object construction.
- Finish the object construction.
- Undo the most recently perform operation.
- Redo the Undo operation.

Most of the above options are also accessible via the TurboCAD main menu or toolbars.

Example 1: Adding three small cylinders of orange color to the large cylinder of gray color



Subtract (Boolean Operation)

This tool allows you to create a solid 3D object by performing the Boolean operation “SUBTRACT” between two solid objects of 3D type in model space (see [Model Space versus Paper Space](#)).

Remember that TurboCAD lets you create solid objects directly (for example, using the Box and Sphere options of the Insert | 3D Object menu) or indirectly (for example, first, using the Prism

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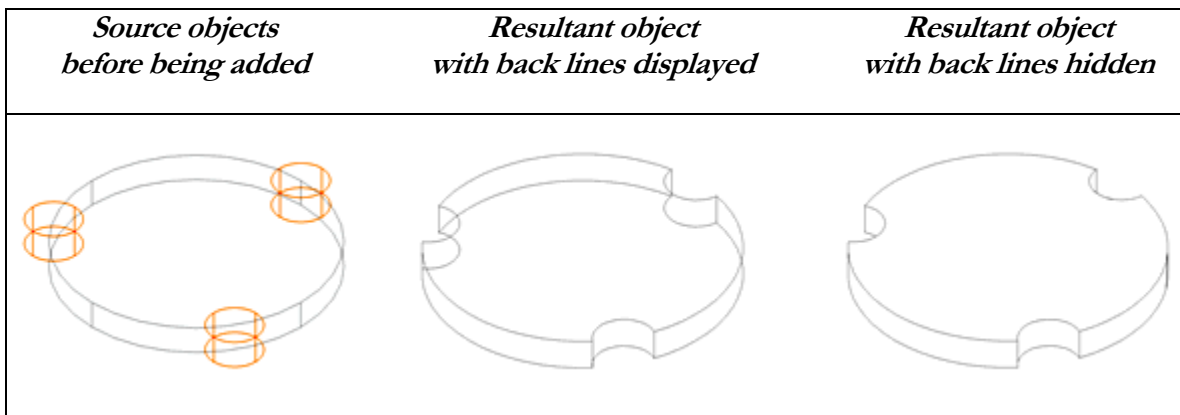
option of the Insert|3D Object menu and then making the prism solid with the help of the Properties dialog).

To subtract two solid objects, do as follows:

1. If necessary, enable the Model Space mode (choose WorkSpace|Model Space).
2. Activate the Subtract tool (Modify|Boolean Operations).
3. Select a minuend (the first solid object of 3D type).
4. Select a subtrahend (the second solid object of 3D type).

TurboCAD creates a 3D object, which is a result of the Boolean subtraction of two selected objects.

Example 2: Subtracting three small cylinders of orange color from the large cylinder of gray color



Intersect (Boolean Operation)

This tool allows you to create a new solid 3D object by performing the Boolean operation “INTERSECT” between two solid 3D objects space (see [Model Space versus Paper Space](#)).

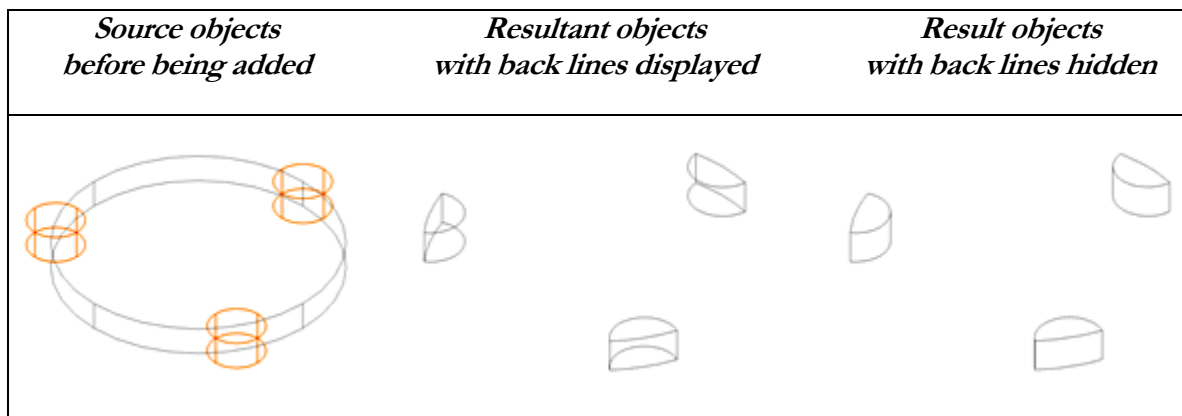
Remember that TurboCAD lets you create solid objects directly (for example, using the Box and Sphere options of the Insert|3D Object menu) or indirectly (for example, first, using the Prism option of the Insert|3D Object menu and then making the prism solid with the help of the Properties dialog).

To intersect two solid objects, do as follows:

1. If necessary, enable the Model Space mode (the WorkSpace|Model Space menu).
2. Activate the Intersect tool (Modify|Boolean Operations).
3. Select the first object.
4. Select the second object.

TurboCAD creates a 3D object, which is a result of the Boolean intersection of two selected objects.

Example 3: Intersecting three small cylinders of orange color with the large cylinder of gray color



3D Slice (Boolean Operation)

This tool allows you to create a new solid 3D object by performing the Boolean operation “INTERSECT” between two solid objects of 3D type in model space (see [Model Space versus Paper Space](#)).

You create a new 3D object by cutting an existing 3D object and removing a specified side. You can retain one or both halves of the sliced object. The sliced objects retain the layer and color properties of the original objects. The default method of slicing a 3D object is to specify two points that define the cutting line and then select which side to retain. Note that when cutting by line, TurboCAD actually cuts with the plane that is perpendicular to the current view and passing through the specified line.

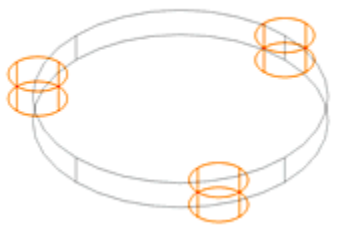
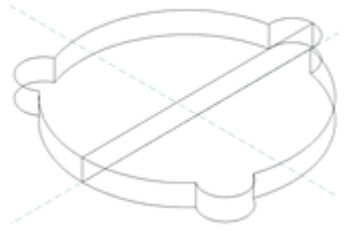
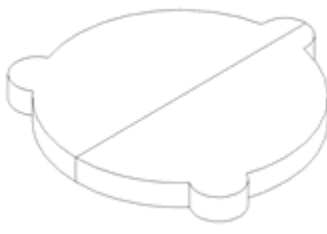
Using the local menu, you can choose any the following options to customize a 3D Slice operation. These options are: Slice by Line (2 points), Slice by Plane (3 Points), Slice by WorkPlane, Create Facet, and Delete a Sliced Part.

To slice a 3D object, do as follows:

1. If necessary, enable the Model Space mode (the Workspace | Model Space menu).
2. Activate the 3D Slice tool (Modify | Boolean Operations).
3. Select a 3D object.
4. Click with right mouse button and make preferable settings for the slice operation.
5. Depending on settings in the local menu, specify the points for slicing.
6. If necessary, select the sliced part to delete.

TurboCAD creates a 3D object, which is a result of the 3D Slice operation.

Example 4: 3D-slicing three small cylinders of orange color with the large cylinder of gray color

<i>Source objects before being added</i>	<i>Resultant object with back lines displayed</i>	<i>Resultant object with back lines hidden</i>
		

<i>Part of resultant object with back lines displayed</i>	<i>Part of resultant object with back lines displayed</i>
