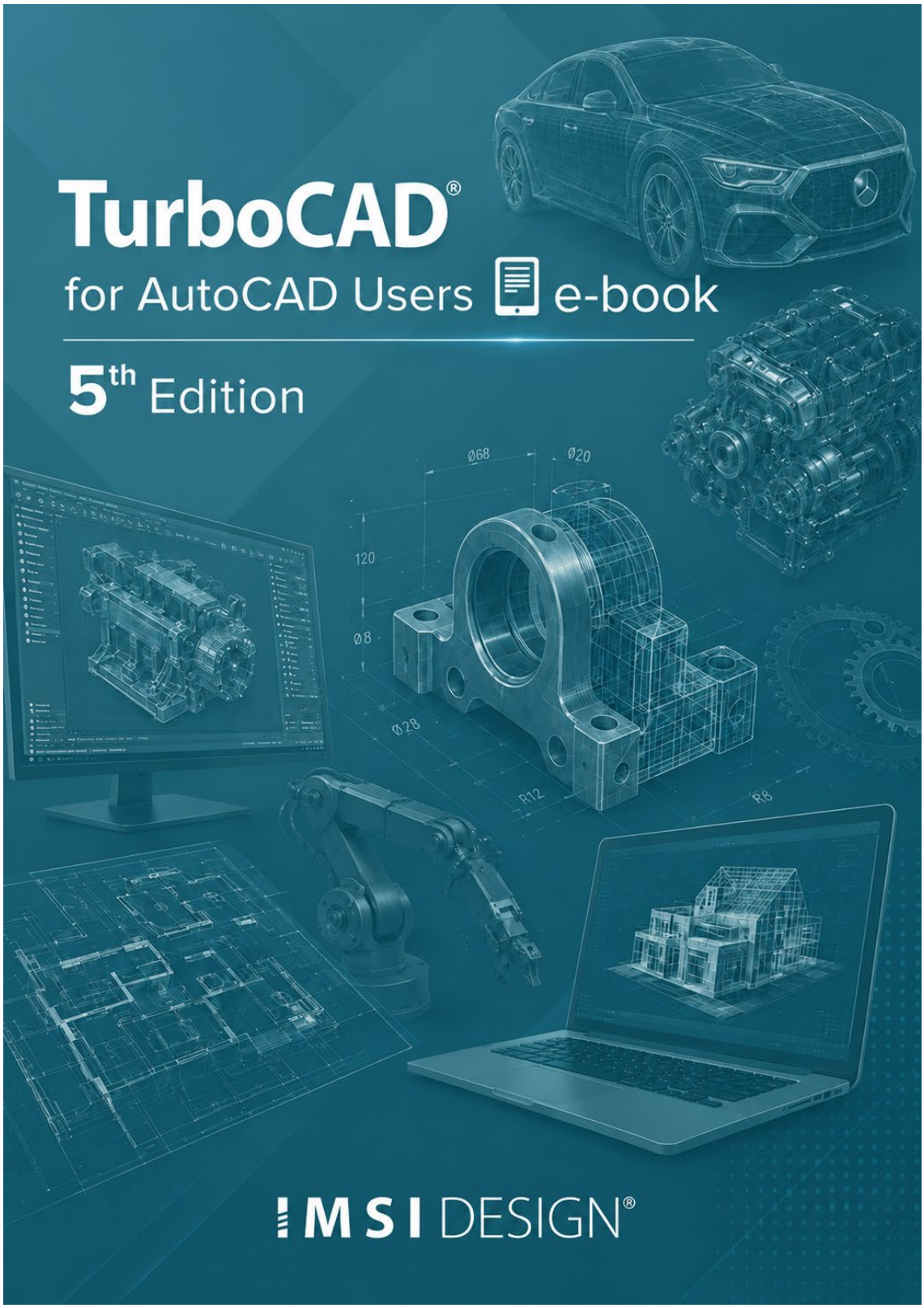
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for AutoCAD Users e-book
5th Edition
IMSI DESIGN®

TurboCAD®

for AutoCAD® Users

5th Edition

by



IMSI Design®, LLC
384 Bel Marin Keys Blvd.
Suite 150
Novato, CA 94949

www.turbocad.com

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Introduction

TurboCAD for AutoCAD Users

If you are an AutoCAD user, then you are probably curious about how TurboCAD compares to AutoCAD. Here we look at some of the similarities and differences between AutoCAD and TurboCAD.

What TurboCAD Offers the AutoCAD User

TurboCAD offers the AutoCAD user many advantages, including DWG compatibility, better handle-based editing, a broader variety of object snaps, and an improved user interface. It offers features not found in AutoCAD, such as built-in raster-to-vector conversion, history for 3D solid models, node editing, and constraint management.

File Compatibility

TurboCAD reads a variety of vector file formats, including many of the same files produced by AutoCAD:

- AutoCAD DWG drawing format.
- Autodesk DXF drawing interchange format.
- Autodesk DWF drawing Web format.
- Spatial Technology SAT, the ASCII version of ACIS solid model files.
- 3D Studio surface models, and STL stereolithography files.

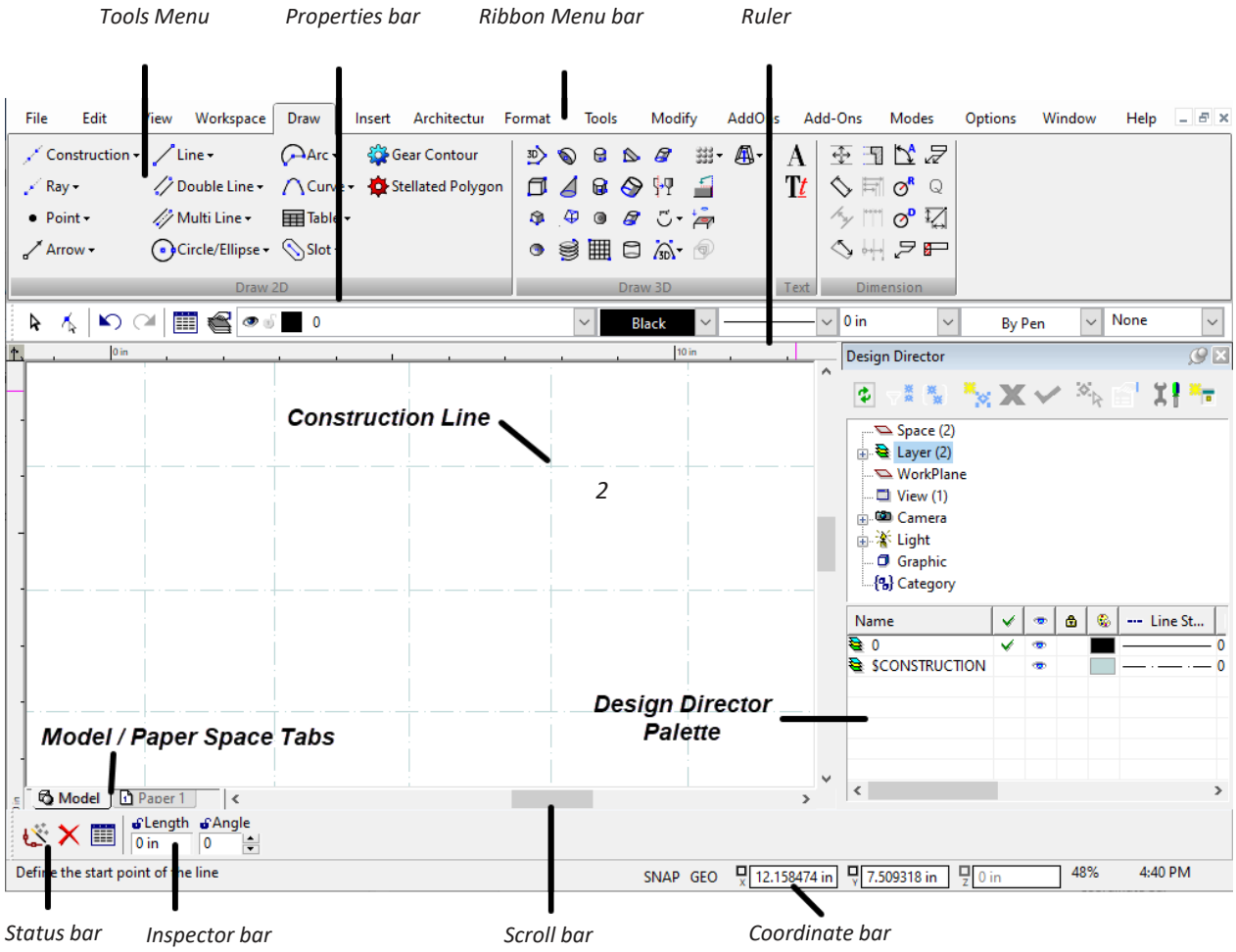
TurboCAD also reads files created by competitors, such as DGN (Bentley Systems), DCD (DesignCAD), and FCW (FastCAD for Windows).

It supports a host of vector formats, such as IGES, STEP, Metastream, HPGL (Hewlett-Packard graphics language), VRML (virtual reality markup language), CGM (computer graphics metafile), and WMF (Windows metafile).

In addition, TurboCAD can import AutoCAD drawings and symbols via the Windows Clip-board and as OLE objects.

The TurboCAD User Interface

Because of Windows, much of TurboCAD's interface is the same as AutoCAD's. You already know how to use the menu bar, toolbars, scroll bars, status bar, and so on. Here is an overview of the TurboCAD Ribbon user interface with classic theme and its primary elements:



TurboCAD's user interface

If you don't care for the interface or theme, select another one from **Tools - Customize - Option - Styles and Themes**, or create your own.

Repeating Commands

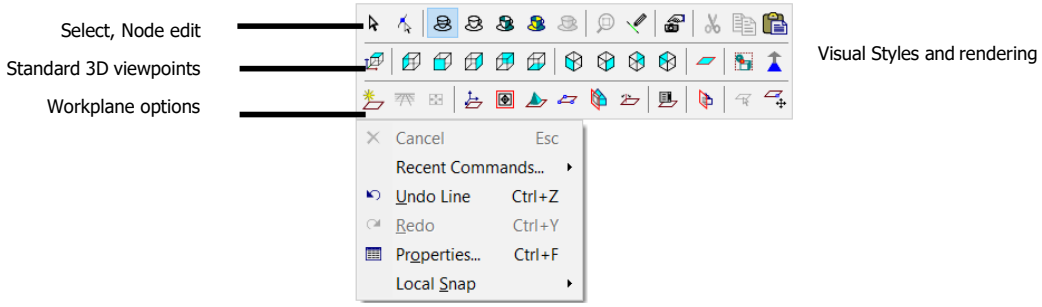
As an AutoCAD user, you know that some commands, such as **Point**, repeat automatically, and that some don't, such as **Circle**. To make non-repeating commands repeat, you need to employ the **Multiple** command or press the spacebar.

In TurboCAD, all drawing and editing commands repeat. More precisely, the current command remains in effect until you select a different one. Start with the **Circle** command, and TurboCAD continues drawing circles — until you choose another command.

AutoCAD's default command is **Select**; TurboCAD's default command is **Line**.

Local Menus

Right-click the drawing to bring up a context-sensitive *local menu* (known as the cursor menu or shortcut menu in AutoCAD). The local menu is related to the current tool in use. It usually contains the **Properties** option to access the properties of the selected object.

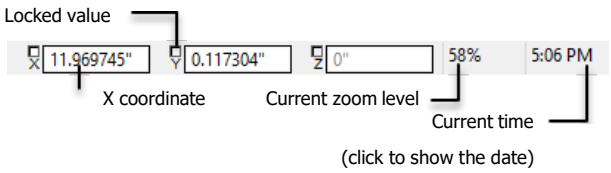


TurboCAD's local menu in node edit mode

At the top of the local menu is a handy toolbar that lets you quickly select rendering modes (first row of icons), switch viewpoints (second row), and control workplanes (third row).

Coordinate Bar

The Coordinate bar displays the current 3D coordinates. To change the value of X, Y, or Z, press **Shift+Tab** to move the focus to the **X** field. The tiny boxes lock the related coordinate when an 'x' appears in them; this is the equivalent to using the. x, y, and. z point filters in AutoCAD.



TurboCAD's Coordinate bar

TurboCAD displays coordinates in absolute, relative, and polar modes. To change quickly between modes, press **Shift+A** for absolute coordinates, **Shift+R** for relative, and **Shift+P** for polar display (distance and angle).

The Coordinate bar can be used together with the Inspector bar to place objects in drawings — all by keyboard entry.

For example, you can draw precise lines with the Line command and the keyboard, as follows:

Step 1: Press **Shift+Tab** to access the Coordinates bar, and then enter the x coordinate.

Step 2: Press **Tab** to enter the y coordinate.

Step 3: Press **Enter** to start drawing the line.

Step 4: Press **Tab** to access the Inspector bar, and then enter the length.

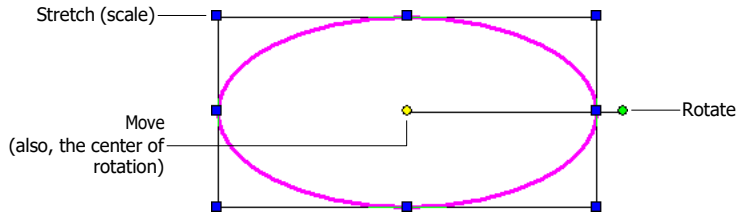
Step 5: Press **Tab** again to enter the angle.

Step 6: Press **Enter** to define the second point.

Editing with Select Cursor

AutoCAD's grips editing emphasizes stretching and moving; performing other editing functions, such as copying and rotating, requires pressing the spacebar or entering option letters.

TurboCAD adds the rotate grip missing from AutoCAD. Use the **Select** cursor to stretch, scale, move, and rotate the selected object(s) through grips. (Speaking of stretching, circles can be stretched into ellipses, and vice versa.)

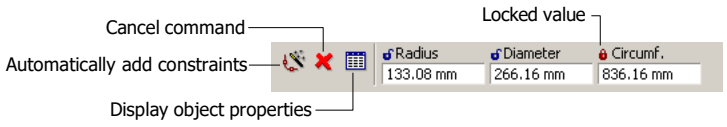


TurboCAD's Select cursor stretches (scales), moves, and rotates objects

Inspector Bar

TurboCAD's Inspector bar is like AutoCAD's Dynamic Input, but located near the status bar. For instance, when you begin drawing a circle, the Inspector bar displays the radius, diameter, and circumference. Click the padlock icon to lock the associated value; this is like AutoCAD's point filters.

The Inspector bar changes the fields and buttons it displays, depending on the object you are drawing. Illustrated below is the Inspector bar displayed when drawing circles.



TurboCAD's Inspector bar

As you move the cursor, the values are updated continuously, unless locked. Press **Tab** to access the Inspector bar and enter new values.

On the right of the Inspector bar are three buttons. These toggle whether constraints are added automatically, the operation cancels, or the Properties dialog box displays.

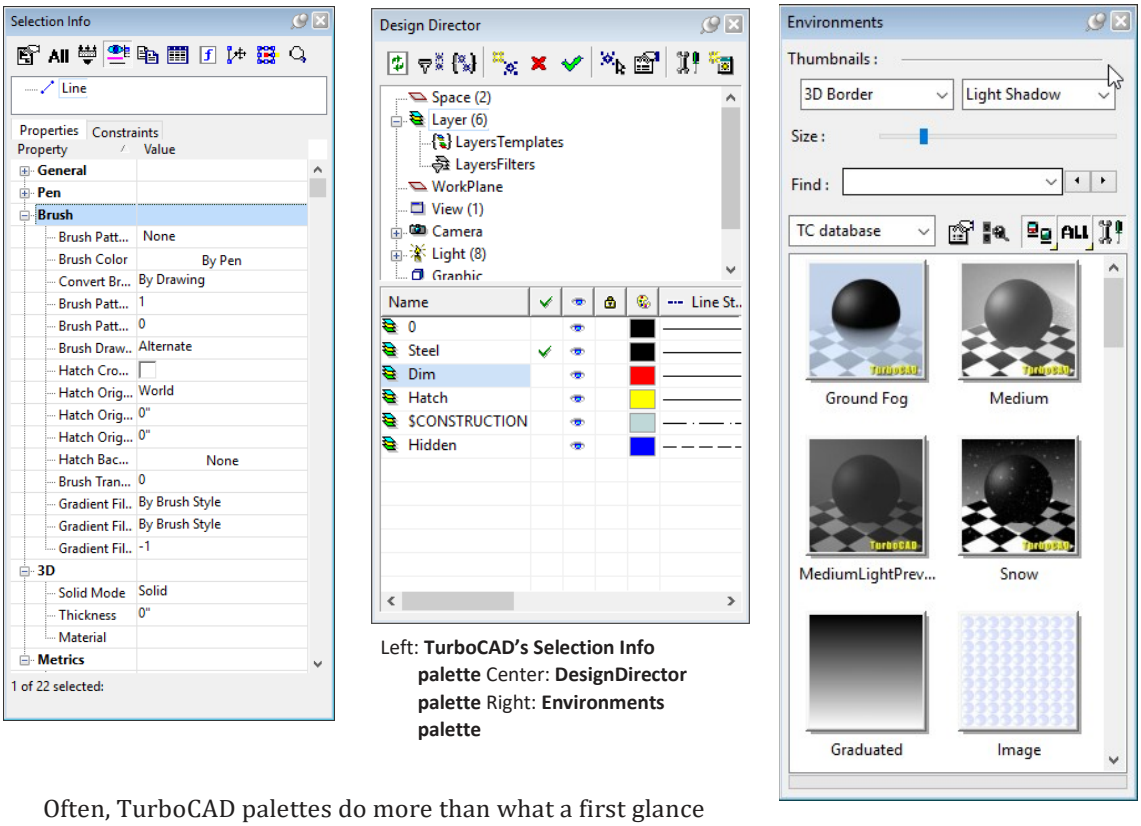
Palettes

Both AutoCAD and TurboCAD offer palettes. You'll find palettes in TurboCAD for blocks, command tools, lights, rendering materials, and more. TurboCAD's Selection palette is like AutoCAD's Properties palette, and the Design Director palette is like DesignCenter.

TurboCAD has some palettes that are not found in AutoCAD, such as for reporting measurement information, viewing part trees, and recording macros. Similarly, AutoCAD has palettes not found in TurboCAD; the table below illustrates the similarities and differences.

AutoCAD Palette	**Equivalent TurboCAD Palette**
Advanced Render Setup	Environment
Dashboard	Tools
DesignCenter (Folders)	TC Explorer
DesignCenter (Open Drawings)	Design Director
Lights	Luminances
Materials	Materials
Properties	Selection Info
QuickCalc	Calculator/Variables
Tool Palettes	Blocks, Symbols
Palettes not in TurboCAD	
dbConnect	*Substitute with **Tools | Database***
External References	*Substitute with **Insert | Create External References** and **Tools | RasterImages***
Info	*Substitute with **Help | TurboCAD Help Topics***
Markup Set	*Substitute with **File | Open**, and then select "DWF" format*
Sheetset	*Sheet sets not supported by TurboCAD*
Visual Styles	*Substitute with **View | Camera | Draft Render***
Palettes not in AutoCAD	
*Substitute the **Color** dialog box*	Color
*Substitute the **SolView** command*	Drafting
*Substitute the **Browser** command*	Internet
Handcode in Notepad	Macro Recorder
*Substitute the **Dist** and **Area** commands*	Measurement Info
Part trees not available in AutoCAD	Part Tree

Illustrated below are three of TurboCAD's 26 palettes.



Left: **TurboCAD's Selection Info palette** Center: **DesignDirector palette** Right: **Environments palette**

Often, TurboCAD palettes do more than what a first glance might reveal. The Calculator palette, for instance, is more than a calculator: it is also used with TurboCAD's Dimensional Constraint Engine to display the driving dimensions created in constrained drawings. Vari-ables can be modified in the Calculator with formulas that describe mathematical relation- ships between the dimensions.

How TurboCAD Differs from AutoCAD

As mentioned, the primary difference you'll notice between TurboCAD and AutoCAD is that *all* commands automatically repeat until you select another command. Some other crucial differences include the following.

What's Missing from TurboCAD

As an AutoCAD user, you have come to expect a certain selection of commands. You might not, however, find a few of them in TurboCAD. In some cases, the commands have different names. For example, gradient fills aren't applied with a **Gradient** command, but with TurboCAD's Pen Brush property. AutoCAD uses visual styles to edit in 3D rendered mode; in TurboCAD, you would turn on Draft Render mode. In Appendix, we provide a comparison of many AutoCAD commands with their TurboCAD equivalent.

Some AutoCAD features are completely missing from TurboCAD, such as sheetsets, DWF markup sets, and a few 3D solid modeling commands.

On the other hand, TurboCAD contains commands lacking in AutoCAD. For example, TurboCAD can import DWF files as entities. AutoCAD isn't able to draw a series of concentric circles as quickly as TurboCAD. Creating 2D views of 3D models with AutoCAD's **SolView** command seems downright pre-historic compared to TurboCAD's Drafting palette. And while AutoCAD illustrates a simple history of edited solid models, TurboCAD shows the complete part tree in the Part Tree palette.

You may find some of TurboCAD's commands to be oddly implemented. For example, there is a command that draws one line perpendicular to another. The AutoCAD user would wonder why a separate command is needed, instead of just using Perpendicular object snap. While TurboCAD does have a perpendicular snap mode, users may like the fact that lines can be drawn perpendicularly without having to turn on a snap mode.

Jargon

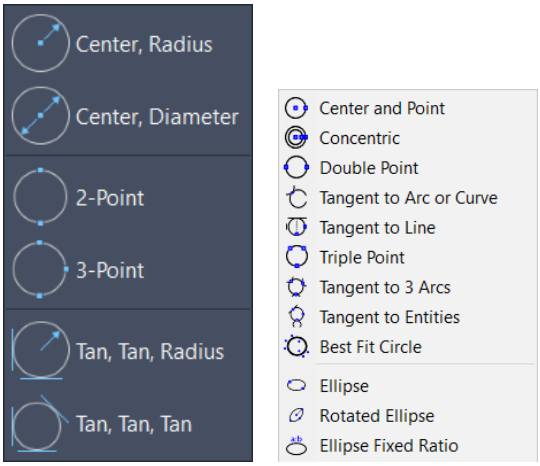
Many terms in TurboCAD are similar to AutoCAD, but there are some differences. You may have already noticed that TurboCAD uses the name "local menu" instead of shortcut menu or cursor menu. AutoCAD's linetypes are TurboCAD's "line patterns"; model space is "world" space.

Whereas AutoCAD has most of its editing commands in the **Modify** menu, TurboCAD has them in the **Edit** and **Modify** menus. AutoCAD's **Draw** menu is like TurboCAD's **Draw** menu. For quick reference, the Appendix includes a concise AutoCAD-to-TurboCAD terminology guide.

TurboCAD Drawing Commands

The primary user interface difference is that TurboCAD commands cannot be typed in, as with AutoCAD. For this reason, TurboCAD tends to have more commands than AutoCAD, because each command is usually limited to fewer options.

For instance, AutoCAD has one command for drawing circles: its **Circle** command has five options for constructing circles by a variety of methods. In contrast, TurboCAD has eight circle commands, one for each primary construction method.



Left: **AutoCAD's circle construction methods**
Right: **TurboCAD's circle construction methods**

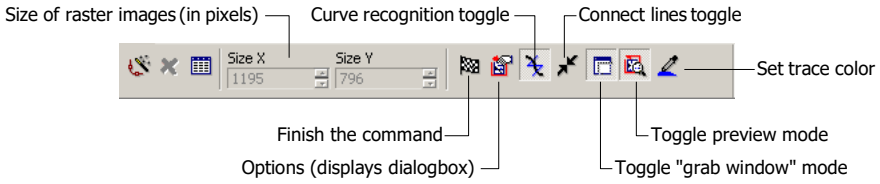
AutoCAD Circle options	Equivalent TurboCAD commands
Center, Radius	Center and point (enter radius on Inspector bar)
Center, Diameter	Center and point (enter diameter on Inspector bar)
2 Points	Double point
3 Points	Triple point
Tan, Tan, Radius	Tangent to Arc or Curve Tangent to Line
Tan, Tan, Tan	Tangent to 3 Arcs Tangent to Entities

Because most TurboCAD commands provides options on the Inspector bar, this multiplies the number of ways that TurboCAD can draw circles, as listed by the table below.

TurboCAD Circle command	Inspector Bar options
Center and Point	(1) Radius, (2) Diameter, (3) Circumference
Concentric	(4) Radius, (5) Diameter, (6) Circumference
Double Point	(7) Radius, (8) Diameter, (9) Circumference, (10) Angle
Tangent to Arc or Curve	(11) Radius, (12) Diameter, (13) Circumference
Tangent to Line	(14) Radius, (15) Diameter, (16) Circumference
Triple Point	(17) *none*
Tangent to 3 Arcs	(18) *none*
Tangent to Entities	(19) Radius, (20) Diameter, (21) Circumference

In general, TurboCAD provides more variations than does AutoCAD. There are more object snap modes, more ways to draw, and more ways to edit.

Many TurboCAD commands also display buttons on the Inspector bar, in place of the command prompts used by AutoCAD. These allow you to change parameters on-the-fly and access dialog boxes with even more options. Illustrated below are the Inspector bar buttons displayed by the **Tools | Trace** command.



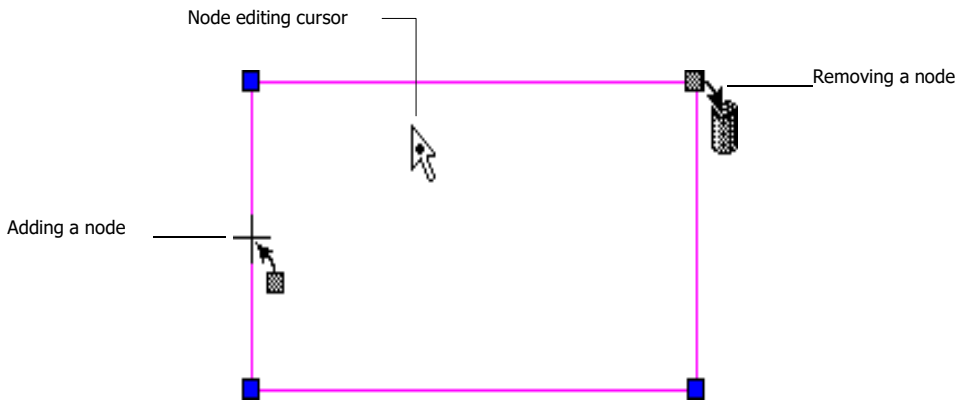
Buttons on the Inspector bar for the Trace command

The Finish flag is a common sight; click it to tell TurboCAD you're done specifying options.

Node Editing

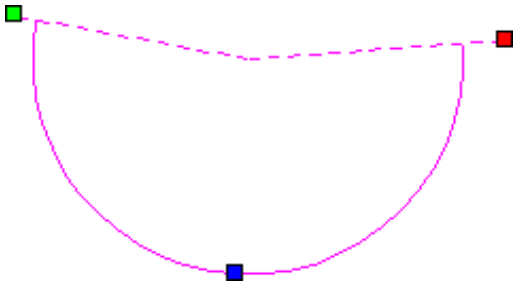
TurboCAD's *node editing* is an enhancement over grips editing. It allows you to move, add, and remove nodes. Node editing is useful for converting objects, such as closing arcs into circles and dividing lines into multiple segments.

To add or remove nodes, hold down the **Ctrl** key while performing the action: click where there is no node to add one; click on a node to remove it. Icons show which action is being performed — add or remove. The figure below shows a rectangle being edited by its nodes. A new local menu item "Highlight end points" (On/Off) has been added to "Edit" tool. The item is enabled if the selected graphic is non-closed and if vertex count > 2.



TurboCAD's node editor

Most nodes are blue; the red and green ones show the start and endpoints of arcs, circles, and ellipses.



TurboCAD's green and red nodes indicate start and stop points

.

What's Unique to TurboCAD

User Interface

AutoCAD provides just few tools for creating mechanical drawings. There's a block library of bolts, a user interface suitable for 3D modeling, and a weak history for editing solid models. In fact, if you want mechanical tools, AutoDesk suggests you purchase a new program such as Inventor. In contrast, TurboCAD Platinum includes significant tools for 2D and 3D mechanicalmodels.

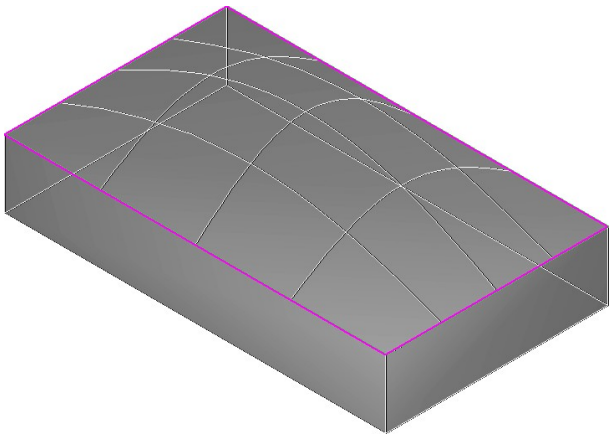
Quick pull
Not just command line, up and running in one hour

Mechanical Tools

AutoCAD provides just few tools for creating mechanical drawings. There's a block library of bolts, a user interface suitable for 3D modeling, and a weak history for editing solid models. In fact, if you want mechanical tools, AutoDesk suggests you purchase a new program such as Inventor. In contrast, TurboCAD Platinum includes significant tools for 2D and 3D mechanicalmodels.

The **Parts Tree** palette displays editable history trees of 3D operations, such as blends, Booleans, and shells. You can use it to apply and change chamfers and fillets; change the size and position of Boolean operations; and change the thickness and open faces of shelled solids.

Facet deform changes the shapes of surface and solid faces. It either applies uniform pressure or specifies deformation points and constraints.



TurboCAD deforming the faces of a 3D solid

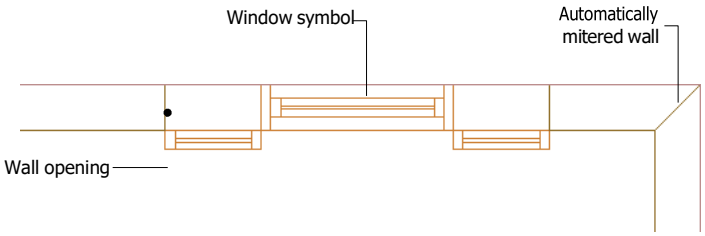
The **Sheet Metal** palette displays editable history trees of 3D operations, such as blends, Booleans, and shells. You can use it to apply and change chamfers and fillets; change the size and position of Boolean operations; and change the thickness and open faces of shelled solids.

Architectural Tools

AutoCAD provides multi-lines for drawing walls, and dynamic blocks for placing doors, windows, and other symbols. Strictly speaking, Autodesk would rather you use their Revit architectural software, which is completely different from AutoCAD.

TurboCAD Platinum includes tools specific to architectural drafting: create and edit walls, add window and door openings, and design roofs. The walls are smart objects: intersections are automatically cleaned up, and openings are made for symbols that orientthemselves automatically.

Openings are represented twice, once each for 2D and 3D. Blocks inserted into openings are seen correctly in orthographic (floor plan) and isometric (3D) views. Roofs can be placed on any 3D object, not just walls.



You access these tools through the **Tools | Architecture** menu. From there, you can draw walls, roofs, openings, windows, and doors. The **Tools | FloorPlan Print Space Maker** menu imports 3D architectural drawings made in FloorPlan.

Data Sharing

AutoCAD provides just few tools for creating mechanical drawings. There's a block library of bolts, a user interface suitable for 3D modeling, and a weak history for editing solid models. In fact, if you want mechanical tools, AutoDesk suggests you purchase a new program such as

The **Data Sharing** palette displays editable history trees of 3D operations, such as blends, Booleans, and shells. You can use it to apply and change chamfers and fillets; change the size

Data Sharing

TurboCAD supports a diverse range of file formats that are not found in AutoCAD or AutoCAD LT, enhancing its versatility in various design and drafting environments. This extensive range of formats underscores TurboCAD's capability to integrate with a wide array of applications and workflows, setting it apart from its counterparts.

The additional file formats that go above and beyond AutoCAD include:

- ACIS v29 and earlier (V20 SAT, SAB, ASAT)
- 3DXLM (3DXML)
- VRML Worlds (3DV)
- 3D Manufacturing Format (3MF)
- Windows Bitmap Format (CGM)
- Collada Model (DAE)
- DesignCAD (DC,DCD)
- Adobe PostScript (EPS)

- Fast CAD (FCD, FCW)
- VRML Words (GEO)
- Graphics Interchange Format (GIF)
- Autodesk Inventor Assembly File (IPT, AIM)
- Google Map Format (KML, KMZ)
- Wavefront (OBJ)
- Adobe Portable Document Format (PDF)
- Adobe 3D Portable Document Format (PDF)
- Adobe PRC Portable Document Format (PDF)
- Plotter file format language (PLT)
- Product Representation Compact for 3D PDF (PRC)
- Shape Format (SHX)
- TurboCAD Drawing Template (TCT)
- TurboCAD Native format (TCW)
- Universal 3D (U3D)
- Windows MetaFile (WMF)
- VDA-FS (VDA)
- VRML Worlds (WRL)
- VRML Worlds (WRZ)
- Terrain Data (XYZ)

Pricing

TurboCAD offers both Permanent License pricing and Annual Subscription pricing, whereas AutoCAD is only available on a Subscription basis.

When You Need More Help

If you need more help, refer to the on-line documentation available in TurboCAD. Select **TurboCAD Help Topics** from the **Help** menu.

In Appendix, at the back of this manual, we provide a list of AutoCAD's commands, along with the equivalent in TurboCAD, and an explanation of differences. Throughout the text, we provide a translation of jargon unique to TurboCAD. The AutoCAD jargon is listed in parentheses following "a.k.a." (short for "also known as"). For example:

line style (a.k.a. linetype)

Appendix provides a TurboCAD-AutoCAD dictionary, listing TurboCAD jargon in alphabetical order with the AutoCAD translation.

1

Setting Up Drawings

Lesson 1: Scale, Units, Layers, and Save

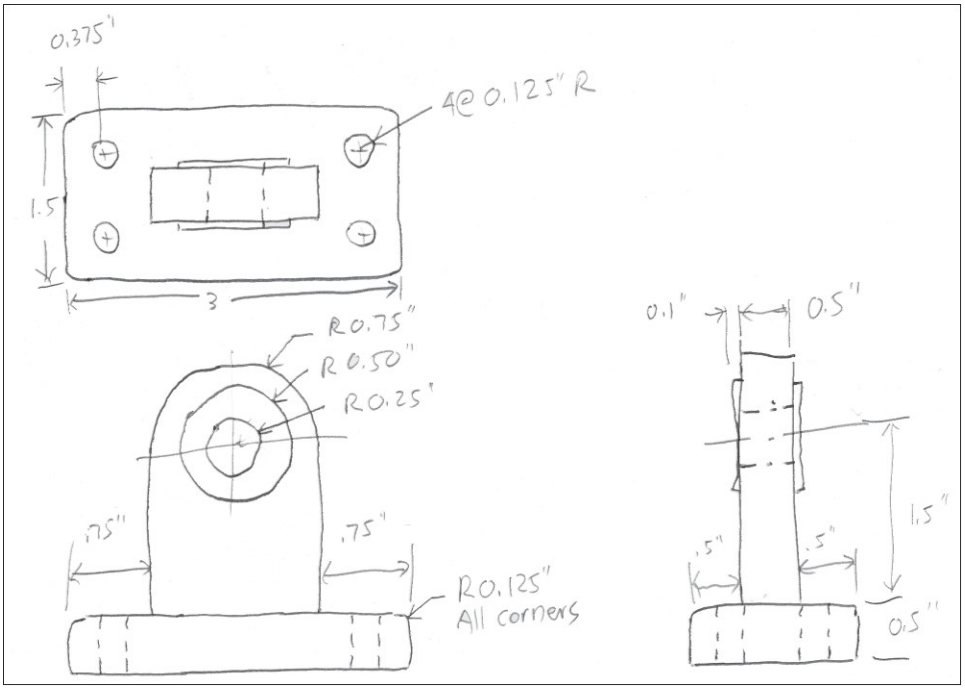
To understand better how TurboCAD compares with AutoCAD, we invite you to sit through a parallel design session. On the left-hand page, AutoCAD's actions are described; on the right side, TurboCAD's. We'll work with the most recent releases available at the time of writing this book: AutoCAD 2026 and TurboCAD 2026.

Because you are most familiar with AutoCAD, we recommend you work through this tutorial with TurboCAD running on your computer. As you work through the lessons on the right-hand "TurboCAD" pages, keep an eye on the left-hand "AutoCAD" pages. That will help alert you to the similarities and differences between the two CAD programs.

About the Tutorials

This book includes two tutorials, one for 2D drafting and one for 3D modeling.

For the 2D tutorial, you will be creating an engineering-view drawing of a mechanical part small enough to fit a standard A-size sheet of paper. This drawing lets you practice placing lines, construction lines, arcs, dimensions, and applying fillets. The drawing is sketched on the next page.



Hand sketch of 2D drawing exercise

Lesson 1: Scale, Units, Layers, and Save

This chapter contains the first lesson. In it, you learn how to set up the drawing in preparation for creating the mechanical part. Setting up new drawings includes these steps:

- Starting the CAD software.
- Using the Page Setup Wizard to set the drawing scale and other parameters.
- Creating the layer names.
- Setting the snap and grid spacing.
- Naming the drawing.

Lesson 2: Drawing, Editing, and Block Creation

In lesson 2, you begin to draw the mechanical part by placing objects, such as rectangles, lines, circles, and arcs. You also learn how to create and place blocks.

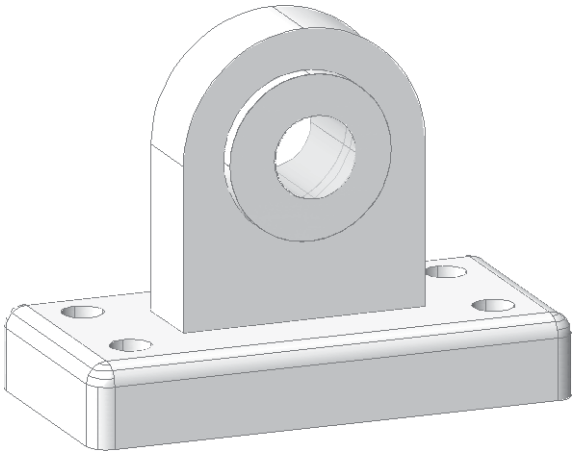
Lesson 3: Dimensioning, Paper Space, Text, and Plotting

In lesson 3, you apply the finishing touches to the 2D mechanical drawing:

- Dimensioning using horizontal, vertical, and radial dimensions.
- Switching to paper space and inserting a title block.
- Placing and modifying text.
- Plotting the drawing with the **Page Setup** and **Print** commands.

Lesson 4: Modeling in Three Dimensions

In lesson 4, you create the same part as a 3D solid model. The model lets you practice extruding objects to 3D solids, and then editing them.



Rendering of 3D modeling exercise

Starting AutoCAD

To prepare the drawing, you use the Advanced Setup Wizard to set drawing parameters.

To start AutoCAD, double-click its icon. (Alternatively, click the Windows Start button, and then select Programs, AutoCAD [version number], and AutoCAD [version number] again.) Notice that AutoCAD appears.

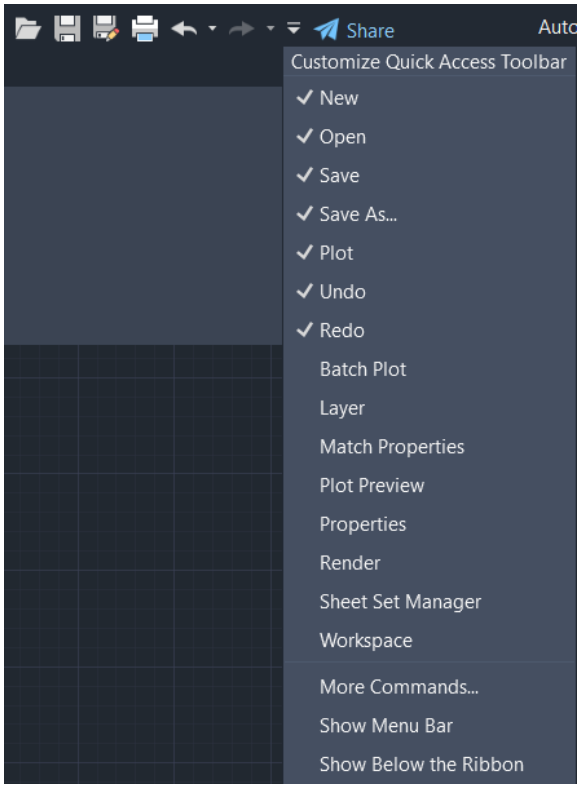
Both AutoCAD and TurboCAD start up by displaying one or more dialog boxes. User can choose to don't show these dialogue boxes at start option for

By Default, Ribbon UI is set as default workspace.

On start, you will land on **Start** Tab which holds Open Files, Open a Sheet Set & Example/last opened files.

You can start by just clicking on Start Drawing button.

For your help, you can use **Menu bar** from Classic UI in Ribbon UI:



AutoCAD: Advanced Setup Wizard

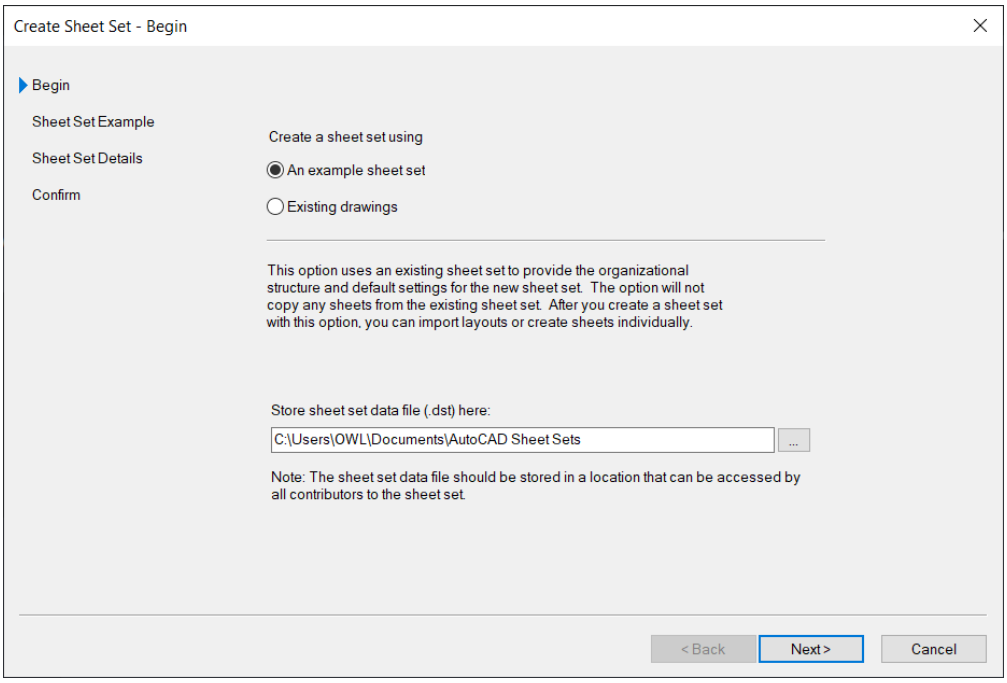
Use the Advanced Setup Wizard to help set the parameters for this new drawing.

1. You can access the wizard by selecting **New** from the **File** menu — provided that the **Startup** system variable is set to **1**. Notice that AutoCAD displays the Create New Drawing dialog box.

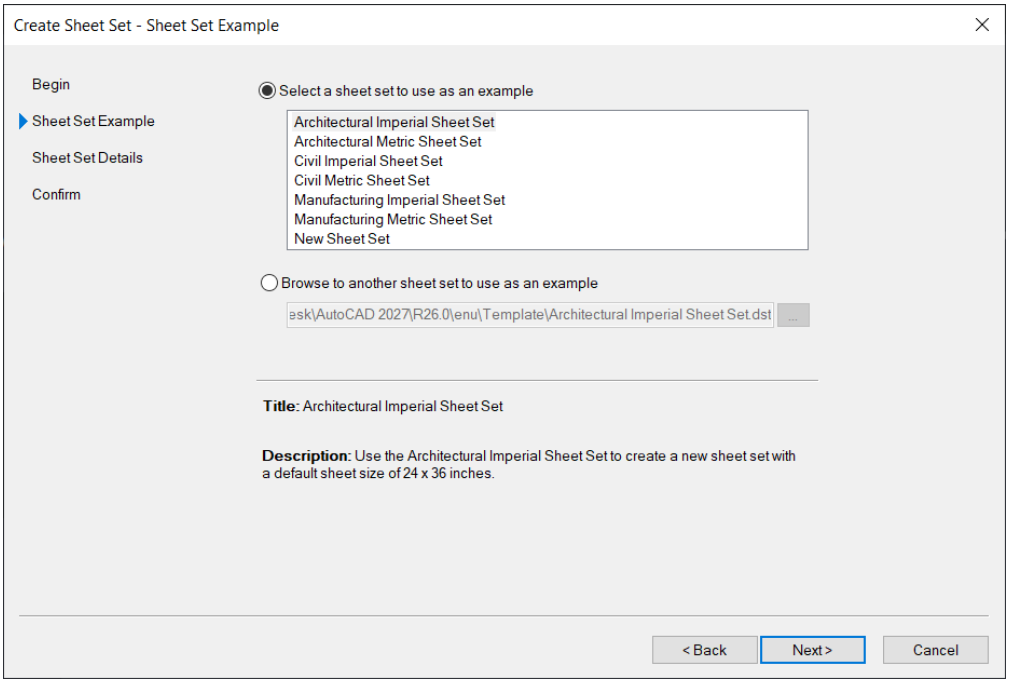
   a. Click the **Use a Wizard** button.

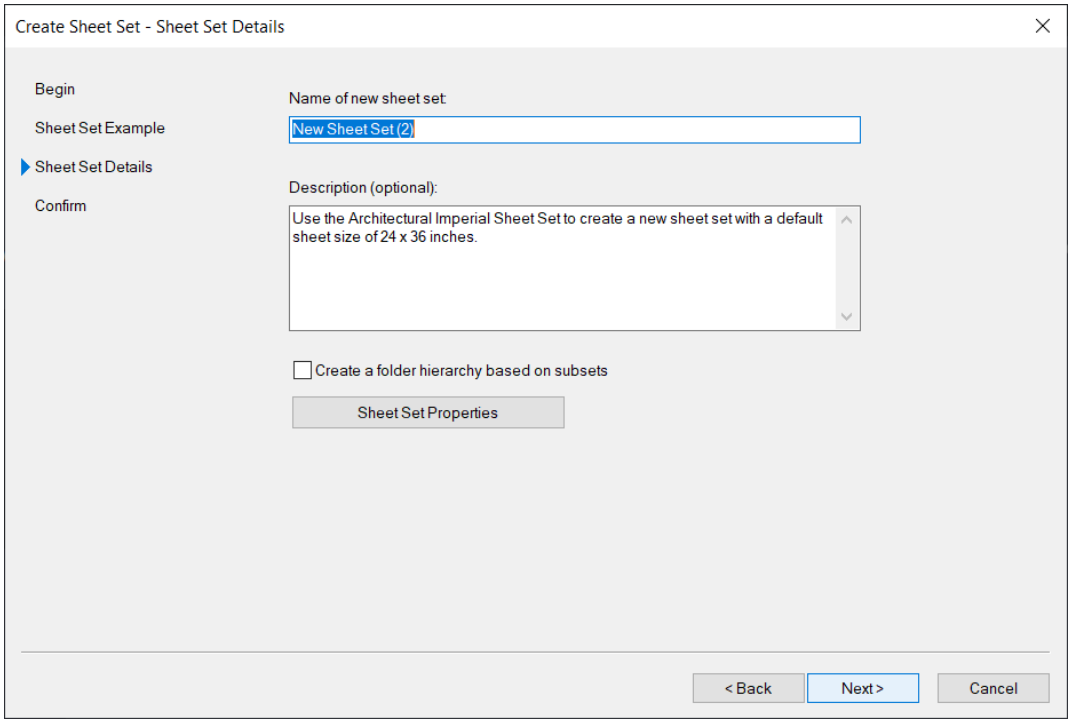
   b. Select **Advanced Setup**, then click **OK**.

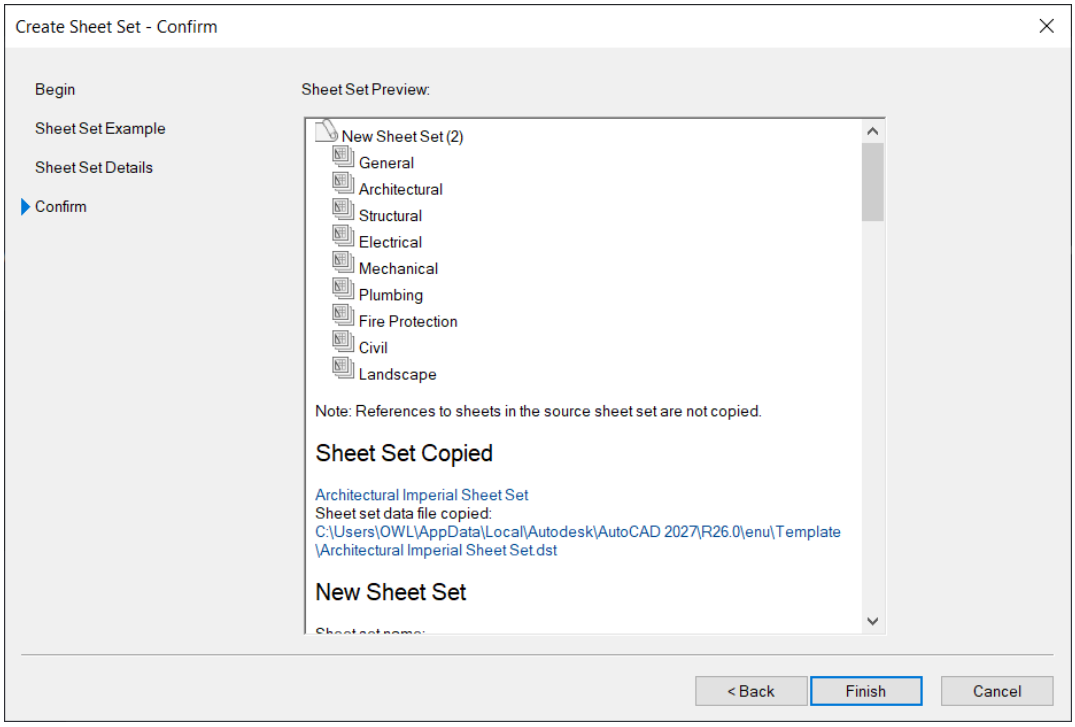
   AutoCAD displays the Units step of the Advanced Setup dialog box.



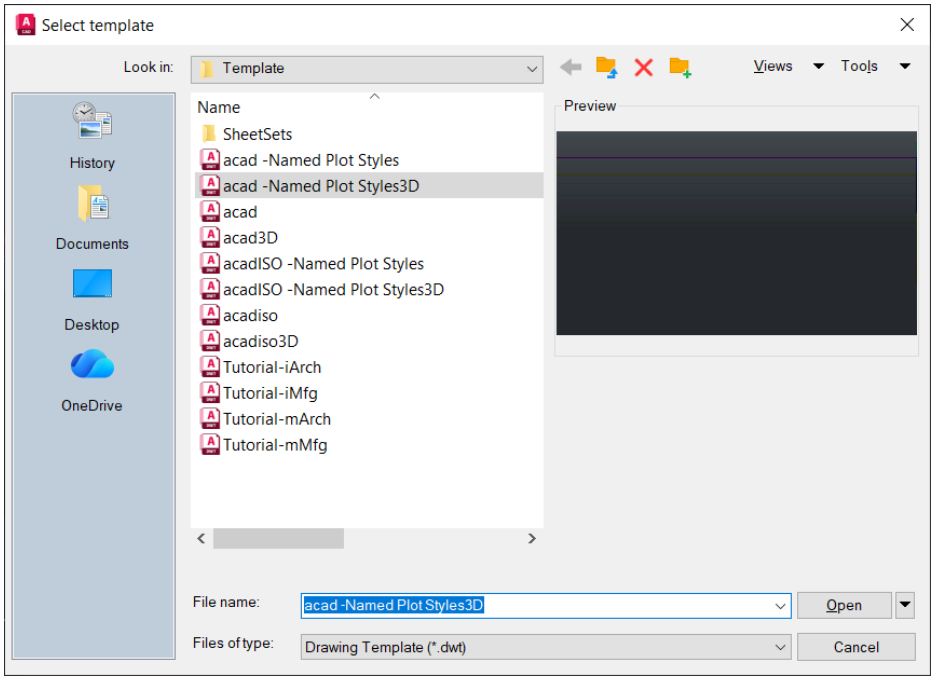
AutoCAD's Create New Drawing dialog box




Create Sheet Set - Sheet Set Details
Begin
Sheet Set Example
Sheet Set Details
Confirm
Name of new sheet set:
New Sheet Set (2)
Description (optional):
Use the Architectural Imperial Sheet Set to create a new sheet set with a default sheet size of 24 x 36 inches.
Create a folder hierarchy based on subsets
Sheet Set Properties
< Back
Next >
Cancel


Create Sheet Set - Confirm
Begin
Sheet Set Example
Sheet Set Details
Confirm
Sheet Set Preview:
New Sheet Set (2)
General
Architectural
Structural
Electrical
Mechanical
Plumbing
Fire Protection
Civil
Landscape
Note: References to sheets in the source sheet set are not copied.
Sheet Set Copied
Architectural Imperial Sheet Set
Sheet set data file copied:
C:\Users\OWL\AppData\Local\Autodesk\AutoCAD 2027\R26.0\enu\Template\Architectural Imperial Sheet Set.dst
New Sheet Set
< Back
Finish
Cancel

For Templates:



Starting TurboCAD

To prepare the drawing, you use the Page Setup Wizard to set drawing parameters.

To start TurboCAD, double-click its icon. (Alternatively, click the Windows **Start** button, and then select go to programs, **IMSI TurboCAD 2026**, and then **TurboCAD 2026**.) Notice that TurboCAD appears.

Both AutoCAD and TurboCAD start up by displaying one or more dialog boxes. User can choose to don't show these dialogue boxes at start from TurboCAD settings. By default, these dialogue boxes don't show.

If these appear, then take the following actions:

- TurboCAD WorkSpace Style dialog box — select "e.g. Ribbon Style"
- New TurboCAD 2023 Drawing Dialogue – Choose "**New from Scratch**"

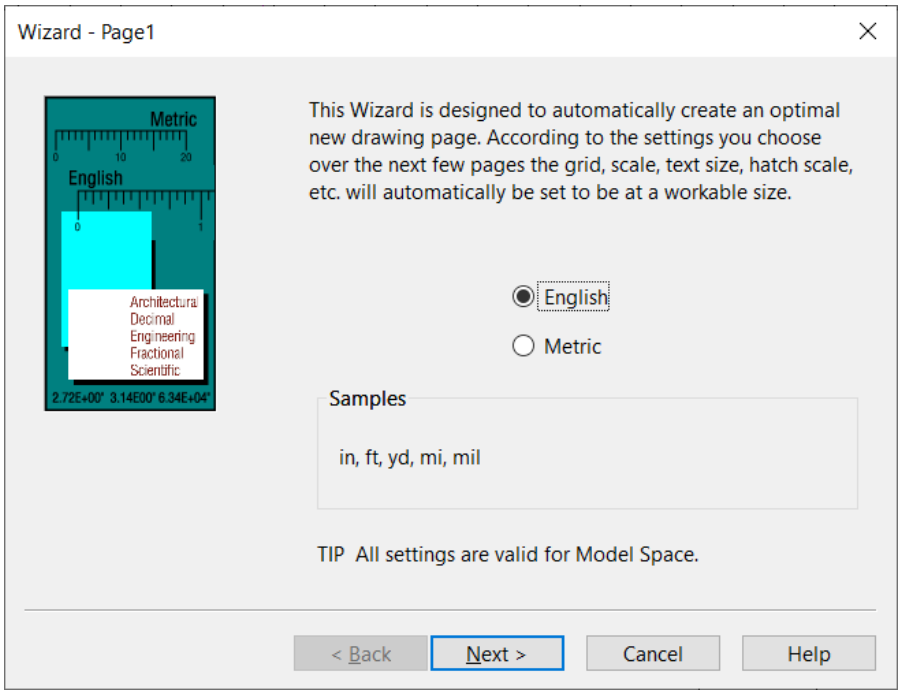
  This dialogue box also has options of **New from Template**, **Page Setup Wizard**, **Open Drawing**, **Open Last Edited, Cancel**

TurboCAD: Page Setup Wizard

Use the Page Setup Wizard to help set the parameters for this new drawing.

1. You access the wizard from the **File** menu: select **Page Setup Wizard**.

   Notice that TurboCAD displays the Wizard dialog box.



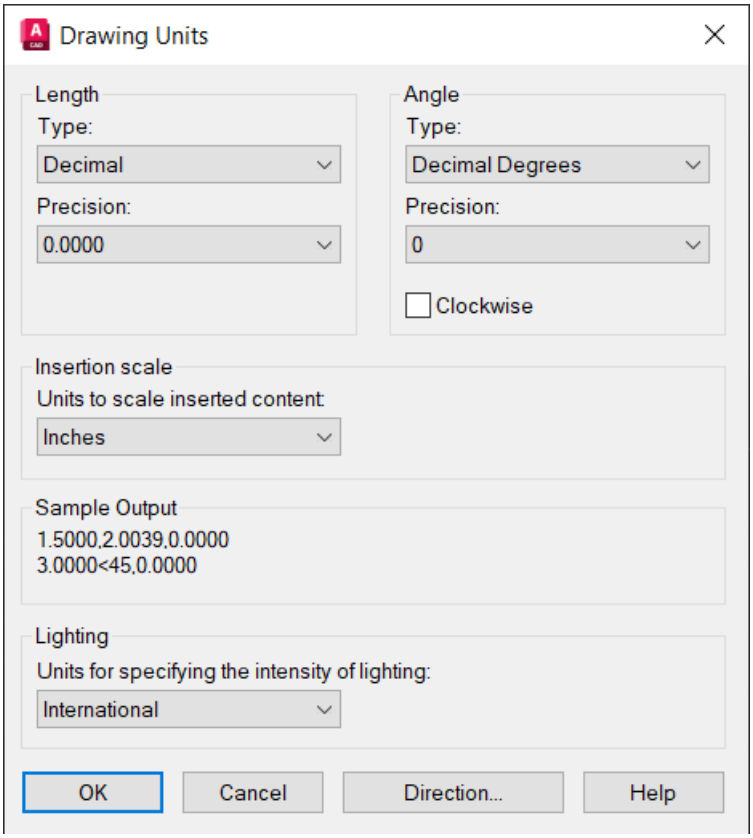
TurboCAD's Wizard dialog box

2. Setup **Units** from Drawing Units window found at **Format | Units** and select the display format of units.

   a. Select **Engineering** units and a precision of **2** decimals places (looks like 0'0.00").

3. Now to you need to setup Angle on same window.

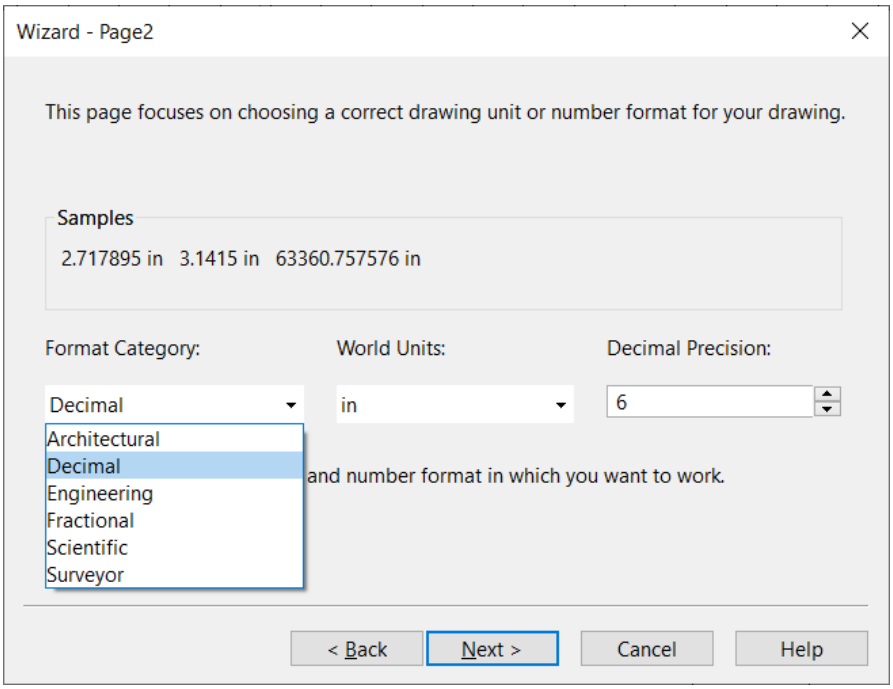
   Accept the default of Decimal Degrees and 0 decimal places by clicking Next. Leave the Clockwise option as unchecked.



AutoCAD's Drawing Unit / Angle dialog box

2. Page 1 selects the display format of units. Select the default of English units. Click **Next**.
3. TurboCAD displays the Wizard - Page 2 dialog box for specifying the drawing units.
   a. Select **Engineering** (ft-in).
   b. Accept the default Decimal Precision of **2** places by clicking Next.

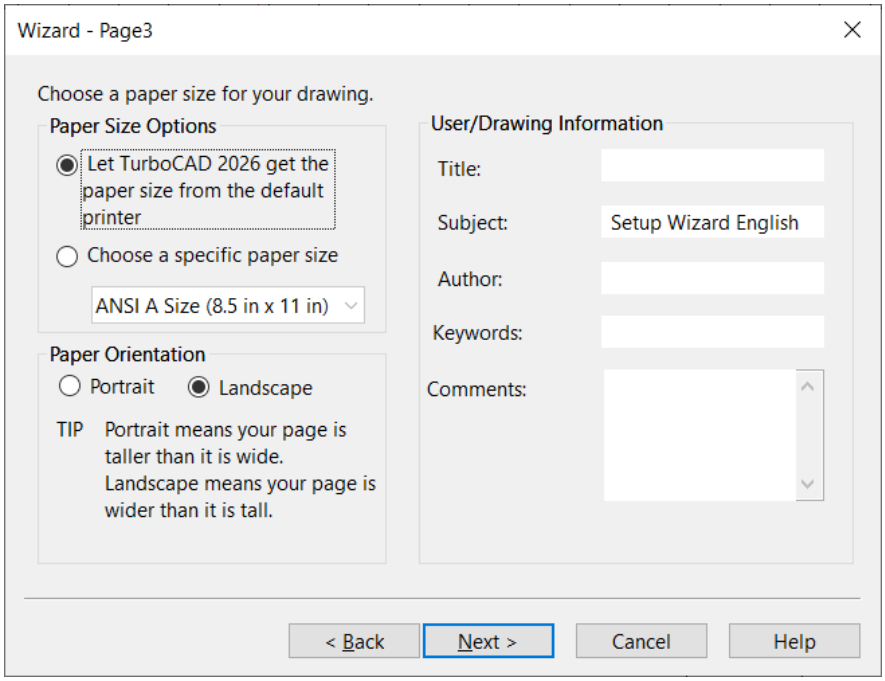
4.



TurboCAD's unit's setup dialog box

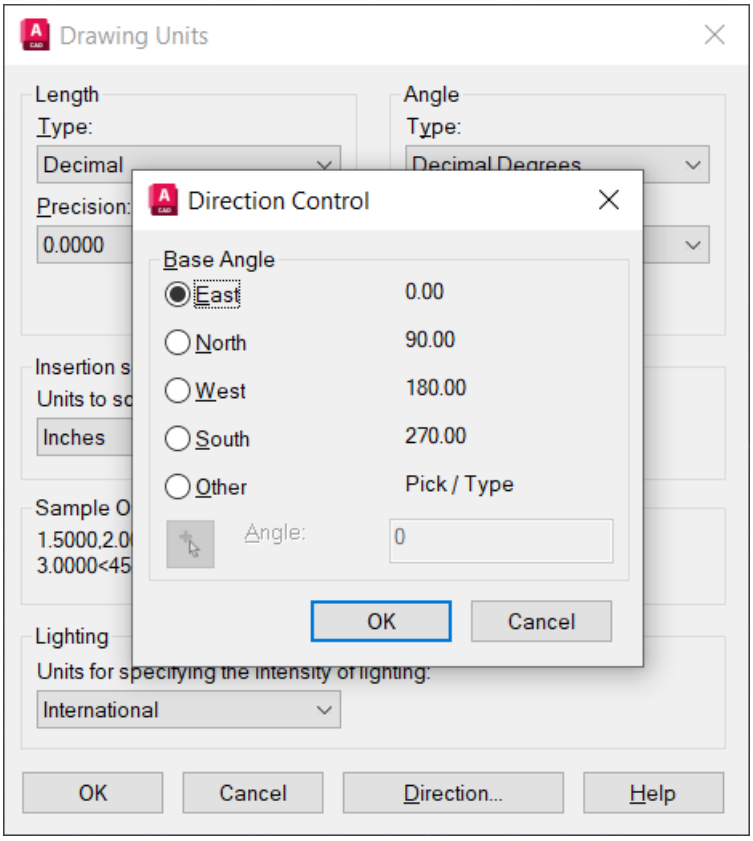
5. TurboCAD displays the dialog box for Page 3.

a. Under Paper Size Options, select **Choose a Specific Paper Size** and then ensure that "ANSI A Size" is showing in the droplist. For Paper Orientation, select **Landscape**.



TurboCAD's paper size dialog box

6. Now you need to set Angle Direction. Click on **Direction** button. Direction Control window will show. Accept the default base angle East.



AutoCAD's Angle Direction Control dialog box

In the User/Drawing Information section, enter the following data:

Section	Example Text
Title	Mechanical Part
Subject	TurboCAD forAutoCAD Users
Author	Your Name
Keywords	Tutorial, Mechanical

Click **Next**.

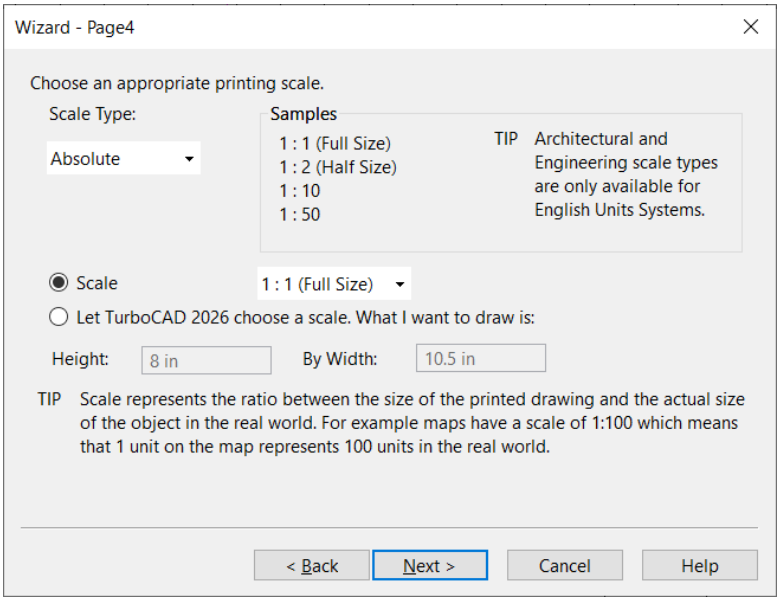
5. Page 4 of the wizard determines the scale. This is perhaps the most important step, because it affects the scale factor of scale-dependent drawing elements, such as text, dimensioning, and hatch patterns. Examine the mechanical part you plan to draw, and then add up the widths and heights of each view. Allowing a half-inch around the edges for dimensioning:

   - The overall width is **6"** (0.5" + 3" + 0.5" + 1.5" + 0.5")
   - The overall height is **6.5"** (0.5" + 3.5" + 0.5" + 1.5" + 0.5")

   a. Since you want to fit the drawing on an A-size sheet, which is 11" wide and 8.5" tall, select the Scale Type of Engineering.

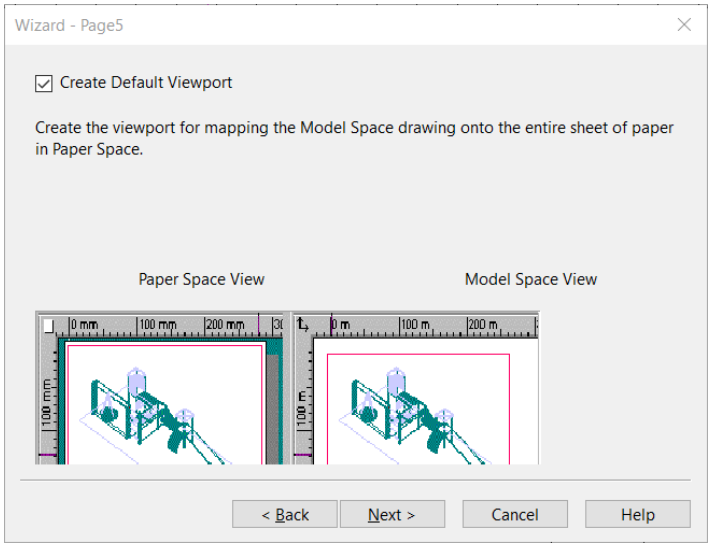
   b. Specify the **Scale** of "1"=1" (Full Size)."

   c. Click **Next**.



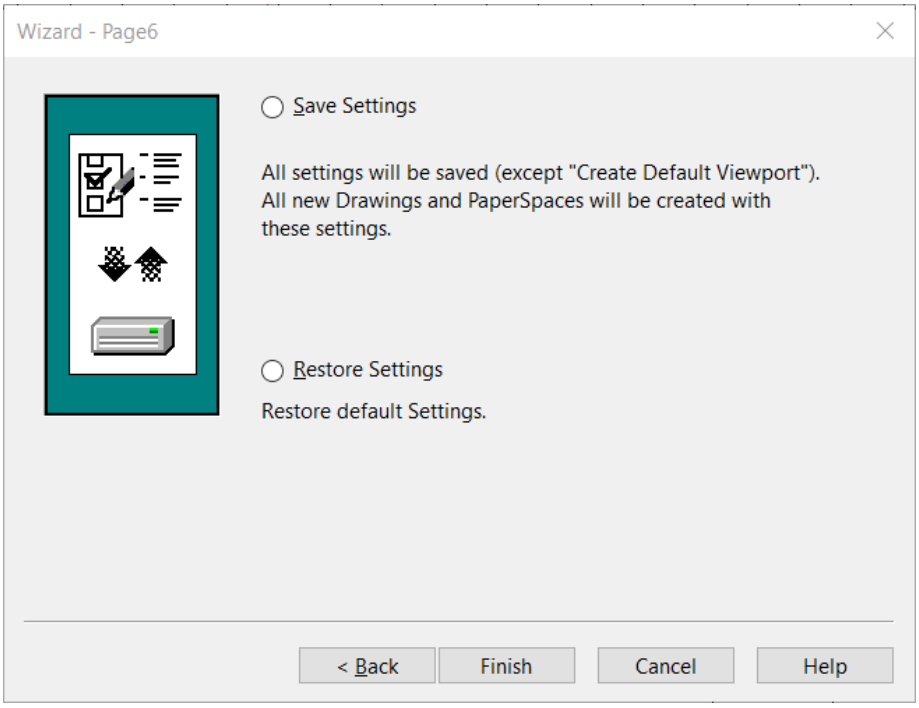
TurboCAD's Wizard - Page 4 dialog box

6. Page 5 of the wizard asks if you would like to create a viewport in paper space. This is useful for lesson 3, so click the **Create Default Viewport** option, and then Next.



TurboCAD's Wizard - Page 5 dialog box

7. Finally, Page 6 of the wizard asks if you would like to save the settings for the next time you start a new drawing. This is beneficial when all your drawings have similar settings.



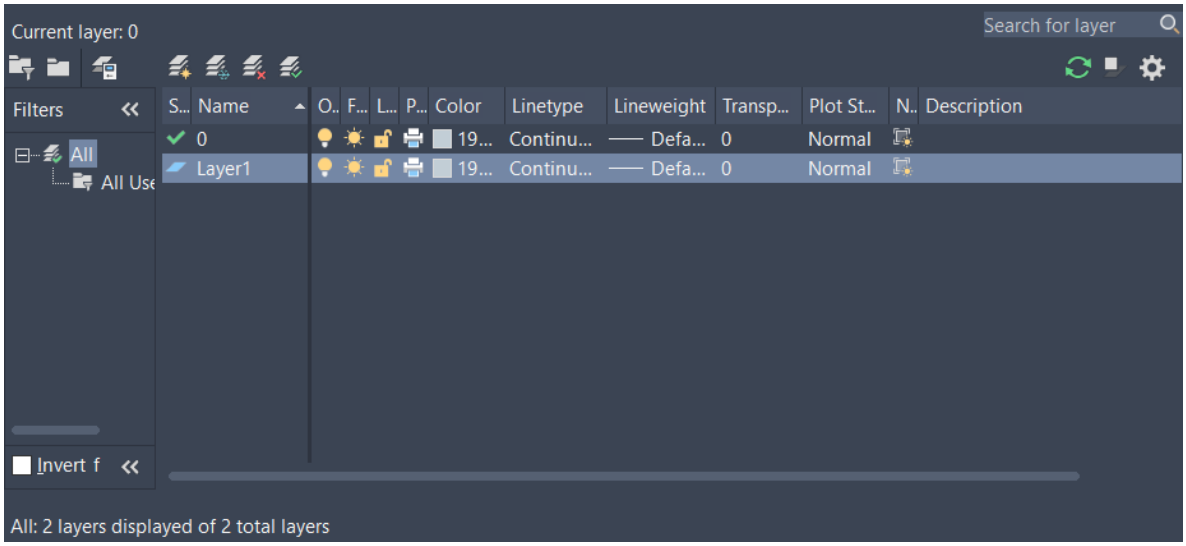
TurboCAD's Wizard - Page 6 dialog box

For this tutorial, just click Finish.

AutoCAD: Creating Layers

The next step in setting up new drawings is adding layer names and properties. In every new drawing, AutoCAD automatically creates a single layer, named "0," just like TurboCAD.

1. Create layers by selecting **Layer** from the **Format** menu. AutoCAD displays the Layer Properties Manager dialog box.



AutoCAD's Layer Properties Manager dialog box

2. Create four new layers with the properties listed in the table below:

New Layer Names	Color	Linetype
Steel	Black	Continuous
Dims	Red	Continuous
Hatch	Yellow	Continuous
Hidden	Blue	Hidden

To create new layers, click the **New Layer** icon. AutoCAD creates a new layer with the default name of "Layer1."

3. To name the layer, select the "Layer1" text, and then replace it with the proper name. In this case, type "Steel," and then press **Enter**.

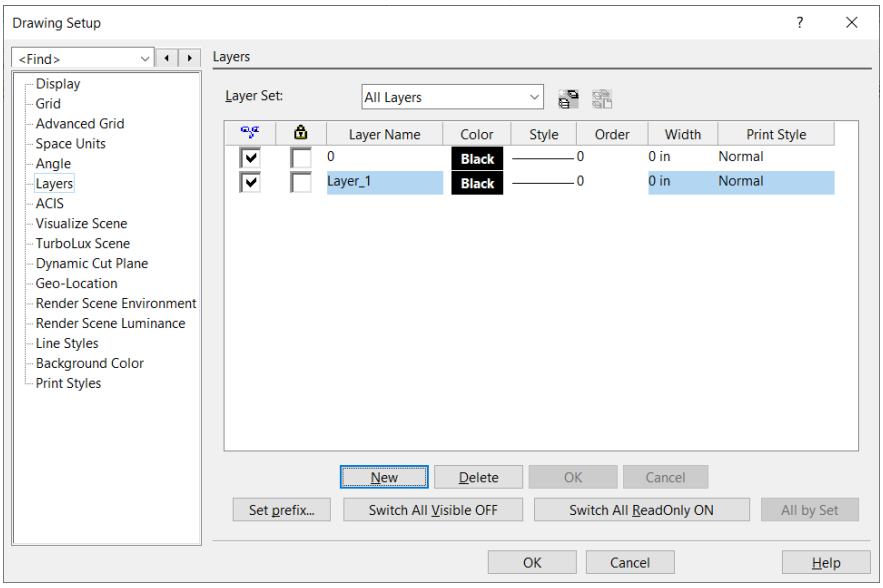
4. To create layer Dims, click **New Layer** and type "Dims."

TurboCAD: Creating Layers

The next step in setting up new drawings is adding layer names and properties. In every new drawing, TurboCAD automatically creates a single layer, named "0," just like AutoCAD.

1. From the **Options** menu, choose **Drawing Setup - Layers**. TurboCAD displays the Drawing Setup dialog box with the Layers view showing.

   Notice that this dialog box handles many aspects of setting up drawings, such as specifying the grid, setting the rendering environment, and creating new line styles.



TurboCAD's Drawing Setup dialog box for layers

2. Create four new layers with the properties listed in the table below:

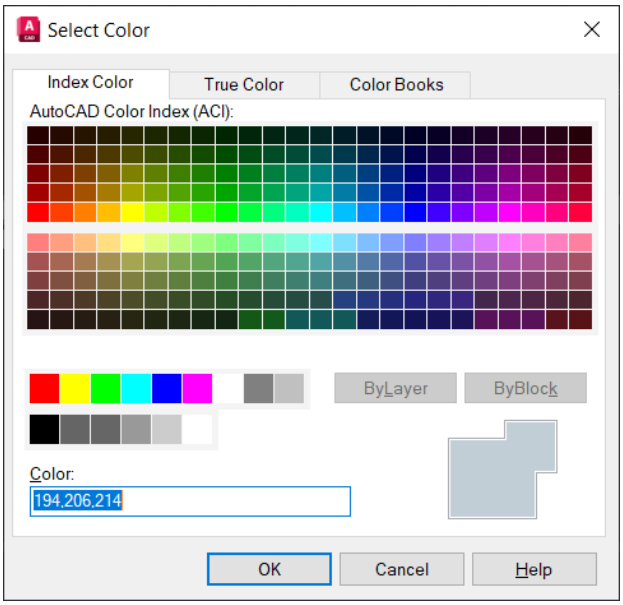
New Layer Names	Color	Style (*Linetype*)
Steel	Black	Continuous
Dims	Red	Continuous
Hatch	Yellow	Continuous
Hidden	Blue	Hidden

   To create new layers, click the **New** icon. TurboCAD creates a new layer with the default name of "Layer_1."

3. To name the layer, select the "Layer_1" text, and then replace it with the proper name. In this case, type "Steel," and then press Enter.

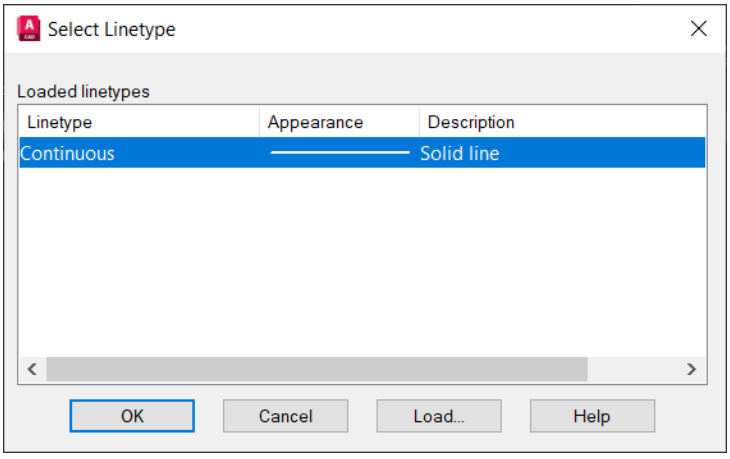
4. To create layer Dims, click **New** and type "Dims."

To change the color of this layer from black to red, click the small black square in the **Color** column. AutoCAD displays the Select Color dialog box:



AutoCAD's Select Color dialog box

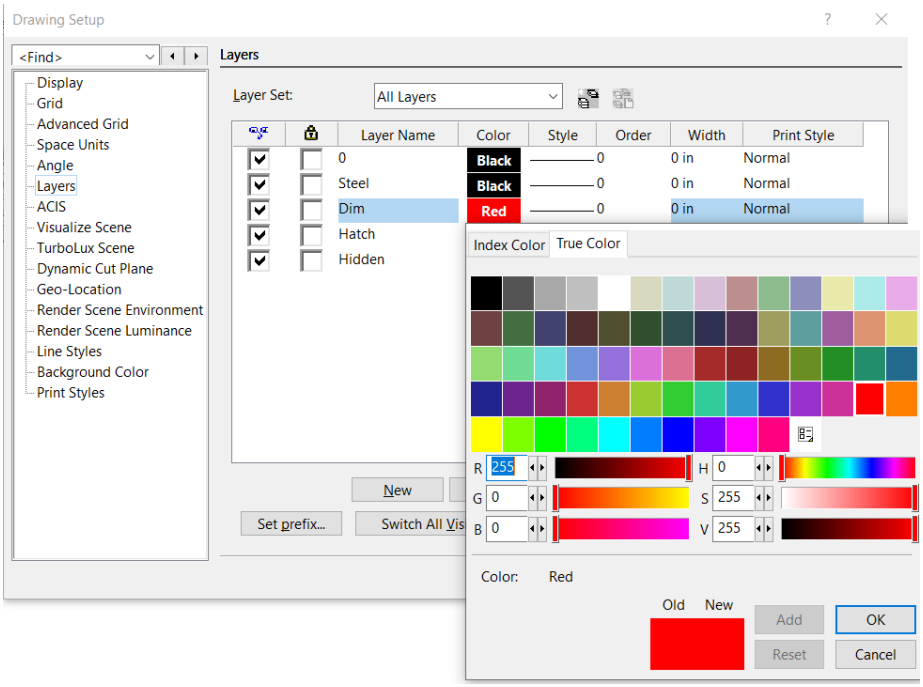
5. Click the red square, then click **OK**. AutoCAD changes the layer color from black to red.
6. Add layer "Hatch," and then change its color to yellow.
7. Add layer "Hidden," and then change its color to blue.
8. To change the linetype to Hidden, click the word "Continuous" in the Linetype column. AutoCAD displays the Select Linetype dialog box.



AutoCAD's Select Linetype dialog box

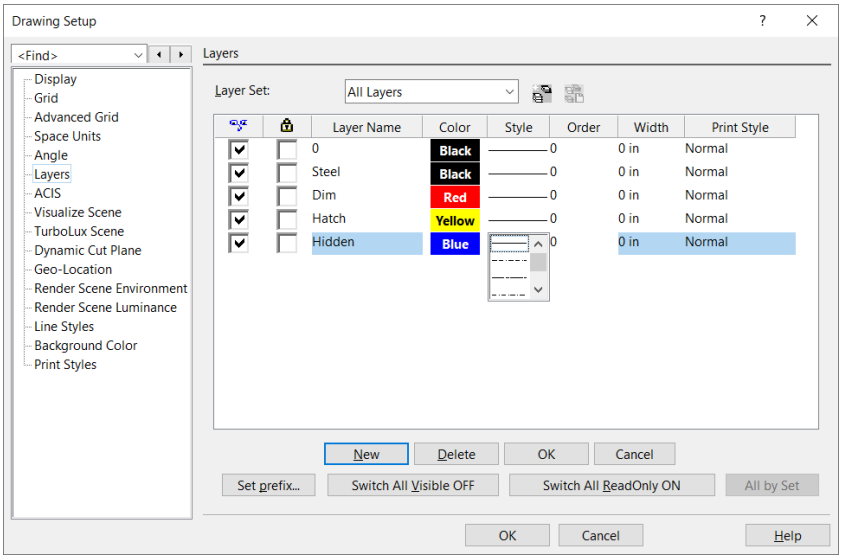
a. No linetypes are loaded in new AutoCAD drawings, so click the **Load** button. AutoCAD displays the Load or Reload Linetypes dialog box

To change the color of this layer from black to red, click the word "Black" in the Color column. TurboCAD displays another dialog box that lists its 16.7 million colors:

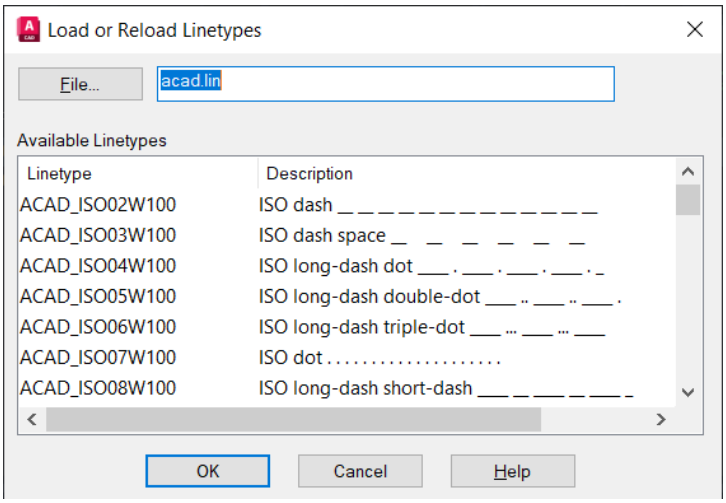


TurboCAD's layer colors

5. Double-click Red. TurboCAD changes the layer color from black to red, and then dismisses the color dialog box automatically.
6. Add layer "Hatch," and then change its color to yellow.
7. Add layer "Hidden," and then change its color to blue.
8. To change the linetype to Hidden:
   a. Click the continuous line in the **Style** (a.k.a. *linetype*) column. TurboCAD displays the linetype droplist:



b. Scroll down and select "Hidden," and then click **OK**. AutoCAD loads the linetype from the *Acad.Lin* linetype definition file into the drawing.

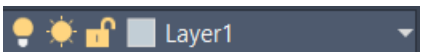


AutoCAD's Load or Reload Linetypes dialog box

c. Back in the Select Linetype dialog box, select "Hidden" and **OK**. AutoCAD sets the hidden linetype pattern to the Hidden la ye r.

9. Back in the Layer Properties Manager dialog box, click "Steel" layer, and then click the **Set Current** button to set it as the working layer.

10. Click **OK** to dismiss the dialog box. Ensure that Steel is the current layer by looking at the Layers toolbar.



AutoCAD's Layers toolbar

Notice that *all* linetypes are loaded in a new TurboCAD drawing, but are unnamed.

b. Scroll down, and then double-click the line style that looks the most like hidden. TurboCAD sets the hidden linetype pattern to the Hidden layer.

9. Click **OK** layer to dismiss the Drawing Setup dialog box.

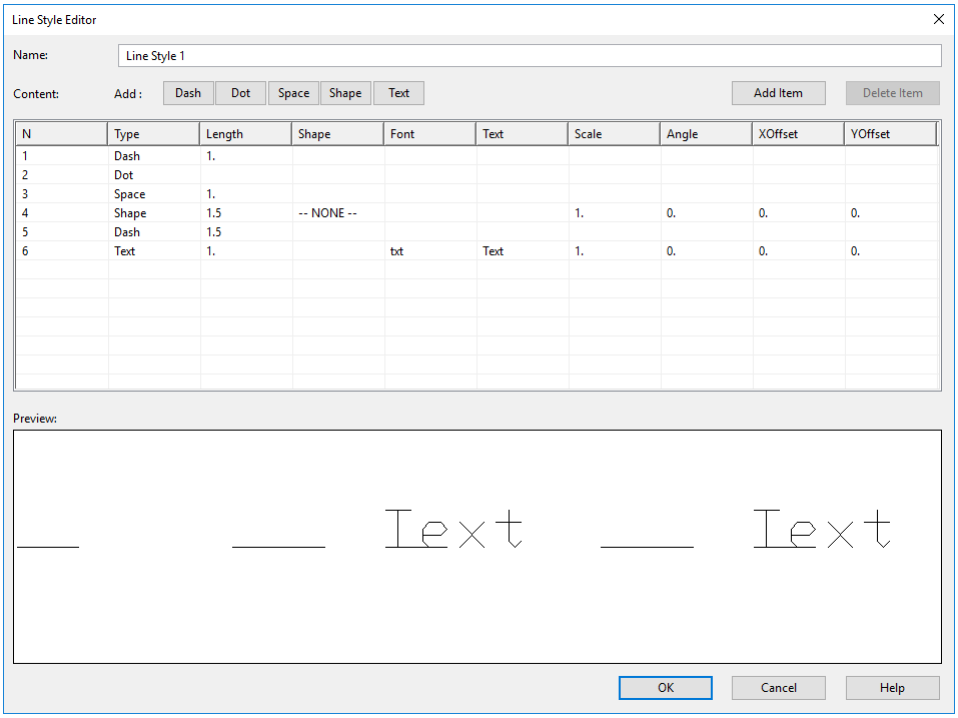
10. To set the "Steel" layer as the active (a.k.a. *current*) layer, select it from the Properties toolbar:



TurboCAD's Properties toolbar

TIPS: *In TurboCAD, layers also control display order, line width, and print styles of entities.*

*Linetypes can be constructed using the **Line Styles | New** option in the Drawing Setup dialog box. See the figure below:*



TurboCAD's built-in linetype customization facility

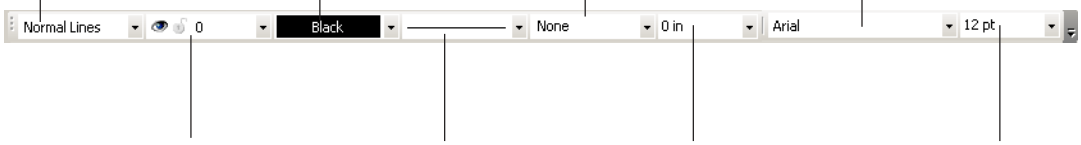
TurboCAD's Property Toolbar

ENTITY STYLE. Applies predefined property settings to a drawing tool; does not change the properties of selected objects. Create entity styles with the **Format | Properties** command.

PEN COLORS. Selects the pen color; as in AutoCAD, color "ByLayer" matches the color to the current layer. "ByBlock" matches the block color.

BRUSH PATTERN (a.k.a. *hatch pattern*): Selects a fill pattern for closed objects; patterns can be customized.

TEXT FONT: Selects a font for text (when Text tool is selected); all SHX and TrueType fonts are available.

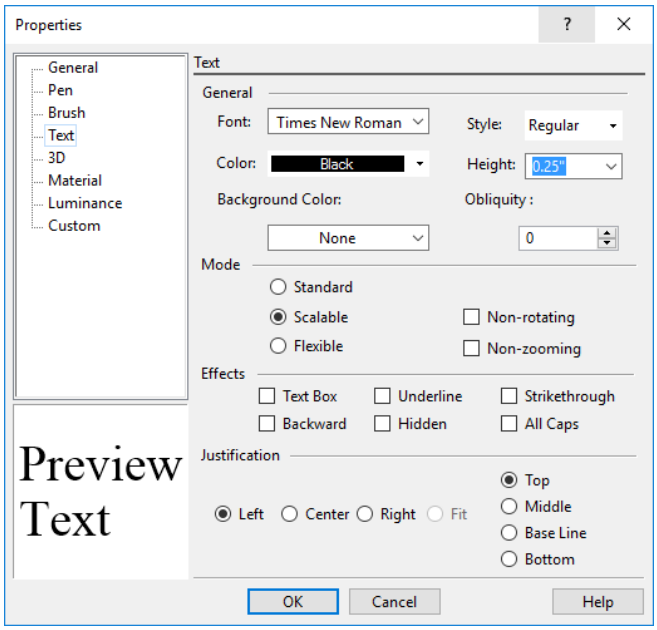


LAYERS. Sets the active (a.k.a. *current*) layer.

PEN STYLE (a.k.a. *linetype*): Selects the style of the line. "ByLayer" matches the line style to the current layer. "ByBlock" matches the block linestyle.

PEN WIDTH: Selects the width for any object.

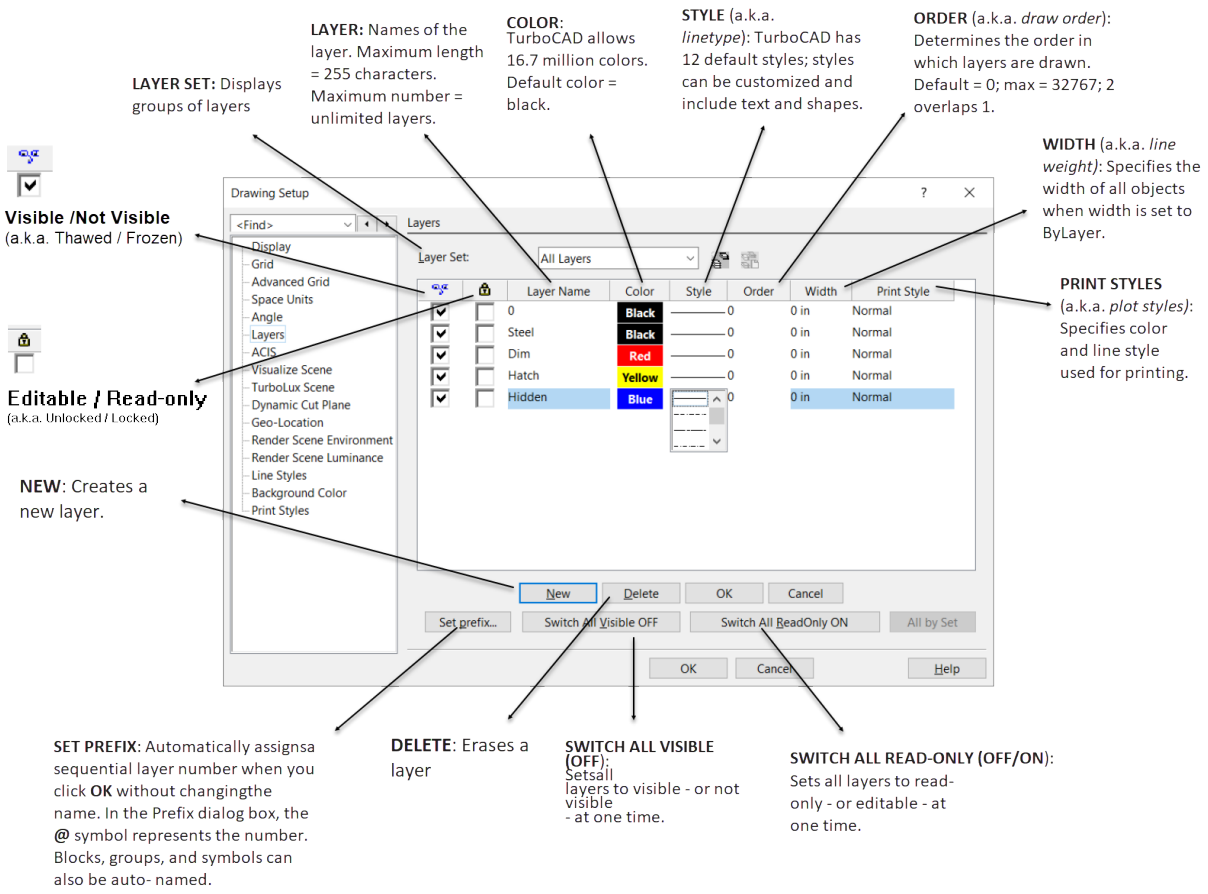
TEXT HEIGHT: Sets the height of text (when Text tool is selected).

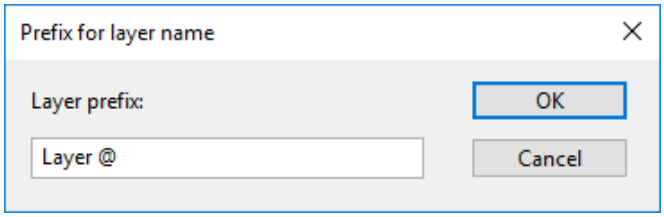


TurboCAD's Properties | Text dialog box

PROPERTIES: Provides additional brush, general, pen, and text settings with **Format | Properties**.

TurboCAD's Layers Drawing Setup





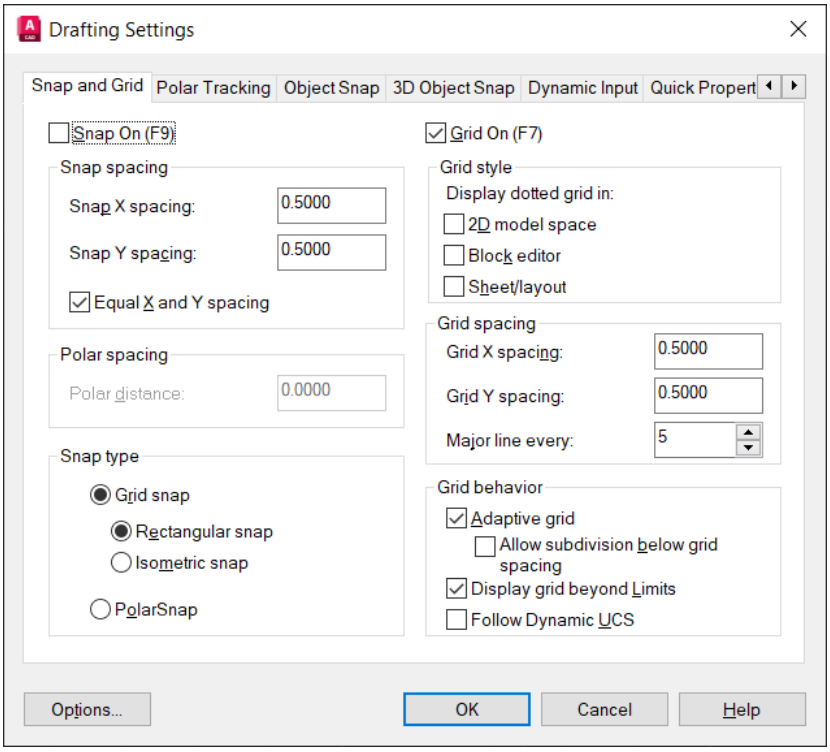
TIP: *To set the active layer, use the **Layers** list box of the **Properties** toolbar.*

AutoCAD: Setting the Snap and Grid Spacing

1. To set the snap and grid spacing, select **Drafting Settings** from the **Tools** menu. AutoCAD displays the tabbed Drafting Settings dialog box. (If necessary, click the **Snap and Grid** tab.)

   Click **Snap On** and **Grid On** to turn on the cursor snap and grid display.

2. Looking at the diagram of the mechanical part, note that you need to draw to the nearest 0.05". For the **X Spacing**, type **0.05**. Press **Tab** and AutoCAD automatically fills in the **Y Spacing** with the same value.



AutoCAD's Drafting Aids dialog box

You want the grid to display every half-inch, so for the **X Spacing** type **.5**.

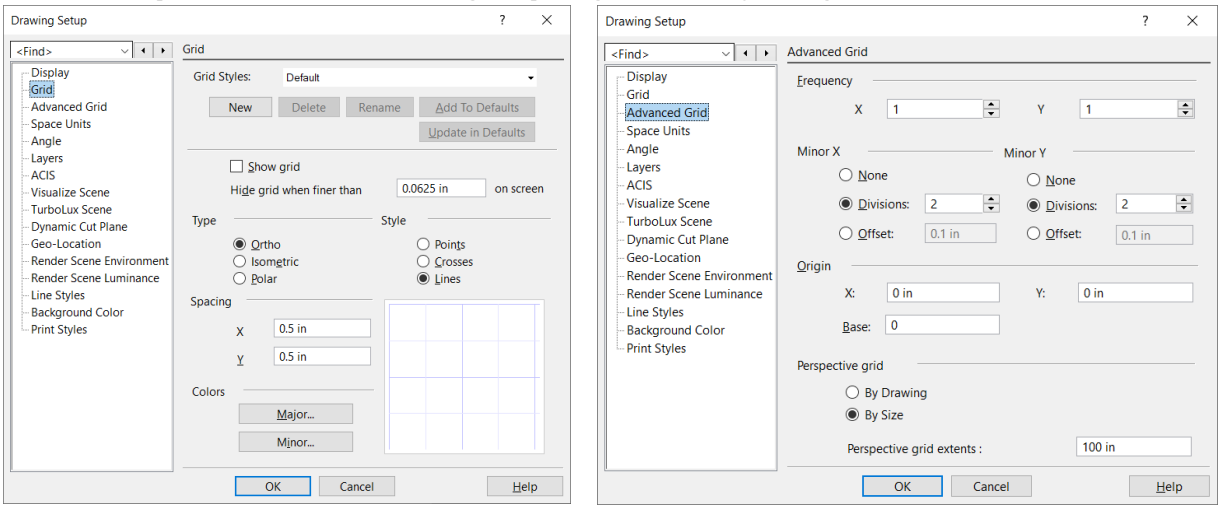
Press **Tab** and AutoCAD automatically fills in the **Y Spacing**.

3. For **Major Line Every**, type 5.

TurboCAD: Setting the Snap and Grid Spacing

1. To set the drawing modes — ortho, snap, and grid spacing — select **Grid** from the **Options** menu; this displays the Drawing Setup dialog box again.

2. Looking at the diagram of the mechanical part, notice that you need to draw to the nearest 0.05". In the **Spacing** section, set **X** and **Y** to **0.05"**. In TurboCAD, this sets the snap distance and the "minor" grid spacing, as shown by the figure below:

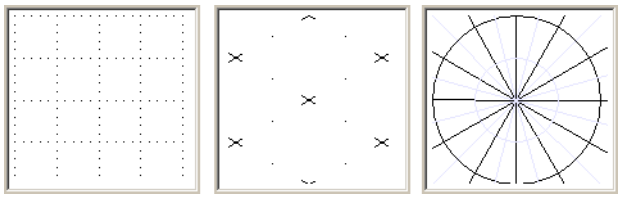


Left: **TurboCAD's Grid setup**
Right: **Advanced Grid setup**

Change the value of **Hide grid when finer than** to **0.04**.

3. To set the "major" grid (a.k.a. *the grid spacing in AutoCAD*), click the **Advanced Grid** item. For **Minor X** and **Minor Y Divisions**, type **5**. This causes TurboCAD to display the major grid at five times the spacing of the minor grid.

TIPS: *While AutoCAD has ortho, isometric, and perspective grids, TurboCAD also displays grid markings as dots, crosses, or lines. It also includes a polar grid.*

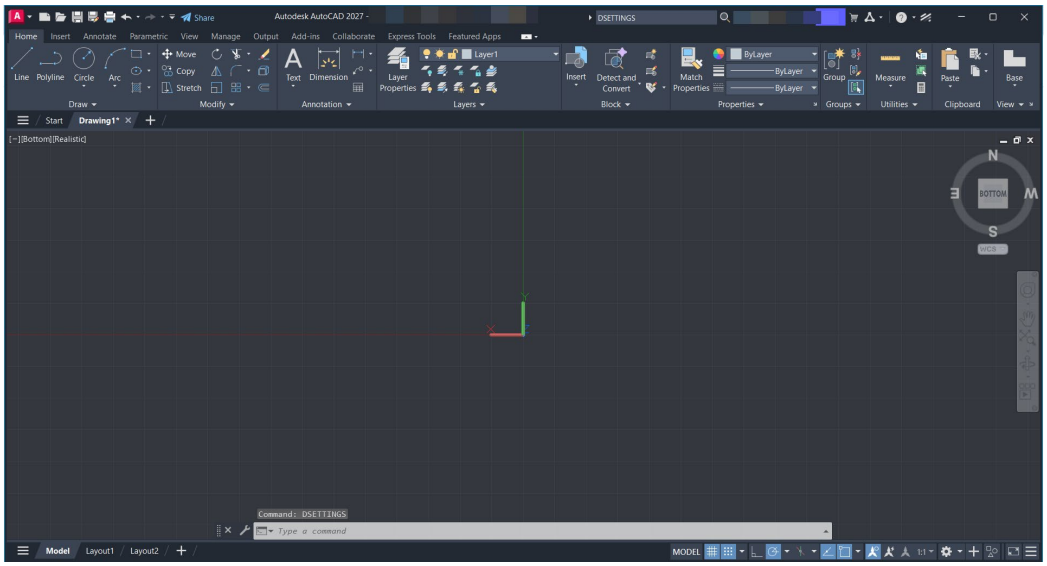


*Unlike AutoCAD, TurboCAD has no "ortho" button on its status bar to toggle orthographic mode. That's because TurboCAD draws in ortho mode any time you hold down the **Shift** key.*

4. Click **OK** to dismiss the dialog box. AutoCAD displays the grid.

   Looking at the status line, confirm that SNAP and GRID are turned on.

   Click the **ORTHO** to turn on orthographic mode.

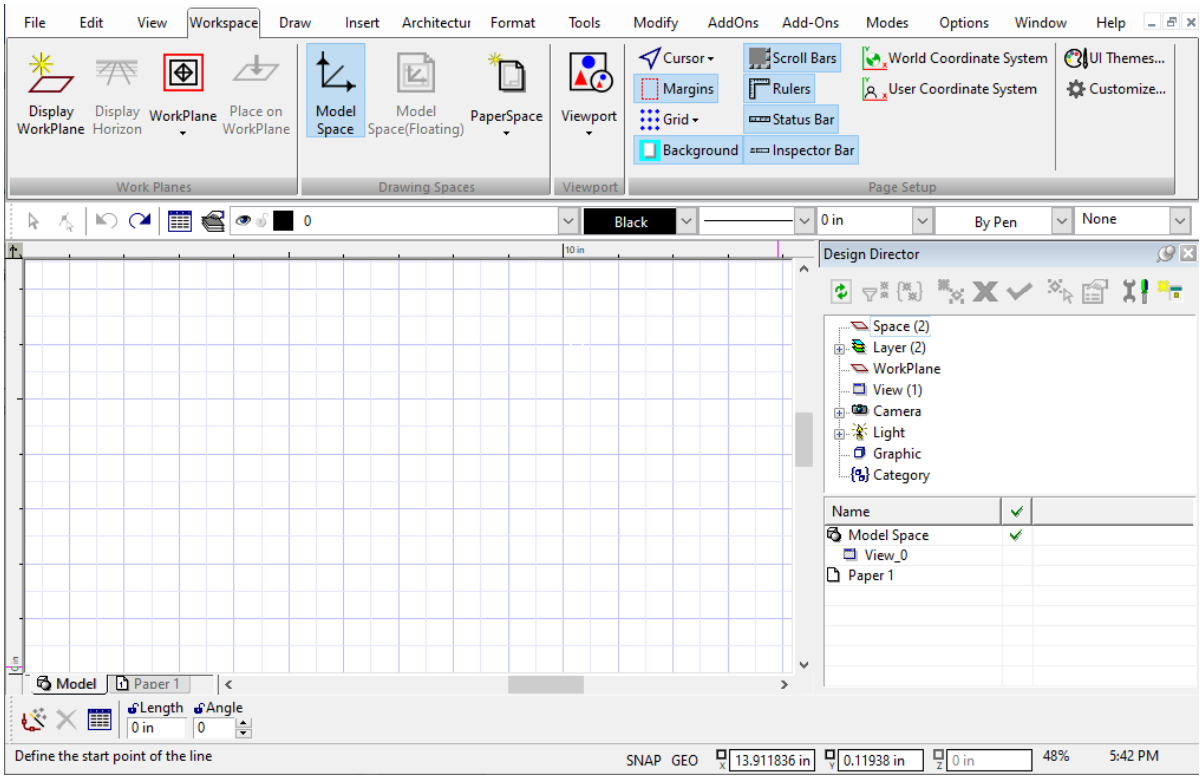


AutoCAD drawing with grid turned on

5. Use the **View | Zoom | Extents** menu pick to ensure you see the entire drawing.

4. Click **OK** to dismiss the dialog box.

   TurboCAD displays the major grid in blue and the minor grid in light blue. Looking at the ruler, confirm the grid spacing.



TurboCAD's drawing with the grid turned on

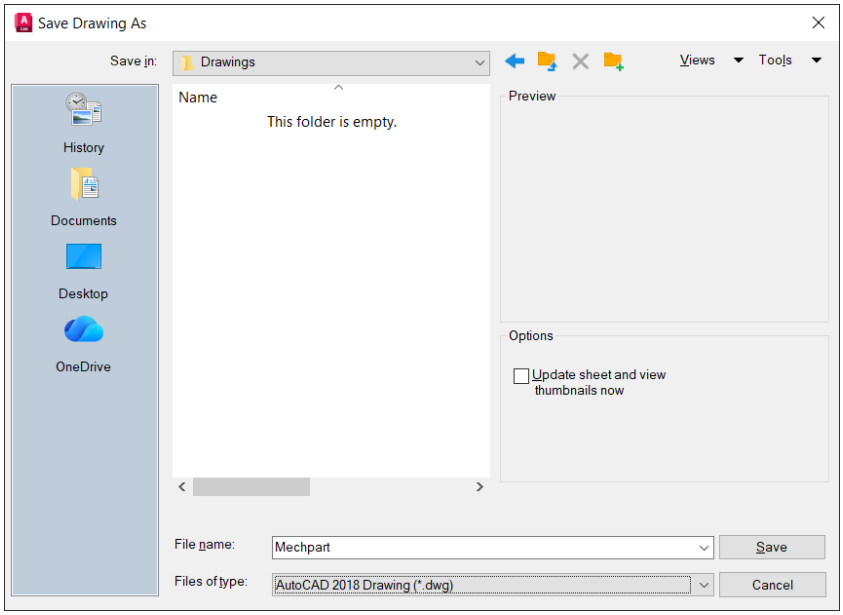
5. Use the **View | Zoom | Extents** menu pick (or press **Ctrl+Backspace**) to ensure you see the entire drawing.

TIPS: *By default, TurboCAD turns off the aperture cursor. To turn it on, select **Options | Preference**. In the Drawing Cursor area, turn on the **Show Snap Aperture** option, and then click **OK**. TurboCAD's aperture cursor is round, as illustrated below.*



AutoCAD: Saving the Drawing

1. To name the drawing, select **Save As** from the **File** menu. AutoCAD displays the Save As dialog box.



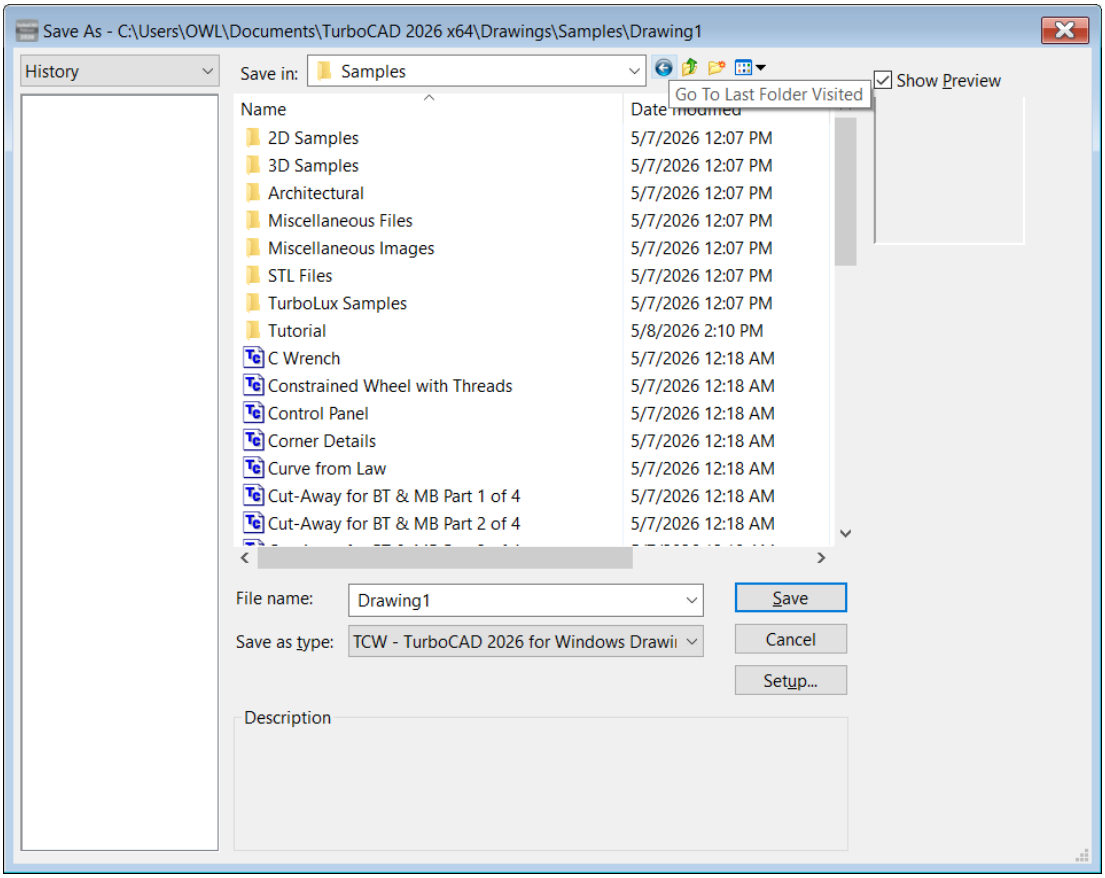
AutoCAD's Save As dialog box

2. Select a convenient folder in the **Save In** droplist.
3. For the **File Name**, type "Mechpart," and then click **Save**.

   AutoCAD saves the drawing.

TurboCAD: Saving the Drawing

1. To name the drawing, select **Save As** from the **File** menu. TurboCAD displays the Save As dialog box.



TurboCAD's Save As dialog box

2. Select a convenient folder in the **Save In** droplist. If necessary, change the type of drawing from Templates to Drawings.
3. For the **File Name**, type "Mechpart," and then click **Save** to save the drawing.
4. Notice that the Summary Info dialog box appears. Click **OK** to dismiss it.

   TurboCAD saves the drawing.

TurboCAD's Snap Modes

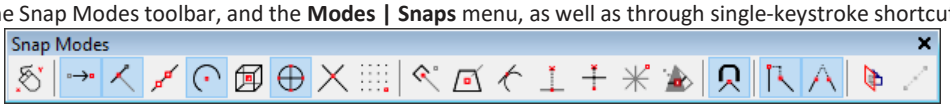
Object snaps are crucial for accurate drawings, and TurboCAD shares many snap modes with AutoCAD, plus provides several unique ones. In addition, TurboCAD has *snap priority*. This is useful when more than one snap mode is turned on, and one needs to be more important than the others.

Same as AutoCAD

The following TurboCAD snap modes are identical to AutoCAD's:

Apparent intersection	Center	Grid	Intersection	Middle
Nearest	Ortho	Perpendicular	Quadrant	
	Tangent			

You can access TurboCAD's snap modes from the local menu's Local Snap item (right-click shortcut menu), the Snap Modes toolbar, and the **Modes | Snaps** menu, as well as through single-keystroke shortcuts.



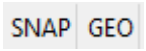
TurboCAD has running snaps (always on) and temporary snaps (on for just one selection).

Additional Snap Modes

TurboCAD provides two classes of snaps:

- **Geo** — extended ortho, apparent intersection, and show magnetic point.
- **Snap** — all other snap modes.

Using buttons on the Edit bar, you can quickly toggle between the two classes of snap modes:



TurboCAD has these snap modes not found in AutoCAD:

Center of extents snaps to the center of the 2D bounding rectangle or 3D box that defines the extents of 2D and 3D objects.

Divide snaps to division points along lines and arcs; you specify the number of segments in the Drawing Aids dialog box (default = 4). Nearest AutoCAD equivalents are M2P (midpoint two points), or the Divide command NODe Osnap.

Extended Ortho snap to points located orthogonally from the current point.

Nearest on Facet snaps to the nearest point on facets, or to projections onto the current workplane.

Opposite snaps to a point mirrored about another object.

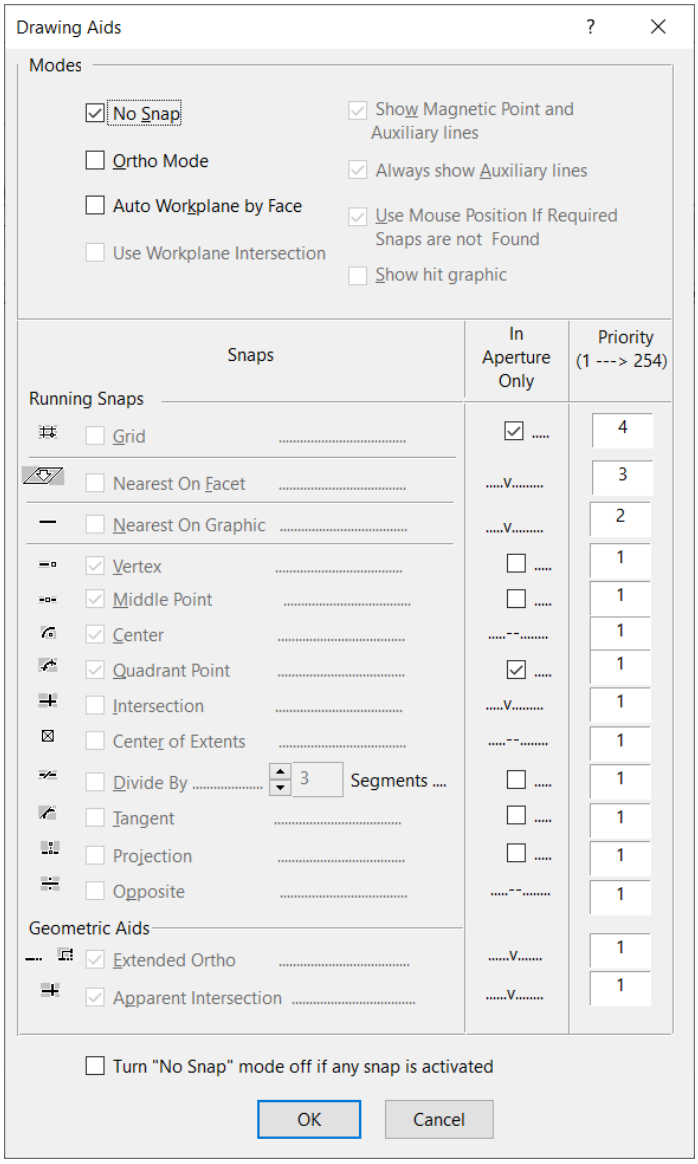
Projection snaps perpendicular to a selected object, from the last point drawn; can be on the selected object or projected from it.

Vertex snaps to endpoints of lines, arcs, and curves, and to the corners of polygons. Nearest AutoCAD equivalent is a combination of Endpoint and Intersection snaps.

Workplane intersection snaps to the intersection of 3D objects and the current workplane.

Snap Priority

TurboCAD provides a way to prioritize snaps. For example, when endpoint and quadrant snaps are both turned on, which should have priority when selecting an arc? To access snap priority, right-click **SNAP** on the Edit bar.



TurboCAD's dialog box for setting snap priorities

The **Priority** column determines which snap mode takes precedence when more than one is available. The values range from 1 to 254, where 1 is the highest priority.

When priority is the same, then the snap point closest to the cursor is selected. When the **In Aperture Only** option is turned on, then the snap mode operates only when a snap point is within the aperture. When this option is off, the snap point need not to be within the snap aperture.

The **Show Magnetic Point** option is similar to AutoCAD's AutoSnap Magnet feature.

Summary

Congratulations! You've set up your new drawing in TurboCAD. Even though the drawing doesn't seem to have anything in it (except many grid lines), it already carries much information.

In this lesson, you learned how to start TurboCAD, set up the drawing parameters, and save the drawing. In the next lesson, you start to draw with TurboCAD. You create the mechanical part with a variety of drawing and editing commands.

At this point, you can exit TurboCAD by selecting **Exit** from the **File** menu. When you want to continue with Lesson 2, double-click the TurboCAD icon. After TurboCAD starts, load the *MechPart.Tcw* drawing file from the **File - Open** menu.

2

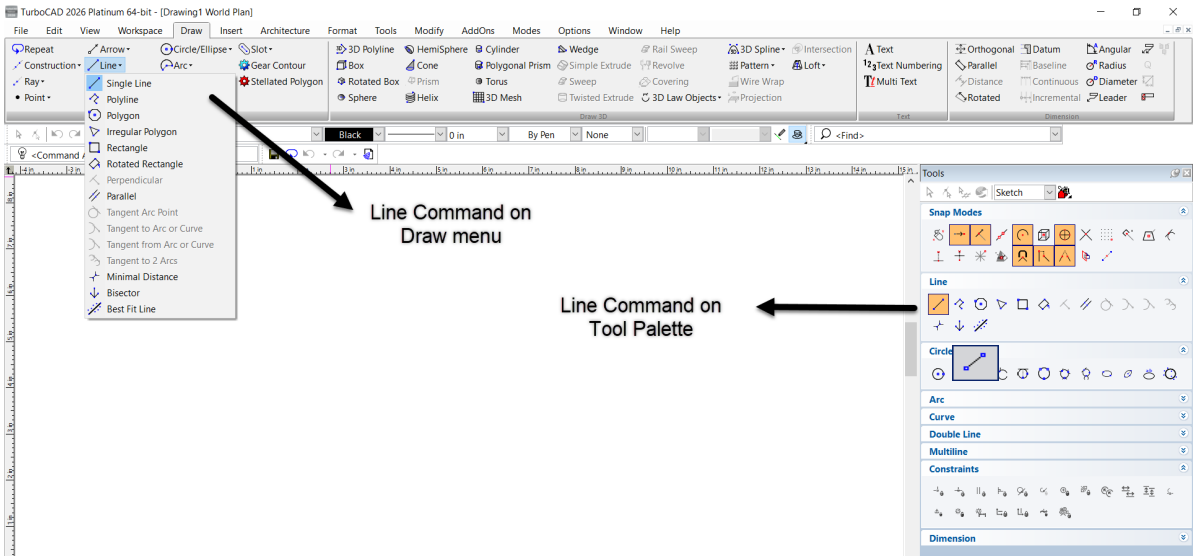
Inserting Objects

Lesson 2: Drawing, Editing, and Creating Blocks

In this lesson, you begin drawing the mechanical part in 2D. TurboCAD drawings are created by *drawing* objects, such as rectangles, lines, circles, and arcs. It also creates and inserts blocks. As objects are drawn, you may enter coordinates, use relative coordinates, turn on object snaps, and *lock values* (a.k.a. point filters). Although you do not type the names of commands, as you can with AutoCAD, you do have the option to override parameter values, such as length and angle.

In TurboCAD, you can start commands in a variety of ways:

- Select the command from the menu bar, such as **Draw | Line | Single.**
- Or, select the icon from the Tools palette.
- In Default UI, click the related icon on one of the left toolbars, such as 



While you cannot type command names, many commands offer keyboard shortcuts. See Appendix for the complete list.

TIP: *Drawing commands are found in the **Draw** menu item. Think of this as inserting object.*

Starting to Draw

If TurboCAD is not running, start it now. From New TurboCAD 2023 Drawing Dialogue, select the *Mechpart.tcw* drawing you created in Lesson 1.

To draw the mechanical part, begin with the front view in the lower-left corner of the drawing. The TurboCAD version of this tutorial continues on the opposite page.

AutoCAD: Drawing Construction Lines

To make it easier to draw the mechanical part, start by placing construction lines. First place horizontal xlines, followed by vertical ones. To place the xlines at precise spacing, ensure snap mode is turned on.

1. Unlike TurboCAD, AutoCAD does not automatically place construction lines on their own layer. Use the **Layer** command to create a new layer named "Construction" with a pale blue color.
2. From the **Draw** menu, select **Construction Line**. AutoCAD prompts you:

   `_xline specify point or [Hor Ver Ang Bisect Offset]: H`

   Type **H** to select the horizontal option, and then press **Enter**.
3. AutoCADprompts you:

   `Specify Through point:`

   Specify the following positions:

   0, 0 0, 0.5 0, 2.0 0, 3.5 0, 5.0
4. To draw the vertical construction lines, select **Draw | Construction Line**, and then type V for the vertical option. Specify the following positions to place the vertical construction lines:

   0, 0 0.75, 0 1.5, 0 2.25, 0 3.0, 0 5.5, 0 7.0, 0

The drawing now contains a grid of construction lines, which will help you to place drawing elements quickly, such as rectangles, lines, and circles.

TurboCAD: Inserting Construction Lines

To make it easier to draw the mechanical part, start by placing a series of *construction lines* (a.k.a. xlines). Construction lines are infinitely long and are displayed but not plotted. TurboCAD's construction lines are automatically placed on layer "$CONSTRUCTION," drawn in a light blue color and dashed *pen style* (a.k.a. linetype).

As in AutoCAD, TurboCAD lets you place vertical, horizontal, and angular construction lines — though not rays. (TurboCAD also has a command to place construction circles.) In fact, you can use any object as a construction object by setting the layer to $CONSTRUCTION prior to drawing the object.

1. Prepare for drawing:



   a. Turn on **Grid Snap** by clicking its icon. This ensures construction lines are placed at precise grid spacing. To do this, go to **Modes - Snaps.**



   b. Turn on **Show Magnetic Point** (a.k.a. the AutoSnap magnet). This forces the cursor to jump to the nearest grid point.

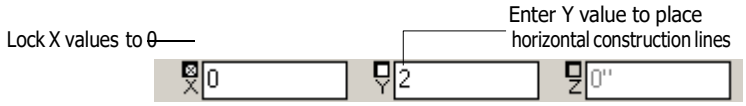


   c. Because it is hard to see construction lines when the grid is on, turn off the grid by clicking off the **Show Grid** button. The grid is now like AutoCAD's snap: invisible but activated. (As an alternative, you could change the color of the construction lines by editing the properties of their layer.)



2. Start to draw the construction lines: from the **Draw** menu, select **Construction | Horizontal Line**. TurboCAD prompts you, "Define the position of the horizontal construction line."

3. On the Coordinate bar, enter **0** for **X**, and then lock the value, as illustrated below:



Locking the x coordinate and entering y coordinates

   Press **Tab** to enter each of the following values for **Y**, followed by pressing **Enter**:

   0 0.5 2.0 3.5 5.0

   Notice that the construction lines are drawn at each y-coordinate position.



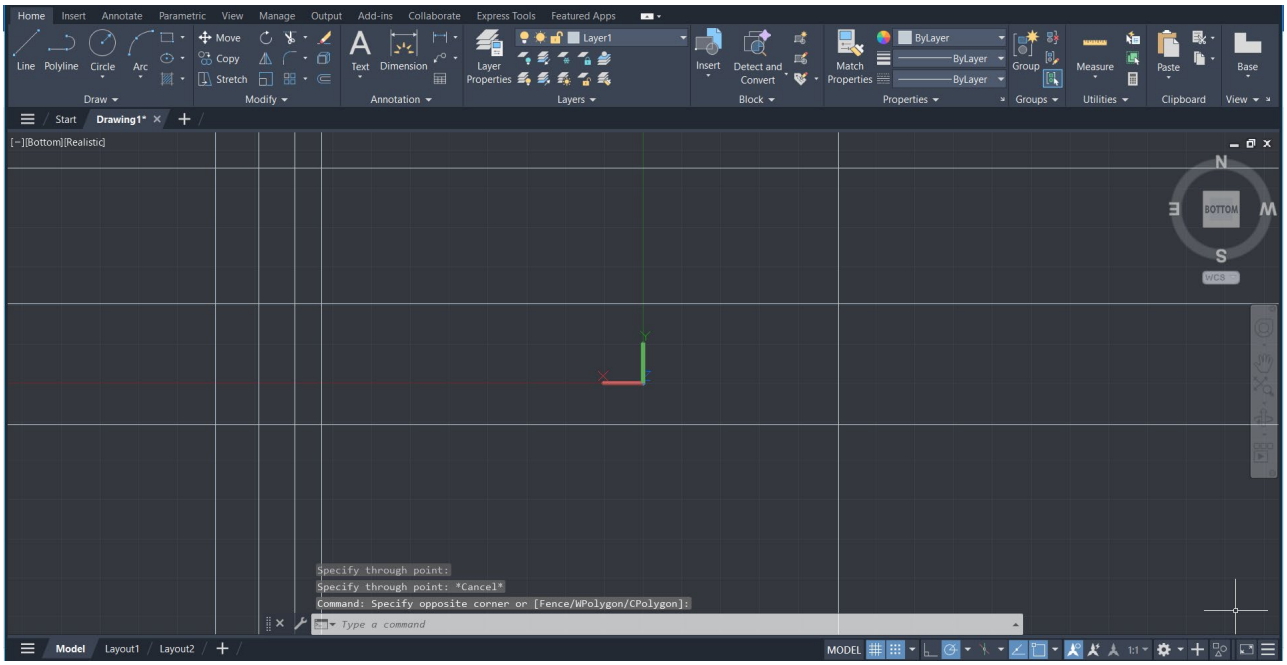
4. To draw the vertical construction lines, select **Draw | Construction | Vertical Line**. Unlock the **X**, and then lock the **Y** value at **0**. Enter the following **X** coordinates to place the vertical construction lines:

   0 0.75 1.5 2.25 3.0 5.5 7.0



   When done, click the racing flag. Remember to unlock the **Y** .

The drawing now contains a grid of construction lines, which will help you quickly place drawing elements, such as rectangles, lines, and circles.



AutoCAD's construction lines

Even though AutoCAD has automatic backup, it is a good idea to save the drawing when you complete a portion. Press **Ctrl+S** to save your valuable work.

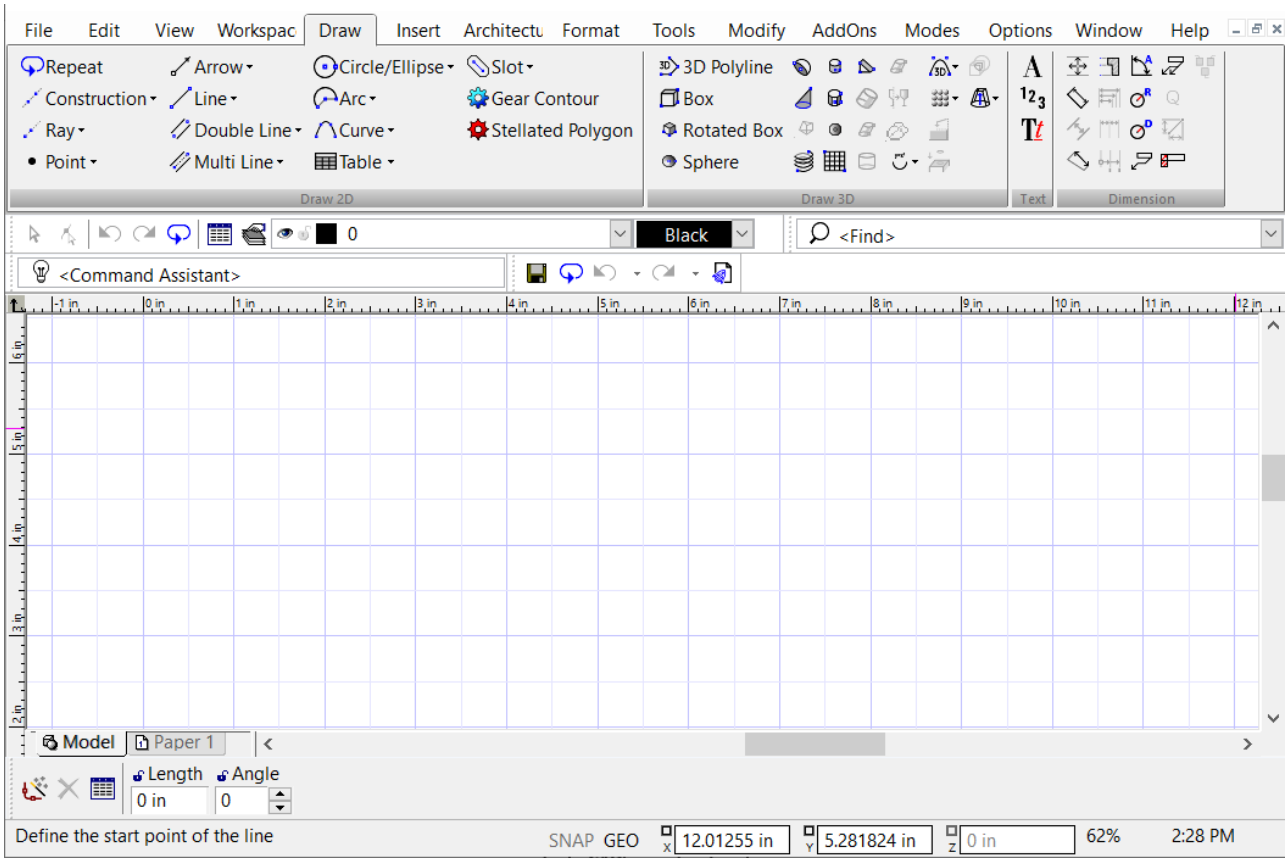
AutoCAD: DrawingRectangles

With the grid of construction lines in place, begin drawing the mechanical part by placing rectangles. You'll employ two methods to draw them accurately: (1) using intersection object snap, and (2) typing coordinates and options.

Intersection Object Snap

The quick and easy way to place rectangles is to snap the corners to the intersection of construction lines.

1. Begin by switching to layer Steel.



TurboCAD's construction lines

Even though TurboCAD has automatic backup, it is a good idea to save the drawing when you complete a portion. Press **Ctrl+S** to save your valuable work.

TurboCAD: Inserting Rectangles

With the grid of construction lines in place, begin drawing the mechanical part by placing rectangles. You'll employ two methods to draw them accurately:

(1) using intersection object snap, and

(2) typing coordinates and options.

Intersection Snap

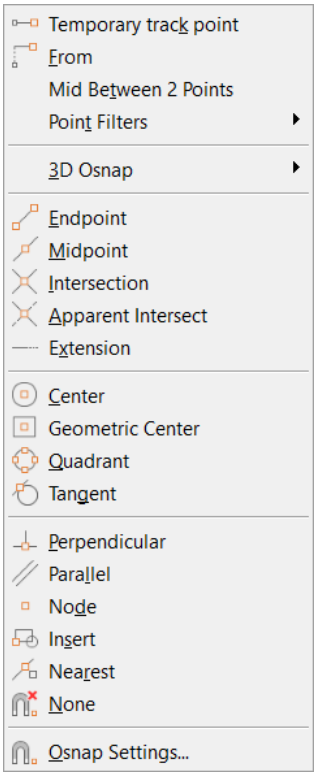
The quick and easy way to place rectangles is to snap the corners to the intersection of construction lines.

1. There is no need to switch to layer Steel, because TurboCAD automatically switches away from layer $CONSTRUCTION when you select another command.

2. From the **Draw** menu, select **Rectangle**. At the command line, AutoCAD prompts you:

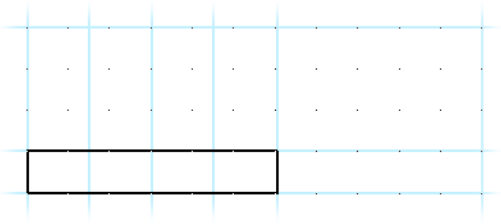
```
_rectang
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:
```

3. Change the object snap mode via the shortcut menu:
   a. While holding down the **Ctrl** key, right-click the mouse button.
   b. From the cursor menu, select **Intersection**.



AutoCAD's shortcut menu of object snaps

4. With the **Rectang** command still active, move the cursor to the origin **0",0"**. Click to define the first corner.
5. Move the cursor to **3",0.5"** by watching the coordinates on the status bar, then click to define the second corner.



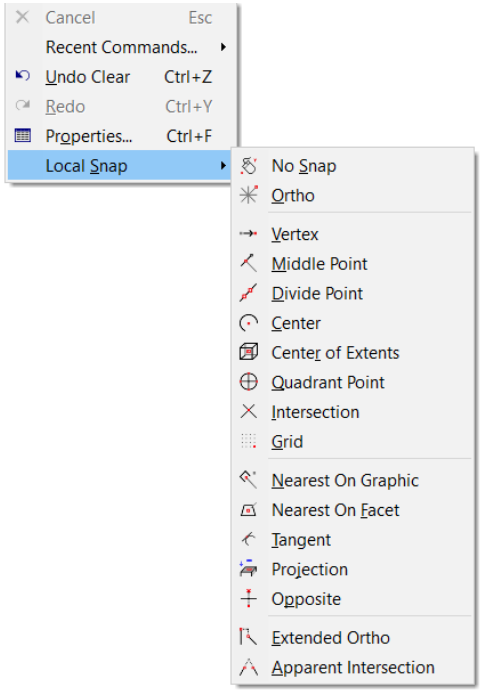
First rectangle in AutoCAD



2. From the **Draw** menu, select **Rectangle**. Notice that the cursor changes to a cross. On the status line, TurboCAD prompts you, "Define the first corner of the box."

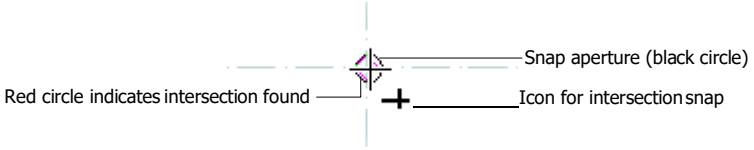


3. Change the snap mode by a different method: instead of using icons on the toolbar, use the right-click menu. When you right click, TurboCAD displays a context-sensitive *local menu* (a.k.a. shortcut menu). From the local menu, select **Local Snap | Intersection**. Or, simply press the letter **Shift+I**.



TurboCAD's local menu of snaps

4. With the **Rectangle** command still active, move the cursor to the origin at **0",0"**. Notice the intersection snap icon displayed by TurboCAD.



TurboCAD's intersection snap icon

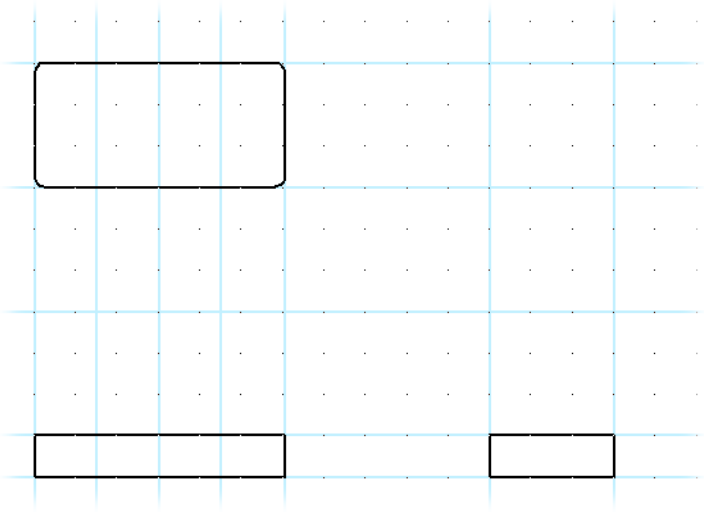
Click to define the first corner.

5. Move the cursor near to **3",0.5"** by watching the Coordinate bar. Local snaps work only for one cursor pick, so press **I** when the cursor is near the intersection of the two construction lines. (Do not click the mouse button — there's no need!) See figure on the AutoCAD side.

   Notice that the Edit bar displays the rectangle's **Size A** (width) as 3" and **Size B** (height) as 0.5".

6. Draw two more rectangles by following the table and the figure shown on the TurboCAD side. Because the second rectangle has all four corners filleted, you can use the **Fillet** option to place a filleted rectangle:

```
Command: rectang
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/
      Width]: f
Fillet radius for rectangles <0.0000>: .125
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/
      Width]: (Pick.)
Specify other corner point or [Area/Dimensions/Rotation]: (Pick.)
```



Three rectangles in AutoCAD

AutoCAD: Typing Coordinates and Parameters

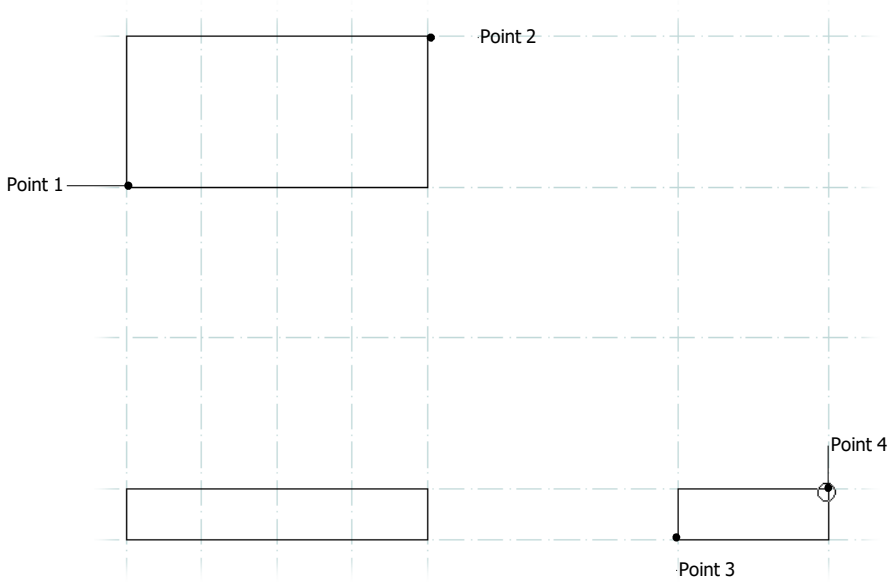
In drawing the first three rectangles, you used Intersection object snap on the xlines. When a rectangle needs to be placed where no construction lines exist, you could draw more construction lines. The drawback is that too many construction lines make the drawing too cluttered. The alternative is to enter x, y coordinates.

1. Start the **Rectang** command again, and remember to turn off filleting:

```
Command: rectang
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/
      Width]: f
Fillet radius for rectangles <0'-0.125">: 0
```

2. Draw two more rectangles by following the points illustrated in the figure below. Remember to press **I** instead of the left mouse button.

Rectangle	Size	Pick Points
First rectangle	3" x 0.5"	...
Second rectangle	1.5" x 3"	1 and 2
Third rectangle	0.5" x 1.5"	3 and 4



All three rectangles in TurboCAD

TIPS: *If, at any time, you need to exit a command, press the **Esc** key, just as in AutoCAD.*

*To undo previous command(s), press **Ctrl+Z**, click the **Undo** icon, or select **Undo** from the **Edit** menu. As in AutoCAD, TurboCAD can undo all the way back to the beginning of the editing session.*

TurboCAD: Typing Coordinates and Parameters

To draw the first three rectangles, you used intersection snap with the construction lines. When a rectangle needs to be placed where no construction lines exist, you could draw more construction lines. The drawback is that too many construction lines clutter up the drawing.

Instead, use the Inspector bar and the Coordinate bar to place the other rectangles precisely.

1. TurboCAD keeps the current command active, so there is no need to restart it.

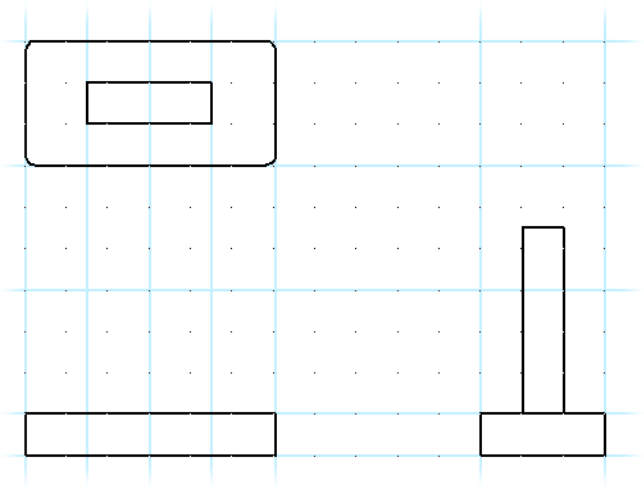
```
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/
      Width]: .75,4
```

2. That fixes the lower-right corner of the rectangle. AutoCAD lets you define the size of the rectangle with the **Dimensions** option:

```
Specify other corner point or [Area/Dimensions/Rotation]: 2.25,4.5
```

This creates the fourth rectangle.

3. To place the final rectangle, use x, y coordinates of **6, 0.5** for one corner and **6.5, 2.75** for the other.



Five rectangles in AutoCAD

AutoCAD: Drawing Circles

With the basic outline formed by rectangles, you can now place the circles.

1. The drawing has a pair of concentric circles, so you'll use the **Circle** command twice. From the **Draw** menu, select **Circle | Center, Radius**.
2. Turn on **Intersection** object snap.
3. Pick at the intersection of the xlines at 1.5", 2".
4. AutoCAD prompts:

```
_circle Specify center point of circle or [3P/2P/TTR (tan
tan radius)]: int of <pick at the intersection of 1.5,2>
Diameter/<Radius>: .25
```

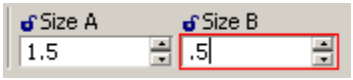
Press **Ctrl+R** to enter the X field, and then enter the x, y coordinates of the rectangle's first corner:

X: **.75**, and then press **Tab**.

Y: **4**, and then press **Enter**.

TIP: *You don't have to enter the inch symbol if you don't want to. TurboCAD automatically uses the default unit set in Units and Scale.*

2. The lower-right corner of the rectangle is fixed. To define the other corner of the rectangle, move the cursor roughly into position, and then press **Tab** to enter the Inspector bar:



TurboCAD's Inspector bar

Size A: **1.5**, and then press **Tab**.

Size B: **0.5**, and then press **Enter**.

That creates the fourth rectangle.

3. Use the same method to place the final rectangle:

Step 1: Press **Ctrl+R** to enter the Coordinate bar. X= **6**, and then press **Tab**.

Step 2: Y= **0.5**, and then press **Enter**.

Step 3: Press **Tab** to enter the Inspector bar. Size A= **0.5**, and then press **Tab**.

Step 4: Size B= **2.25**, and then press **Enter**.

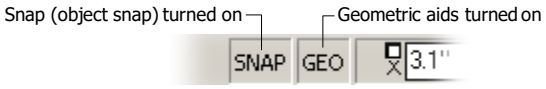
The result looks like the figure on the AutoCAD side of the page.

TurboCAD: Inserting Circles

With the basic outline formed by rectangles, you can go on to place the circles.



1. Because the drawing has a pair of concentric circles, from the **Draw** menu select **Circle | Concentric**. TurboCAD prompts you, "Define the center point of the circles."
2. Select the **Intersection** snap icon. Ensure the **SNAP** button is turned on.



Toggling snaps and aids

3. Click at the intersection of **1.5,2**.
4. TurboCAD prompts, "Define the radius of the circle."

Press **Tab**, specify Radius of **0.25**, and then press **Enter** to insert the first circle.

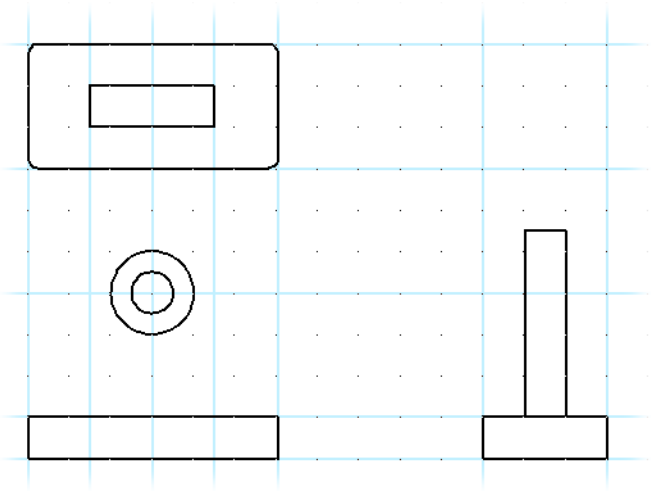
5. Press **Enter** to repeat the **Circle** command.

```
_circle Specify center point of circle or [3P/2P/TTR (tan tan
radius)]:
```

Enter **0.5"** for the Radius. AutoCAD draws the second circle.

```
Diameter/<Radius>: 0.5
```

6. Save your drawing with **Ctrl+S**.



Concentric circles added

AutoCAD: Copying and Mirroring Entities

The top view has four bolt holes. The obvious way to draw them is to draw one circle, and then copy it three times.

1. Draw a circle with center at **0.375", 4.625"** with radius **0.125"**.
2. There is no need to run the **Select** command (as in TurboCAD), because AutoCAD always reverts to select mode after most commands.
3. Click the circle you just completed. It turns dashed, and is surrounded by five blue squares.
4. Select **Copy** from the **Modify** menu. AutoCAD prompts you:

```
_copy specify base point or [Displacement mode]:
```

5. Specify the displacement by entering **2.25<0**.

5. Press **Tab**, and then enter **0.5"** for the Radius of the second **circle**.
6. Save your drawing with **Ctrl+S**. See the figure on the AutoCAD side for the result.

TIPS: *Notice that TurboCAD allows you to specify circles by radius, diameter, or the length of its circumference — the latter is not available in AutoCAD.*

*TurboCAD has many ways to place circles, as found under the **Draw | Circle** menu selection. Contrasted with AutoCAD, these are:*

TurboCAD Circle Commands	Equivalent AutoCAD Circle Option
Center and Point, radius	Center and Radius
Center and Point, diameter	Center and Diameter
Center and Point, circumference	*not available*
Concentric	*not available*
Double Point	2P
Tangent to Arc or Curve	*use TANgent object snap*
Tangent to Line	*use TANgent object snap*
Triple Point	3P
Tangent to 3 Arcs	TTR
Tangent to Entities	*use TANgent object snap*

TurboCAD: Copying and Mirroring Entities

The top view has four bolt holes. The best way to create them is to insert one circle, and then copy it three times.

1. Draw a circle at **0.375, 4.625** with radius **0.125**.



2. From the toolbar, click the **Select** icon. The cursor changes to the standard Windows pointer arrow.
3. Click the circle you just completed. It turns pink and is surrounded by colors grips.

Circle selected 



4. From the **Modify** menu, select **Array | Linear**. TurboCAD prompts you, "Define the step position of the reference point."
5. Press **Tab**, and then change X Step from **0** to **2.25**.

6. Press **Enter** at the next prompt:

   ```
   Specify second point or <use first point as displacement>: (Press Enter.)
   ```

   Notice that AutoCAD copies the circle.

7. To place the two lower bolt holes, you will use the **Mirror** command. First, select the first bolt hole, and then select the second one.

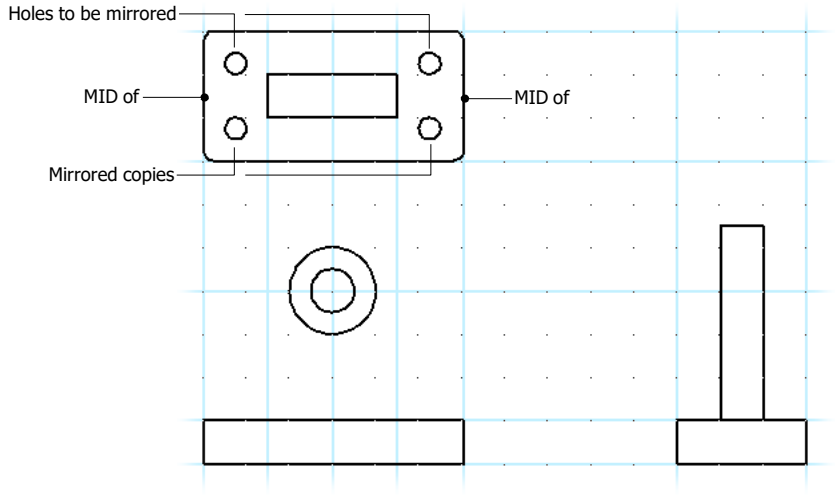
8. Select **Mirror** from the **Modify** menu. AutoCAD prompts you:

   ```
   Command: _mirror 2 found
   First point of mirror line: mid
   ```

   Use **MIDpoint** object snap to mirror the holes accurately.

9. Click the midpoint of the vertical line, as indicated by the figure below.

10. At the "Second point" prompt, click the other vertical line. When AutoCAD asks, "Delete old objects?" answer **N**. Notice that AutoCAD inserts the lower two bolt holes. There is no need to press **Esc** to remove the selection set.



AutoCAD drawing with four holes

AutoCAD: Drawing Arcs

Finish off the rounded portion of the mechanical part by placing an arc. Use the "SCE" method to draw the arc, as follows:

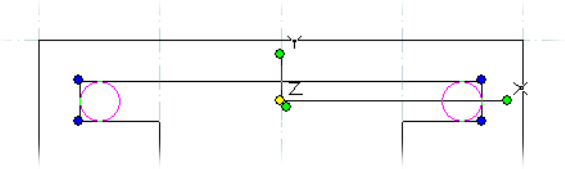
1. Select **Arc | Start, Center, End** from the **Draw** menu.
2. AutoCAD prompts you to indicate the arc's starting point:

```
_arc specify start point of arc or [Center]: int
of (Click at the intersection of xlines at 2.5", 2.0".)
```

6. Set Y Step = **0**, and then press **Enter**.

   TurboCAD copies the circle. Press **Esc** to remove the selection box.

7. To place the two lower bolt holes, you will use the **Mirror** command. First, select the first bolt hole, and then hold down the **Shift** key while selecting the second bolt hole.



Press Shift to add to selection sets



8. From the **Modify** menu, select **Copy Entities | Mirror**. TurboCAD prompts you, "Define the first point of the mirror line."



9. Turn off grid snap, and then turn on **Middle Point** object snap. (It is far quicker in TurboCAD to press the **M** key for Middle Point snap, then to change snap modes all the time.)

10. Click the midpoint of the vertical line, as indicated by the figure on the page opposite. (Or, move the cursor to the line, and then press **M**. That snaps to the middle of the line without clicking the mouse.)

11. At the "Define the second point of the mirror" prompt, click the other vertical line. TurboCAD inserts the lower two bolt holes.

    Press **Esc** to remove the *selection box* (a.k.a. selection set).

TIP: *The original selection stays selected. To perform a mirror — rather than a mirror copy — press* ***Del*** *to remove the original selection.*

TurboCAD: Inserting Arcs



Finish off the rounded portion of the mechanical part by placing an arc. TurboCAD has eight ways to draw arcs as shown by the **Draw | Arc** menu selection. It is difficult to compare TurboCAD's eight methods with those of AutoCAD, because TurboCAD uses a different methodology. In AutoCAD, you draw most arcs by specifying a start, center, and/or endpoint; in TurboCAD, you usually define the start and end angles. To draw this arc, use the double-point method, as follows:

1. From the **Draw** menu, select **Arc | Double Point**. (Notice that the Edit bar lets you specify the arc's diameter, circumference, angle, start angle, end angle, and length.)

2. TurboCAD prompts you, "Define the first point on the arc or circle." Ensuring **Intersection** snap is turned on, click at the intersection of construction lines at **2.5", 2.0"**.

3. At the next prompt, specify the arc's center:

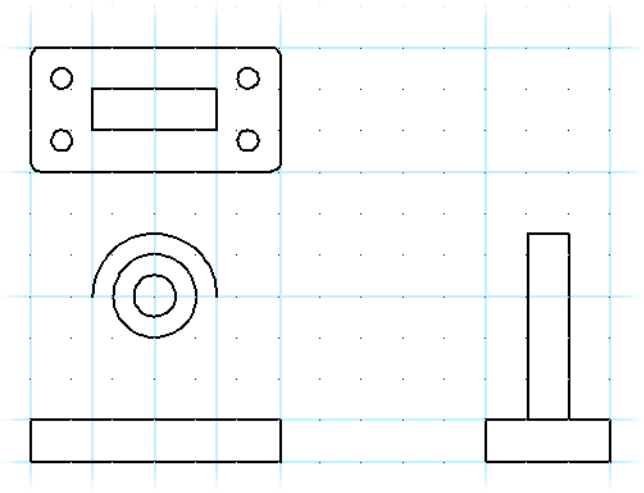
   ```
   Center/End/<Second point>: _c Center: int
   of (Click at the intersection of xlines at 1.5", 2.0".)
   ```

   Notice that AutoCAD draws an arc. You now complete the definition of the arc by specifying the endpoint.

4. AutoCAD prompts you to complete the arc:

   ```
   Angle/Length of chord/<End point>: int
   of (Click at the intersection of xlines at .75", 2.0".)
   ```

Like TurboCAD, AutoCAD draws arcs in the counter-clockwise direction.



AutoCAD drawing with arc

AutoCAD: Drawing Lines

To connect the arc with the base rectangle, use the **Line** command.

1. Select **Line** from the **Draw** menu bar:

   ```
   _line from point: (Pick the endpoint of the arc you just drew.)
   ```

2. Complete the line when AutoCAD prompts:

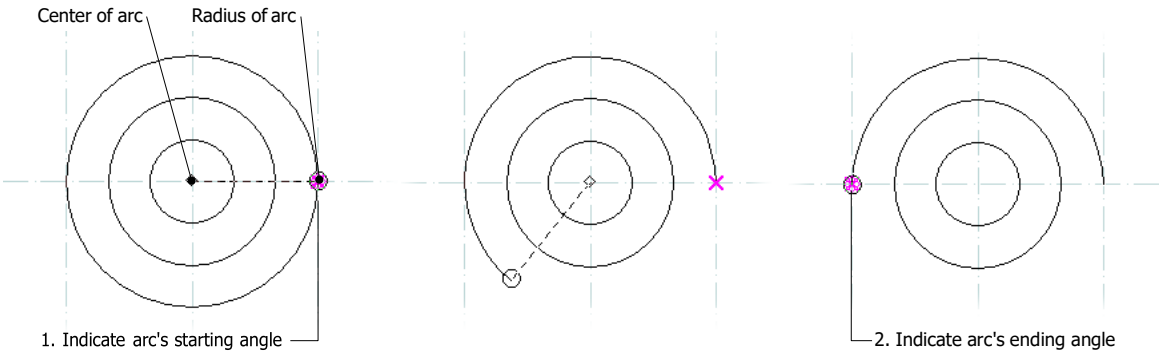
   ```
   To point: 1.5<270
   ```

3. Repeat for the second line connecting the arc's other endpoint.

At this point, the new part of your drawing should look like the figure below. Save the drawing.

3. At the "Define a second point on the arc or circle's circumference" prompt, click at the intersection of construction lines at **1.5, 2.0**.

   Notice that TurboCAD draws a circle. You define the arc by specifying its starting and ending angles.



TurboCAD's arc starts as a circle

4. TurboCAD prompts you, "Define the start angle of the arc." Like AutoCAD, TurboCAD draws arcs in the counter-clockwise direction, so click at the starting point to the right. See figure above.

   At the "Define the end angle of the arc" prompt, click at the ending point to the left. As you move the cursor, notice how TurboCAD sweeps the arc. When you click, the arc is complete.

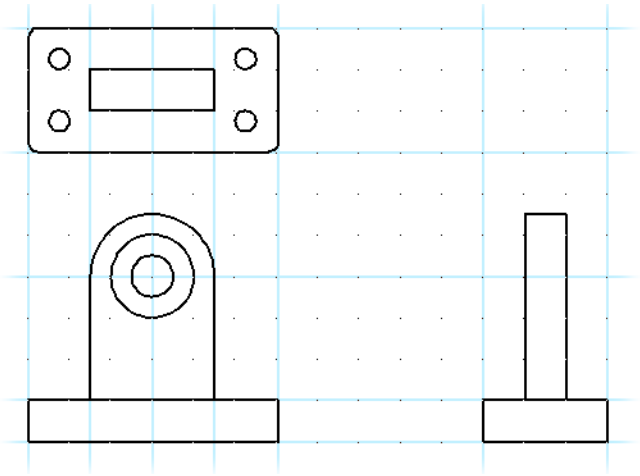
   See figure on the AutoCAD side for the completed arc.

TurboCAD: Inserting Lines

To connect the arc with the base, use the **Line** command.

1. From the **Draw** menu, select **Single** line. TurboCAD prompts you, "Define the start point of the line." Pick the endpoint of the arc you just drew. (Or, press the **V** key for Vertex snap.)
2. When TurboCAD prompts, "Define the end point of the line," move the cursor straight down to connect with the base. (If you prefer, you can type **1.5** in the Length field and **270** in the Angle field.)
3. Repeat for the second line connecting the arc's other endpoint.

At this point, your drawing should look like the figure on the next page. Save the drawing.



Vertical lines added

AutoCAD: Filleting Corners

The base of the mechanical part has a 1/8" fillet on all top corners. Earlier, you drew a rectangle with fillets at its four corners. Here you fillet the two remaining rectangles with the **Fillet** command.

1. Filleting works best when object snap mode is set to **Nearest**.
2. From the **Modify** menu, select **Fillet**.

```
_fillet
Current settings: Mode = TRIM, Radius = 0'-0.13"
Polyline/Radius/Trim/<Select first object>: r
```

3. Type **R** to change the radius from 0.5" to **0.125"**:

```
Enter fillet radius <0'-0.00">: 0.125
```

4. Now fillet twolines:

```
Select first object or [Undo/Polyline/Radius/Trim/Multiple]:
      (Pick one line.)
```

5. At the "Select second object" prompt, click an adjacent line. AutoCAD applies the fillet.

```
Select second object or shift-select to apply corner: (Pick
      another.)
```

Zooming In and Out with TurboCAD

To get a closer view of what you are working on, you may find these two TurboCAD icons useful:



*The **View / Zoom Window** icon works exactly like AutoCAD's Zoom Window command. Pick two points and TurboCAD zooms into that rectangle.*



*The **Zoom Extents** icon displays the entire drawing, as in AutoCAD. The keyboard shortcut for Zoom Extents is **Ctrl+Backspace**.*

*TurboCAD's zoom commands are found in the **View | Zoom** menu:*

TurboCAD Zoom Option	TurboCAD Shortcut Key	Equivalent AutoCAD Command
Zoom In	+	Zoom 2x
Zoom Out	-	Zoom 0.5x
Zoom Window	...	Zoom Window
Extents	Ctrl+Backspace	Zoom Extents
Full View	Shift+Backspace	Zoom All
Printed Size	...	Preview
Aerial View	...	DsViewer
Previous View	...	Zoom Previous
Pan to Cursor	Ctrl+End	Pan

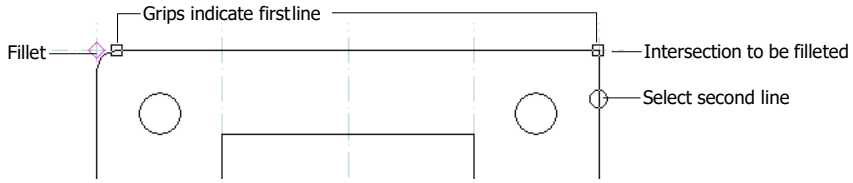
TurboCAD: Filleting Corners

The base of the mechanical part has a 1/8" fillet on all top corners. That means you apply the **Fillet** command eight times: four in the top view, and twice on each of the front and side views.

1. The Fillet command works best when snap mode is set to **Nearest on Graphic**. Click the icon or right-click to display the local menu.

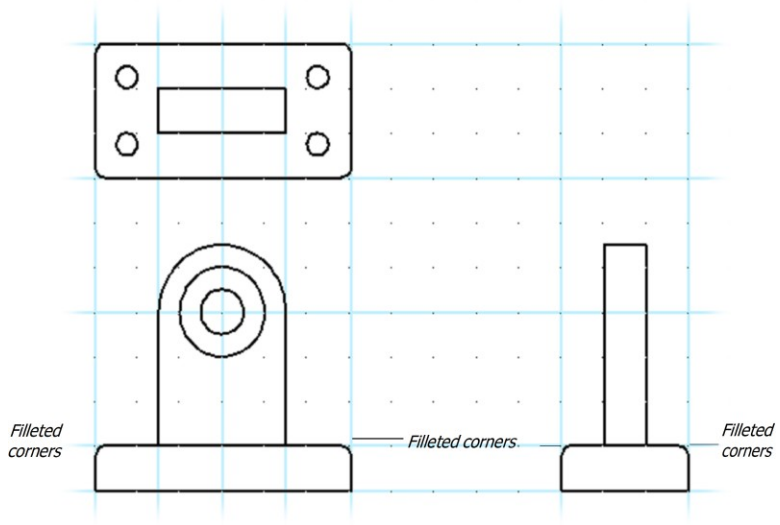


2. From the **Modify** menu bar, select **Fillet**.
3. Press **Tab**, and then change the Radius from **1"** to **0.125"**.
4. TurboCAD prompts you, "Select the first entity." Click one line, as shown in the figure. A pair of square handles indicates the selected line.



Selecting the first entity for filleting

5. At the "Select the second entity" prompt, click an adjacent line. TurboCAD applies the fillet.



Corners filleted

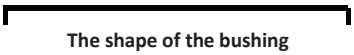
6. Continue filleting corners, as shown in the figure.

   Save your work.

AutoCAD: Making and Inserting Blocks

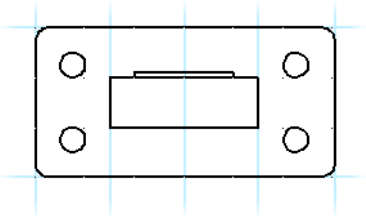
This mechanical part has a round bearing surface. Since this appears four times in the drawing, you create it as a block, and then insert the block in the drawing.

1. Ensure the layer is Steel and the linetype is Continuous. Add more xlines, if you think you will find them helpful.
2. Draw the 0.1" thick bushing face with **Draw | Polyline**.



The shape of the bushing

3. To end the **Polyline** command, press **Enter**.

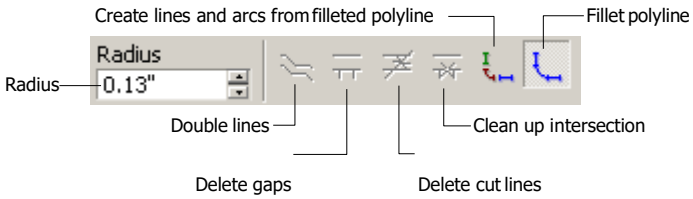


The bushing drawn as a polyline

6. Continue filleting corners, seven more times, as shown in the figure on the AutoCAD side. Notice how quick the filleting process is (as contrasted with AutoCAD), because TurboCAD automatically repeats the **Fillet** command.

   See the figure on the AutoCAD side for the finished result. Save your work.

TIP *TurboCAD's Inspector bar provides these filleting options:*



Fillet polyline *fillets all corners of polylines.*

Create lines and arcs *breaks a filleted polyline into lines and arcs.*

The ***Fillet*** *command also works on double lines.*

TurboCAD: Creating and Placing Blocks

The mechanical part has a round bearing surface. Since it appears four times in the drawing, create it as a block, and then place it in the drawing. TurboCAD has three forms of grouping objects together — in contrast with AutoCAD's two:

TurboCAD	AutoCAD Equivalent
Block	Block
Group	Group
Symbol	...

A *symbol* differs from a block in these two ways: (1) it can be stored on disk in any file format, whereas a block cannot, because it is an integrated part of the active drawing; and (2) each time a symbol is placed, it is copied, whereas a block is referenced.

1. Add more construction lines, if you find them helpful.

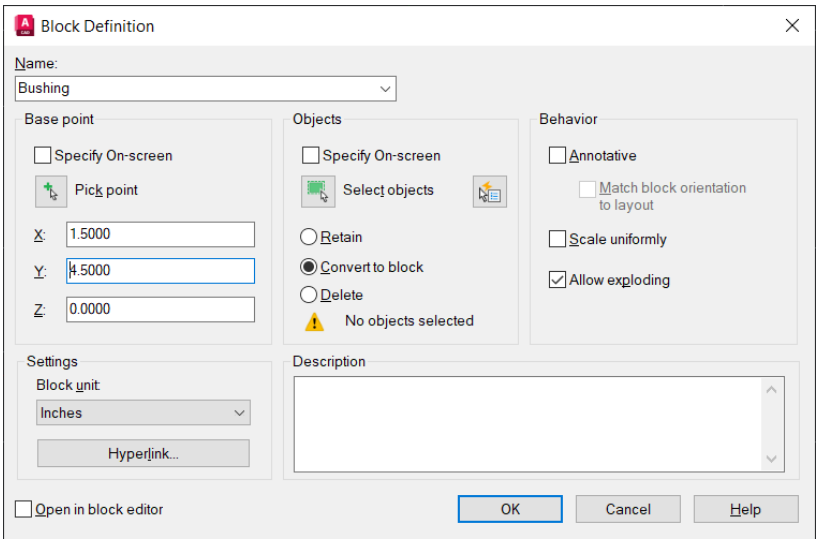


2. Draw the 0.1" thick bushing face with **Draw | Polyline**. Hold down the **Shift** key to draw lines horizontally and vertically. Enter values in the Inspector bar, if you find it helpful.



3. To end the **Polyline** command, click the **Finish** flag on the status bar. (Don't press **Esc**, because that removes the polyline from the drawing.)

4. To create the block, select the polyline. AutoCAD highlights it with handles.
5. From the **Draw** menu, select **Block | Make**. Notice that the Block Definition dialog box appears.



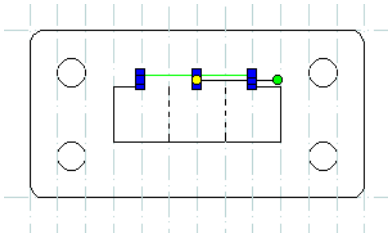
AutoCAD's Block Definition dialog box

6. To make it easier to place the block, change the base point. Under Base point, click the **Pick Point** button. The dialog box disappears and AutoCAD prompts:

```
_bmake Insertion base point: (Pick the MIDpoint of the polyline.)
```

4. To create the block, click the **Select** icon. Select the polyline that you just drew. (Or press the **F7** key, which is a shortcut for selecting the last-drawn object.)

   Notice that TurboCAD highlights it with handles.



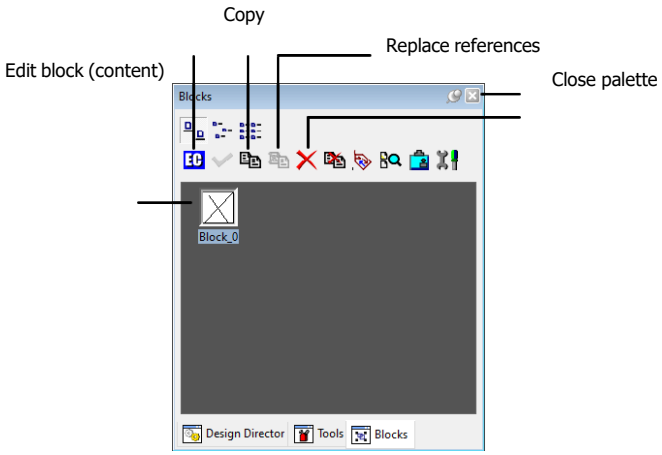
TurboCAD's polyline highlighted with handles

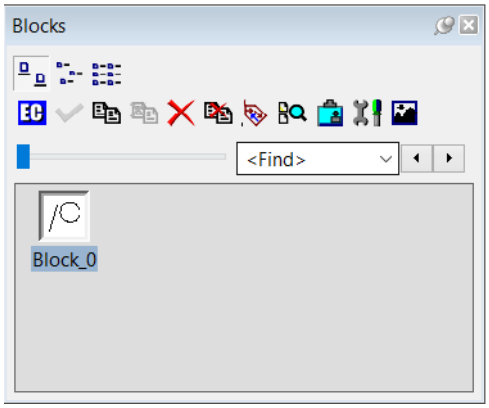


5. From the **Format** menu, select **Create Block**. The bushing disappears.

   Alternatively, you could drag it by its reference point into the Blocks palette.

6. To see the bushing block, select **Blocks (Ctrl+Shift+B)** from the **Tools** menu. This displays the Blocks palette.

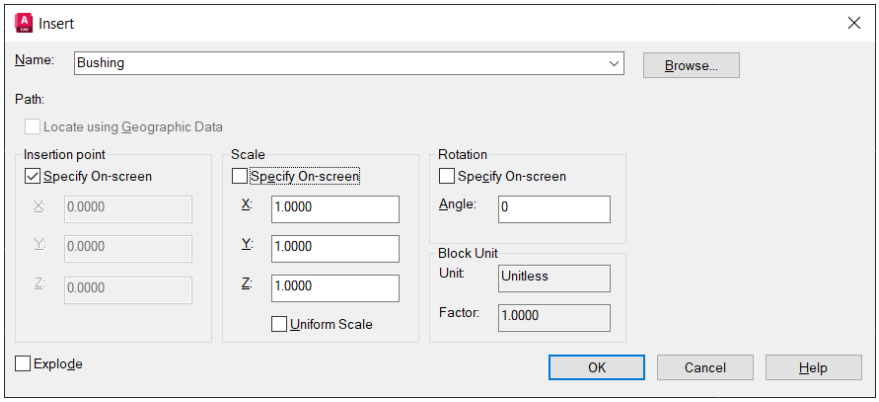




TurboCAD's Blocks palette

TIP: *When the palette appears, notice that TurboCAD automatically gives the block the name of "Block_0." You can change the automatic name that TurboCAD gives blocks, groups, and symbols with the* ***Options | Auto- Naming*** *command. The* ***Program Setup*** *dialog box displays the* ***Auto-Naming*** *tab.*

7. To name the block, type "Bushing" for **Block Name**. Ensure that the **Convert to Block** option is turned on. Click **OK**.
8. To place the block, select **Block** from the **Insert** menu. When the Insert dialog box appears, ensure "Bushing" is shown in the Name droplist.

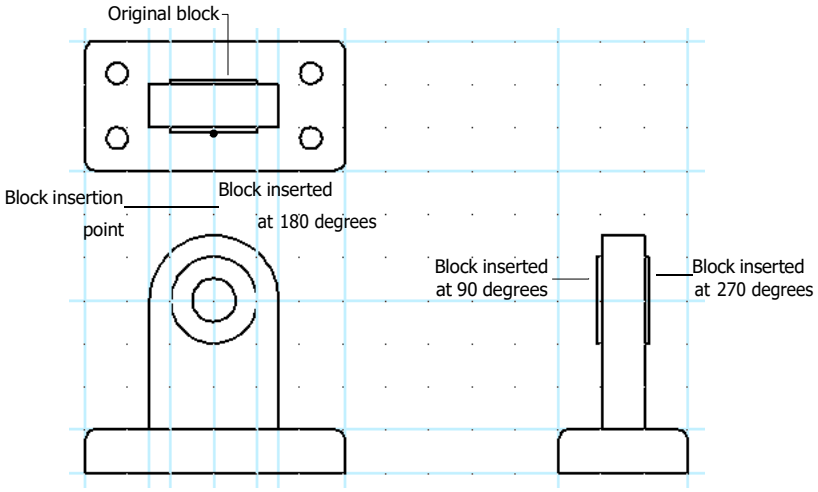


AutoCAD's Insert dialog box

Set the following insertion parameters:

- Insertion point **Specify On-screen**
- Scale **X = Y = Z = 1.0**
- Rotation **180**

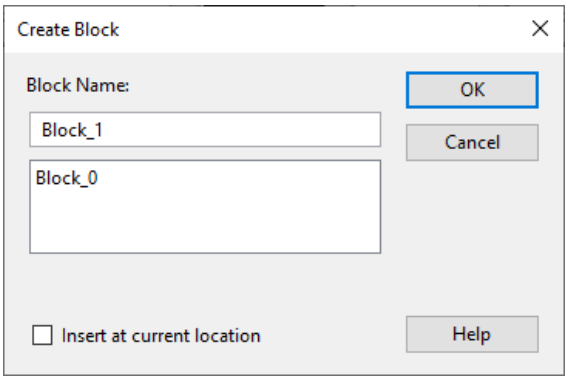
Click **OK**, and then place the bushing block in the drawing. The block's insertion point lines up with the construction lines. (You may find it easier to place when osnap is turned off.)



Blocks inserted in drawing

9. Place the block three more times. When required, use the **Rotation** parameter to rotate the block by 90 and 270 degrees. Save your work.

7. To change the name of the block, double-click it and TurboCAD displays the Edit Block dialog box. Change the name to "Bushing," and then click **OK**.



TurboCAD's Edit Block dialog box

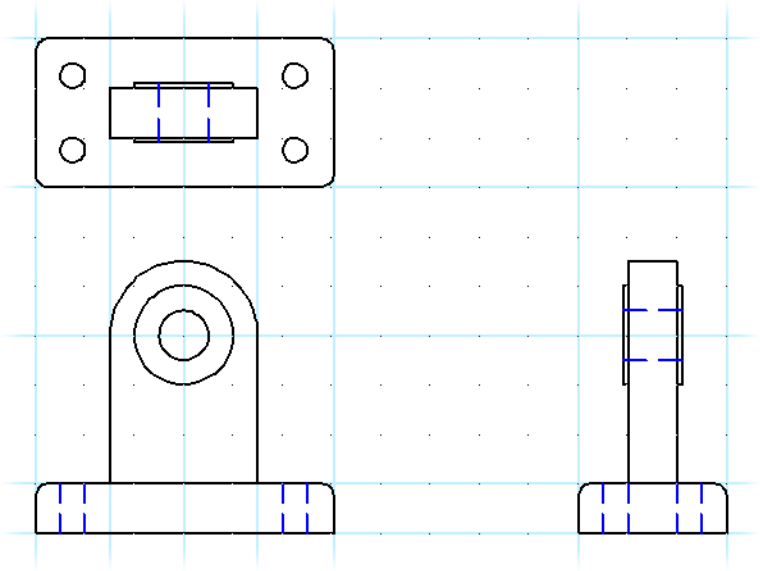
8. Ensure **Vertex** snap is turned on. Place the block by dragging it from the palette into the drawing.
9. Place the block three more times, as illustrated by the figure on the AutoCAD side.

   When required, use the **Rot Z** parameter on the Inspector bar to rotate the block by **90** degrees. (Alternatively, use the rotation handle; hold down the **Shift** key to rotate the block in 90-degree increments.) Save your work.

AutoCAD: Changing Linetypes and Layers

The five holes in the mechanical part show up as hidden lines in all three views. First, draw the hidden lines in the top view.

1. Place a pair of xlines on layer **Construction** to help draw the hidden lines, as shown in the figure. Use **INTersection** object snap to place the lines accurately.
2. From the Layers toolbar, select layer "Hidden," and ensure linetype Hidden is active.
3. Select **Draw | Line**, and then draw the two hidden lines, as shown in the figure.



Hidden lines

4. Repeat to draw the remaining hidden lines.

   Save your work.

TurboCAD: Changing Linetypes and Layers

The holes in the mechanical part show up as hidden lines in all three views. First draw the hidden line in the top view.

1. On the Properties toolbar, select layer Hidden and linetype Hidden (looks like a short-dashed line).



TurboCAD's Property toolbar

2. Place a pair of construction lines to help draw the hidden lines. Remember to use **Intersection** snap to place the lines accurately. Note how the construction lines retain their own properties; they do not take on the properties you set above in step 1 .
3. Ensure the **Line** tool is still selected and draw the two hidden lines, as shown in the figure. See the figure illustrated on the AutoCAD side.
4. Repeat to draw the remaining hidden lines.

   Save your work.

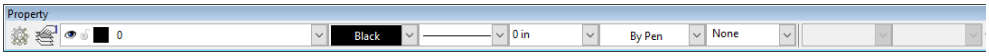
TIP *To remove construction lines, you cannot simply erase them. Instead, use the **Edit | Clear | Construction** menu item, and then pick the construction lines to remove. By having a separate erase command for construction lines, you can erase objects in drawings (or even all objects) without affecting construction lines.*

TurboCAD's Entity Styles

In AutoCAD, you can create named objects for views, fonts, layers, and so on. These named objects (a.k.a. tables) predefine viewpoints, text styles, layer properties, etc.

TurboCAD takes that concept a step further by allowing you to create styles for entities. Just like styles in a word processor, TurboCAD's entity styles predefine the properties of objects.

Entity styles are independent of layer settings. Unfortunately, you cannot apply a style to selected objects, nor can you define a style from a selection set. Entity styles can be created in **Style Manager** Palette in **Tools**:



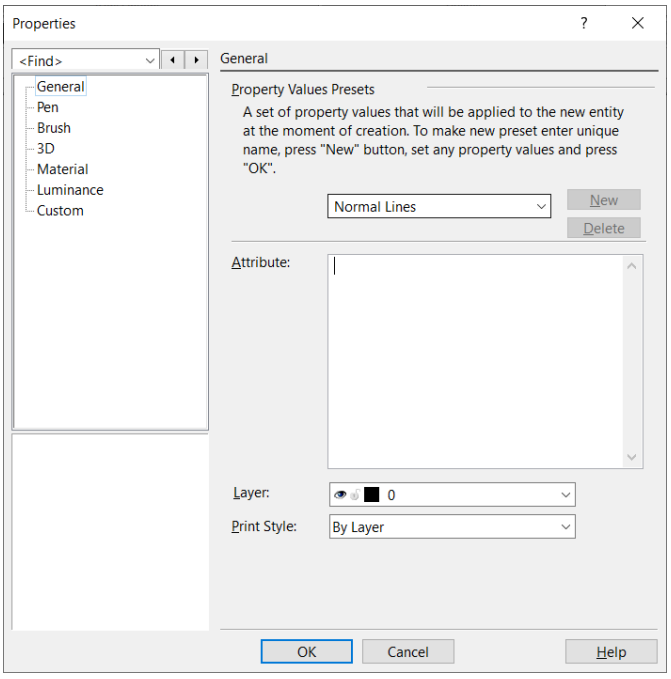


You create entity styles with the Properties dialog box Properties in inspector bar or Local Menu using these steps:

1. Set the properties.
2. Click the **Style** list box.
3. Enter the style name.
4. Click **New**. TurboCAD adds the style to the Properties toolbar and makes it the default.

General Properties

The **General** page sets the following properties:

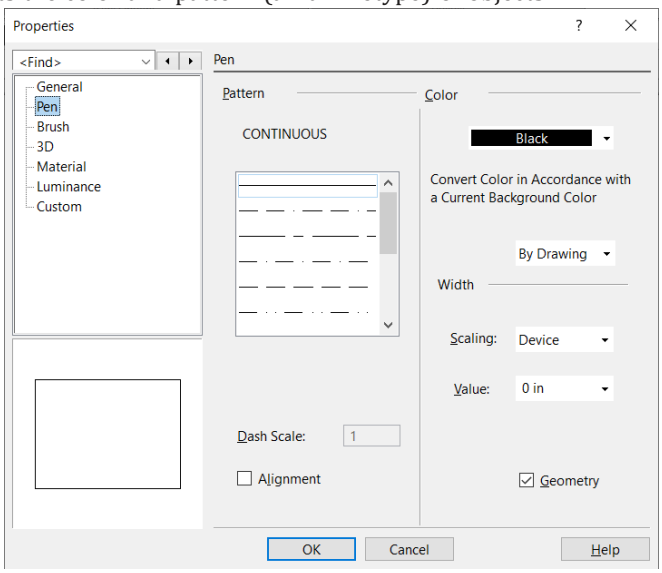


General page of the Properties dialog box

- **Attribute** includes any sort of text with the object (a.k.a. extended entity data).
- **Layer** specifies a previously-defined layer.
- **Print Style** specifies a previously-defined print style (color and line style).
- **Style** creates the new style name; simply type the name in the text box; default = "Normal Lines."

Pen Properties

The Pen page sets the color and pattern (a.k.a. linetype) of objects:



Pen page of the Properties dialog box

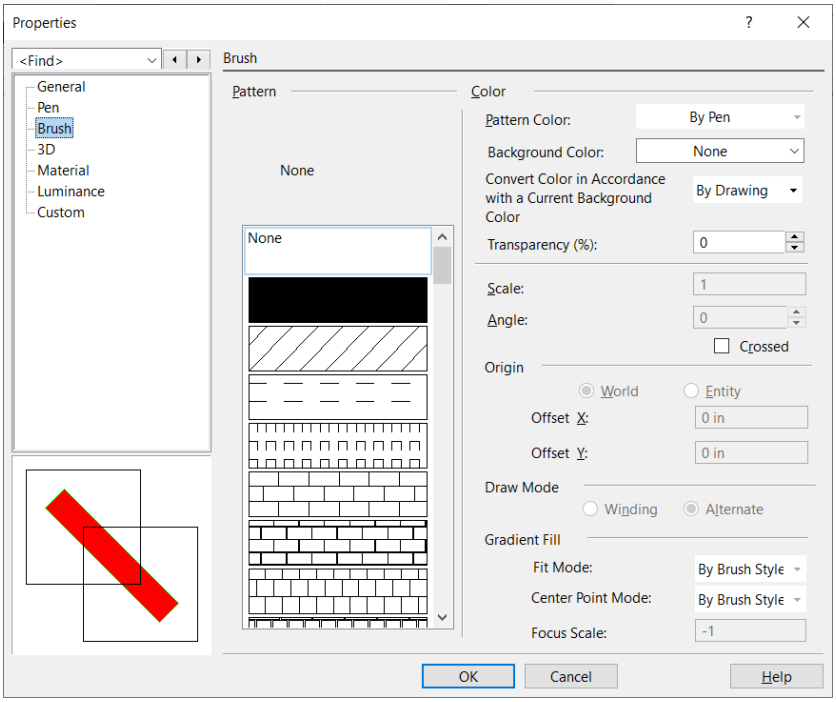
- **Pattern** selects the line type; note that one is named $INVISIBLE.
- **Dash Scale** sizes the pattern; equivalent to AutoCAD's *linetype scale*.
- **Alignment** forces the pattern to fit along the line's length.
- **Color** specifies the color.
- **Width** (*line weight*) gives a width to any object using **World** (absolute) or **Device** (relative) scaling.
- **Value** specifies the width of lines, from 0" to 2".
- **Geometry** applies thickness to 2D objects.

Brush Properties

The Brush page sets the hatch pattern parameters:

- **Pattern** select a predefined hatch pattern, solid fill, or none; you can define custom patterns (brush styles) as vector hatches, bitmaps, or gradients.
- **Color** specifies the color of the brush.
- **Scale** specifies the size of vector patterns.
- **Angle** specifies angle of the pattern.

- **Crossed** repeats the pattern 90 degrees to the first one.
- **Background color** specifies the underlaying color of the pattern.
- **Origin** specifies **World** (uses WCS origin) or **Entity** (lower-left corner of object being hatched) for the origin of the hatch pattern.
- **Offset X** and **Y** move the hatch pattern's origin from 0,0.
- **Draw Mode** specifies Winding (pattern covers overlapping areas) or Alternate (pattern skips alternating overlapping areas); equivalent to *island detection* in AutoCAD.
- **Transparency (%)** specifies translucency of solid and gradient fills; 0 = opaque, 100 = invisible.



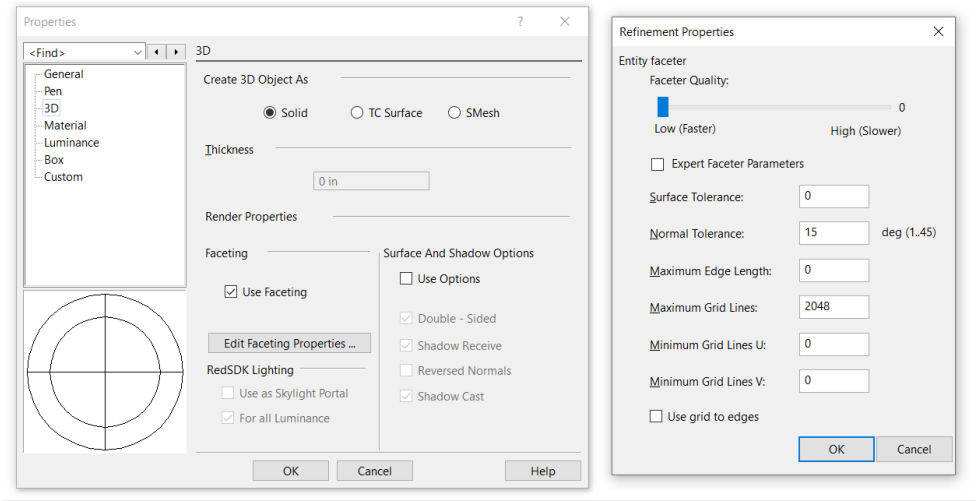
Brush page of the Properties dialog box

TIP: *Gradient fills must be defined using Brush Styles.*

3D Properties

The 3D page sets parameters for creating 3D objects:

- **Create 3D Object As** specifies **Solid** (3D solids created by the ACIS modeler) or **TC Surface** (3D surfaces created by TurboCAD's own modeler); AutoCAD uses the ACIS-based ShapeManager modeler.
- **Thickness** specifies default thickness for 2D objects.
- **Object Specific Faceting** can be controlled on an individual object basis via the Properties of the object. Object specified faceting overrides the global setting in the ACIS settings.

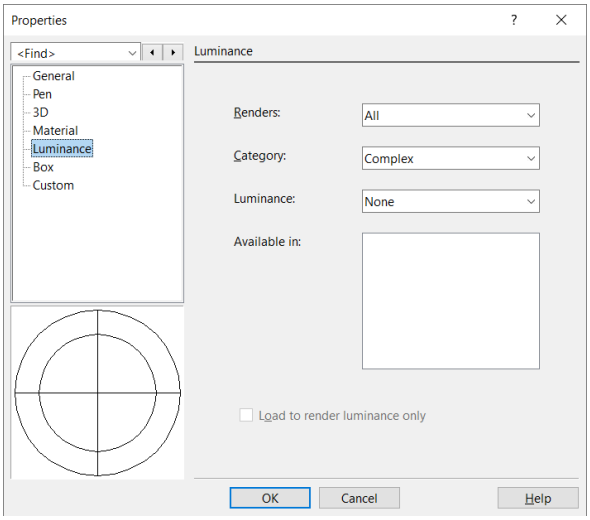


3D page of the Properties dialog box

Luminance Properties

The Luminance page sets parameters for placing lights in 3D scenes:

- **Renders:** Select which render engine you wish to select a material from: RedSDK, LightWorks, or both.
- **Category:** Groups of Luminances such as complex, single, sky, or the name of an open drawing.
- **Material:** This field depends on the category. For example, if the Category is sky, etc.
- **Edit Luminance:** The list of the Luminances available in the selected category.
- **Available In:** Shows which of the render engines have definitions for the selected luminance. RedSDK, LightWorks, or both.
- **Load to render luminance only:** If on, the geometry of the luminance object is hidden and only the illumination effects are shown.



Luminance page of the Properties dialog box

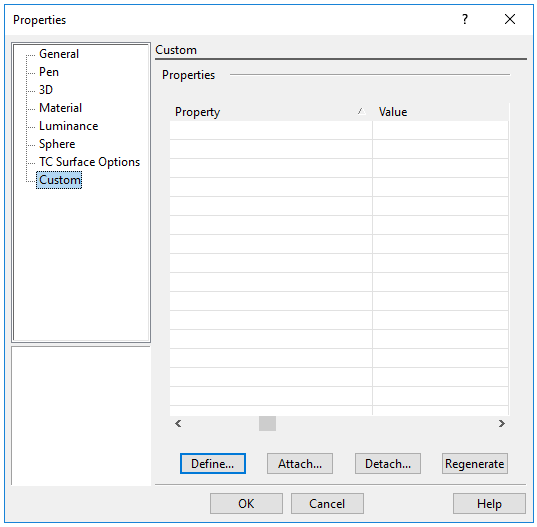
Material Properties

Properties related to material for the 3D objects. They are not enabled for 2D objects.

Custom Properties

The Custom page records TurboCAD parameters selected through the Define Property button:

- **Property** lists the name of the property.
- **Value** specifies value for the property.
- **Define** defines the types of information to be recorded.
- **Regenerate** updates the database.



Custom page of the Properties dialog box

Summary

Congratulations! In this lesson, you did a lot of drafting. You inserted rectangles, circles, arcs, lines, and polylines. You used TurboCAD's snaps, Edit bar, and Inspector bar. You learned how to create and insert blocks.

In the next lesson, you learn how to dimension with TurboCAD, place text, and plot the drawing.

At this point, you can exit TurboCAD by selecting **File | Exit** from the menu bar. When you want to continue with the next lesson, double-click the TurboCAD icon. After TurboCAD starts, open the *MechPart* drawing.

3

Applying the Finishing Touches

Lesson 3: Dimensions, Paper Space, Text, and Plots

In this lesson, you apply finishing touches to the mechanical part drawing.

Activities in this Lesson

- Dimension using horizontal, vertical, and radial dimensions.
- Switch to paper space.
- Place and modify text.
- Plot the drawing with the **Page Setup** and **Print** commands.

If TurboCAD is not running with the drawing from Lesson 2 loaded, start TurboCAD now, and load the *MechPart* drawing. Alternatively, you can double-click the *MechPart.Tc*w file in Windows Explorer; this action loads both TurboCAD and the drawing.

Dimensioning with TurboCAD

TurboCAD's dimensioning capability is similar to AutoCAD's. Dimensions are associative. TurboCAD's associative dimensions move and stretch when the associated object is moved and stretched. For example, rotate an object and the dimension rotates with it.

In addition to the dimension commands, you can create dimension styles in TurboCAD: right- click and select the **Properties** dialog box for modifying the dimension style. You can also do this from Style Manager palette.

TurboCAD Dimension Commands	AutoCAD Equivalent
Orthogonal	DimLinear
Parallel	DimAligned
Distance [1]	...
Rotated	DimRotated
Datum	DimOrdinate
Baseline	DimBaseline
Continuous	DimContinue
Incremental	...
Angular	DimAngular
Radius	DimRadius
Diameter	DimDiameter
Leader	Leader, QLeader
Quick	QDim
... [2]	DimCenter
... [3]	Tolerance

[1] Places constrained dimensions between parallel lines; not available in AutoCAD.
[2] Not available in TurboCAD.
*[3] Select from **Dimension | Tolerance** or **Customized > Commands > Tolerance***

Creating Dimensions in TurboCAD

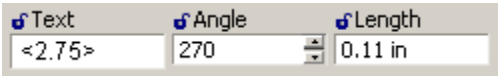
Dimensioning in TurboCAD is similar to AutoCAD:

Step 1: Start a dimensioning command.

Step 2: Select points or entities to locate extension lines.

Step 3: Pick a point to locate the dimension line.

You can change the dimension text, angle, and length of the extension lines in the Inspector bar. The **<** angle brackets **>** indicate the default text as measured by TurboCAD. The padlock icons allow you to place dimensions with fixed text, at a fixed angle, and with fixed extension lines.



TurboCAD's Inspector bar for orthogonal dimensions

Lesson 3 **Applying the Finishing**

TurboCAD contains a variety of dimension tools that you can use to display the measurements of lines and angles.



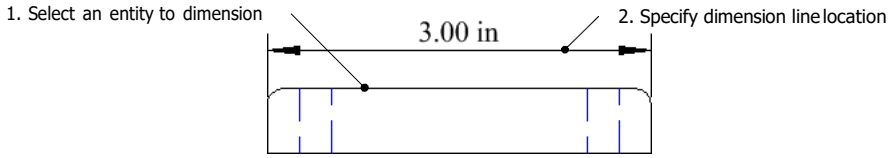
Dimensions consist of three basic components:

Dimension lines, with arrows pointing to either end of the dimension. A linear dimension can have an interior dimension line or two exterior dimension lines. Exterior dimension lines can be supplemented with an optional interior line.

Extension lines, which connect the dimension line to the object being dimensioned. Extension lines have optional line segments that continue the extension outward beyond the dimension line (extensions to the extension).

Dimension text, typically displaying the distance being dimensioned in World units.

- **Entity** — applies dimensions to entire entities:



Pick one entity and locate the dimension line

- **Quick** — applies dimensions to all related geometric features:



Pick two points and locate the dimension line

- **Constraint** — uses dimensions to constrain the positions of entities.

TIP: *Since the drawing has a scale of 1:1, it is not necessary to set the dimension scale. If you need to change the scale of dimensions in work space (a.k.a. model space), change the value of* ***Dimension Size Scale*** *in the* ***Advanced Format*** *tab of the* ***Properties*** *dialog box. To have paper space dimensions displayed at a different scale, change the value of* ***Linear Measurement Scale*** *in the* ***Advanced Text*** *tab.*

AutoCAD: Drawing Dimensions

AutoCAD: Object Dimensions

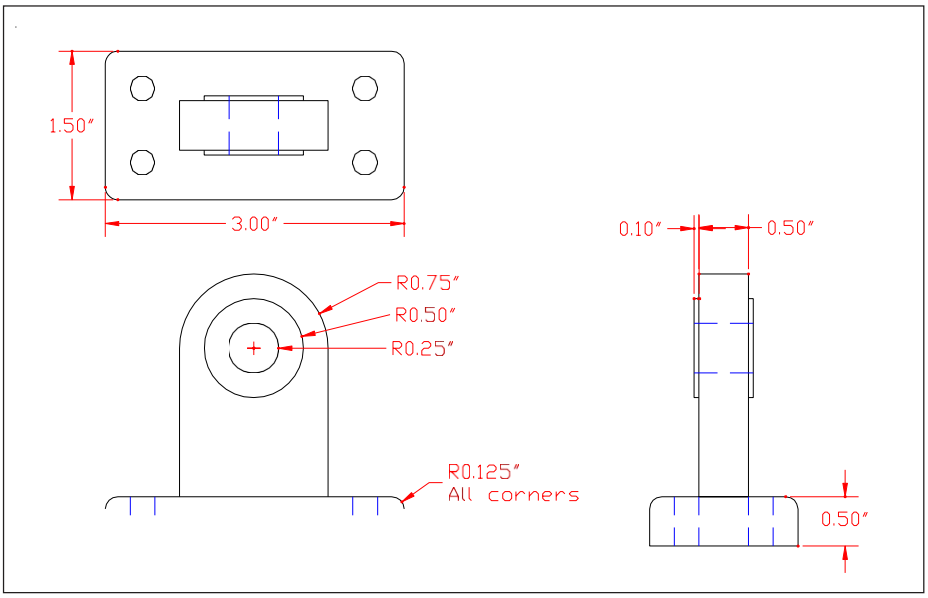
1. Freeze layer "Construction," because you don't need the construction lines anymore. Switch to layer "Dims" by selecting the name from the Layer toolbar.
2. From the **Dimension** menu, select **Linear.**
3. You enter AutoCAD's "entity" dimensioning mode when you press **Enter** at the prompt, '...press ENTER to select', as follows:

   ```
   _dimlinear
   First extension line origin or press ENTER to select: (Press Enter.)
   ```

4. Pick the bottom line of the base, as shown in step 4 on the TurboCAD side.

   ```
   Select object to dimension: (Pick line.)
   ```

5. Move the mouse downward so that the dimension pulls away from the base.
6. Click the mouse button to place the horizontal dimension.
7. Complete the entity dimensioning with the **Dimension | Linear** and **Dimension | Radius** commands.



Mechpart drawing with entity dimensions

TurboCAD: Inserting Dimensions

TurboCAD: Entity Dimensions

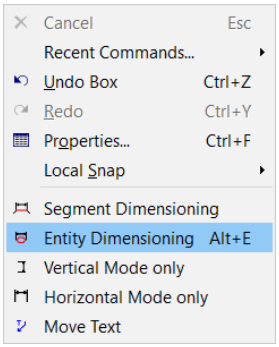
1. Freeze layer "$Construction," because you don't need the construction lines anymore. Switch to layer "Dims" by selecting the name from the Properties toolbar.



2. Select **Dimension | Orthogonal**. This command inserts horizontal and vertical dimensions.

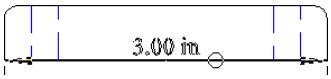


3. To turn on entity dimensioning, right-click to display the local menu. Select **Entity Dimensioning** so that the icon (to the left) darkens.



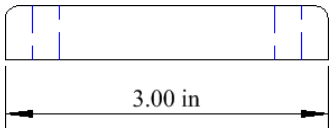
TurboCAD's local menu toggles dimensioning modes

4. Select the bottom line of the base, as shown in the figure.



TurboCAD's entity dimensioning

5. Move the mouse downward, so that the dimension pulls away from the base.

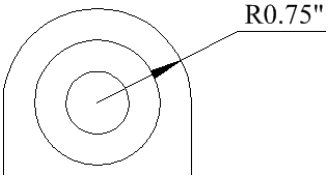


TurboCAD positioning the dimension line

6. Click the mouse button to place the dimension.



7. Continue dimensioning the horizontal and vertical portions of the drawing. Use the **Dimension | Radius** command for round parts.



TurboCAD's radial dimensioning

AutoCAD: Segment Dimensions

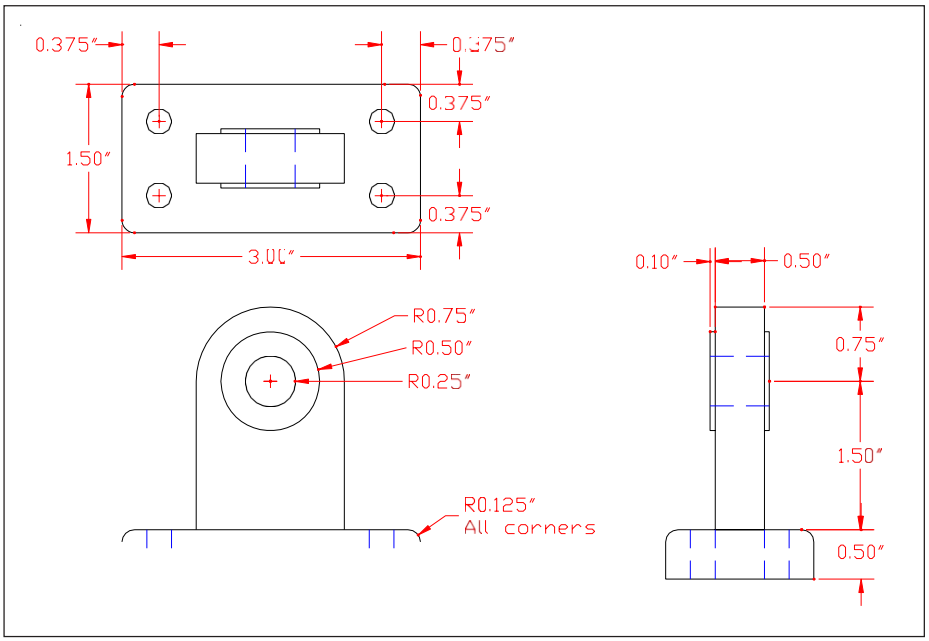
To dimension the distance between the mounting holes on the base:

1. From the **Dimension** menu, select **Linear**.
2. While holding down the **Ctrl** key, right-click and then select **Center** osnap mode.
3. When AutoCAD prompts you, pick the first circle.

   ```
   _dimlinear
   First extension line origin or press ENTER to select: (Pick circle.)
   ```
4. Then, pick the second circle:

   ```
   Second extension line origin: (Pick circle.)
   ```
5. Drag the dimension away from the base, and then click the mouse button to place the dimension.

   ```
   Dimension line location (Mtext/Text/Angle/Horizontal/Vertical/
         Rotated): (Place dimension.)
   ```
6. Press the **Enter** key to repeat the **Dimlinear** command, and then dimension the vertical distance between the mounting holes.
7. Save your work with **Ctrl+S**.

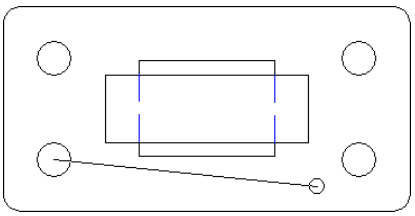


Mechpart drawing with completed dimensioning

TurboCAD: Segment Dimensions

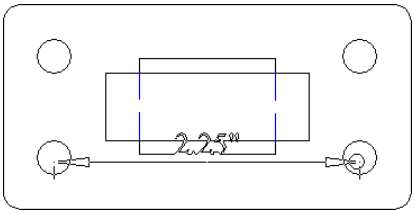
One area of the mechanical part cannot be entity-dimensioned: the distance between the mounting holes on the base.

1. Select **Dimension | Orthogonal**.
2. Right-click and deselect (turn off) **Entity Dimensioning**.
3. TurboCAD prompts you, "Define the first extension line origin or use local menu to select." Place the cursor near the first circle, and then press **C**. TurboCAD snaps the first extension line to the center of the circle.



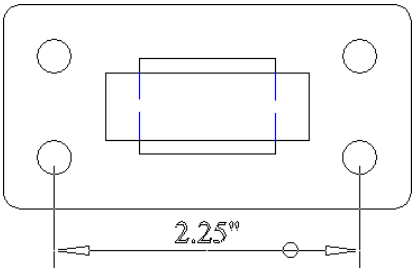
TurboCAD snapping to the first circle

4. At the "Define the second extension line origin" prompt, move the cursor to the second circle, and press **C** again.



TurboCAD snapping to the second circle

5. When TurboCAD prompts, "Define any point on the dimension line," drag the dimension away from the base, and then click.



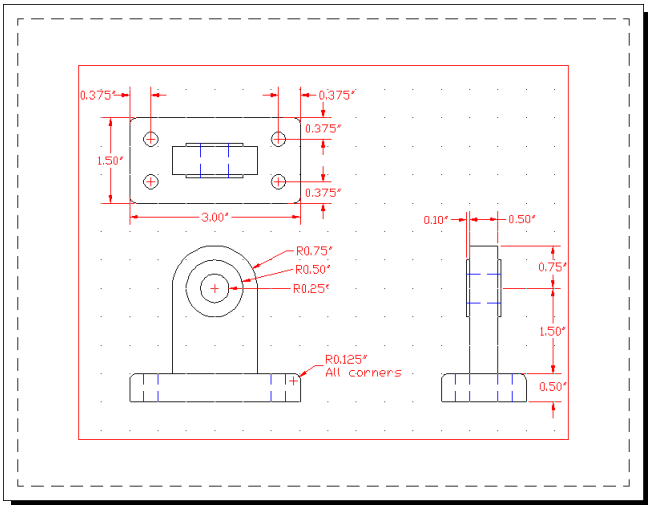
TurboCAD positioning the dimension line

6. Dimension the vertical distance between the two holes.
7. Save your work with **Ctrl+S**.

AutoCAD: Switching to Paper Space

Before placing text and plotting the drawing, switch from model space to paper space (layout mode).

1. Click the **Layout1** tab. (If the Page Setup Manager dialog box appears, click **Close**.) Notice that the model appears in a viewport.



AutoCAD model appears in layout viewport

TurboCAD: Switching to Paper Space

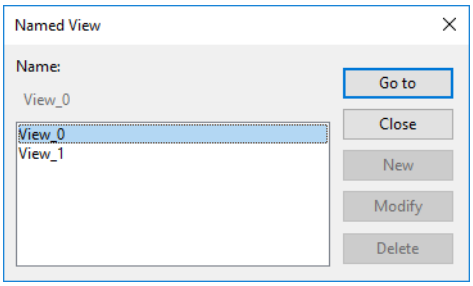
Before placing text and plotting the drawing, switch the *working space* (a.k.a. model space) to paper space. TurboCAD handles paper space somewhat differently from AutoCAD. In both CAD programs, you first create one or more viewports in paper space. In AutoCAD, you then pan and zoom the model to get the view you want in each viewport; in TurboCAD, you select a named view to display in each viewport.

1. Let's take a look at how this works:
   a. Click the **Paper 1** tab. Notice that TurboCAD opens a blank paper.



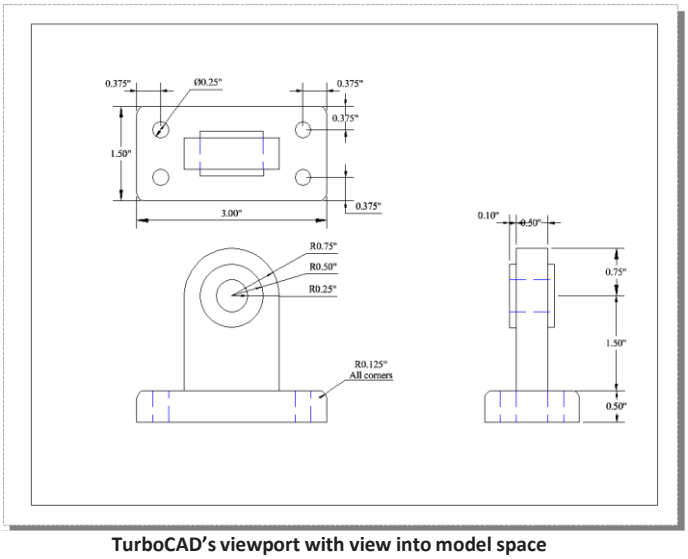


   b. From the **View** menu, select **Viewport**. TurboCAD prompts you to pick the two points that form the viewport's corners.
   c. Notice that the Named View dialog box appears. Choose **View_0**, and then click **Go To**. "View_0" is TurboCAD's generic name for the entire model view. You can create additional named views for showing details.



TurboCAD's Named View dialog box

Notice that the viewport fills with the model space view.



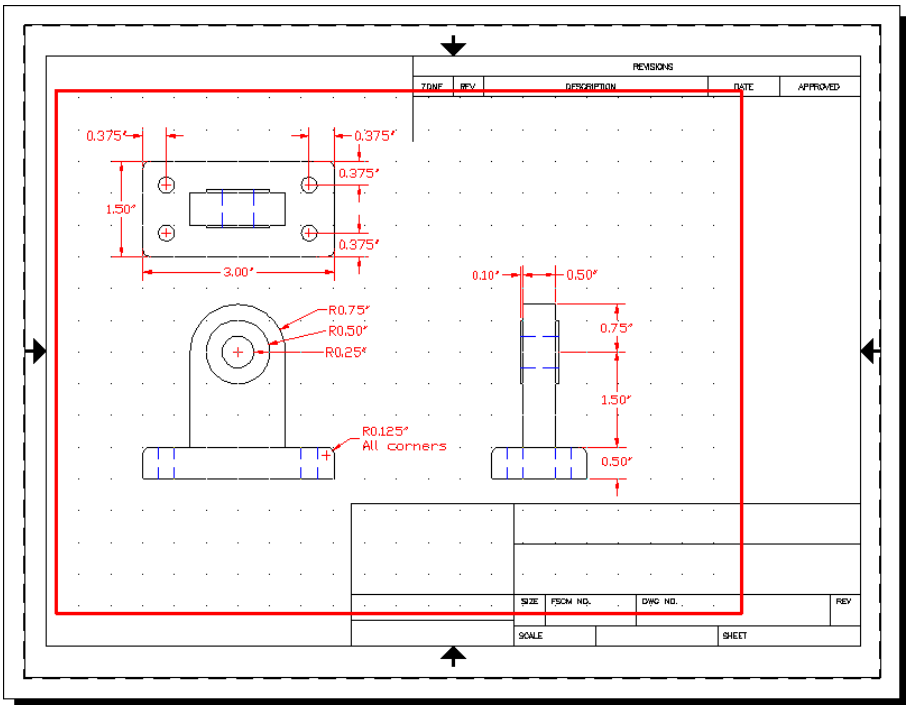
TurboCAD's viewport with view into model space

2. To add the title block and drawing border, use the **Insert | External Reference** command. AutoCAD does not make it easy to attach title block drawings. With the **Look In** droplist, make your way over to the following folder:

   `C:\Users\<login name>\AppData\Local\Autodesk\AutoCAD`

   `<version number>\<release number>\enu\Template`

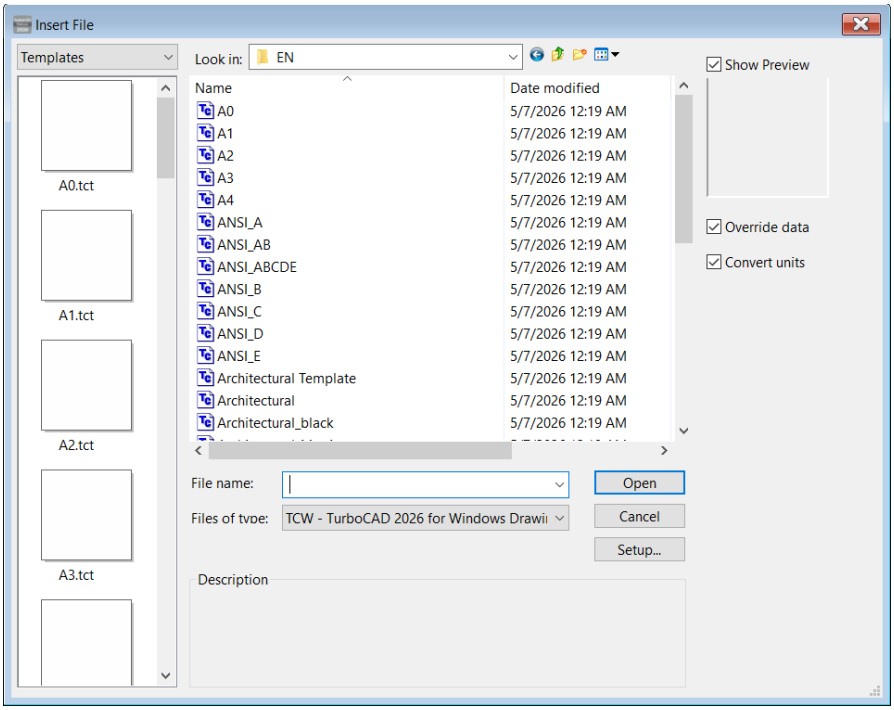
   (Autodesk chose to place the drawing border files in a hidden folder: Local Settings. If you cannot find it, you need to turn off the **Do Not Show Hidden Files and Folders** option in Windows Explorer — found in the **Tools | Folder Options | View** dialog box.)
3. Select "Generic 24in x 36in Title Block.dwg" file, and then click **Open**.
4. Position the title block drawing in model space.
   If the drawing interferes with the title block, you may need to reposition the viewport and scale the drawing.



AutoCAD model positioned outside of title block



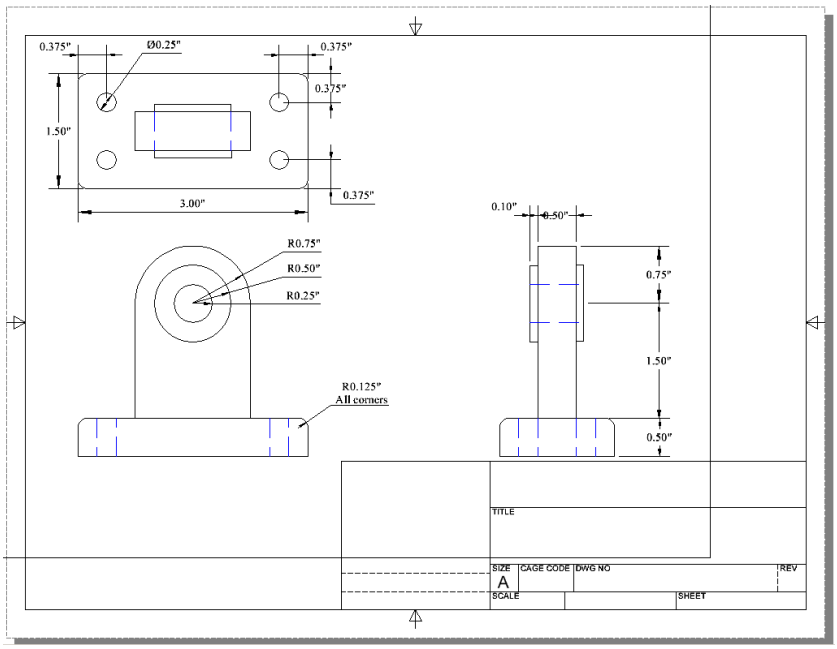
2. To add the title block and drawing border, use the **Insert | File** command. TurboCAD makes it easy to attach title block drawings: select **Templates** from the droplist.



TurboCAD's Insert File dialog box

3. Select "ANSI A .tct" from path `C:\Program Files\IMSIDesign\TCW<version number>\Template`, and then click **Open**.
4. Position the title block drawing in model space.

If the drawing interferes with the title block, you may need to reposition the viewport and scale the drawing.



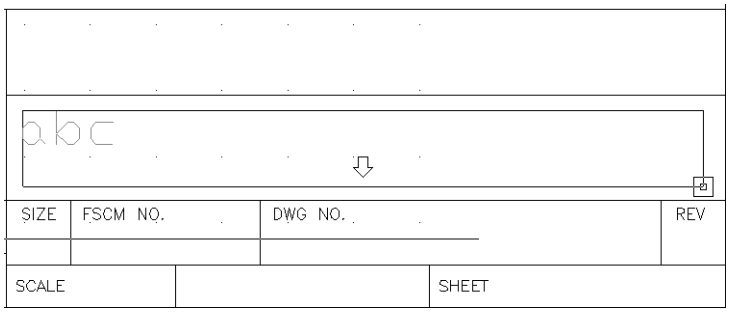
TurboCAD's title block and drawing border

AutoCAD: Drawing Text

Fill in the title block by following these steps:

1. Zoom in to the title block area with the **Zoom Window** command.
2. Click the **Draw | Text | Multitext** icon. AutoCAD starts the **MText** command:

   ```
   _mtext Current text style: STANDARD. Text height: 0.2000
   ```



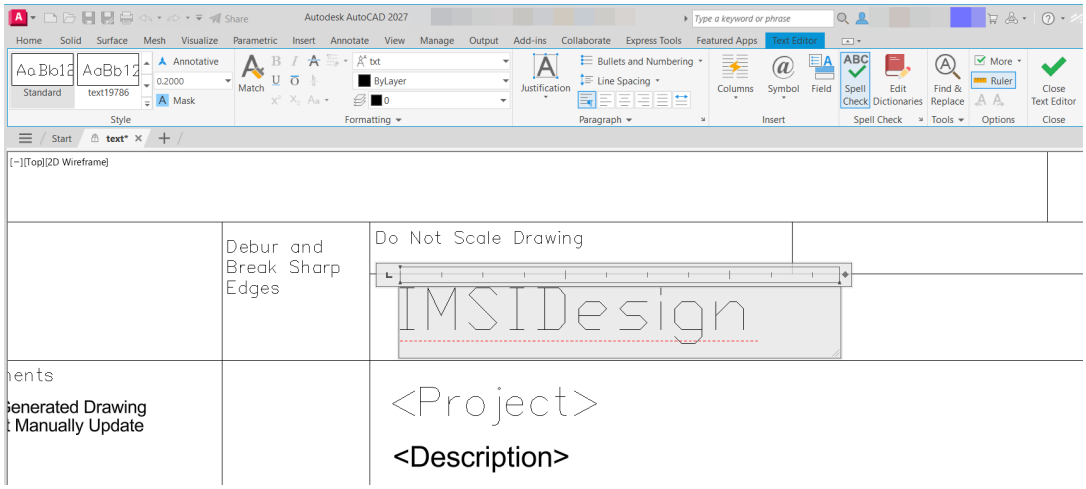
AutoCAD specifying the Mtext rectangle

3. Pick two points to specify the rectangle in which to place the text:

   ```
   Specify first corner: (Pick a point.)
   Specify opposite corner or [Height/Justify/Rotation/Style/Width]:
   (Pick another point.)
   ```

4. When the Multiline Text Editor appears, type "Mechanical Part."
5. If you need to make the text larger, use the mouse to select the text, which highlights it. Change the default text height from 0.2. (If you don't first select the text, AutoCAD does not change its size.)
6. Change the font from the ugly-looking Txt font to something bolder, such as Arial Black

or any other font found on your computer.



AutoCAD adding text to the title block

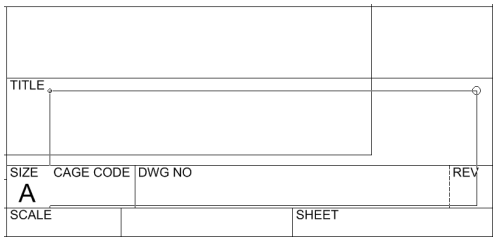
7. When finished with the text, click **OK**, and AutoCAD places the text in the drawing.

TurboCAD: Inserting Text

Like AutoCAD, TurboCAD uses TrueType and SHX-format fonts for text in its drawings. You can use all of AutoCAD's fonts in TurboCAD drawings. (To create AutoCAD-like text styles, you go through the process described in the previous chapter for creating entity styles.)

Fill in the title block by following these steps:

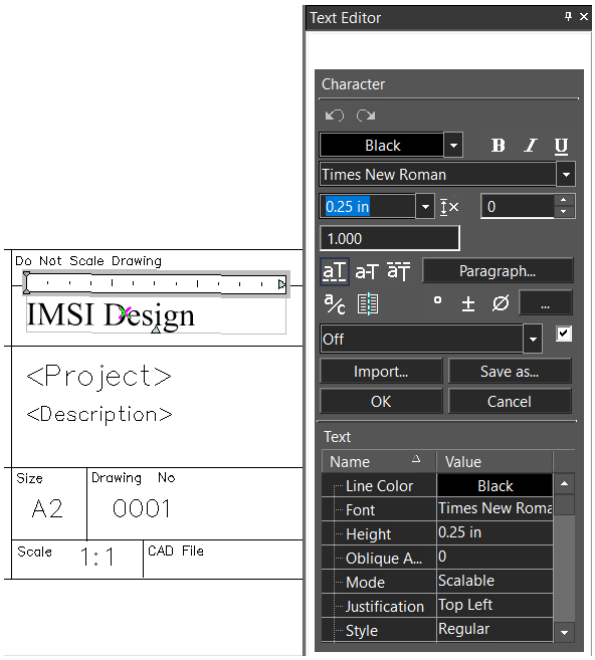
1. Zoom in to the title block area with the **View | Zoom Window** command.
2. On the toolbar, click the **Multi Text** or go to **Draw | Text | Multi Text** icon.
3. Pick two points to specify the rectangle in which to place the text:



TurboCAD specifying the multi text rectangle

```
Define the insertion point for the text (Pick a point.)
Define the text width and angle: (Pick another point.)
```

4. When the Multiline Text Editor appears, type "Mechanical Part."
5. If you need to make the text smaller, use the mouse to select the text, which highlights it. Change the default text height to 24pt. (If you don't first select the text, TurboCAD does not change its size.)
6. Change the font to Arial Black or any other font found on your computer.



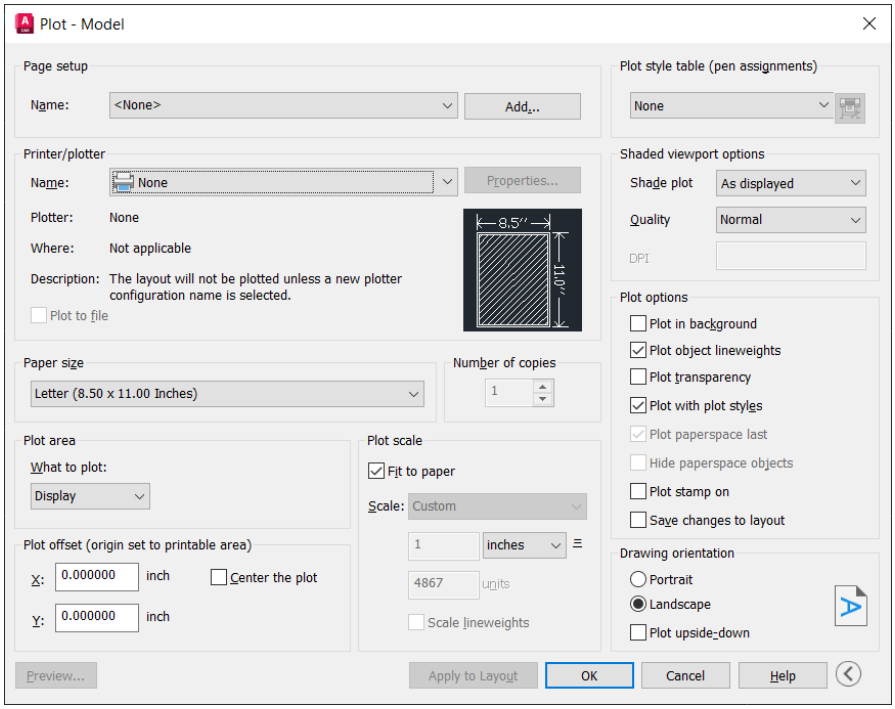
TurboCAD adding text to the title block



7. When finished with the text, click the **Finish** flag icon. Notice that TurboCAD places the text in the drawing.

AutoCAD: Plotting Drawings

1. To set up the drawing for plotting, select **File | Print**.



AutoCAD's Plot dialog box

2. Change parameters to make the drawing fit your printer's paper:
   - **Printer/plotter - Name** — select a printer or plotter from the droplist.
   - **Plot area - What to plot** — select "Layout" from the droplist.

TurboCAD: Plotting Drawings

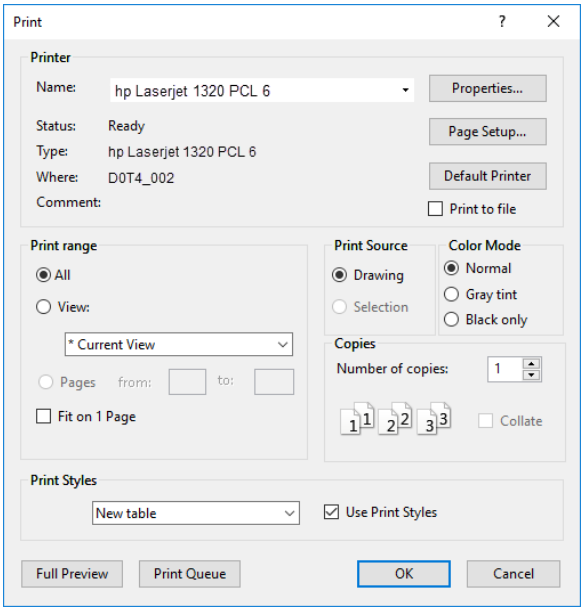
To plot a drawing in TurboCAD, follow these steps:

a. Choose **File | Print** to set up the printer.
b. Click **Page Setup** to set up the paper.
c. Click **Print** to plot the drawing.

Let's follow these stepsnow:



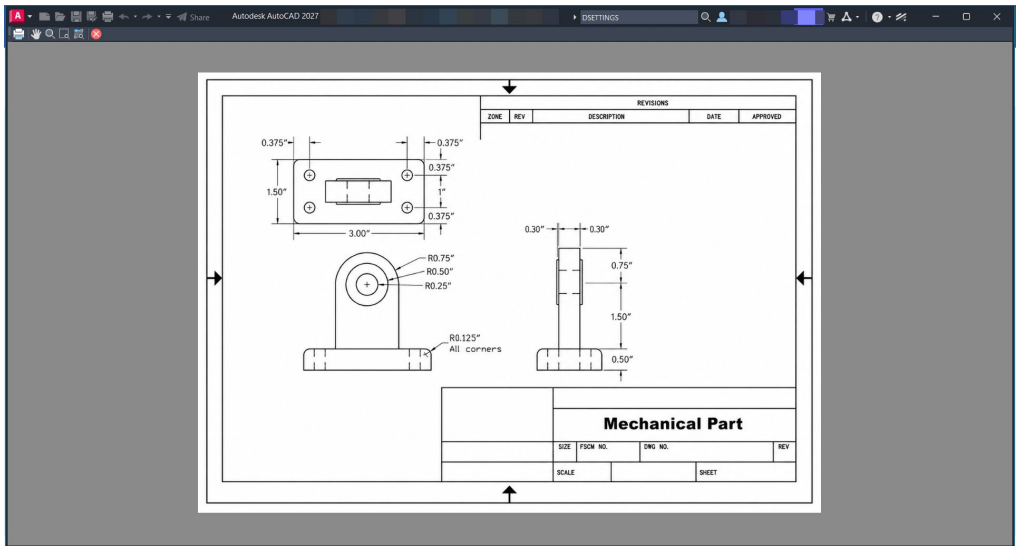
1. From the **File** menu, select **Print**. The Print dialog box contains options that primarily affect the printer.



TurboCAD's Page Setup dialog box

2. Choose parameters to make the drawing fit your printer's paper:
   - **Printer - Name** — select a printer or plotter from the droplist.
   - **Print range - View** — select "*Current View" from the droplist.
   - **Fit on 1 page** — select to turn on (check mark shows).

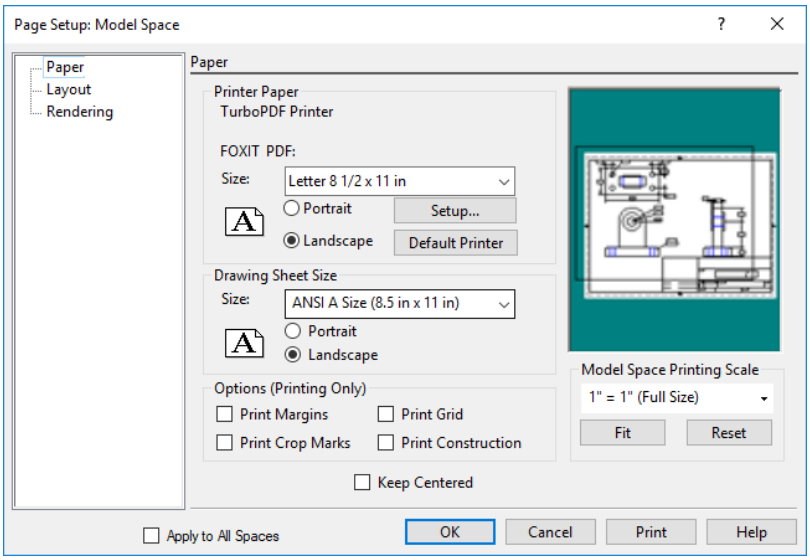
3. Preview the plot by clicking the **Preview** button.



AutoCAD's Plot dialog box

4. Click the printer icon on the toolbar to produce a hardcopy of your drawing.
5. Press **Ctrl+S** to save the drawing, and then exit AutoCAD with **File | Exit**.

3. Preview the plot by clicking the **Page Setup** button. The dialog box also controls the size, orientation, and optional markings of the paper. You have the option of letting TurboCAD print the following:
   - A rectangular margin around the drawing (quick substitute for drawing borders).
   - Crop marks at the four corners (useful for commercial printers).
   - Grid lines.
   - Construction lines.



TurboCAD's Page Setup dialog box

The preview window lets you see what's happening.

If the drawing does not fit the page, make the following changes to parameters:

- **Printer Paper** — change orientation to landscape.
- **Paper Space Printing Scale** — choose "1"=1" (Full Size)".

4. Click the **Print** button to produce a hardcopy of your drawing.
5. Press **Ctrl+S** to save the drawing, and then exit TurboCAD with **File | Exit**.

Summary

In this lesson, you learned how to dimension the drawing, switch from working space to paper space, place text, and plot the drawing.

In the following chapter, you learn how AutoCAD and TurboCAD produce 3D solid models of the same mechanical part.

4

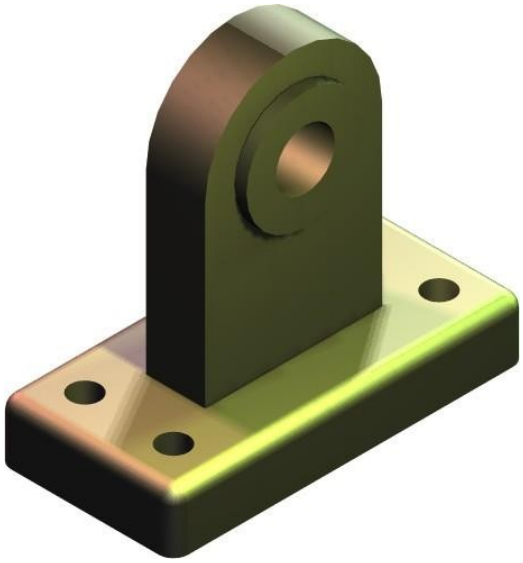
3D Design

Lesson 4: Designing a Three-dimensional Model

CAD software provides several approaches to creating 3D models, including the following:

- Inserting primitives, and then editing them into shape.
- Extruding 2D drawings, and then modifying them.

In this lesson, you use the second approach, because you already have the 2D drawing. The result will be a 3D model that looks like the figure illustrated below.



Completed 3D model designed by TurboCAD

AutoCAD: Preparing for 3D Modeling

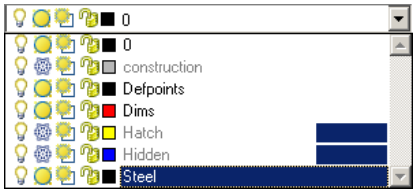
Prepare to create the 3D model by following these steps:

1. Three-dimensional modeling is always performed in model space. (Paper space is strictly for 2D layouts.) Return to model space by clicking the **Model** tab.



AutoCAD: Switching from layout mode

2. You no longer need the construction lines, dimension lines, and hidden lines. Turn off the related layers:
   a. On the Layers toolbar, open the layers droplist by clicking on it.
   b. Click the lock icon next to the following layers:
      - Dims
      - Hidden
      - Construction



AutoCAD: Turning off unneeded layers

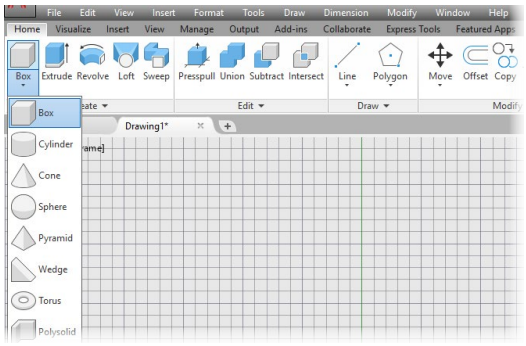
3. For 3D modeling tools. Go to **Tools** menu, select **Toolbars | AutoCAD | Modeling.**



AutoCAD: 3D Make panel

4. AutoCAD also provide Workspace option, through which selected workspace appear. For 3D Modelling go to **Tools** (click on Show Menu Bar if it's hidden) **| Workspace | 3D Modelling.** Menu bar will hide after completion.

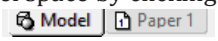
5. 3D Basics workspace has basic tools for 3D drawing.



TurboCAD: Preparing for 3D Modeling

Prepare to create the 3D model by following these steps:

1. Three-dimensional modeling is always performed in model space. (Paper space is strictly for 2D layouts.) Return to model space by clicking the **Model** tab.



TurboCAD: Switching from paper space

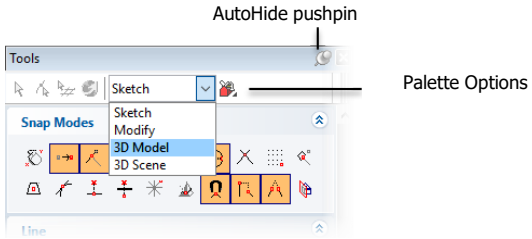
2. You no longer need the construction lines, dimension lines, and hidden lines. Turn off the related layers:
   a. On the Object Properties toolbar, open the layers droplist by clicking on it.
   b. Click the eye icon next to the following layers:
      - Dims
      - Hidden
      - $Construction



TurboCAD: Turning off unneeded layers

3. The Tools palette is useful for accessing 3D modeling tools. To turn it on:
   a. Select **Tools** from the **View** menu.

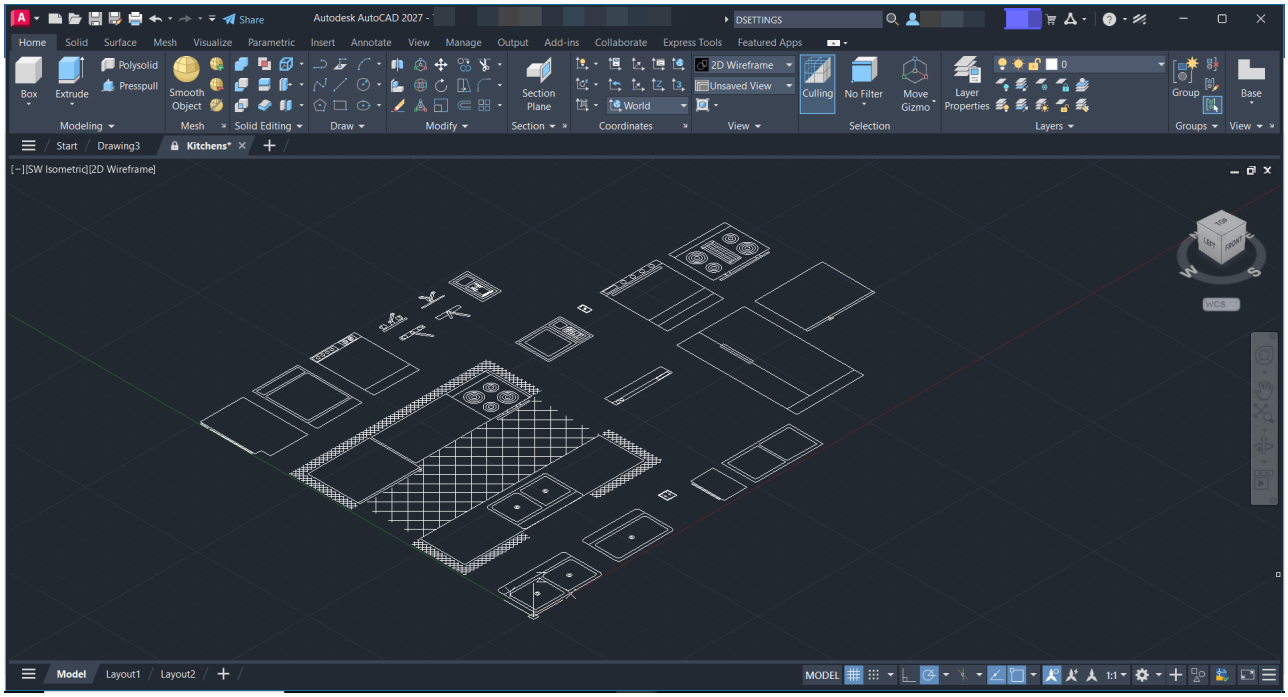
      Keep it displayed for the duration of the modeling session by clicking the **AutoHide** pushpin icon; when the pushpin points down, the palette no longer hides automatically.



TurboCAD: Locking the palette into place

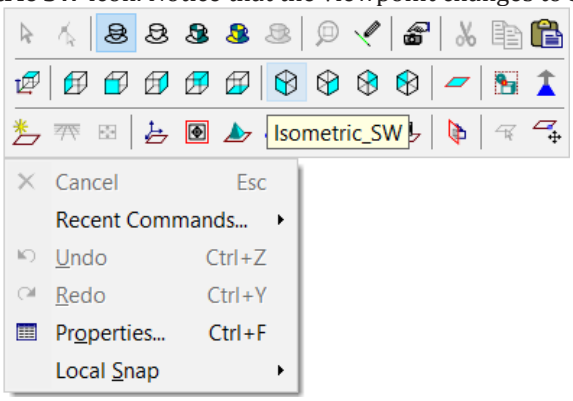
   b. From the Sketch droplist, select **3D Model**. This displays all 3D modeling-related commands.

4. From the **View** menu, select **3D Views**, and then select **SW Isometric**. Notice that the viewpoint changes to 3D.
5. Save your work with **Ctrl+S**.



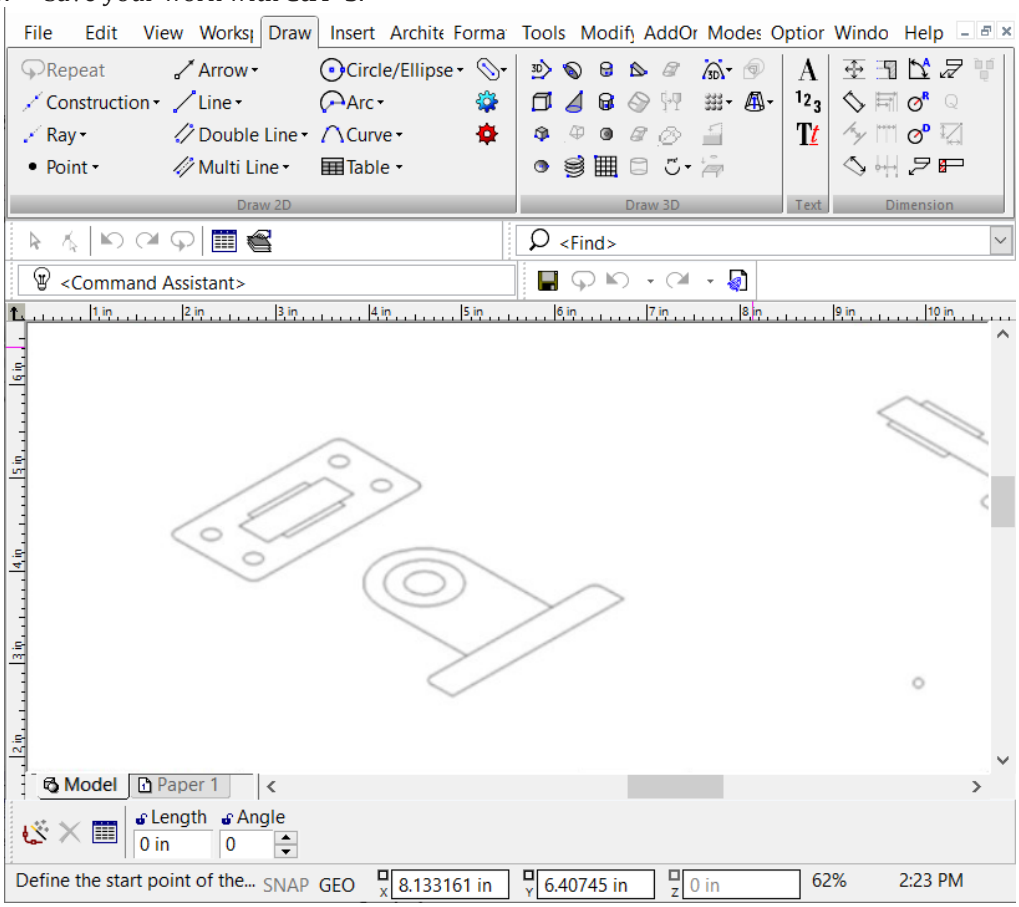
AutoCAD: 2D drawing displayed in 3D viewpoint

4. Change the viewpoint to 3D. Right-click the drawing, and then from the local menu, click the **Isometric SW** icon. Notice that the viewpoint changes to 3D.



TurboCAD: Selecting the isometric viewpoint

5. Save your work with **Ctrl+S**.



TurboCAD: 2D drawing displayed in 3D viewpoint

AutoCAD: Simple Extrusion

To convert the 2D drawing to a 3D solid, use the **Extrude** command. First, apply it to the mechanical part's base, as found on the former top view.

1. In **Home|Create** tab click **Extrude**. Notice that AutoCAD prompts you:

   ```
   Command: _extrude
   Current wire frame density:  ISOLINES=4
   ```

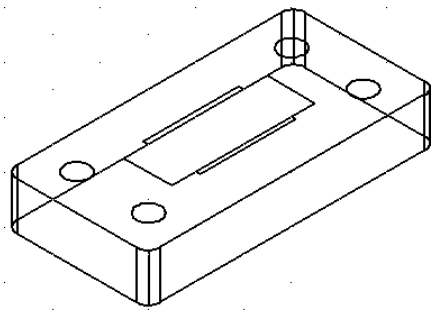
2. Select the base, and then press **Enter** to end object selection.

   ```
   Select objects to extrude: (Select the base.)
   Select objects to extrude: (Press Enter.)
   ```

3. When prompted for the extrusion distance, enter **-0.5**, and then press **Enter**.

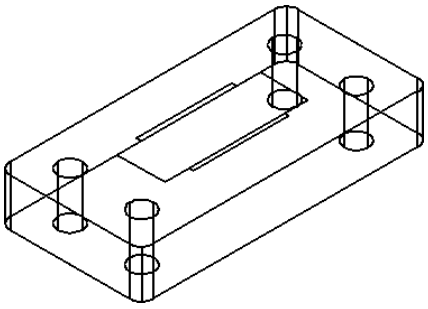
   ```
   Specify height of extrusion or [Direction/Path/Taper angle]
         <0'-1.00">: -.5
   ```

   Notice that the base extrudes downward by a half-inch. (The reason for extruding it downwards is to make it easier to assemble the model later.)



AutoCAD: Extruded base

4. Repeat for the four holes, extruding them downward by 0.5 inches.



AutoCAD: Extruded base and holes

TurboCAD: Simple Extrusion

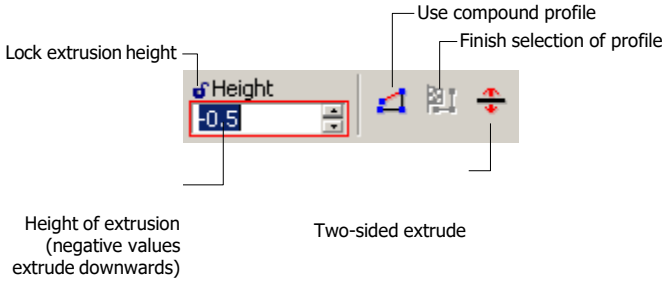
To convert the 2D drawing to a 3D solid, use the simple extrusion tool. First, apply it to the mechanical part's base, as found on the former top view.



1. In the Tools palette, click **Simple Extrude** in 3D Object section (found in the 3D Object section). Notice that TurboCAD prompts you:

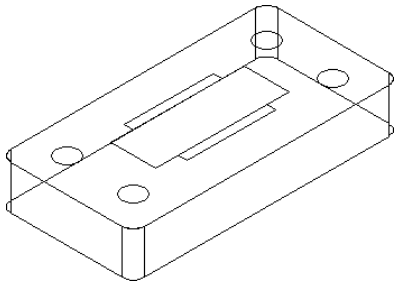
   `Select an entity to extrude` ***(Select the base.)***

2. Select the base, and then press **Tab** to access the Inspector bar.



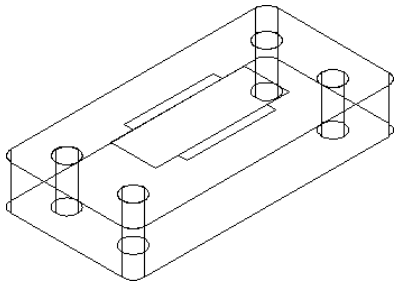
TurboCAD: Inspector bar for the Simple Extrusion tool

3. Enter **-0.5**, and then press **Enter**. Notice that the base extrudes downward by a half-inch. (The reason for extruding it downward is to make it easier to assemble the model later.)



TurboCAD: Extruded base

4. Repeat for the four holes, extruding them downward by 0.5 inches.



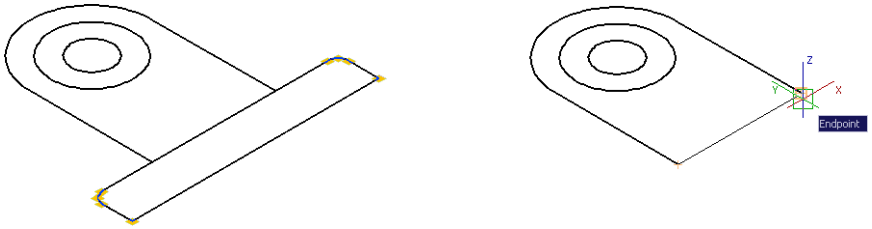
TurboCAD: Extruded base and holes

AutoCAD: Extrusion of Regions

Before you can extrude the next part, you need to modify the 2D drawing.

1. Erase the parts shown highlighted (the profile of the base), and replace them with a single line segment.

   From the **Draw** menu, select **Line**. It easiest to use the **ENDPoint** keyboard osnap mode to connect lines to existing geometry accurately. Use OSNAP command.



Left: **AutoCAD erasing the lines and arcs making up the profile of the base**

Right: **Replacing them with a single line**

2. AutoCAD lacks TurboCAD's abilities to use compound profiles or to make two-sided extrusions. The workaround is to create regions and to move the solids, respectively.
   a. From the **Draw** menu, select **Region**.
   b. Select the profile (outline) you created in the previous step. AutoCAD creates a region that can be extruded later as a solid.
   c. Execute the **Extrude** command using a height of 0.5".



Left: **AutoCAD's one-sided extrusion**
Right: **Circles extruded as cylinders**

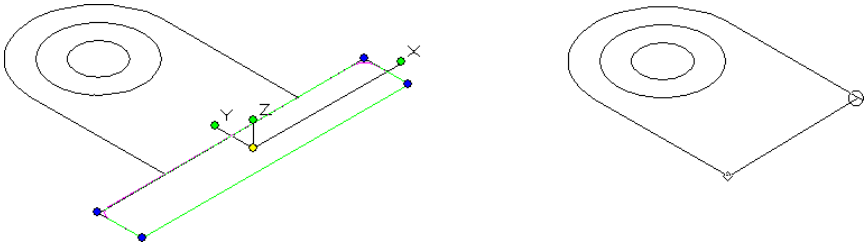
   d. Use the **Move** command to move the solid in the z-direction up a distance of 0.1".
3. Repeat the extrusion for the two circles using a height of 0.7".
4. Save your work with **Ctrl+S**.

TurboCAD: Two-sided Extrusion of Compound Profiles

Before you can extrude the next part, you need to modify the 2D drawing:

1. Erase the parts shown highlighted (the profile of the base), and replace them with a single line segment.

   From the **Draw** menu, select **Single**. I find it easiest to use the **V** (vertex) key- board shortcut to connect lines to existing geometry accurately.



Left: **TurboCAD erasing the lines and arcs making up the profile of the base**

Right: **Replacing them with a single line**

2. For this extrusion, use the compound profile and two-sided extrude options:
   a. From the Tools palette, select **Simple Extrude**.
   b. On the Inspector bar, click the **Compound Profile** button to turn on the option.

      Click the **Two Sided Extrude** button, as well.
   c. Click the profile (outline) you created in the previous step.
   d. For Height, enter **0.25** (half the thickness). Notice that TurboCAD extrudes in both directions from the profile.



Left: **TurboCAD's two-sided extrusion**
Right: **Circles extruded as cylinders**

3. Repeat the two-sided extrusion for the two circles. Lock the extrusion height to **0.35**.
4. Save your work with **Ctrl+S**.

AutoCAD: Union and Subtract Solids

In 3D solids modeling, holes are created by subtracting cylinders from the rest of the solid. The base has four small holes, the upright part has one large hole.

1. To create the holes in the base, select **Subtract** from the **Create** section in **Home** Tab.
2. Notice that AutoCAD prompts you:

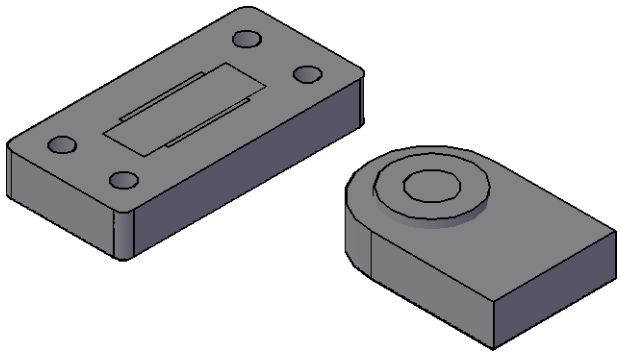
   ```
   Command: _subtract
   Select solids and regions to subtract from ..
   Select objects: (Pick the base.)
   ```

   Select the base, the part from which the cylinders will be subtracted.
3. Press **Enter** to end object selection.

   At the next prompt, select the four cylinders.

   ```
   Select solids and regions to subtract ..
   Select objects: (Pick the cylinders.)
   ```
4. Press **Enter** when done. The model won't look any different.

   ```
   Select objects: (Press Enter.)
   ```
5. To confirm that the cylinders have been removed, select **View |Visual Styles | Conceptual**. (There is no hole in the tombstone-shaped part, yet.)



AutoCAD: Draft rendering of base with holes

Return to wireframe mode: select **View | Visual Styles | Wireframe**.

6. To create the hole in the tombstone part, first union the tombstone to the outermost cylinder, and then subtract the innermost cylinder, as illustrated on the TurboCAD side.

TurboCAD: Adding and Subtracting Solids

In 3D solids modeling, holes are created by subtracting cylinders from the rest of the solid. The base has four small holes, the upright part has one large hole.

TIP: *IMSI/Design provides optional Mechanical Pack for TurboCAD that includes a fully parametric Hole tool which simplifies the hole-making and enables you to specify typical hole parameters, such as threading and counter-sinking.*

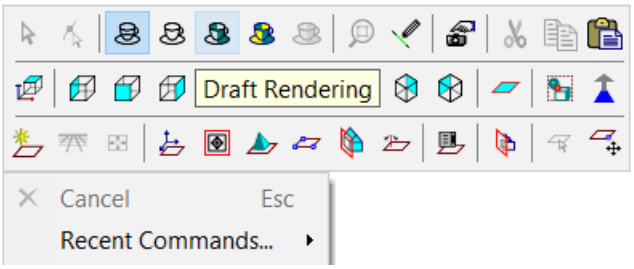


1. To create holes in the base, select **3D Subtract** from the Boolean Facet section of the Tools palette.
2. Notice that TurboCAD prompts you:

   `Select the first of 3d entities for subtract` ***(Pick the base.)***

   Select the base, the part from which the cylinders will be subtracted.
3. At the next prompt, select one of the cylinders.

   `Select next operand(s) by click or drag` ***(Pick a cylinder.)***
4. Repeat the process for each cylinder: pick the base, and then another cylinder. The model won't look any different.
5. To confirm that the cylinders have been removed, right-click the drawing, and then select **Draft Render**. There is no hole in the tombstone-shaped part, yet.

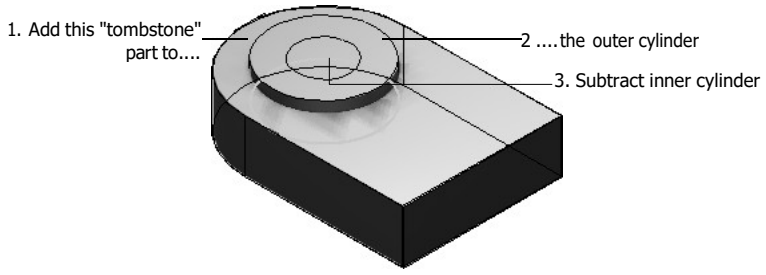




Left: **TurboCAD selecting the Draft Render command**
Right: **Draft rendering of base with holes**

Return to wireframe mode: right-click, and then select **Wireframe**.

6. To create the hole in the tombstone, first *add* (a.k.a. union) the tombstone part to the outermost cylinder, and then subtract the innermost cylinder, as illustrated below.



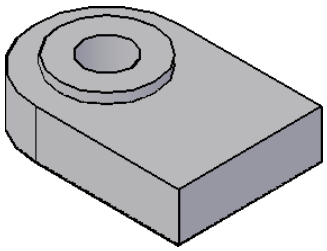
TurboCAD: The order in which solids will be added and subtracted

a. Click **Union** in Home **|** Edit tab. AutoCAD prompts you to pick two solids: `Select objects:` ***(Select the "tombstone" and the outermost cylinder.)*** `Select objects:` ***(Press Enter.)***

After a moment, the two parts are joined into a single part.

b. Select **Subtract**, and then subtract the innermost cylinder.

c. Apply a visual style to check the result.



AutoCAD: Result of adding and subtracting parts

7. Save your work with **Ctrl+S**.

AutoCAD: Model Assembly

The next stage is to assemble the two parts: (1) rotate the tombstone around the x axes by 90 degrees, and (2) move it into place on top of the base.

1. Here's how to rotate the tombstone in 3D space:

   a. Enter the **3DRotate** command.

   `Command:` **`3drotate`**

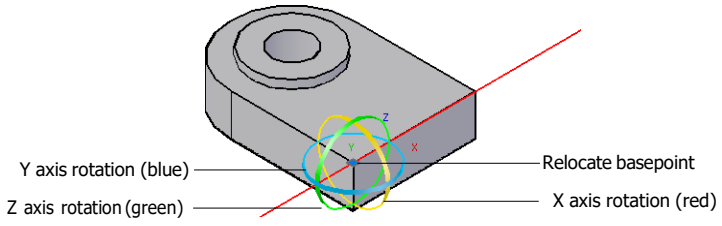
   `Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0`

   `Select objects:` ***(Select the tombstone.)***

   `Select objects:` ***(Press Enter to end object selection.)***

   b. Select the tombstone part. Notice the tri-circle grip tool.

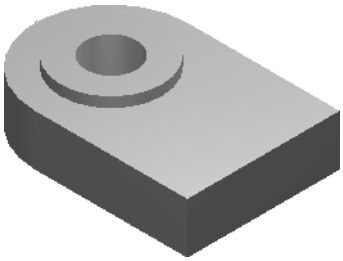
   `Specify base point:` ***(Pick a corner on the tombstone.)***



AutoCAD: Selecting the X axis handle



a. In the Tools palette, click **3D Add**. TurboCAD prompts you to pick two solids:
   `Select the first of 3D entities for add` ***(Select the "tombstone")***
   `Select next operand(s) by click or drag` ***(Select the outer cylinder)***
   After a moment, the two parts are joined into a single part.
b. Select **3D Subtract** from the Tools palette, and then subtract the innermost cylinder.
c. Do a quick render to check the result.



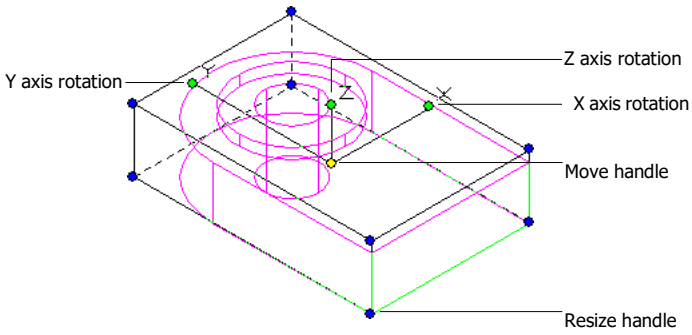
TurboCAD: Result of adding and subtracting parts

7. Save your work with **Ctrl+S**.

TurboCAD: Model Assembly

The next stage is to assemble the two parts: (1) rotate the tombstone around the x axis by 90 degrees, and (2) move it into place on top of the base.

1. Here's how to rotate the tombstone in 3D space:
   a. Press the spacebar to switch to **Select** mode.
   b. Select the tombstone part. Notice the 3D handles.



TurboCAD: Selecting the X axis handle

c. Click the red circle, which represents the X axis.

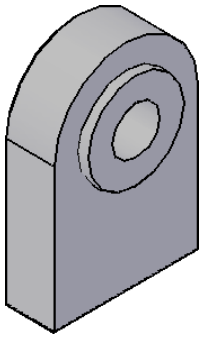
```
Pick a rotation axis: (Pick red circle.)
```

Notice that it turns yellow, and that a red axis line appears.

d. Specify the angle:

```
Specify angle start point: 90
```

Enter **90**, and then press **Enter**. Notice that the part rotates and becomes upright.



AutoCAD: Part rotated upright

2. The second step is to align the tombstone with the base. This is done with AutoCAD's **3dMove** command, as follows:

a. From the **Home** menu, in **Modify** tab select **3D Operations | 3D Move**.

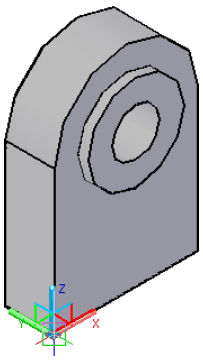
b. When AutoCAD prompts you to select the object, select the tombstone, and then press **Enter**:

```
Command: _3dmove
Select objects: (Pick tombstone, and then press Enter.)
```

c. You need to tell AutoCAD the basepoint from which to start moving:

```
Specify base point or [Displacement] <Displacement>: endp
of (Pick a bottom corner.)
```

Enter **ENDPoint** object snap, and then pick a corner, as illustrated below.

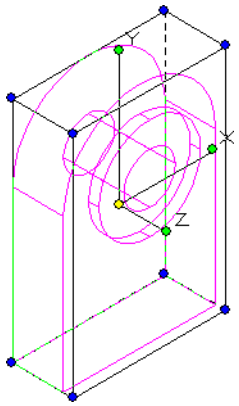


AutoCAD: Selecting the corner of the base

c. Click the green ball at the end the X axis handle.

d. Press **Tab** until the **Rot X** field is highlighted on the Inspector bar.

Enter **90**, and then press **Enter**. Notice that the part rotates and becomes upright.



TurboCAD: Part rotated upright

2. The second step is to align the tombstone with the base. This is done with TurboCAD's assembly feature, as follows:



a. From the 3D **Modify** menu in tools palette, select **Assemble by Edge and Point**.

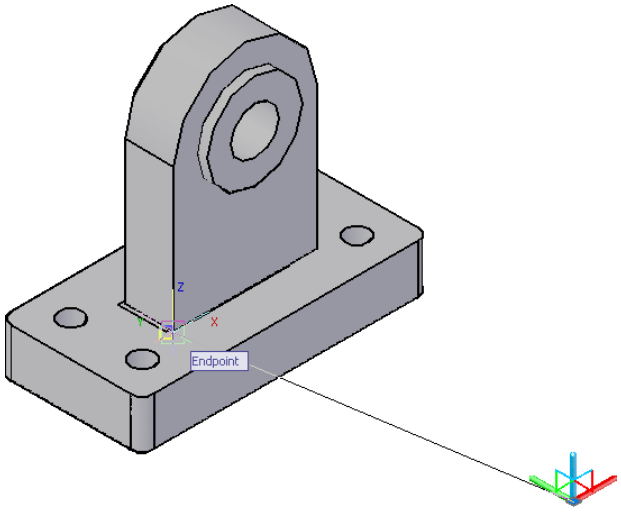
b. When TurboCAD prompts you to select the part, select the tombstone:

`Select an entity whose position you wish to change` ***(Pick tombstone.)***

c. You need to tell TurboCAD which surfaces to mate:

`Select the location of the source line (edge)` ***(Press C, and then pick a bottom line.)***

Notice that AutoCAD previews the move with a line.

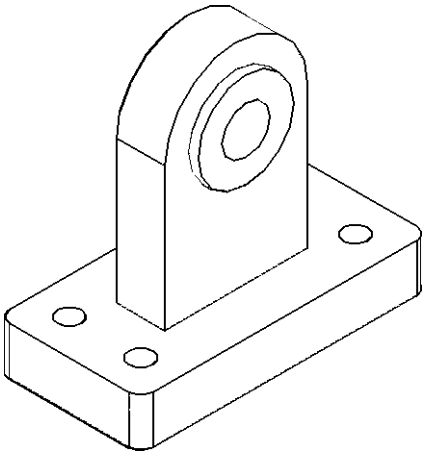


AutoCAD previews the move with a line

d. In the same way, specify the destination on the base:

```
Specify second point or <use first point as displacement>: endp
of (Pick the associated corner on the base.)
```

Enter **ENDPoint** osnap again, and then pick the equivalent corner on the base, as illustrated below.

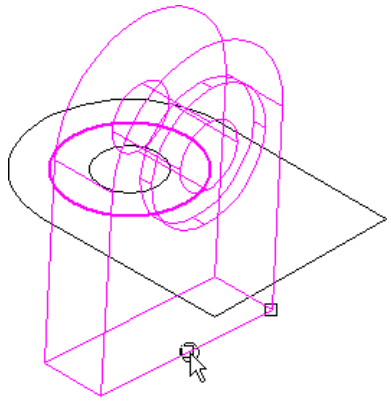


AutoCAD: The model shown with hidden lines removed

e. Use the **Union** command to join the two parts into a single 3D model.

To clean up the drawing, erase any 2D lines and arcs that are no longer needed.

Hold down **C** (or Shift+C) to invoke the Center object snap, and then pick a line at the bottom, as illustrated below.

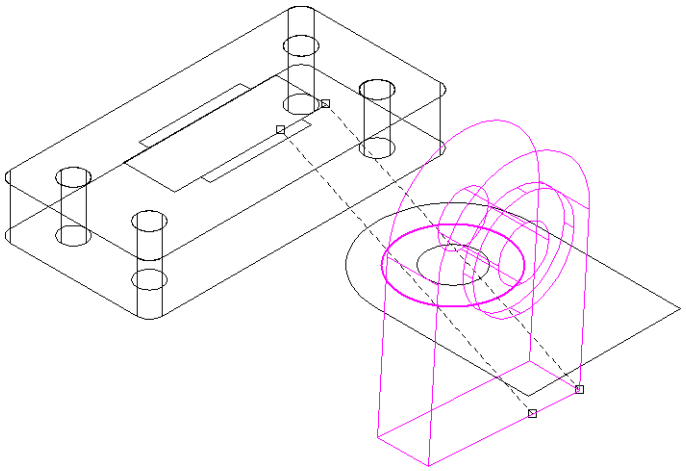


TurboCAD: Selecting the center of a base line

d. In the same way, specify the destination on the base:

`Select the location of the destination line (edge)` ***(Press C, and then pick the associated line on the base.)***

Hold down **C** again, and then pick the line on the base, as illustrated below. Notice that TurboCAD previews the assembly with dashed lines.



TurboCAD previews the assembly with dashed lines

To complete the assembly, click the **Finish** flag icon. Notice that TurboCAD moves the tombstone onto the base, as illustrated on the AutoCAD side.



e. Use the **3D Add** command to join the two parts into a single 3D model.

To clean up the drawing, erase the 2D lines and arcs that are no longer needed.

AutoCAD: Filleting the Edges

The final stage is to fillet the top edges of the base.

1. From the **Modify** menu, select **Fillet** (available in dropdown menu). Notice that AutoCAD prompts you to select the model:

   ```
   Command: _fillet-
   Current settings: Mode = TRIM, Radius = 0'-0.0"
   Select first object or [Undo/Polyline/Radius/Trim/Multiple]: (Select the model.)
   ```

2. Next, specify the fillet radius, 0.125":

   ```
   Enter fillet radius <0'-0.0">: 0.125"
   ```

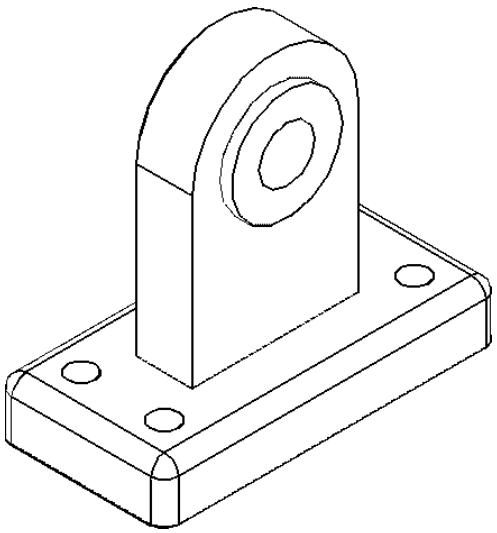
3. Use the **Chain** option to select the entire top edge of the base, as follows:

   ```
   Select an edge or [Chain/Loop/Radius]: c
   Select an edge chain or [Edge/Radius]: (Select the top edge of the base.)
   ```

4. Press **Enter** to apply the fillet.

   ```
   Select an edge chain or [Edge/Radius]: (Press Enter.)
   ```

   Notice that the edges are blended.



AutoCAD: Filleted edges

TurboCAD: Blending/Fillet the Edges

The final stage is to *blend* (a.k.a. fillet) the top edges of the base.

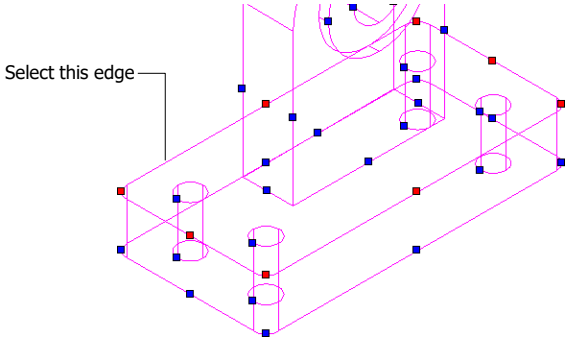
1. From the Tools palette, select **Fillet Edges** — found in the 3D Modify section. Notice that TurboCAD prompts you to select the model:

   `Select a 3D model for edge fillet` ***(Select the model.)***

2. Next, select the edge to be blended:

   `Select an edge to fillet` ***(Select the top edge of the base.)***

   Notice that the red handles show the selected edge(s).



TurboCAD selecting the edges to blend/Fillet

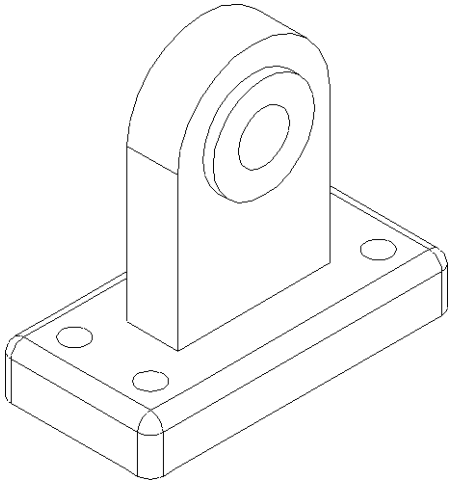
3. Press **Tab** to access the Inspector bar, and then enter the following values:

Start Radius	End Radius	Setback 1	Setback 2
0.125 in	-1 in	0.125 in	0.125 in

TurboCAD: Inspector bar for the Blend command



4. Click the **Finish** flag icon. Notice that the edges are blended.



TurboCAD: Blended edges

Summary

In this lesson, you learned how AutoCAD and TurboCAD produce 3D solid models of the same mechanical part. The methods of extruding, subtracting, filleting, and blending, are quite similar in the two programs.

In the final chapter, you learn how to bring AutoCAD drawings into TurboCAD, and the sorts of translation issues to watch out for.

5

DWG Import & Export

Translating Drawings from AutoCAD

TurboCAD imports and exports drawings made by CAD and other graphics packages. Specifically, TurboCAD imports and exports AutoCAD files with the extensions of DWG (short for "DraWinG") and DXF (the abbreviation for "Drawing interchange Format"). In general:

- You import DWG files when the drawings come from AutoCAD.
- You would use DXF files when the drawings come from another CAD package. (There is no point to importing DXF files from AutoCAD, because that requires the additional step of translating from DWG to DXF.)

Similarly, save TurboCAD drawings in DWG format to be read by AutoCAD. Save TurboCAD drawings in DXF format when they will be read by software other than AutoCAD.

In addition to DWG and DXF, TurboCAD also imports and numerous other file formats, as listed later in this chapter.

In this chapter, we look in detail at how TurboCAD reads and writes DWG files. The process is identical for DXF files. We also look at using the Windows Clipboard and OLE mechanism to place drawings and other documents in TurboCAD.

Accurate DWG Translation

The translation of a drawing is only considered 100% accurate when no objects (or their properties) are erased or changed. Unfortunately, this is never possible between AutoCAD and TurboCAD — or between any other CAD packages you care to name.

Fortunately, TurboCAD does an excellent job of translating drawings; there are errors only among very few objects. The following is a concise summary of objects changed when importing drawings.

Formats Imported and Exported by TurboCAD

TurboCAD imports and exports a far larger number of formats than does AutoCAD:

Note: File formats availability with respect to different variants of TurboCAD. Following file formats are those available in Platinum:

Import (Open)	Export (Save As)	Notes
Vector file formats		
3DS	3DS	Autodesk's 3D Studio
3DV	...	VRML Worlds
3DM	3DM	Rhino format
3MF	3MF	3D printing format
ASAT	ASAT	ACIS®
ASC, PCD, PCG	...	Point Cloud Data
...	BMP	Windows® Bitmap format
CGM	CGM	Computer graphics metafile
DAE	DAE	COLLADA Model
DC, DCD	...	DesignCAD® format
DGN	DGN	Bentley Systems' MicroStation
DWF	DWF	Autodesk's Design WebFormat
DWG	DWG	Autodesk's AutoCADDrawings
DXF	DXF	Autodesk's Drawing Interchange Format
EPS	EPS	Adobe's Encapsulated PostScript
FCD	...	Evolution Computing's FastCAD for DOS
FCW	...	Evolution Computing's FastCAD for Windows
FBX	FBX	3D asset exchange format
GEO	...	VRML Worlds
...	GIF	Graphics Interchange format (w/ Alpha Channel Support)
IFC	IFC	Industry Foundation Classes
IGS	IGS	Initial Graphics Exchange Specification
...	JPG	JPEG
KML, KMZ	...	Google Map Format
OBJ	OBJ	OBJ Drawing
PDF	PDF	Adobe's Portable Document Format
...	PDF	Adobe® 3D Portable Document Format
PLT	PLT	Hewlett-Packard's HPGLPlot
...	PNG	Portable Network Graphic (w/ Alpha Channel Support)
...	PRC	Adobe PRC-Based Portable Document Format
Raster	Raster	(Reference 2D multiple formats)
SAT	SAT	Spatial Technologies' Save As Text (ACIS)
...	SHX	Shape file format
SKP	SKP	Trimble SketchUp® format
STEP	STEP	STEP Drawing
SAB	SAB	ACIS®

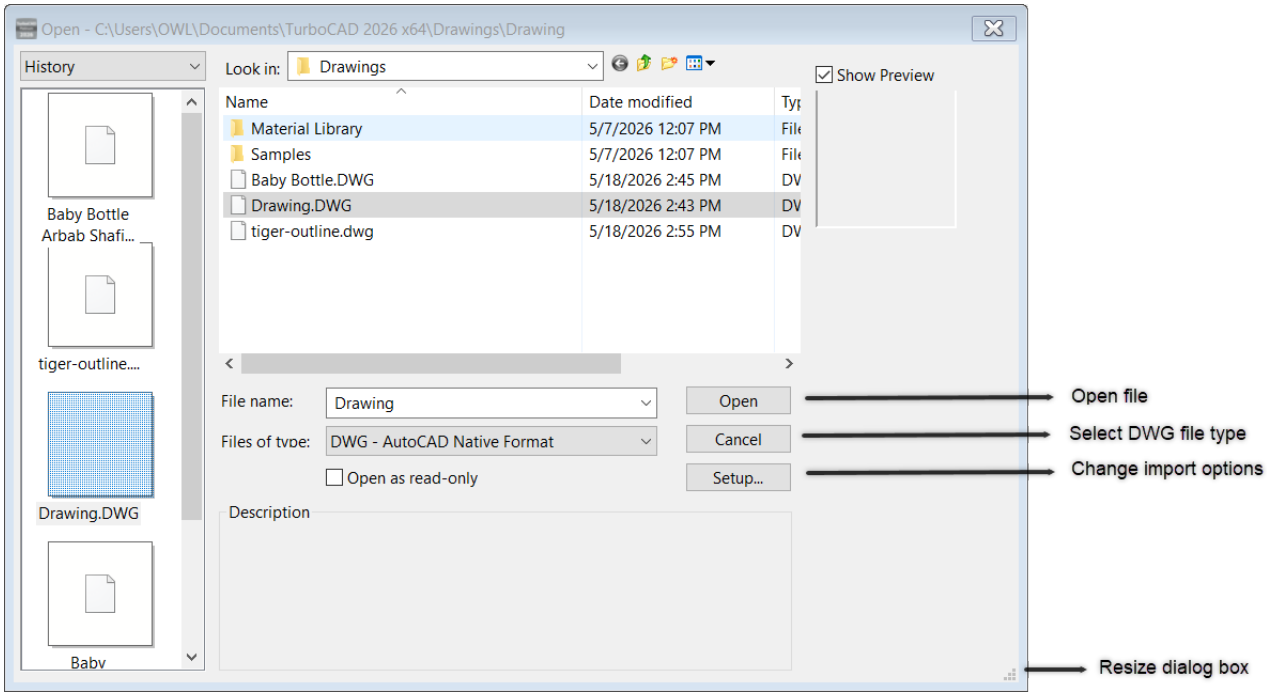
SAT	SAT	ACIS®
STL	STL	Stereolithography format
STP	STP	STEP Drawing
...	SVG	Scalable Vector Graphics
TAP	...	TurboApps® packaged file format
TCT	TCT	TurboCAD® drawing template
TCW	TCW	TurboCAD® native format
...	U3D	Universal 3D
WMF	WMF	Microsoft's Windows Metafile Format
WRL / WRZ	WRL / WRZ	VRML Worlds
XYZ	...	Terrain Data
Raster file formats		
BMF	...	Floorplan for Windows bitmaps
BMP[1]	BMP	Microsoft's Windows bitmap
GIF[1]	GIF	CompuServe's Graphics Interchange Format
JPG[1]	JPG	Joint Photographic Experts Group
PNG[1]	PNG	Portable Network Graphics

*[1] These raster files are placed in drawings with the **Image Manager** and **Insert Picture** commands; TurboCAD can also insert images in TIFF, PCX, icon (ICO), and Targa formats.*

Importing AutoCAD Drawings



To bring AutoCAD drawings into TurboCAD, use TurboCAD's **File | Open** command to display the Open dialog box.e



TurboCAD's Open dialog box

When the dialog box appears, follow these steps:

1. Select "DWG - AutoCAD Native Format" from **Files of Type** droplist. TurboCAD displays all files with the .DWG extension in the current folder.

2. If necessary, change to another folder and drive with the **Look in** droplist. TurboCAD remembers the file extension and folder you select for the next time you use the **Open** command.

3. Select the AutoCAD drawing file by name.

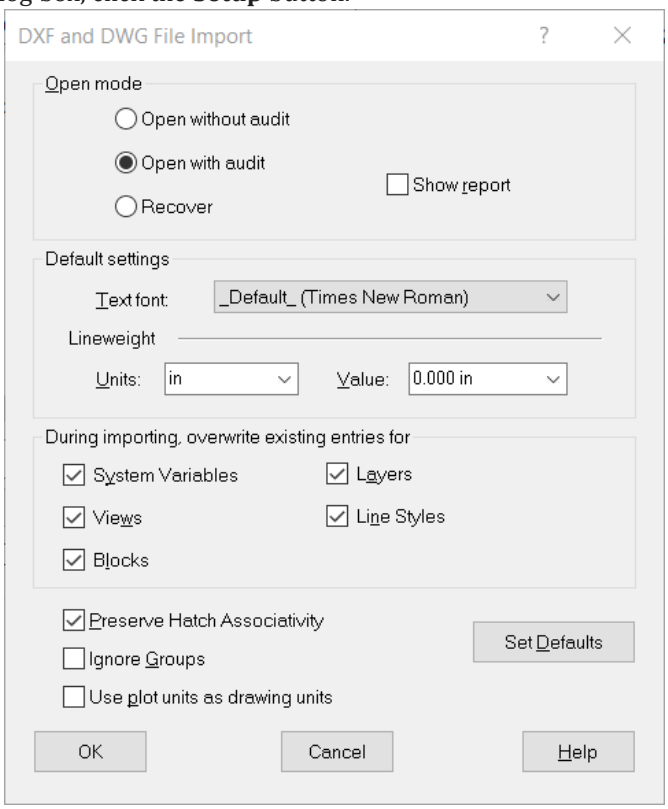
4. *(Optional)* If you want to change the import options, click the **Setup** button. TurboCAD displays the DXF and DWG File Import dialog box. In most cases, options are set for the most accurate translation. You might want to turn on the **Create** option of Model Space Viewports. (More details in the section following.) Click **OK** to dismiss this dialog box.

5. Back in the Open dialog box, click the **Open** button to begin importing the drawing. After a moment or two, TurboCAD displays the AutoCAD drawing.

At this point, you should carefully check the translated drawing to make sure it is accurate. More details appear later in this chapter's "Accurate DWG Translation" section.

DWG File Import Options

The DXF and DWG File Import dialog box is available when importing DWG and DXF files. In the **Open** dialog box, click the **Setup** button.



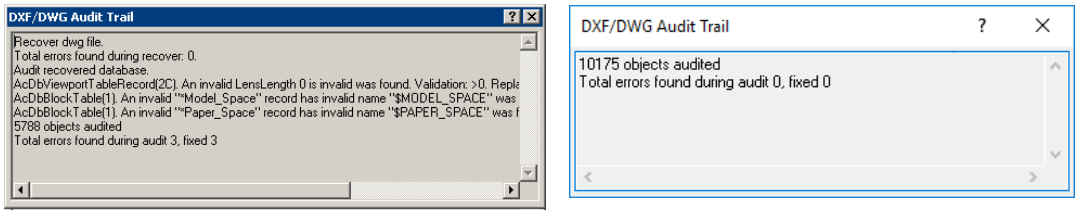
TurboCAD's DWG and DWF File Import dialog box

Open Mode

TurboCAD's DWG/DXF translator can check drawing files for errors:

- **Open without audit** opens drawing files without checking for errors. Not recommended, because when the file has errors, it is not opened by TurboCAD.
- **Open with audit** fixes some errors when possible, and then displays a report if errors are found. *Caution*: Some files may be too corrupt to be repaired, and are not opened by TurboCAD.
- **Recover** corrects more errors than simply auditing.
- **Show report** displays a dialog box listing the errors found, as illustrated below. Close

the dialog box to finish loading the drawing.



TurboCAD's DWG/DWF File Audit Trail dialog box

File Units

AutoCAD does not use units for its drawings; unit indicators, such as " (inches) are converted by AutoCAD from the scientific units it uses internally, and displayed on the run. For this reason, you need to specify the units TurboCAD should use for the drawing. For example, if the AutoCAD drawing is in Architectural units (feet and inches), select “ft” from the droplist.

If you are not sure, select the **By Drawing** option. If the resulting drawing has the wrong scale, then re-import it using one of the units listed below.

Unit	Meaning	Equivalency
English (perial) units		
mil	mils	1 inch = 0.01 mils
in	inches	
"	inches	
'	feet	12 inches = 1 foot
ft	feet	
yd	yards	3 feet = 1 yard
mi	miles	5,280 feet = 1 mile
Metric (international) units		
u	micron	1 micron = 0.001 mm
A	Angstrom	1 angstrom = 0.00000001 mm
mm	millimeter	1 mm = 2.54 inches
cm	centimeter	10 mm = 1 cm
m	meter	100 cm = 1 m
km	kilometer	1,000 m = 1 km
Typesetting Units		
pt	point	72.72 points = 1 inch
pica	pica	6 picas = 1 inch

Default Text Font

If the drawing contains a font that your computer does not have, chose a replacement font from the Default Text Font droplist. You can choose any TrueType (TTF) or AutoCAD (SHX) font on your computer. The default is Times New Roman.

Overwrite Entries

The following options are meant for use when you insert an AutoCAD drawing into an existing TurboCAD drawing with the ***Insert / File*** *command.*

When AutoCAD and TurboCAD share the same settings, then this section overrides TurboCAD's settings with those of AutoCAD.

- **System variables** are where AutoCAD stores many settings. Some of these are similar to those in TurboCAD, such as units, numerical display format, and angle format. (You cannot directly change TurboCAD's system variables, as you can in AutoCAD with the **SetVar** command.)
- **Views**, **Blocks**, **Layers**, and **Line Styles** are identified by name in AutoCAD drawings, as well as in TurboCAD. Turn on these options to have TurboCAD's named entities overwritten with the values of AutoCAD's.

For example, if the existing TurboCAD drawing has a layer named "Airport" and is colored green, inserting an AutoCAD drawing with an "Airport" layer colored red overrides the TurboCAD setting, changing the layer color from green to red.

Set Defaults

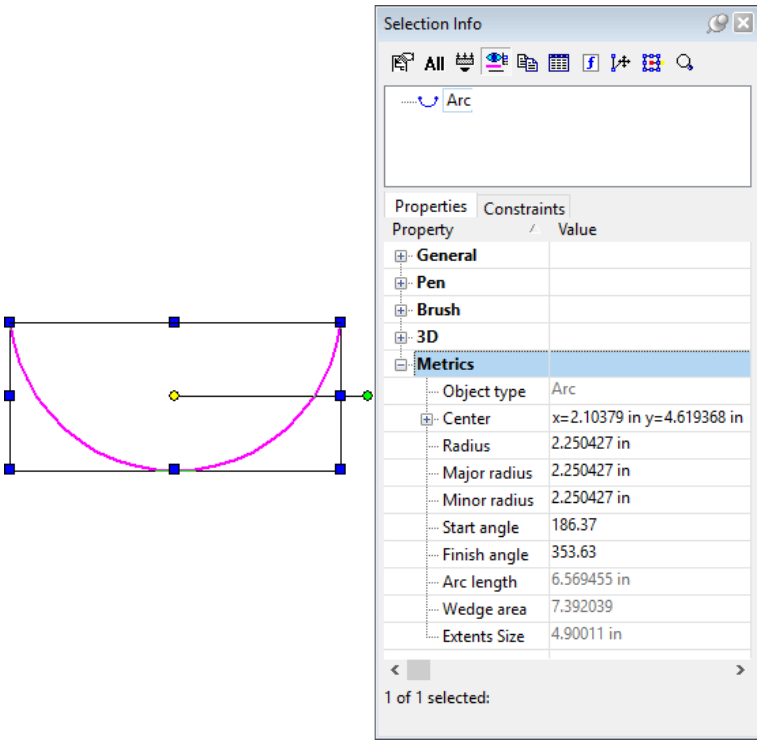
After you've made changes to this dialog box, you can quickly return the settings to their original values by clicking the **Set Defaults** button.

Post-translation Checking

If, after translation to TurboCAD, you find objects in the drawing that puzzle you, use the **Select** cursor and the Selection Info palette as your investigative tools.



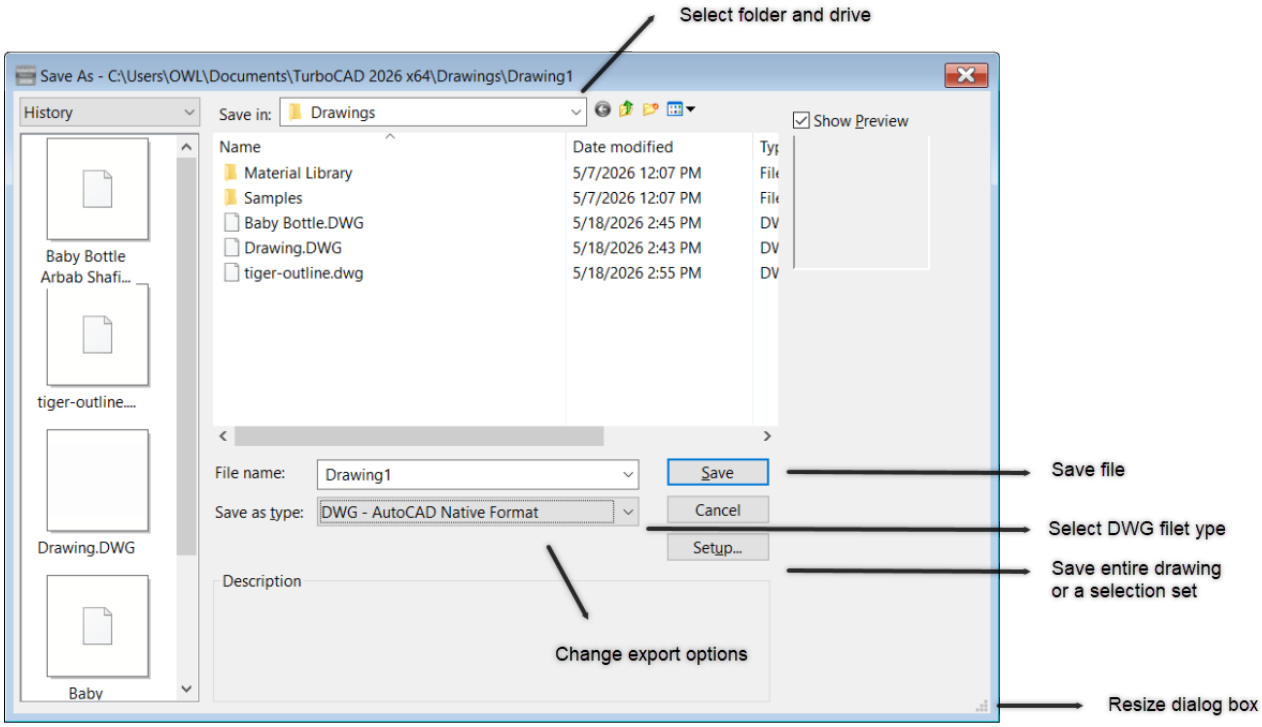
Simply click an object – or window a group of objects – and the Selection Info palette (equivalent to AutoCAD's Properties palette) tells you everything TurboCAD knows about the object(s).



Examining unknown objects with the Selection Info palette

Exporting in DWG Format

AutoCAD cannot read drawings created by TurboCAD. To bring TurboCAD drawings into AutoCAD, you must translate them first from within TurboCAD by saving drawings in DWG format. When you select **Save As** from the **File** menu, TurboCAD displays the Save As dialog box.



TurboCAD's Save As dialog box

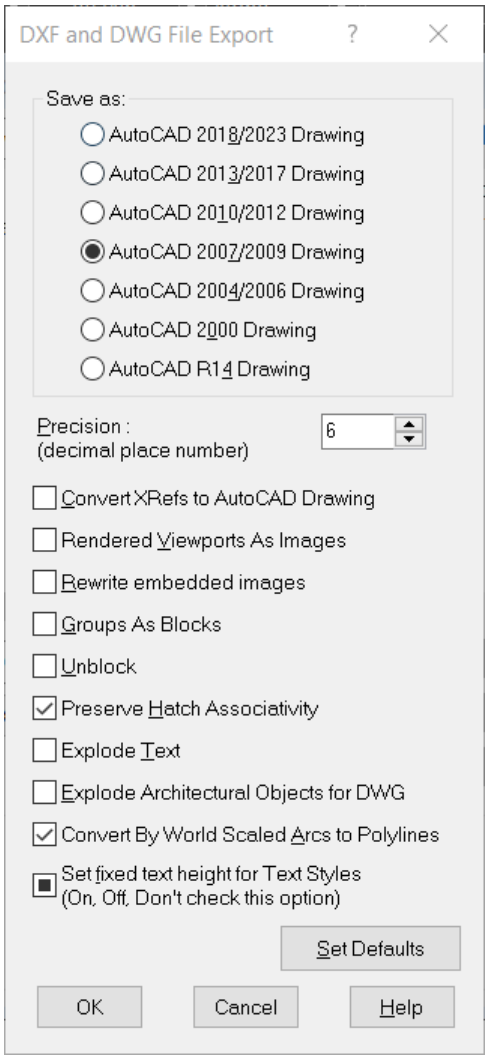
1. In the **Save as type** droplist, change the file type to "DWG - AutoCAD Native Format." (If another CAD package needs to read the TurboCAD drawing, save it in DXF format by selecting "DXF - Drawing eXchange Format" instead.)
2. If necessary, change to another folder or drive using the **Save in** droplist.
3. Although it is not usually necessary, you can change the file name in the **File name** text box.
4. Click the **Setup** button to display the DWG File Export dialog box. (The setup for DXF files is identical). Choose **OK** to exit setup.
5. Click **Save** to save the drawing in DWG (or DXF) format.

TIPS: *TurboCAD allows you to save the entire drawing or just a selection set of objects. When you choose **Selection**, TurboCAD prompts you to select the objects to export after you click **Save**.*

TurboCAD does not save a preview image when saving to DWG and DXF formats.

DWG File Export Options

The DXF and DWG File Export dialog box provides the following options:



TurboCAD's DWG and DXF File Export dialog box

Save As

With some exceptions, older releases of AutoCAD cannot read drawings created by newer releases, and so TurboCAD lets you save the drawing in the many release groupings. Following are few:

- **AutoCAD 2018 Drawing** saves drawings for AutoCAD up to version 2026
- **AutoCAD 2013/2017 Drawing** saves drawings for AutoCAD 2013, 2017
- **AutoCAD R14 Drawing** saves drawings for AutoCAD Release 14.

Caution: Older releases do not understand objects unique to newer releases and so may erase or change objects in order to be compatible.

Other Options

Precision applies only to DXF files, which can be in the range from 1 to 16 decimal places.

TurboCAD for AutoCAD

Save As: Select the AutoCAD version.

Precision: The number of decimal places for numerical values of the exported file.

Convert Xrefs to AutoCAD Drawing: Converts attached Xrefs into DWG files.

Rendered Viewports as Images: All viewports that have renderings will be converted to images, to preserve the appearance of the renderings.

Rewrite embedded images: Relevant for files that contain image objects. Choose whether to replace images.

Groups as Blocks: Displays objects included in groups as blocks when imported into AutoCAD.

Unblock: Explodes blocks into its constituent objects.

Preserve Hatch Associativity: Retains hatch association in the resulting DWG file.

Explode Text: Explodes any text into its constituent parts.

Explode Architectural Objects for DWG: Explodes any TurboCAD Architectural objects into data comprehensible to AutoCAD.

Convert By World Scaled Arcs into Polylines: All arc using the By World setting for line scaling will be converted into arc segments of polylines with a supported AutoCAD scaling method.

Set Fixed Height for Text Styles: (On, Off, don't check this option): Locks Height for Text Styles.

DWG and DXF file formats do not support unit types, only a designation as Imperial or Metric. All Imperial units are converted to inches in DWG, all Metric units to mm.

Embedding and Linking Objects

There is an alternative to translating drawings between TurboCAD and AutoCAD, and that's not to translate at all. You can place wireframe representations of drawings in the other CAD package, and they will look exactly correct, because they are not translated.

The drawings are embedded as objects, with a link back to the originating application using a Windows technology named OLE, short for "object linking and embedding."

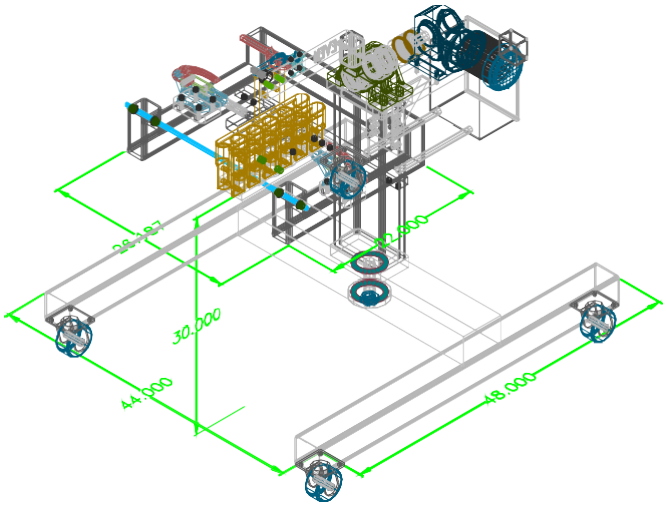
The method described below works only when both CAD programs are installed on the same computer. You can use this method with other software as well, such as placing CAD drawings in Word documents, or Visio diagrams in TurboCAD.

Using OLE

Let's see how object linking and embedding works:

1. Open a complex drawing in AutoCAD, such as illustrated below. Ensure the drawing is in Model tab.

   If necessary, use the **-SHADEMODE** command to change the shading to **2dwireframe**, because beautifully rendered models don't work through OLE. Even hidden-line removal is reduced to regular wireframe mode.



AutoCAD's drawing

2. Press **Ctrl+A** to select all objects in the drawing.

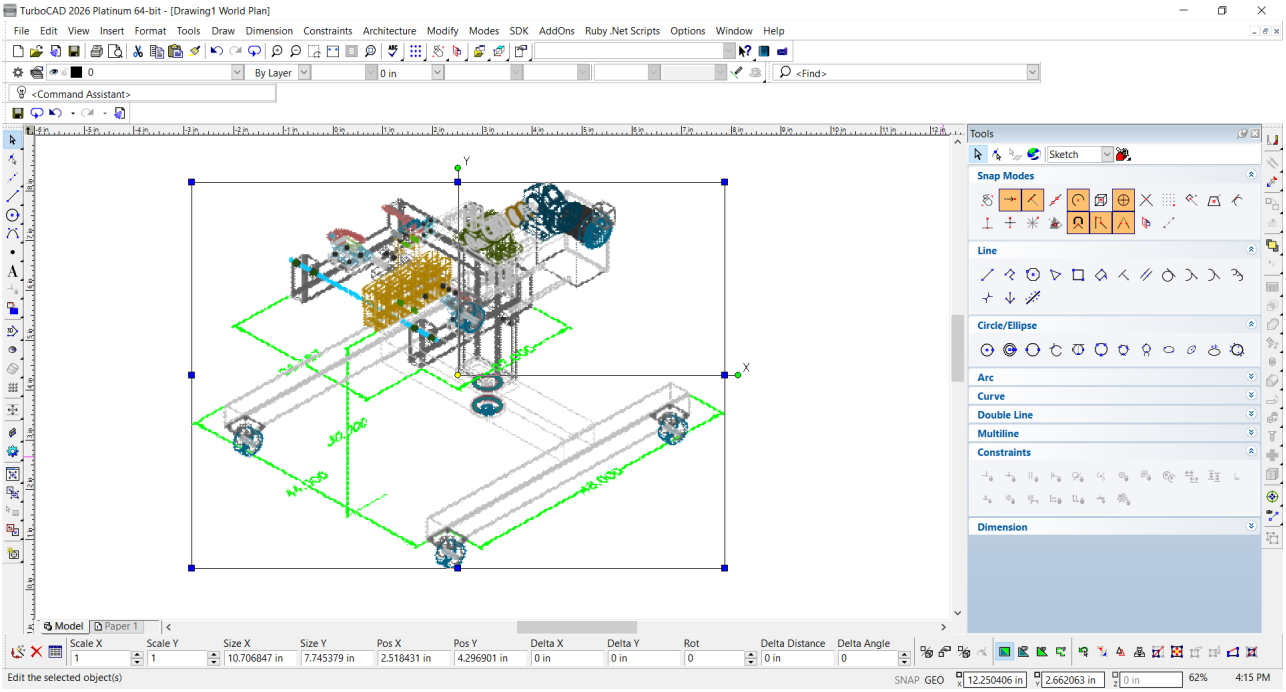


Press **Ctrl+C**, the universal keyboard shortcut for copying objects to the Windows Clipboard. (Ignore messages about objects on locked layers.)

3. Switch toTurboCAD.



4. Press **Ctrl+V**. This pastes the AutoCAD drawing from the Clipboard into the drawing. Notice that the drawing appears in TurboCAD.

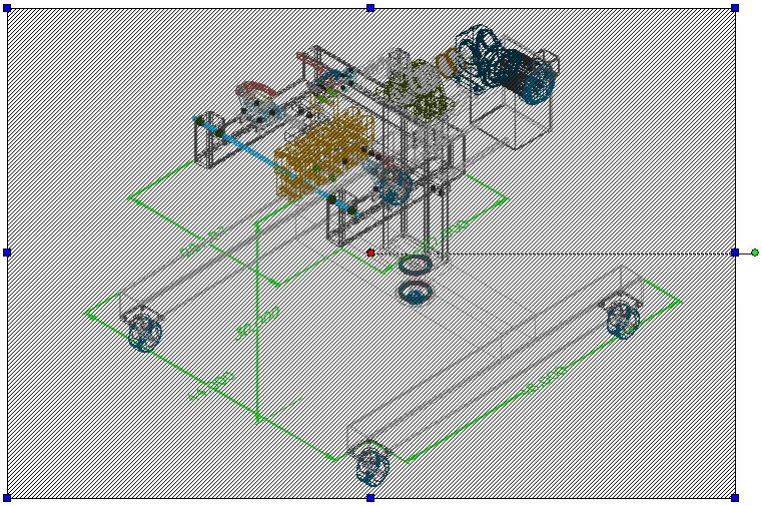


TurboCAD drawing pasted into TurboCAD

5. You can use handle editing to move and resize the AutoCAD drawing — technically, an object, and not a drawing. Curiously, you cannot rotate it or perform any other editing operation, except for erase.

6. To edit the object, you double-click it. This causes Windows to open the original drawing in AutoCAD.

   Notice that the object is now cross-hatched in TurboCAD; this means that it is open in another application and is unavailable until the other application (AutoCAD) exits the drawing.



AutoCAD drawing "busy" in TurboCAD

7. Press **Alt+Tab** to switch to AutoCAD. Notice that the drawing appears in a new window, and that it can be fully edited.

In AutoCAD's **File** menu, a new item appears: **Update TurboCAD**. As you edit the drawing, you can periodically update the drawing back in TurboCAD.

For example, change the colors to blue, and then select **File | Update TurboCAD**. Notice that the drawing's lines are now all blue in TurboCAD. The only changes that won't appear are those that involve shading, rendering, and hidden-line removal.

8. When you are finished editing, close the drawing in AutoCAD by selecting **Close** from the **File** menu. AutoCAD asks if you wish to have the changes updated in TurboCAD.

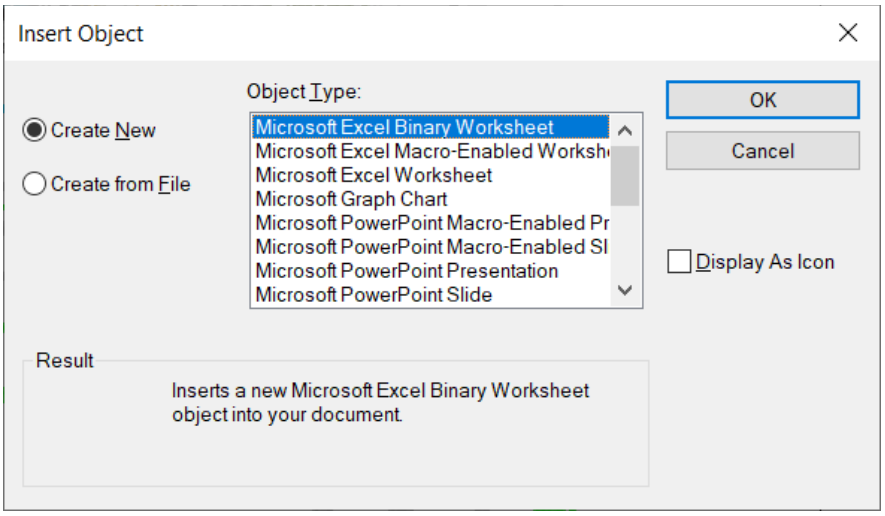
Prompt message will be: Update TurboCAD 2023 Platinum 2023 64-bit before closing the object.

9. Answer **Yes** or **No**, as you wish.

Inserting Objects

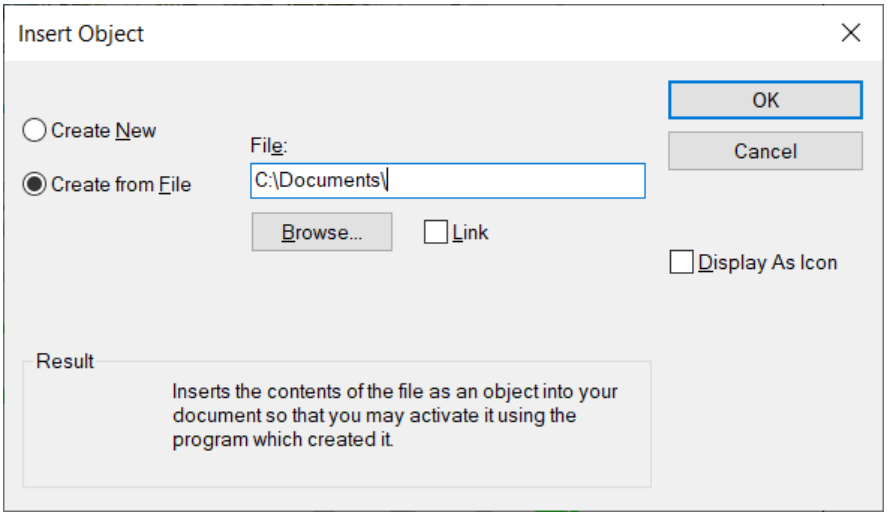


As an alternative to Ctrl+C/Ctrl+V method, you can choose **Object** from the **Insert** menu. TurboCAD displays the Insert Object dialog box, which lists all the kinds of documents that Windows knows about on your computer. (Strictly speaking, this dialog box is generated by Windows, and looks identical on any software that has the **Object | Insert** command.)



Windows Insert Object dialog box

Choose **Create from File**, and then click the **Browse** button. This lets you select a file name, which Windows then places into the TurboCAD drawing.



Create from File option

The files you select are not limited to AutoCAD drawings; they can be just anything: Acrobat PDF files, movie clips, Word documents, Excel spreadsheets, digital photographs, and so on. Some display as you expect; others show an icon instead. In either case, you can double-clickthe object/icon to edit it in its originatingapplication.

The **Link** option creates a live link. When the drawing or document is changed in the original application, the changes are made to the copy in TurboCAD automatically.

If you need to change the linked objects, use TurboCAD's **Edit | Links** command. This displays a dialog box, again common to all Windows applications.

You can set each linked object so that the update happens automatically, or so that the update must be done manually.

Following are the contents of Edit Links window.

Links box: contains all the links in the active drawing. You can highlight multiple links by pressing Shift or Ctrl.

Update Option: Sets whether the selected links are updated automatically or manually.

Update Now: Updates the current link. This is how to update sink that are set to update manually.

Open Source: Opens the file in its source application. This also enables you to compare the linked object to the source file.

Change Source: Enables you to choose a new source for the object. Once the object is linked to the new file, its contents will change to match the new file.

Insert PDF as Underlay: The PDF Underlay tool allows you to insert a page of a PDF file into your drawing to use as a reference or tracing layer. To use properly as a tracing layer, the file must be a vector PDF, rather than a bit-mapped PDF.

Summary

Despite some difficulties, drawing translation works out well in most cases. With each new released version, TurboCAD is made more and more compatible with all new features in latest AutoCAD versions. Your typical 2D floorplan is going to translate smoothly. Lines, circles, colors, line styles — they all convert effortlessly.

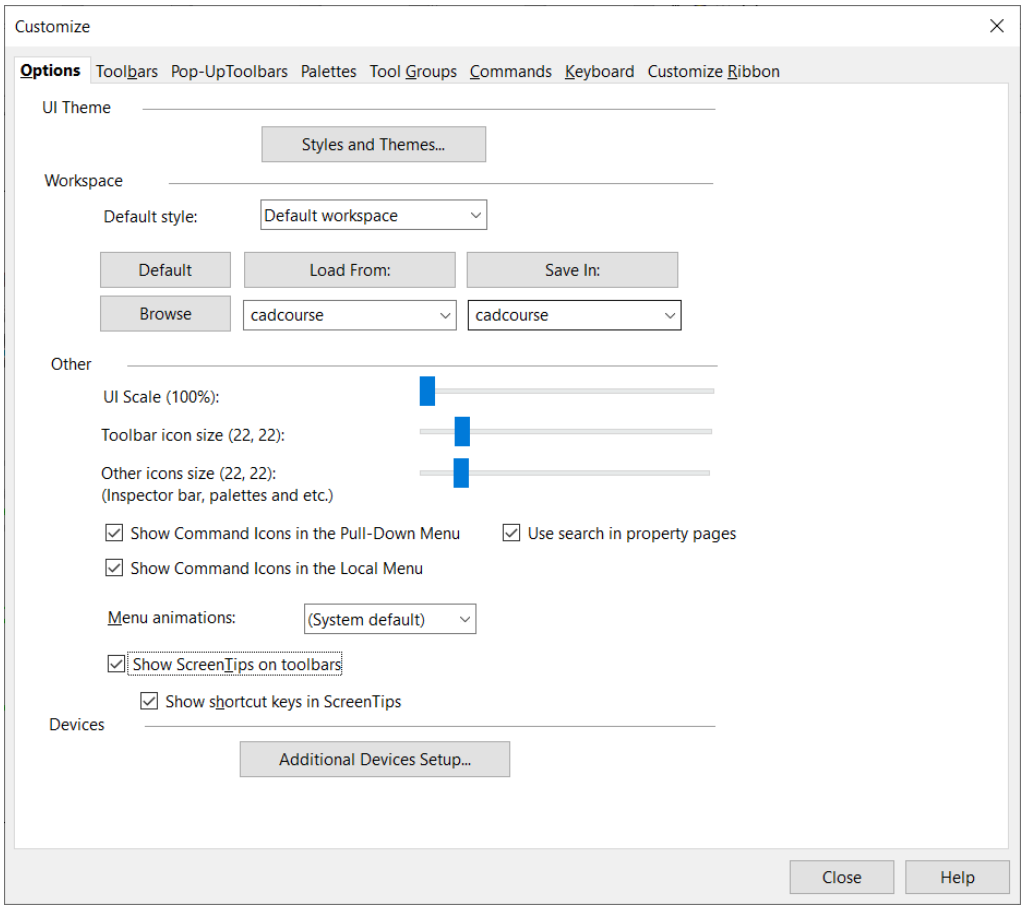
6

Advanced TurboCAD Tools

TurboCAD: *Customizing the UI*

You can customize toolbars, menus, keyboard, and your IntelliMouse (if you use one). You can invoke the Customize window at any time by right-clicking in any blank toolbar area.

Note: If you prefer an older interface for customizing toolbars and menus, select UI Themes, and select the classic interface.



TurboCAD Customize Window (Ribbon UI - Classic Theme)

Customize Commands

The Commands page enables you to customize toolbars by adding and removing icons. Display the toolbar you want to change, and open the Commands page. Locate the commands you want to add to the toolbar, and drag it from the Commands list to the toolbar itself.

Search Option:

You can also search a command. Type a command in "find" option to search, the first suitable item is highlighted. To browse through the commands, buttons next to the input field highlight and can be used to show the previous or next suitable item. Successful search options are added to the list and saved between sessions.

TurboCAD: Customize Keyboard

The Keyboard page enables you to view current keyboard shortcuts (hotkeys), to modify them, or to create new ones.

Whenever you make changes to the UI settings it is best to save the Workspace file (under the Options tab accessed via Tools | Customize) in order to accurately retain your changes.

Tip: Select Help / Keyboard for a list of all shortcuts.

Multiple Keyboard Shortcuts

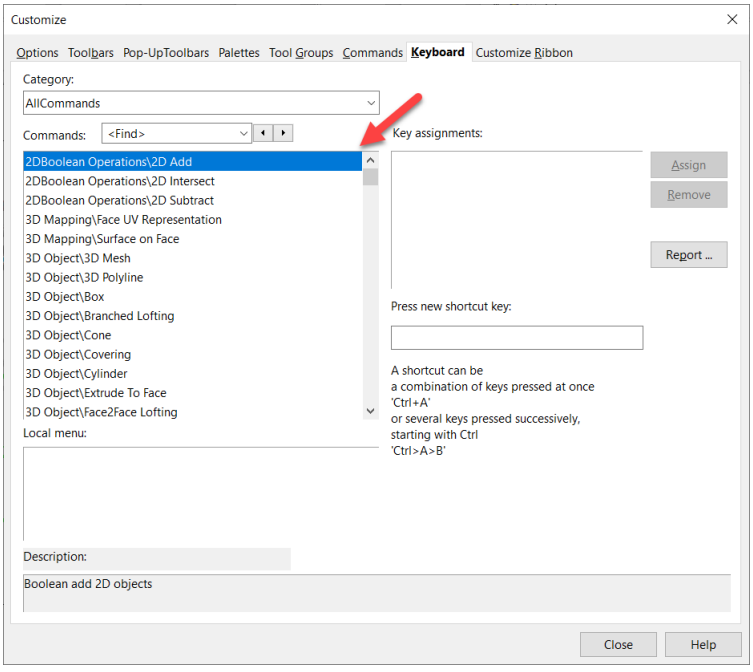
You can create two/three (or more) key customizable keyboard shortcuts. For example, pressing "aa" might invoke the double point arc if you have assigned this shortcut to it.

Another usefulness of this feature is that the user doesn't have to do a key combination which is difficult when only using one hand. The ideal case would be that the user uses left hand to generate the keyboard shortcut and the right-hand to control the mouse.

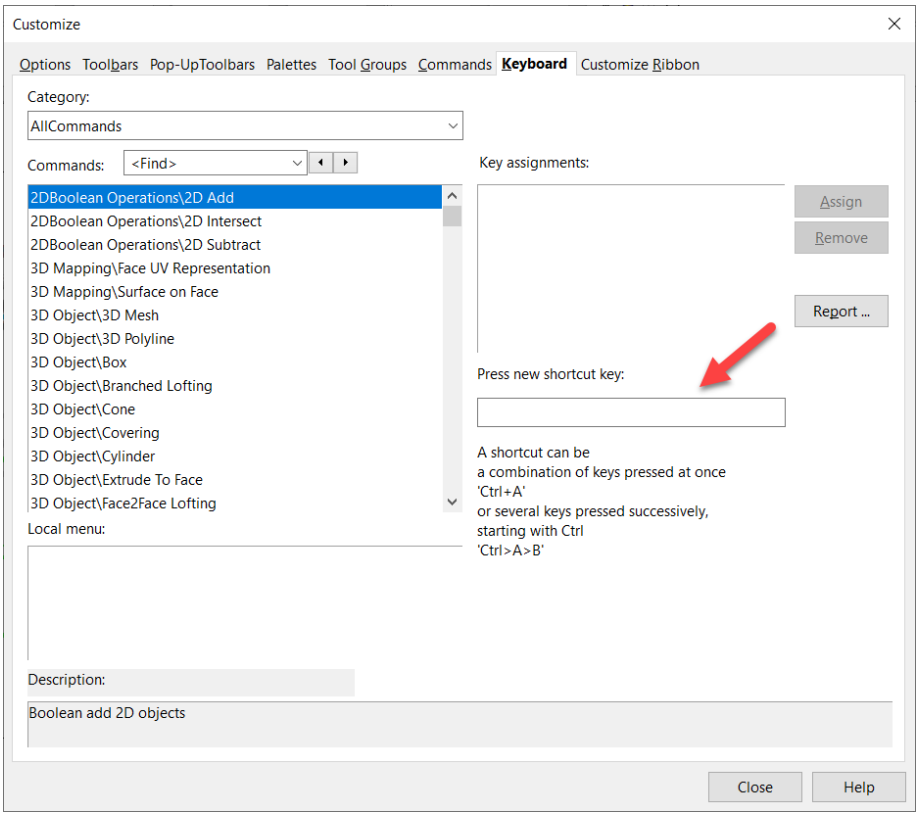
Example:

To assign a multiple keys shortcut to any command:

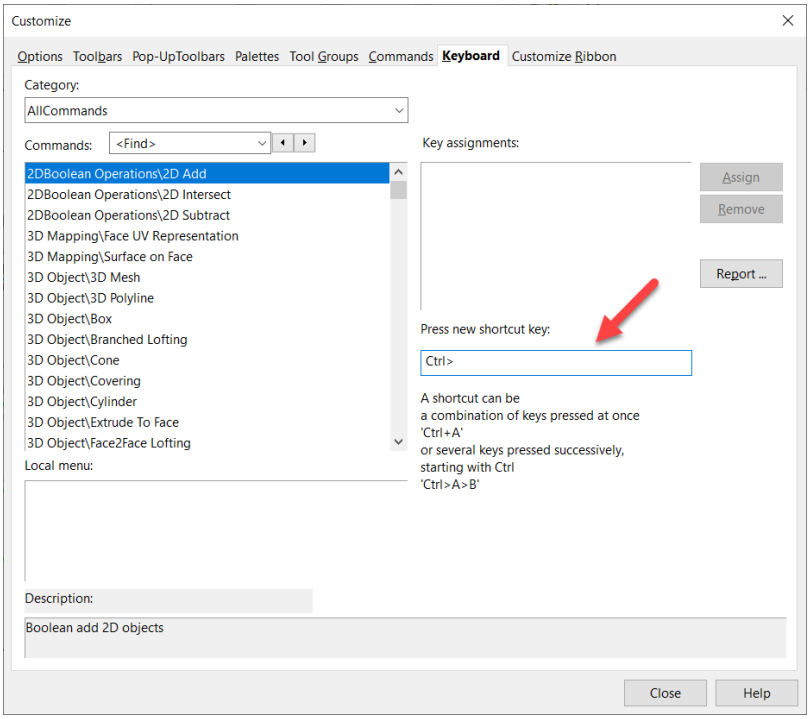
1. Select the command in customize keyboard.



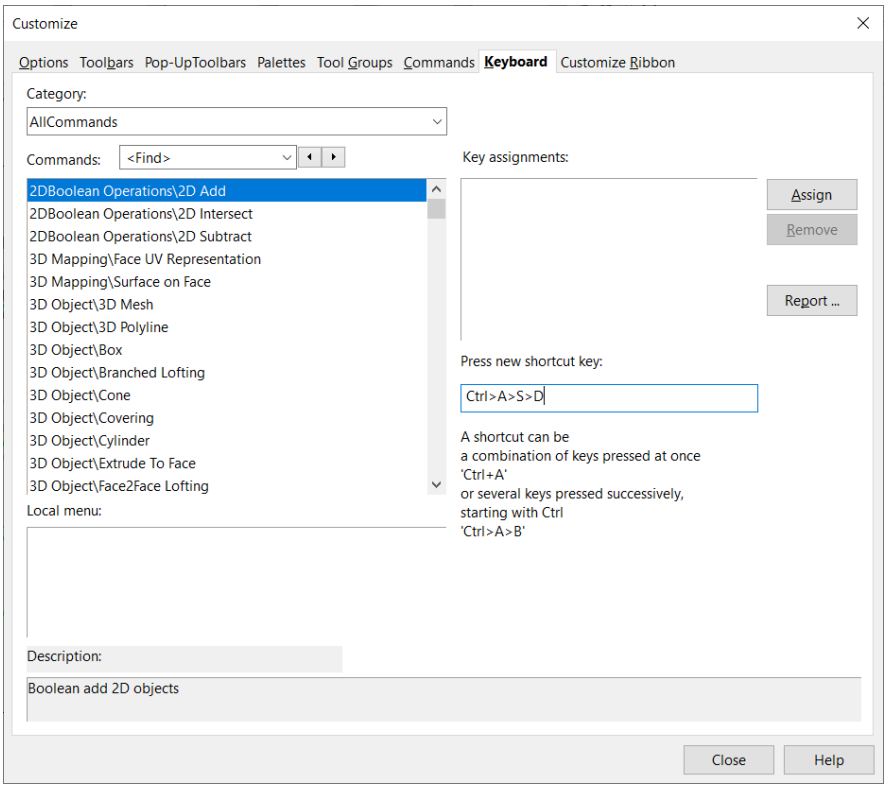
2. Select shortcut key box.



2. Press and release Control key. You will see a "Ctrl>" sign.



3. Now press multiple keys.



4. Press Assign button and Close.
   To invoke the command using shortcut
   1. Press and release control key.
   2. Press multiple keys in a sequence included in the assigned shortcut.

   Note: You can only use CTRL key to assign multiple keys shortcuts to commands. If you want to assign a single key shortcut with CTRL, keep CTRL pressed and select another key.

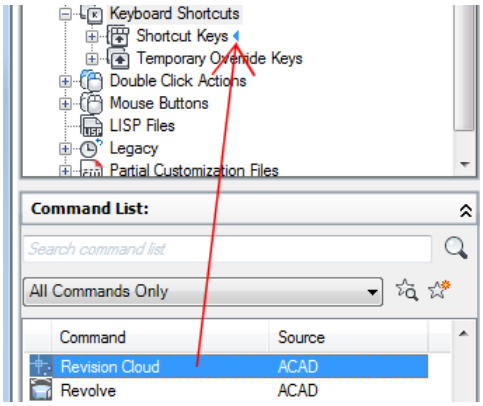
Report:
You can see a detailed report of keyboard shortcuts by pressing "Report" on customize keyboard. A webpage will open in your default internet browser.

AutoCAD: Shortcut Keys Customization

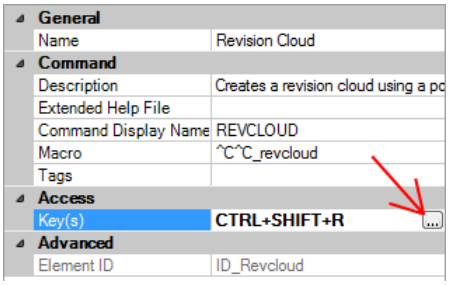
Click Manage tab | Customization panel | User Interface.

Create or Modify a Shortcut key

1. In the Customize tab, Customizations In <file name> pane, click the plus sign (+) next to the Keyboard Shortcuts node to expand it.
2. Click the plus sign (+) next to the Shortcut Keys node to expand it.
3. Do one of the following:
   - To create a shortcut key, in the Command List pane, drag a command to the Shortcut Keys node in the Customizations In <file name> pane.



   - To modify a shortcut key, select a shortcut key from under the Shortcut Keys node.
4. In the Properties pane, click in the Key(s) box, and then click the [...] button to open the Shortcut Keys dialog box.



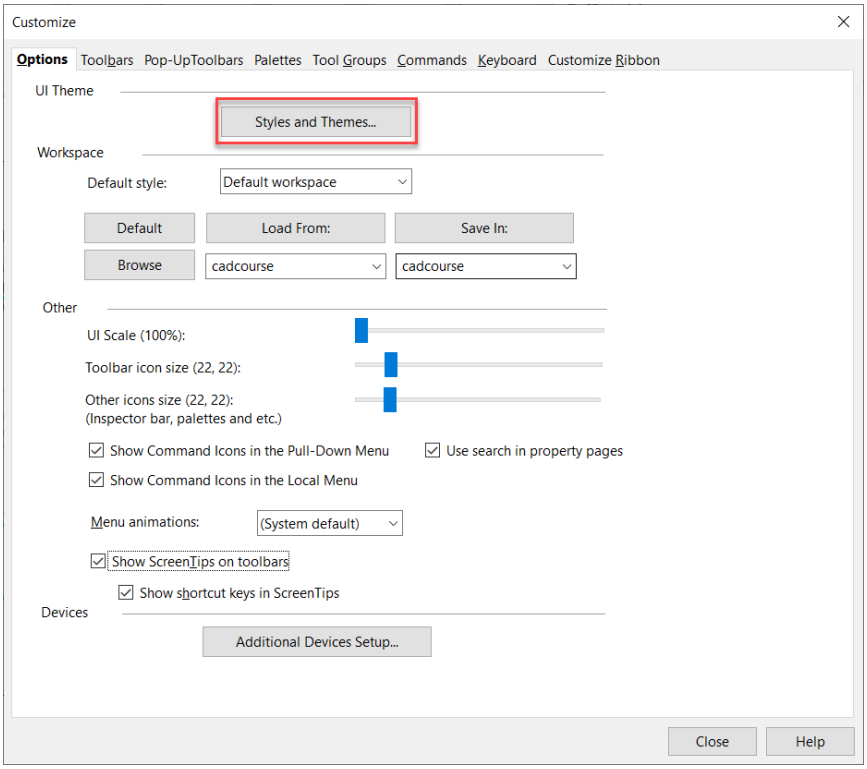
5. In the Shortcut Keys dialog box, hold down the modifier key Ctrl with a combination of Shift, Alt, or Shift and Alt along with the desired letter, number, function, or virtual key such as F1 or Insert.

   Valid modifier and key combinations include
   - Function (Fn) keys containing no modifiers
   - Number Pad (NUMPADn) keys containing no modifiers

- Ctrl+letter, Ctrl+number, Ctrl+function, Ctrl+virtual key

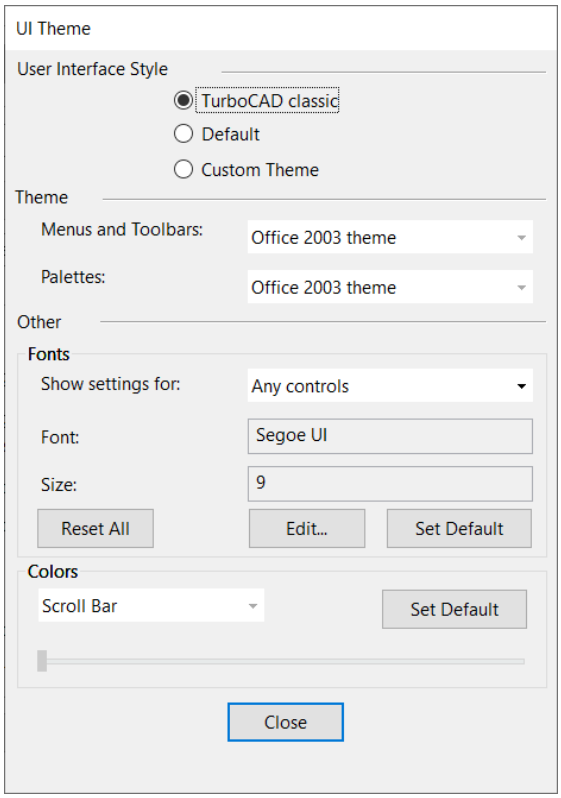
TurboCAD: Customize Options

Enables you to manage the configuration (Config) files, control the look of the TurboCAD window, and specify external devices.



Styles and Themes

These functions allow you to select from several different Workspace/UI looks, or "skins".



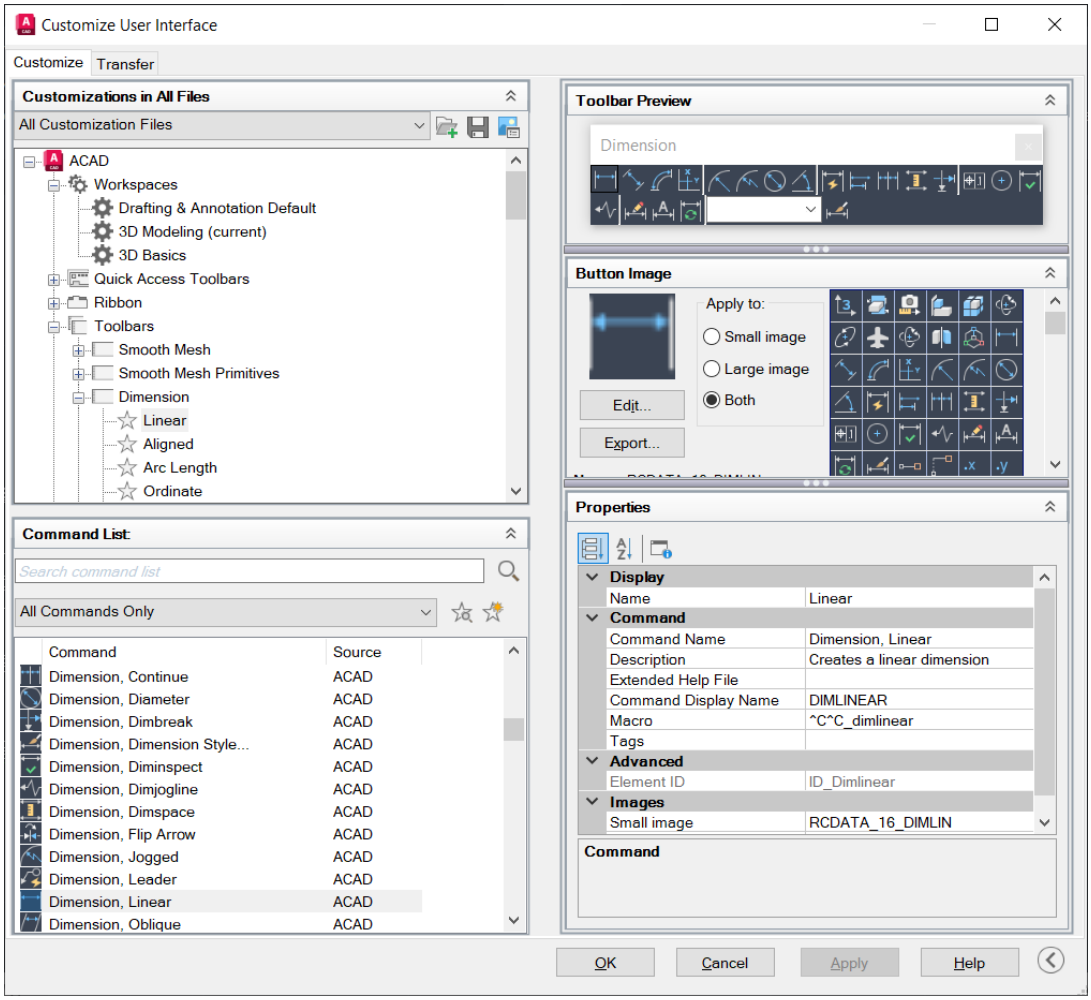
Default: uses the "new" (TC28) dark interface with new icons.
TurboCAD Classic: uses a light background with TurboCAD's classic icons.
Custom Theme: uses the new icon set, but allows you to choose a skin to customize the look and feel of the interface.
Other: Various settings for how icons and tool-tips are displayed.

Fonts: Select the font and its size for the selected control.

Colors: Set color for the selected control.

AutoCAD: Customize UI

This tool is available at Tools > Customize > Interface



Provides an interface for customizing workspaces, toolbars, menus, ribbon panels, shortcut menus, and other user interface elements.

TurboCAD: Program Setup

Default UI Menu: Options/Program Setup

Auto-Naming

Controls how groups, blocks, and symbols are named.

Prompt for name: You will receive a prompt each time a new item is created.
Generate names: Names will be automatically assigned.
Prefix: If names are automatically generated, you can enter a string that appears before the item name. The "@" character is a placeholder for the automatic number.
Insert blocks when creating: Each block will be inserted into the drawing once it is created.
Prompt for library location: If not checked, all saved symbols will be stored in the default folder.

Note: If you turn off both the **Prompt for name** and **Generate names** options, names will not be assigned to groups. Names will be assigned to blocks and symbols, however.

Color Palette

Enables you to add, modify, or delete colors from the TurboCAD palette.

To add a new color, type the name and click **New**. You will then be prompted to select a color from the color wheel.
You can modify a color by selecting it from the list and clicking **Modify**. Delete a color by selecting it and clicking **Delete**.
When you have the colors you want; they can be displayed for easy selection in the Colors and Brushes palette.

The Color Wheel

You can set colors by RGB values, or by Hue, Saturation, and Value numbers.
Red, Green, Blue: Sets the amount of each color in the light. Values can be a maximum of 255.
Hue: The color value, where 0 is red, 60 is yellow, 120 is green, 180 is cyan, 200 is magenta, and 240 is blue. If you change the hue, the values for red, green, and blue will be changed to match.
Sat: The saturation level for the color (amount of color), up to a maximum of 240.
Lum: The luminosity (brightness) of the color.

Constraints

Settings and controls for geometric and dimensional constraints.
Auto Add Constraints: If checked, the **Auto Add Constraints** tool is enabled. Any constrainable geometry you create while this is active will have constraint automatically assigned.
Update Connected Objects while Dragging: Dynamically updates the position, shape and size of constrained objects as you drag them within the **Edit** tool.

Connected Objects Editing:
DCM Default: Changes to any part of a set of constrained objects can affect all objects equally.

Use Priority Level: Constraint changes to any part of a set of objects will affect that part first, with minimal changes to the remaining objects.
Incremental Evaluation: Constraint changes are constantly checked and changes are being made. If disabled, the results will be checked only after changes are made. For large scale changes, this option should be enabled.

Constraint Markers and Dimensions

Place marker to the same layer with object: Constraint markers are placed on the same layer as the object that is constrained. Otherwise they are placed on their own layer.
Show variable name in dimension text: Displays the variable name in parentheses after the dimension value.
Printing: Choose whether constraint markers and constrained dimensions will be included when printing.

Desktop

Controls what appears on the screen.

Workspace: Controls the display of the working area
Status Bar: Displays or hides the Status Bar.
Scroll bars: Displays or hides the scroll bars.
Rulers: Displays or hides the rulers.

Inspector bar

Inspector Bar: Displays or hides the Inspector Bar.
Classic: Set Inspector bar fields and labels to be in the classic side by side orientation,
Compact: Set Inspector bar fields and labels to be in the compact over/under orientation,
Field width: Specify the field width.

Desktop default scheme

Desktop Scheme: Customizes the color of paper and background, and floating palettes.
Click **Modify** to change an element's color.

Tip: You can also click the different elements in the preview window.

Show margins: Displays page (printer) margins.
Show background: If checked, the background behind the paper will be displayed. If unchecked, the background will be white.
Apply to opened drawings: Apply changes in the scheme to all opened drawings.
Apply to active drawing: Apply changes to the currently active drawing.

Tip: You can switch off the paper until you're ready to print, then use the **Page Setup** to place your drawing on the page.

OpenGL Hardware Acceleration: Available if your video card is equipped with an accelerator, take this characteristic into account while performing a render.
Render new window: When you render your drawing, the render view will open in a new drawing window.

Background Render: With this option, TurboCAD tools work faster in Render mode; renderingwill "interfere" less with other tools. This setting can only be applied when working with one window.

Multithreading quality render

Use automatic number of threads: If checked the system automatically use the full number of available threads.
Threads: Specify the number of threads to be used for rendering if **Use automatic number of threads** is off.
Multithreading LightWorks: If checked multithreading will be used for LightWorks.

File Locations

Enables you to specify the folders where TurboCAD places several types of program files.

Templates (.tct)* - Files in which you save settings as well as block libraries, symbol libraries, and scripts, that you can use to start a new drawing.
Backups (.bak)* - Backups that are created if you check **Make Backup Copies** option in the **General** page.
AutoSave Files (.asv)* - Files that are saved automatically, if this option is turned on in the **General** page. These files allow you to restore work after a system crash.
Symbol Library Files (.slw)* - Sets of related symbols, saved to one common *.slw file, or a set of drawing files saved to a specified folder.
Drawings (.tcw)* - Drawings and models you create in TurboCAD.
Publish to HTML Files - Files created using **File / Publish to HTML**.
Shapes (.shx):* Shape files (*.shx) used in the creating of line styles. (This is not the same as *.shx font files.)

Tip: If you have two disk drives (or access to a network drive), it is wise to have your backup files automatically saved to a second drive.

General Setup

Sets file opening and saving parameters, as well as user information
New document from: Select how files will be opened when using **File / New**.
Default template for new files: If no template is specified when creating a file, a default template will be used. This option sets the default template.
Prompt for template name: You will be prompted for a template each time you start a new file. If you want to use the default template without being prompted, clear this box.
Open all files as read-only: If checked, files cannot be edited.

Make backup copies: If checked, select **On First Save** or **On Every Save** to determine how often backups will be made. Backup files have the extension .bak. By default, backup files are saved in the same folder as your primary drawing files, but you can change this on the File Locations page.
Save drawings every: If checked, autosave copies (.asv files) will be made at the specified intervals. After a system crash, TurboCAD will open the autosave version of your drawing as soon as you launch TurboCAD.
Prompt for summary information: The **Summary Information** window enables you to record user information connected to your drawing. If checked, this window will be displayed the first

time that you save any drawing to disk, and whenever you use Save As to save a new copy of the file.
Healing of SAT insertions: For imported files, heals gaps between faces and holes within faces.
Save desktop on exit: If checked, the desktop settings (toolbars, etc.) will be saved when you exit the program.
Load at startup: You can choose to load the last working file and the last window layout each time you start TurboCAD.
Help Settings: Specifies how help will be accessed.
User Information: Basic user information

Preferences

General program options, such as cursor shape, axis color, zoom factor, and display of coordinate system icons.

Coordinate system icon:

You can choose to display the User CS and/or World CS symbol/s.

For both, you can set the color of the axes and size of the icon.

Cursor Shape: Select one of the three shapes provided.

Show crosshairs: Overrides the normal cursor, displaying a screen wide horizontal and vertical line intersecting at the cursor position.

Show Isometric: Overrides the normal cursor, displaying all three axes.

Show Snap Aperture: Displays a circular area around the cursor whose radius is the Aperture Size. Aperture is used while snapping, to detect objects.

Aperture size: Specify the radius of the aperture in pixels.

Undo/Redo

Undo/Redo depth: Sets the number of operations stored in the Undo buffer.

Include Model / Paper space switch in Undo buffer: If checked, switching between Model and Paper spaces will be included in the Undo buffer.

Selection

Use Choice if several entities in aperture while selection: If more than one object lies within the aperture area during a selection, a small window appears from which you can select the desired object.

Visual Effect Settings: Settings for the various "highlighting" methods. See Selection Visual Effects Settings.

Toolbars and Menu

Open the **Customize** window to the **Toolbars** page to modify toolbars.

Warning Dialogs

Controls the display of TurboCAD warning messages. If you received a warning message with the "Don't show this message again" checkbox, the warning message will appear in this table. Use the **Show** column to show or hide messages. UI enhancements of the Warning dialogs page
Show/hide list columns.
Right click on the header of the list to open the local menu. This make it possible to show/hide list columns.

Native Draw

The Native Draw dialog allows you to specify which display/drawing engine will be used. you can select between standard GDI and Redsdk (Addon Plug-in). By default, under Native Draw, only GDI mode is available. GDI uses standard Windows CPU based drawing technology and CPU based OpenGL hardware acceleration. Redsdk is an OpenGL accelerated drawing engine that can substantially accelerate the speed at which entities are drawn.

GDI Options

Flicker Free Draw: When this option is turned on the redrawing of entities is made smoother, however this can marginally slow down zooming and panning of drawings.

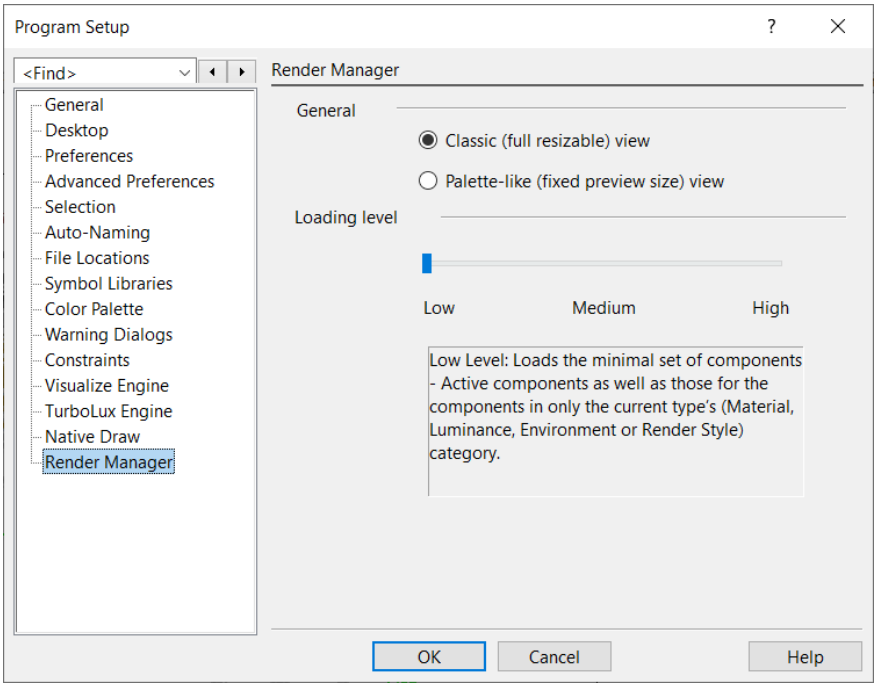
Redsdk Options - Only available after installing its add-on plugin. Not included in TurboCAD by default.

Hardware: Creates an anti-aliasing effect on 2D graphic elements. The values available are: Off, 1x, 2x, 4x, 8x, 16x. Turning this on will make the display smoother, but lowers the performance.
Vertical Synchronization: Makes the display looking better, while moving, however it may higher memory usage.
Text Smoothing: Turning this on will make the display of text smoother, but lowers the performance.
Engine:

- **Wireframe -** Sets the engine anti-aliasing in Wireframe mode. Range – [1-5]. By default – 1. Enabled if Redsdk is On.
- **Hidden Lines** - Sets the engine anti-aliasing in Hidden Line mode. Range – [1-5]. By default – 2. Enabled – if Redsdk is On.
- **Draft** - Sets the engine anti-aliasing in Draft Render mode. Range – [1-5]. By default – 2.

Show Video System Info: Pressing this button will display the current video card setup on your computer. If your current driver for the video card is not the optimal driver for Redsdk you will be prompted to download the correct driver via a link within the window. If you do not download the optimal Redsdk will use a default driver.

Render Manager Options



General

You can specify how the Render Manager is displayed by selecting either Classic or Palette-Like

Classic

- Variable sized preview.
- Tree can be hidden.
- Each category of components is displayed on separate tabs.

Palette-like

- Fixed square preview area (240x240)
- The tree cannot be hidden.
- While resizing the dialog, only the tree's height and the properties area size change.
- All properties of the main shader are shown on one page. Components of sub shaders are shown on separate pages.

Advanced Preferences

Advanced options for program control.

3D Boolean operations

Transfer material from operand on its facet: When checked, material from one object is

transferred onto the other during a 3D Boolean operation (3D add , 3D subtract, etc.). When unchecked, the materials are not affected on either object.

Shader Sets

Copy shaders sets to the application database: Sometimes is will be desirable to copy shaders from a drawing into the Application Shaders Manager. To resolve this issue there is an option on the 'Preferences' page. By default, this option is off. When this option is active, when a file is opened all shaders sets will not only be added to the Model Shaders manager but also will be copied to the Application Shaders Manager.
Shared shaders sets: This option defines the scheme of work with the render attributes (material, luminance, environment, render style). By default, it is off.
Note: After changing of this option it is necessary to restart TurboCAD.

If this option is ON, then all operations with the render attributes occur according to the "old scheme", which was used before TurboCAD 2016. Namely, the changing of an shader attribute in one drawing leads to the changing in all drawings and in the application database.

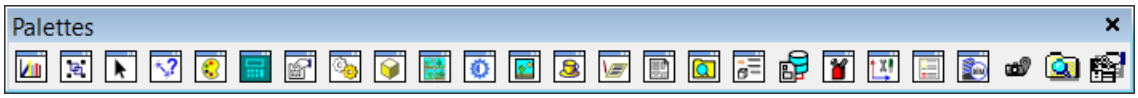
Intelligent Scale

Hatch: When a hatch is created it is automatically scaled to an appropriate size relative to the current view. This means in particular to the level of zoom.
Text: When a text is created it is automatically scaled to an appropriate size relative to the current view. This means in particular to the level of zoom.
Dimension: When a dimension is created it is automatically scaled to an appropriate size relative to the current view. This means in particular to the level of zoom.
Custom Coefficient: This is a scaling multiple that is applied to the intelligent scaling. For example, 2 would double the size whereas 0.5 would halve the size.

TurboCAD: Palettes

TurboCAD palettes are convenient screen areas used for performing common tasks and for obtaining information. By default, the palettes are displayed on the right side of the screen, and contain tabs for easy display and switching.
The palettes commands are available on the **Tools** menu in Default Interface. In Ribbon Interface, Palette toolbar is not available but these palettes can be access from Tools > Palettes.

You can also display the **Palettes** toolbar by right-clicking on any toolbar area and selecting **Palettes**.



Blocks Palette

A block is a collection of objects combined into a single object. Blocks are useful for storing complex, common drawing objects that you will use more than once. Blocks are stored in the drawing's internal block library and a block reference, and not the actual object, is inserted into the drawing. When a block is edited, all instances of that block in the drawing are updated.

Calculator Palette - Variables Palette

Enables you to make numerical calculations within TurboCAD, and to define variables.



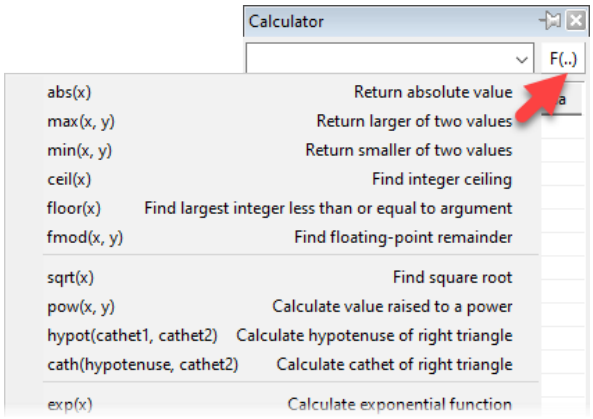
You can also use this palette to assign constraints to dimensions.

Mathematical Calculations

To perform a calculation, enter the values in the top field of the palette, using parentheses as needed to create inner expressions. No spaces are allowed. The four mathematical operators should be entered as +, -, * (multiply), and / (divide).

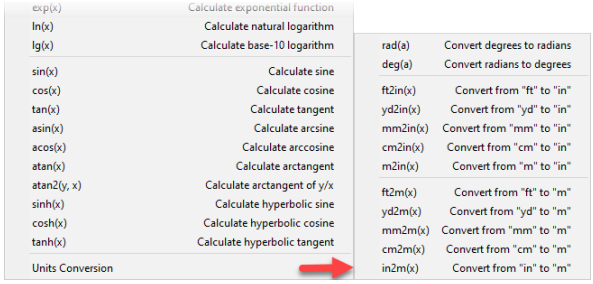
To calculate the result, press Enter.

You can also use these commonly-used values in place of numbers:
pi =3.1415926 e =2.71828
The function (**F**) button enables you to perform a function on the value in the parentheses. For example, select **sin** to get the sine of an angle.



sin appears in the calculator field (you also could have entered this expression yourself manually).

Sine is calculated for radians, so if you have a value in degrees, it must be converted. Look in the functions menu under **Units Conversion** and select rad(a), which converts degrees to radians.



Arrange the expression so that the parentheses are in the correct places, and include the angle in degrees (45 in this case):



Press Enter to obtain the result.

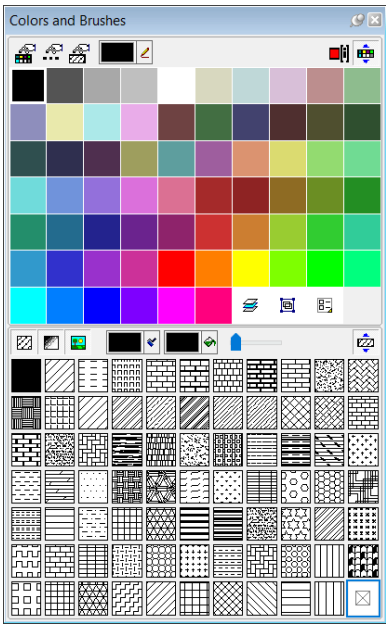


Some functions require two values, such as exponentials. **Pow (x, y)** raises x to the y power.

Include the values and press Enter to get the result.

Colors and Brushes

Displays all colors and brushes currently available. By default, the Colors and Brushes palette shows colors on the top and brushes (hatch patterns) on the bottom.



The color selected at the top and the brush pattern selected at the bottom appear in the **Property** toolbar. These properties become the default for the current tool and other tools in the tool group. 

For example, if **Line** is active and you set a color, this color will become the default for **Rectangle**, **Polyline**, etc. The icons at the top of the palette are as follows:

Design Director

A convenient place to perform commands related to layers, workplanes, views, cameras, and objects or object groups.

Docking Palettes

Palettes can be docked just like toolbars. In addition, palettes can be docked onto other palettes to create palette groups. Drag a palette onto the desktop by its named tab (the tab that contains the name of the palette. The palette must actually be displayed before it can be dragged onto another palette group. To dock a palette to any of the four edges of the TurboCAD window, drag the palette toolbar to the relevant arrow, either displayed at the center of the drawing area, or along the desired edge.



Note: These arrows are displayed only if **Docking Stickers** is checked on the **Palettes** page of the **Customize** window (**Tools / Customize**).

To combine palettes into a group, drag more palettes onto a docked palette.

To replace palettes, simply drag their tabs back to the palette area.

Drafting Palette-Creating Standard Views

The Drafting Palette enables you to insert standard views of your model into Paper Space.

Environments Palette

Provides easy and fast access to environments
Categories



Set the Environment category.
Find Environment
To switch the environment display from thumbnails to list format, click **View Thumbnails** and select a different view.



TurboCAD has LightWorks rendering engine available by default. The Render Engine Filter allows you to specify which components will be displayed in the palette. ALL sets the palette to show all elements from all installed rendering engines. LW will show only LightWorks elements.

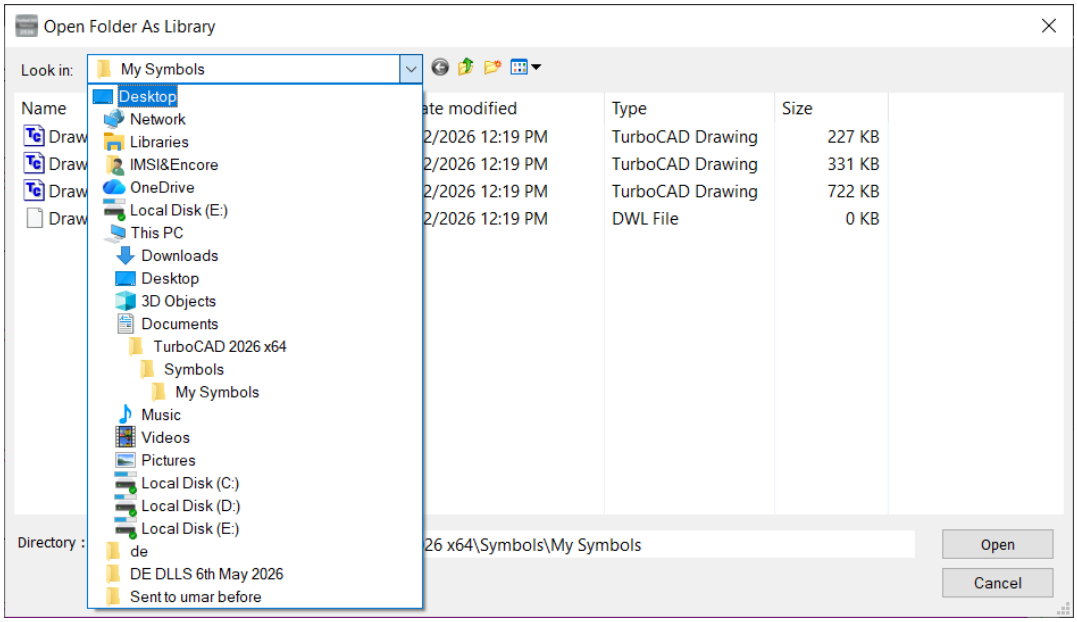
Search Option:

You can also search a environment in the palette. Type a word in "find" option to search, the first suitable item is highlighted. To browse through the environments, buttons next to the input field highlight and can be used to show the previous or next suitable item.

Library Palette

The TurboCAD library is where you can access symbols and parts. Symbols and parts, like blocks and groups, are objects available for repeated use. But while groups and blocks are internal to a drawing, library items are external files. Generally, each symbol or part is stored as a file, and libraries of files are stored in specified library folders. Any Windows folder can be recognized as a library folder, and any vector drawing file (not just TurboCAD files) can be used as a symbol.

A symbol or part is placed into a drawing as a group; it has a unique name and is selected and manipulated as a single object. Because they are placed as groups, each instance of a part or symbol creates additional drawing data, and inserting numerous library items can increase your drawing size significantly.
TurboCAD provides several symbols and parts, organized in folders by category. Each symbol is a separate *.tcw file, and parts are *.ppm files. These files are installed, by default, to the "Symbols" folder in the TurboCAD installation folder.



You can modify these symbols (though it is preferable to modify copies and leave the originals intact), and also create and save your own symbols. Symbol manipulation is done via the Library Palette.

Luminances Palette

Provides easy and fast access to object luminance.
Categories



Specify the luminance category.
Find Luminance.

To edit a luminance, select the item in the palette, then right-click and select **Edit Luminance**. Or double-right-click on the item, or click **Edit Luminance**.

To switch the luminance display from thumbnails to list format, click **View Thumbnails** and select a different view.

Right-click on the Luminance palette will open a local menu that provides a wide array of options.

MacroRecorder Palette

Creates and plays scripts - scenarios of creating and editing objects and manipulating their properties. You can also record object transformation - moving, copying, scaling, and rotating.

Scripts can be saved for future play.

Materials Palette

Provides easy and fast access to object materials.

All materials that have been defined are displayed here, and can be assigned to objects and edited.
Find Material of Selection.

To edit a material, select the item in the palette, then right-click and select **Edit Material**. Or double-right-click on the item, or click **Edit Material**.

You can use **Select by material** to select the object by applied material. Select the material in the materials palette and click **Select by material** the object having the material applied will be selected. Make sure that Active Drawing is selected in the categories.

To switch the luminance display from thumbnails to list format, click **View Thumbnails** and select a different view.

TurboCAD has Lightworks rendering engine available. The Render Engine Filter allows you to specify which components will be displayed in the palette. ALL sets the palette to show all elements from all installed rendering engines. LW will show only LightWorks elements.

Dragging and Dropping Materials

Another way to apply materials to object is to select a material in the Material palette, then drag-and-drop it onto the desired object.

Drag on Facet

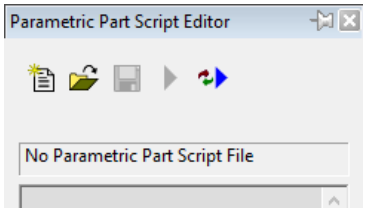
Selecting the **Drag on facet** setting in the local menu of the **Material** palette will allow you to drag a material to a single facet of an object.

Measurement Info

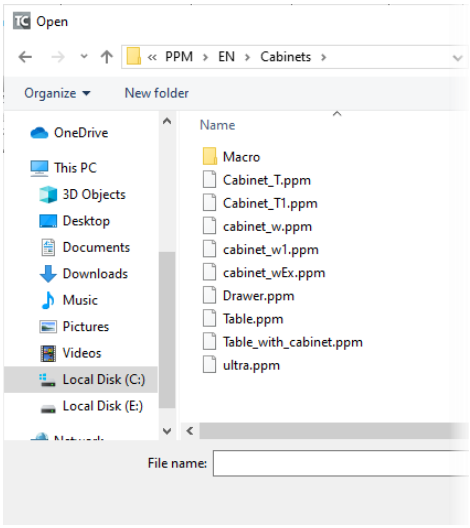
Displays measurements calculated by the **Measure** tools. You can measure point coordinates, distance and perimeter, angle, and area.

Other than creating a parametric part from within TurboCAD, the other way to create a part is to write a script. The parts provided in folders such as "Cabinet" are created this way.
If you want to write your own script, or edit an existing script, you can use the Parametric Part Script Editor.

1. To load an existing script into the editor, click **Open**.



2. Locate one of the scripts in the "Cabinet" folder.

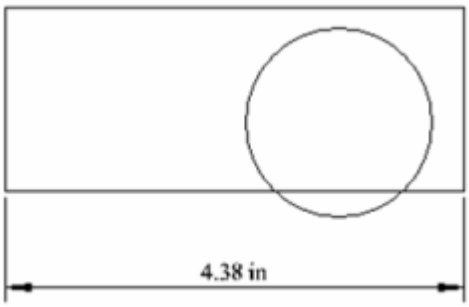


3. The script appears in the editor.

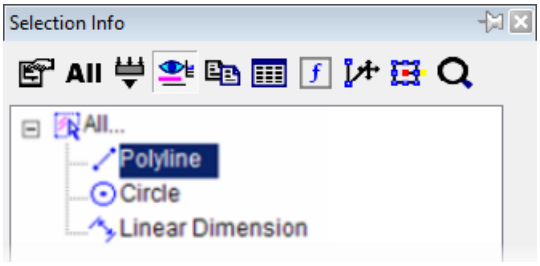
Selection Info Palette

Selection Info- Properties

This example consists of a rectangle, a circle, and a linear dimension.

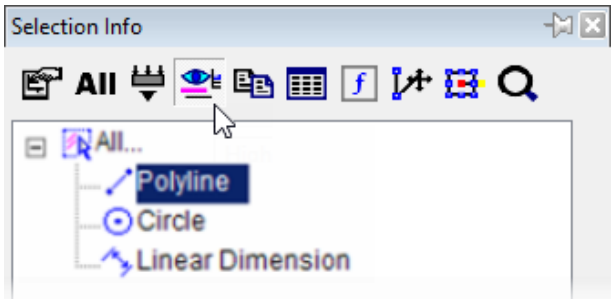


Open the Selection Info palette to see the three objects listed.

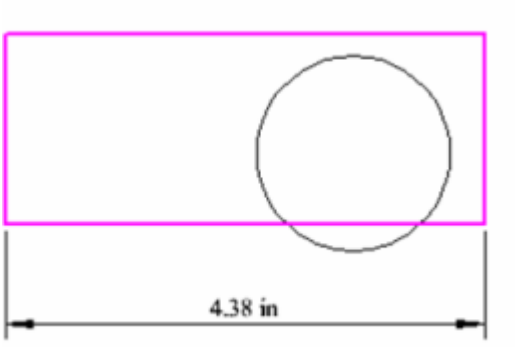


Tip: The number of selected objects must be less than or equal to **Maximum Selection Info Entities,** which is set in the **Program Defaults** page of the TC Explorer Palette.

In the Selection Info toolbar, click **Highlighting**, and make sure Polyline is selected.

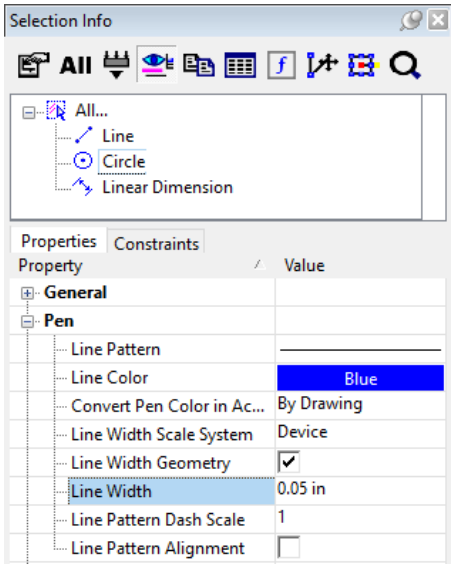


This highlights the rectangle in the drawing window.



Conversely, if you select objects in the drawing area, they will be highlighted in the Selection Info palette.

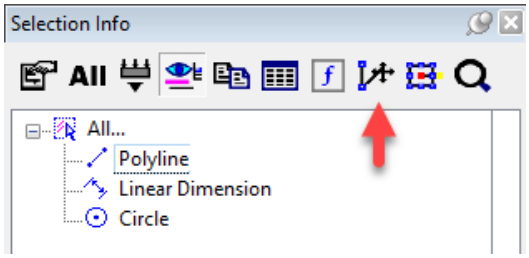
You can use this palette to change an object's properties. Highlight the circle, and the lower section of the palette contains the categories of properties for the circle. In this example, the color and line weight were changed in the **Pen** category.



Note: For 3D ACIS objects, physical properties are not automatically listed. Click the **Physical Metrics** icon in the palette toolbar if you want to see engineering properties *(volume, moments of inertia, etc.).*

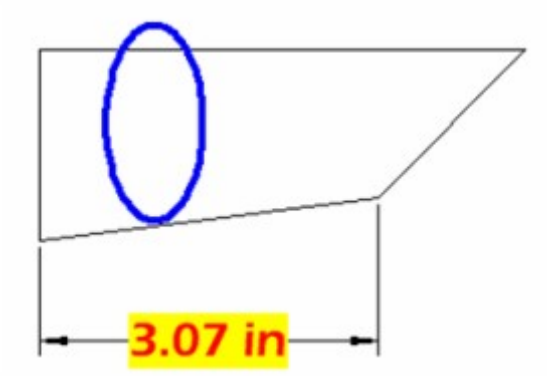
The new properties appear in the drawing area. (You may have to turn off the **Highlight** icon in the toolbar to see the changes.)

To make some changes to the circle, click **Select**.

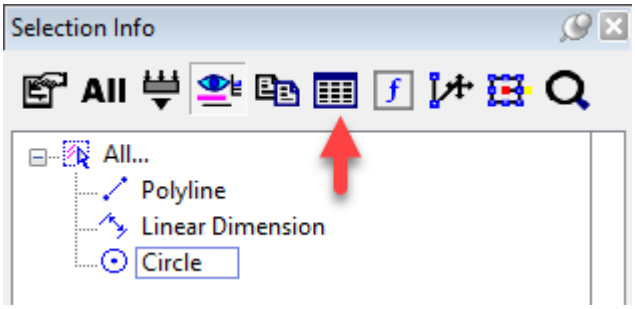


Move one or more nodes to change the shape.

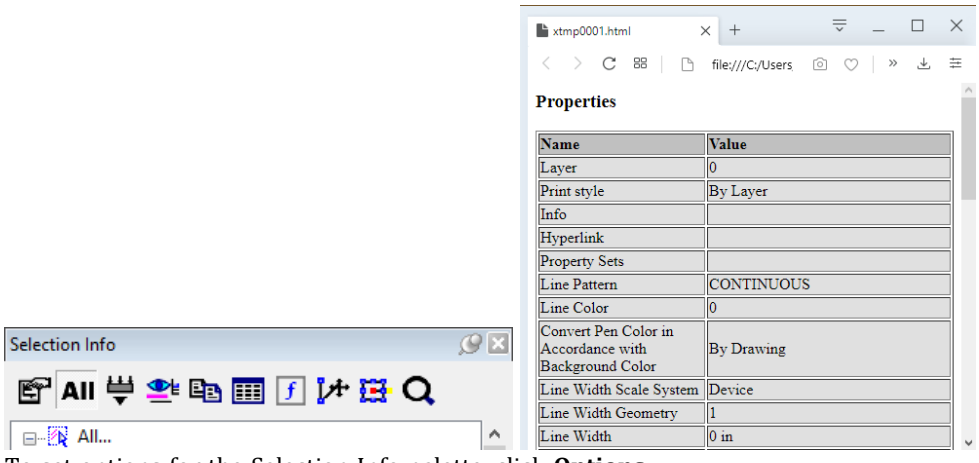
The changed dimension appears in the drawing area.



Note: You can also edit an object's properties by clicking the **Properties** icon in the palette toolbar. This opens the **Properties** window for the object.



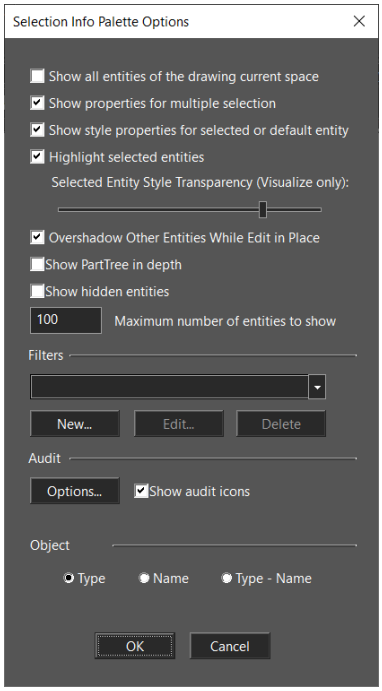
The **Copy** icon is used to view an object's properties in HTML format They will appear in your default browser. The HTML file is stored in the ...Profiles/Built-in folder.

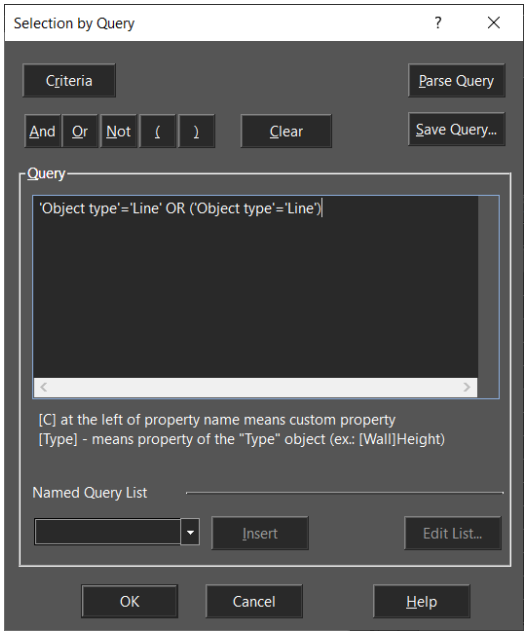


To set options for the Selection Info palette, click **Options**.

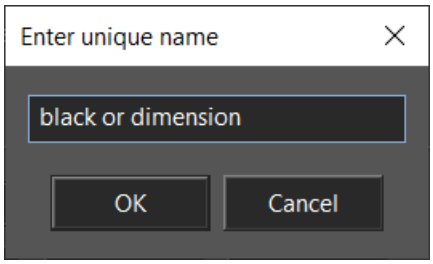


The options at the top control highlighting and fading for objects that are not being edited. To seta filter for what objects are displayed in the palette, click **New**. This options the **Selection by Query** window; in which you can specify types of objects you want to display. In this example, all objects that are either black or dimensions will be displayed in the palette.





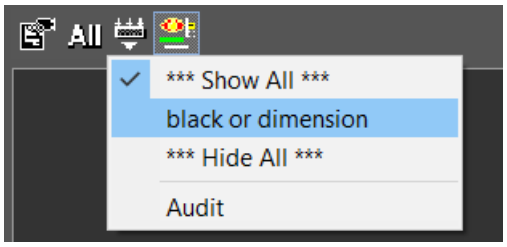
1. Each filter must be assigned a name.



2. To apply this filter, click **Filter** in the toolbar.

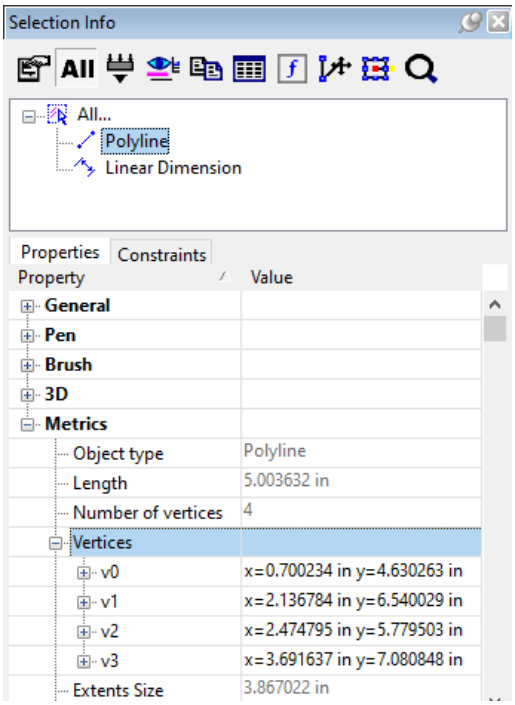


3. Select the filter you just created.

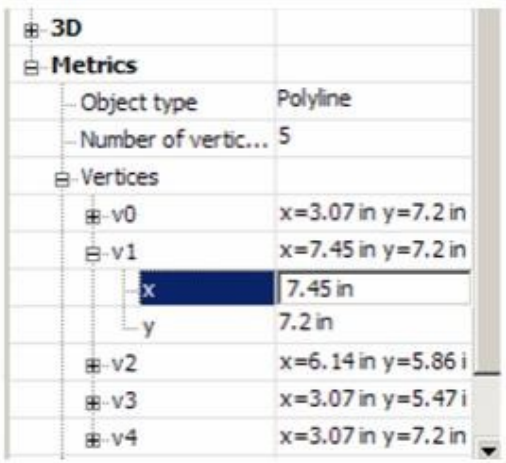


Now only the polyline and dimension appear in the palette.

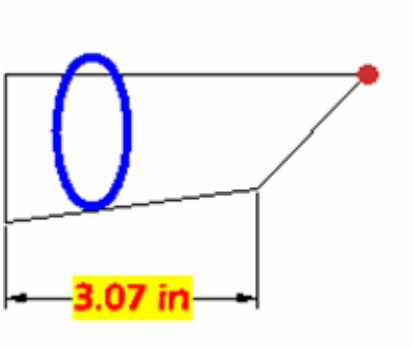
4. You can change geometry by modifying geometry in the **Metrics** category. Open **Metrics** for the Polyline - the coordinates for each vertex are listed.



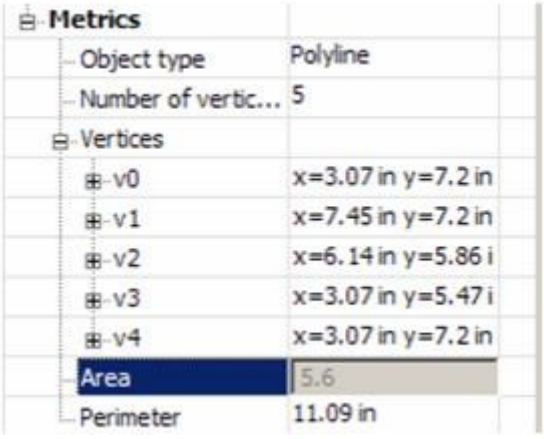
5. Open the branch for a vertex to see its X and Y coordinates. These values can be changed.



The vertex highlighted in the palette is also highlighted in the drawing area.

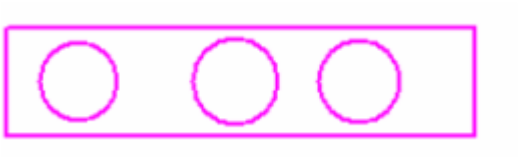


Some values cannot be edited, such as Area and Perimeter. These values depend on the coordinates of the vertices, and are grayed out



.

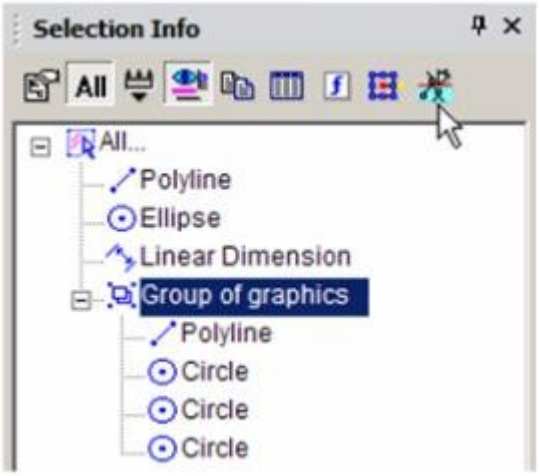
6. To see options for another type of object, create several objects and group them (select them, then select **Format / Create Group**).



The Extents Size parameter has been added to the selection Info palette properties. Extents Size parameter also works with groups, blocks and insertions. You can use the 'Extents size' property to select entity according to size valuation. Select objects that you think are big or small, or object in any range you want.

You can also select objects on the basis of extents size parameter using Select by Query.

1. This object is a "Group of Graphics," and you can open it to see each object that comprises the group. To edit the group, click **Edit Content**.



2. The group's contents fill the screen. Make some change, such as adding or removing objects from it.

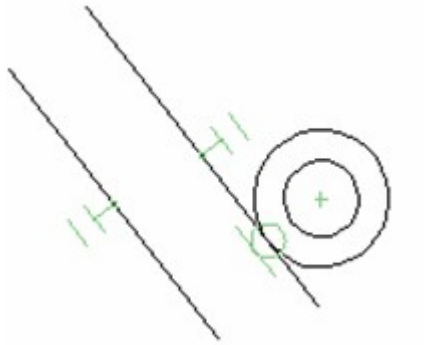


3. When finished, click **Finish Edit Content**.

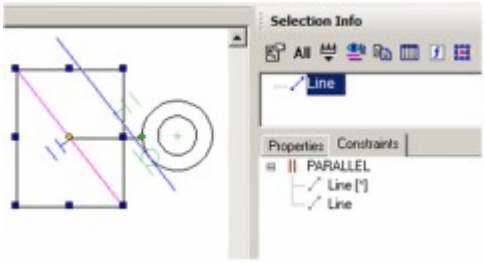


Selection Info- Constraints

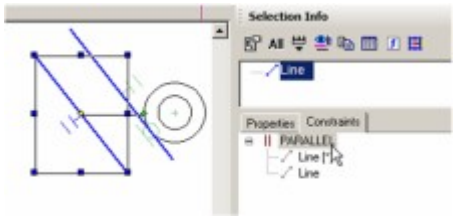
The **Constraints** tab of the Selection Info palette shows what constraints have been applied, and to which objects they are applied.
This example consists of two lines and two circles. Three constraints were applied: the two lines are constrained to be parallel; the circles are constrained to be concentric, and a tangent constraint was applied to the outer circle and adjacent line.



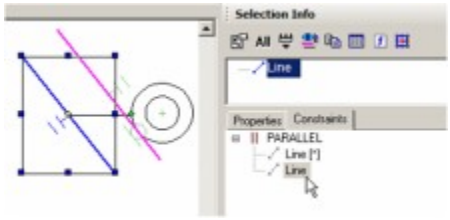
1. Make sure the **All** icon is disabled at the top of the Selection Info palette. Select the outer line, which only has one constraint (parallel). In the **Constraints** tab, the PARALLEL constraint is listed, below which are listed the two lines to which the constraint applies.



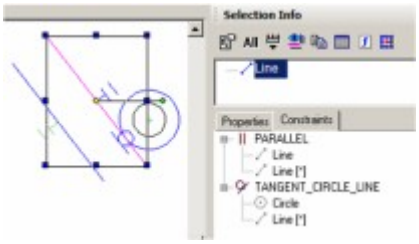
2. Move the cursor over the PARALLEL constraint in the **Constraints** tab, and the two lines that have this constraint are highlighted.



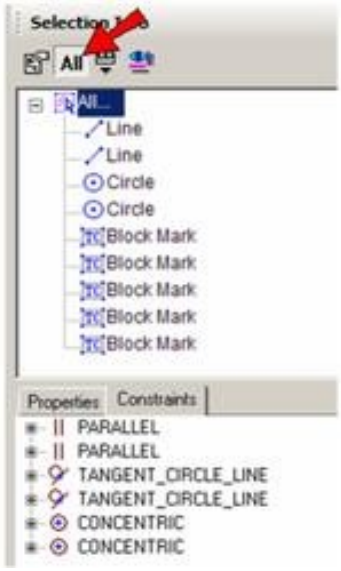
3. Move the cursor over either line below PARALLEL. The relevant line is highlighted.



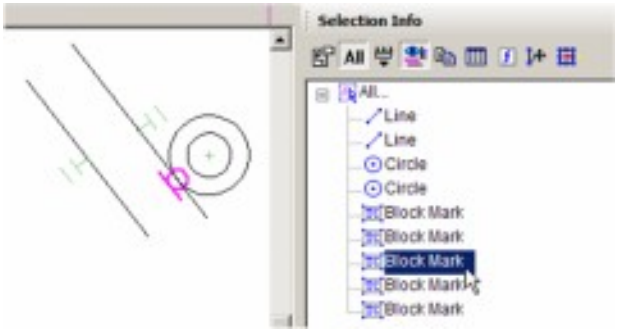
4. Now select the inner line, which has two constraints applied to it. These two constraints are listed: PARALLEL and TANGENT.



5. Now click **All**. The four objects (two lines, two circles) are listed, along with several block marks. These are the constraint markers themselves, which can be selected and deleted if you want to remove the constraint. In the **Constraints** tab, two items are listed for each constraint - one for each of the constrained objects.

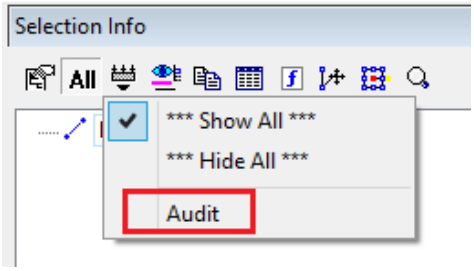


6. Select any blockmark in the palette to see it highlighted onscreen (**Highlighting** must be enabled).



Audit System:

"Audit" is added to menu "Filters". Audit System is used to detect a 'bad' object in drawing. A 'bad' object triggers at least one audit conditions.



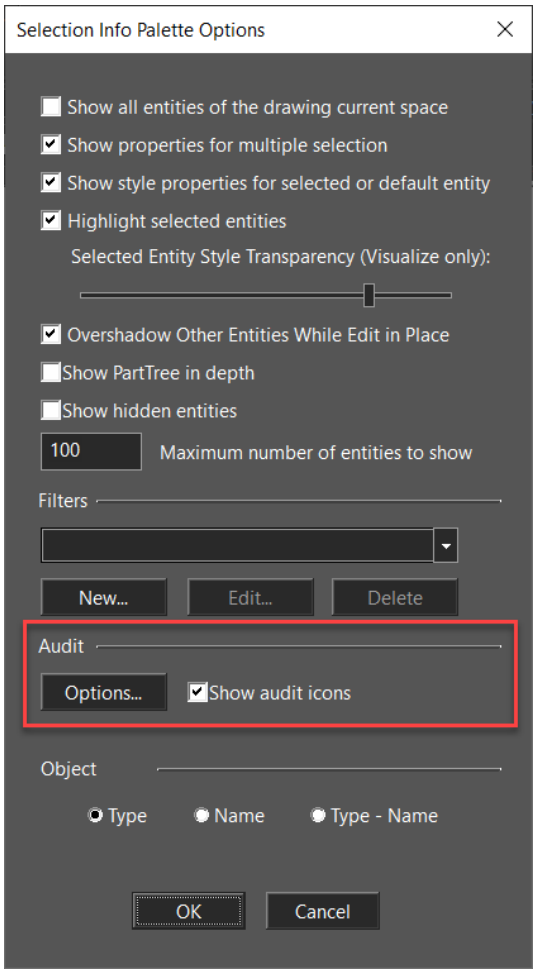
Audit Report:

We can also generate an audit report by clicking on audit report icon on menu bar of selection info palette.

"Audit" report (HTML page).

Audit Options:

Audit icons which are shown in selection info palette with objects can be enabled or disabled by Selection Info Palette Options dialogue box. To open Selection Info Palette Options dialogue box select options icon from selection info menu bar.



Select "Options..." from Selection Info Palette Options.

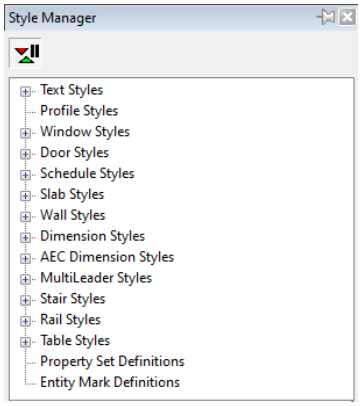
“Audit” section

The Style Manager Palette

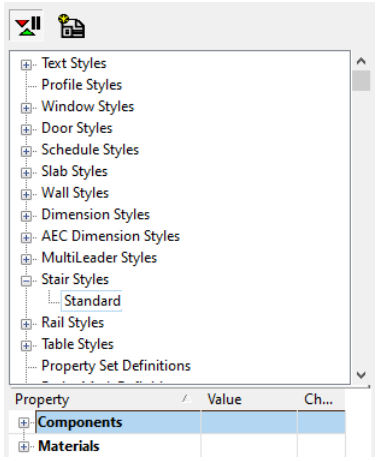
The Style Manager enables you to define styles for the following commonly-used objects:

- Text
- Profiles (for modifying windows and doors)
- Windows
- Doors
- Walls
- Schedules
- Slabs
- AEC Dimensions
- Dimensions
- Stairs
- Rails
- Tables

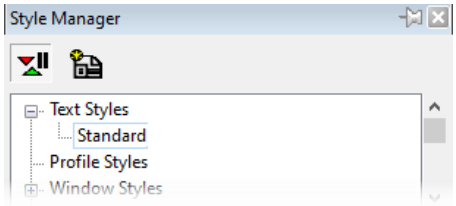
You select assign a style to an object through the object properties on the General page. Simply select the style in the Style drop-down box at the bottom of the page. You can open the Style Manager by selecting **Tools / Palettes / Style Manager Palette**.



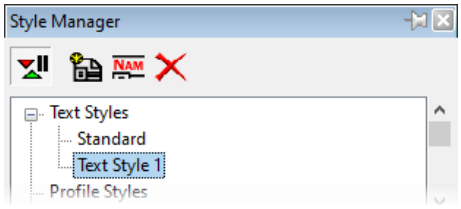
Each category in the Style Manager contains one pre-defined style, called "Standard." For each of these styles, you can define relevant properties.



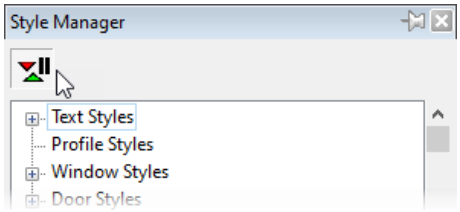
To create a new style, highlight a style you want to base the new style on, and click **Create New Style**.



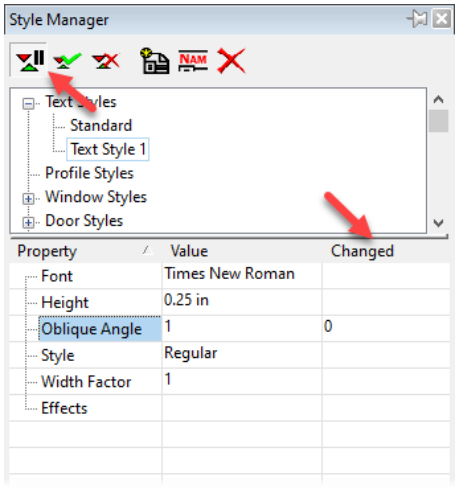
Once you have new style, there are icons at the top you can use to **Rename** or **Delete** it.



The Delay Style Modification* icon can be enabled if you don't want to update your style with each property change.



If **Delay Style Modification** is enabled, you can view both old and new properties in the lower pane. At the top is an icon for updating the entire style at once, or for clearing changes not yet implemented.



The TC Explorer Palette

This palette enables you to perform many operations in one place. It serves as a file browser with previews, and displays components and editable parameters for all open drawings.

Event Viewer

The Event Viewer palette creates a log detailing Error and Warning events effectively improving access to the TurboCAD diagnostic information.

Local menu

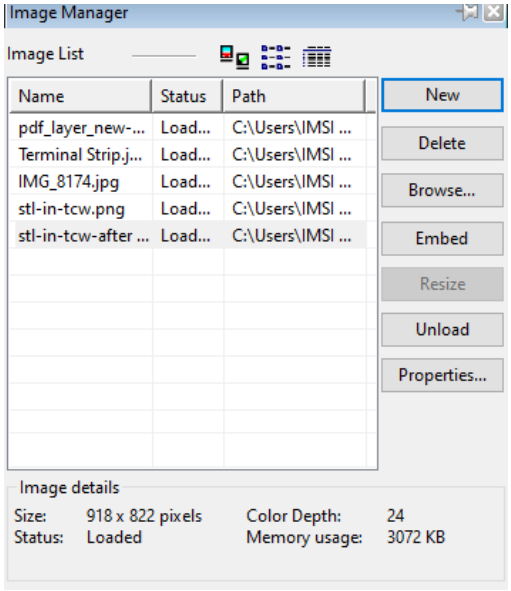
- **Clear** – clears all events
- **Write to Log File** – turns on/off the recording to the log file.
- **Open Log File** – opens the log file in a text editor.

Level

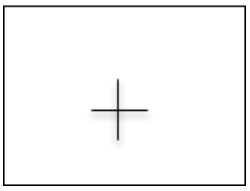
The icon shown under the 'Level' field can have 3 colors with the values: YELLOW = Warning, RED = Critical, and GREEN = Info
If the 'Show' parameter of a warning dialog is set to 'Off', the icon in the Event Viewer will be displayed with a strikethrough X.

Image Manager Palette

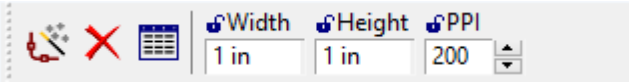
Enables you to manipulate all the raster images in the current drawing. You can insert files directly from this window using **From Image List**. You can also select Multiple files by using "New" button and importing multiple images in the Image Manager palette.



Insert
Insert a new image by drag and drop.



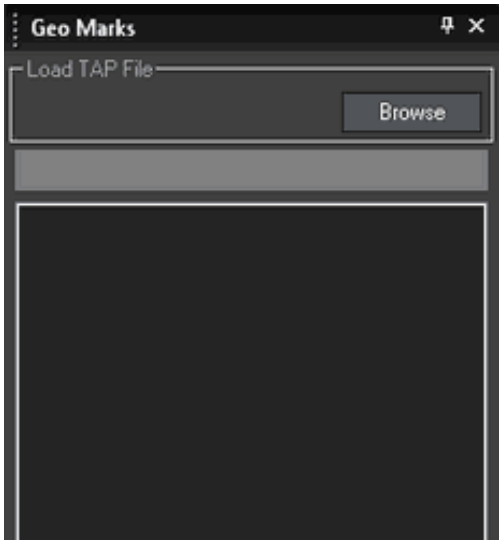
Use the PPI (pixel per inch) option or the Width/Height options from the Inspector bar to set the size of the image.
The PPI image property is also available from the Selection Info Palette



The toolbar at the top of the window can be used to change the image list as thumbnails, list, or detailed list.

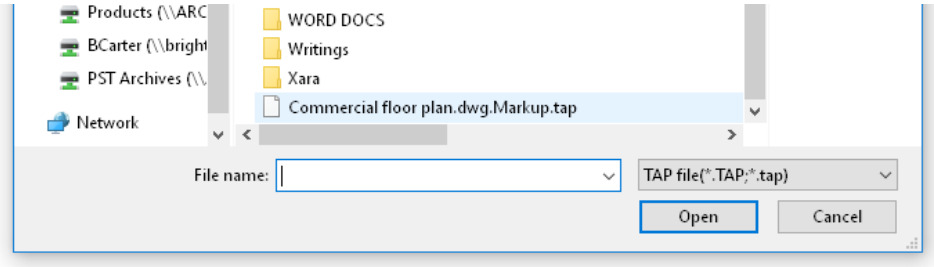
GeoMarks and TAP files

The Geomarks palette allow you to open TAP file created in IMSI/Design TurboApps products.

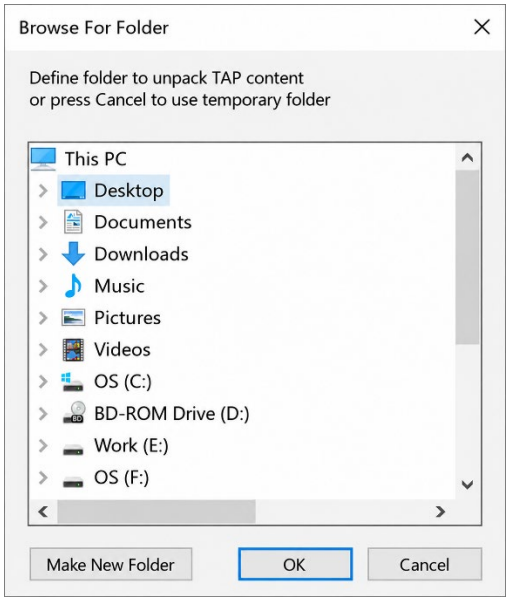


To open a TAP file:

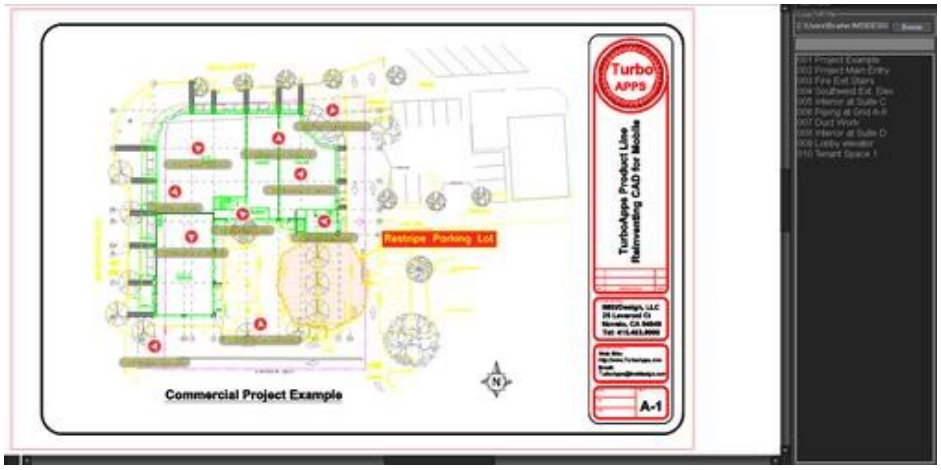
1. Open the GeoMarks palette.
2. Click the Browse button.
3. Browse to the file you wish to open.



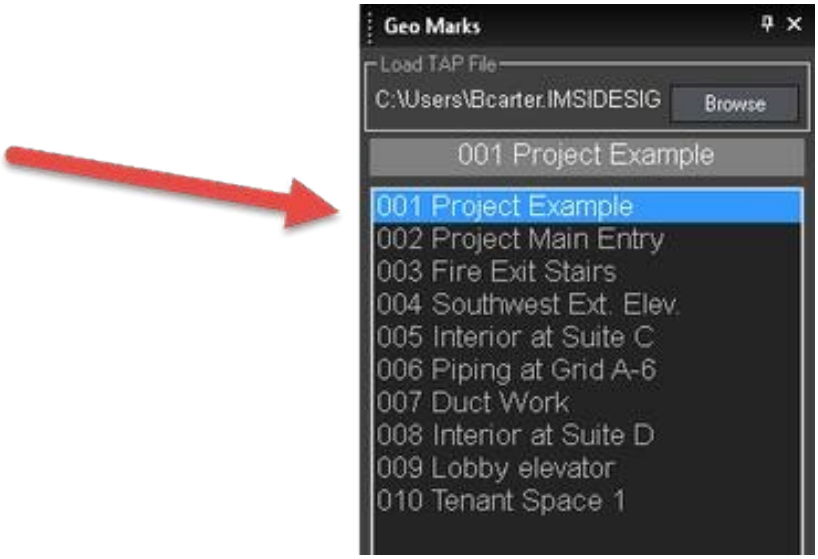
4. Select the file and click **Open**.



5. The contents of the TAP must be unpacked. Select a folder to unpack the TAP content, or click Make New Folder to create a new folder, or press Cancel to use a temporary folder.



You can click on any link to view its content.



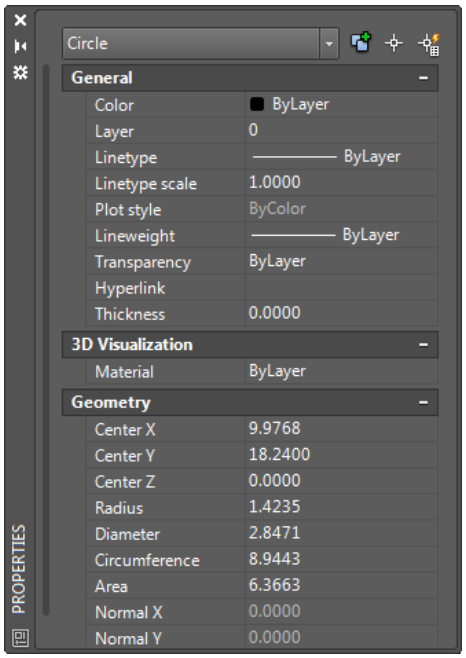
The controls are explained below:

If a TAP file has been created in TurboApps with markups, then TurboCAD now has the ability to display these mark-ups when a TAP file is imported and opened into the program.

AutoCAD: Palettes

Properties Palette

PROPERTIES (Command)  Find



When more than one object is selected, only those properties common to all selected objects are displayed. When no objects are selected, only the current settings of general properties are displayed.

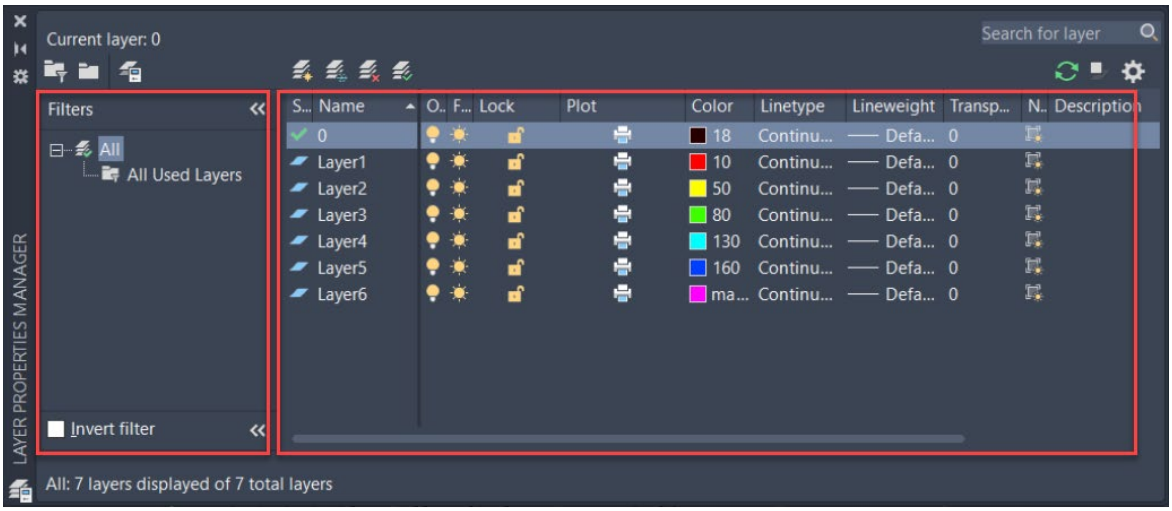
You can specify a new value to modify any property that can be changed. Click the value and use one of the following methods:

- Enter a new value.
- Click the down arrow at the right and select a value from the list.
- Click the Pick Point button to use the pointing device to change a coordinate value.
- Click the QuickCalc calculator button to calculate a new value.
- Click the left or right arrow to increase or decrease the value.
- Click the [...] button and change the property value in a dialog box.

Layer Properties Manager

You can add, delete, and rename layers, change their properties, set property overrides in layout viewports, and add layer descriptions.

LAYER (Command)  Find



Layers and Layer Properties

New Layer 

Creates a layer with a default name that you can immediately change. The new layer inherits the properties of the currently selected layer in the layer list.

New Layer Frozen VP In All Viewports 

Creates a layer and freezes it in all existing layout viewports. This button is accessible from the Model tab or layout tabs.

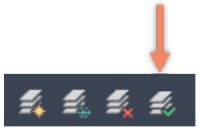
Delete Layer 

Deletes selected layers. The following layers cannot be deleted:

- Layers 0 and Defpoints

- Layers containing objects, including objects in block definitions
- The current layer
- Layers used in an external reference
- Layers in a partially opened drawing (Not applicable to AutoCAD LT.)

Note: Be careful about deleting layers if you are working on a drawing in a shared project or one based on a set of layering standards.

Set Current 

Sets the selected layer as the current layer. New objects are automatically created on the current layer. (CLAYER system variable).

Tools Palette

Tool palettes are used to organize tools that range from blocks to commands to hatch patterns, and are displayed as part of the Tool Palettes window (TOOLPALETTES command).

You customize tool palettes by dragging and dropping files from Windows Explorer or File Explorer, objects from the drawing area, contextual menus of the Tool Palettes window, Customize User Interface (CUI) Editor, or Customize dialog box. Each tool palette that is created represents a tab on the Tool Palettes window.

QuickCalc Calculator

Performs a full range of mathematical, scientific, and geometric calculations, creates and uses variables, and converts units of measurement.

External References Palette

Opens the External References palette.

Note: The FILEDIA system variable is ignored when you attach files from the External References palette.

Sheet Set Manager

Displays and organizes named collections of drawing sheets

SHEETSET (Command)

The Sheet Set Manager is not fully functional if:

- The command is active;
- No drawing is open;

- The sheet set is locked by another user; or
- A lock icon displayed in front of the sheet set name in the Sheet Set Manager indicates that the sheet set is locked. Hold your cursor over the lock icon to view a tooltip that shows who has the sheet set locked.

Markup Set Manager

Displays information about loaded markup sets and their status.

You can show or hide markups and their originating drawing files in your drawing area. You can change the status of individual markups and add notes and comments to them.

DesignCenter

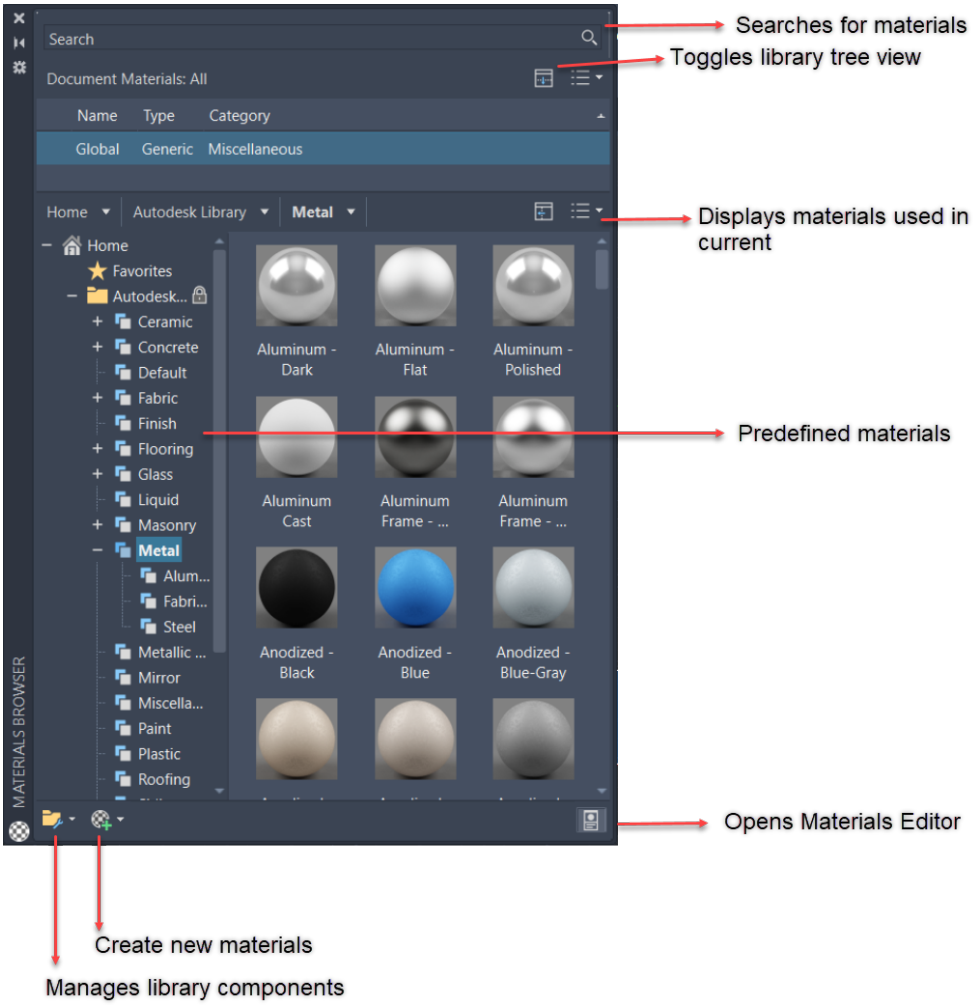
Organizes access to drawings, blocks, hatches, and other drawing content.

With DesignCenter, you can

- Browse for drawing content such as drawings or symbol libraries on your computer, on a networked drive, and on a web page
- View definition tables for blocks and layers in any drawing file and then insert, attach, or copy and paste the definitions into the current drawing
- Update (redefine) a block definition
- Create shortcuts to drawings, folders, and Internet locations that you access frequently
- Add content such as xrefs, blocks, and hatches to a drawing
- Open drawing files in a new window
- Drag drawings, blocks, and hatches to a tool palette for convenient access
- Copy and paste content, such as layer definitions, layouts, and text styles between open drawings

Materials Browser

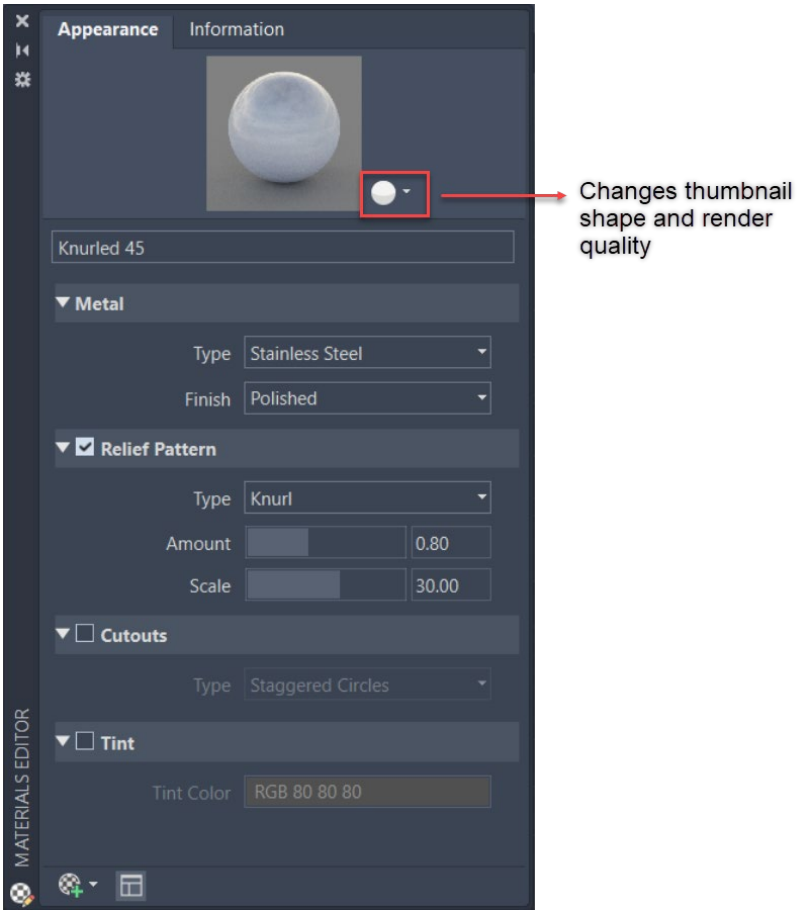
The Materials Browser allows you to navigate and manage materials.



Manage material libraries provided by Autodesk, or create custom libraries for specific projects. Use the filter buttons to change which materials are displayed, the size of the thumbnails, and the amount of displayed information.

Materials Editor

Edits the material selected in the Materials Browser.



The configuration of the Materials Editor changes depending on the type of material that is selected.

Visual Styles Manager

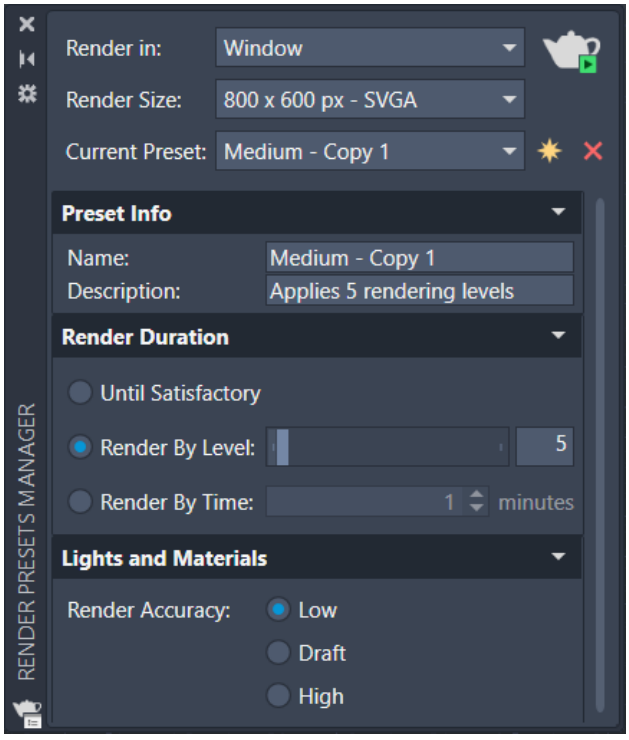
Creates and modifies visual styles.
The Visual Styles Manager has a panel of sample images of the visual styles available in the drawing, and the following properties panels:

- Face settings
- Environment settings
- Edge settings
- Lighting settings

Note: Changes that you make in the Visual Style panel on the ribbon create a temporary visual style, *Current, * that is applied to the current viewport. The settings are not saved as a named visual style.

Render Presets Manager

Controls all the primary settings for rendering.

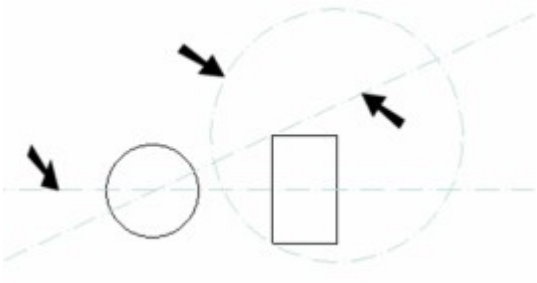


You use the Render Presets Manager to specify the primary settings to use when rendering. You can also use the controls located on the ribbon to change some general rendering settings or set a named render preset current.

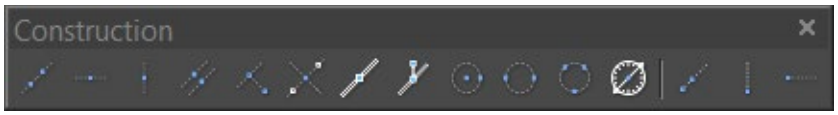
TurboCAD: Drawing Aids

Construction Geometry

Construction geometry tools enable you to place temporary lines and circles in your model. Construction geometry are not drawing objects; they are used as references.



Snaps can be used on construction geometry for creating model geometry, as well as other construction geometry. However, because construction lines are infinite, the **Vertex** and **Middle Point** snaps are not available. You can display the **Construction** toolbar by right-clicking in any toolbar area and selecting **Construction**.



These tools are also available on the fly-out toolbar from the **Drawing Tools**.

Note: You can use construction entities with the Trim tool to trim other drawing entities, however you cannot trim construction entities themselves.

Clearing and Hiding Constructions

Default UI Menu: Edit / Clear / Construction, Edit / Clear / All Constructions

You can clear construction objects via the **Edit** menu.
Edit / Clear / Construction enables you to delete a single construction object.
Edit / Clear / All Constructions deletes all construction geometry.
To hide constructions, you can hide the $CONSTRUCTION layer. Open the **Layers** window and uncheck the visibility box for this layer.

Construction Geometry Properties

Construction objects are placed on layer "**$CONSTRUCTION**". By default, the layer color is light blue, and the line style is dash-dot. You can change construction geometry color and line styles via the layer manager.

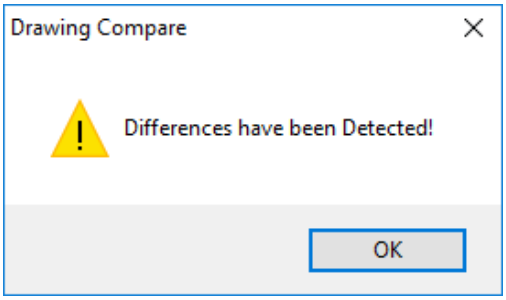
TurboCAD: Drawing Compare

Drawing Compare is an easy to use utility which allows you to compare two drawings and find the differences between them. You can use 2CD, DWG, TCW, and BMP files.
The process is simple, just load two drawings and press the Compare button. The Vector drawing files are converted to a raster image map and pixel testing detects changes between the two images.

Comparing Drawings

Comparing drawings is a simple process:

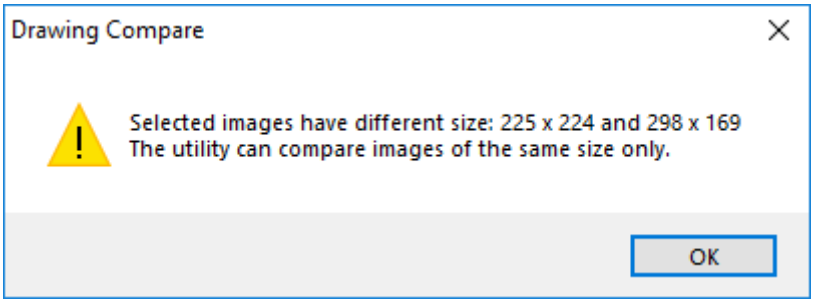
1. Use the **Select File 1** button or path to specify the first file for comparison.
2. Use the **Select File 2** button or path to specify the second file for comparison.
3. Press the **Compare** button.
4. If there are differences between the two drawings you will see the following message:
5.



1. Press the **OK** button. Your Drawings will now be loaded for comparison
2. Use the Zoom and Show buttons to examine your files and the differences.

Drawing Compare File Types

TCW: (TurboCAD Native Files)**:** You can use any TCW file.
DWG: (AutoCAD Native Files)**:** You can use any DWG file.
BMP: You can use any standard Windows BMP file. However, both BMP files must be the same size. If the images are of different sizes you will get an error message.



PNG: You can use any standard Windows PNG file. However, both PNG files must be the same

size. If the images are of different sizes you will get an error message.
JPG: You can use any standard Windows JPG file. However, both JPG files must be the same size. If the images are of different sizes you will get an error message.
PDF: Adobe Portable Document Format
SKP: SketchUp file format.

Show & Compare buttons

 **Show First File:** Shows only the first file.

 **Show Second File:** Shows only the second file.



Show the Differences: Shows only the differences between the two files.



Blink Differences: Blinks between the two files to illustrate the changes.



Compare: Compares the two files and generates the difference images.

AutoCAD: Drawing Compare

The COMPARE command provides you with a visual comparison between different revisions of a drawing or different drawing, and then review those changes. During the comparison process, it identifies objects that have been modified, added, or removed in the review drawing. The default two-color comparison mode uses green and red to highlight the differences.

The **DWG 1** and **DWG 2** are inserted as blocks into the comparison drawing. In the comparison drawing, the differences are highlighted, as change sets, using revision clouds. Revision clouds are created on a new layer, in the comparison drawing, named 0-Markups.

The number of differences in the two drawing files determines the number of revision clouds. By default, rectangular revision clouds are used to highlight the differences. You can use these various comparison options for reviewing the comparison drawing:

View - You can change the display order and object visibility in the comparison drawing from the compare ribbon contextual tab.

Filter - You can use the text and hatch filters to show or hide these objects in the comparison drawing. By default, text objects are always included in the drawing comparison.

Change set - You can change the type of revision cloud for your change set, control the offset distance between the bounding box to the rectangle used for creating the revision cloud, and navigate through the various change sets in the comparison drawing.

Summary of AutoCAD Keyboard Shortcuts

Function Keys

AutoCAD Function Keys	Meaning in AutoCAD	Meaning in TurboCAD
F1	Displays online help	Displays online help
F2	Toggles text-graphics windows	Shows Calculator palette
F3	Toggles osnap mode	Toggles palette
F4	Toggles tablet mode	Selects rectangular area for aerial view
F5	Cycles through isoplanes	Redraws active view
F6	Toggles dynamic UCS	Selects next object in draw order
F7	Toggles grid display	Selects next object in back order
F8	Toggles ortho mode	...
F9	Toggles snap mode	...
F10	Toggles polar tracking	...
F11	Toggles osnap tracking	...
F12	Toggles dynamic input	...
Ctrl+F4	Closes drawing	Closes drawing
Ctrl+F6	Switches to next drawing window	Switches to next drawing window
Alt+F4	Closes AutoCAD	Closes TurboCAD
Alt+F8	Runs VBA macros	...
Alt+F11	Opens VBA IDE window	...

Other Keys

AutoCAD Other Keys	Meaning in AutoCAD	Meaning in TurboCAD
Ctrl	Overrides Lock UI Selects 3D faces Cycles through selection	...
Del	Deletes objects	Erases selected objects
Enter	Executes commands Accepts options Repeats last commands	Executes coordinate values
Esc	Cancels current command Cancels grips	Cancels command in progress
Shift	Overrides ortho mode	Enforces ortho mode

Control Keys

AutoCAD Control Keys	Meaning AutoCAD	Meaning in TurboCAD
Ctrl+A	Selects all objects	Selects all objects
Ctrl+B	Toggles snap mode	...
Ctrl+C	Copies to Clipboard	Copies to Clipboard
Ctrl+D	Toggles dynamic UCS	Places camera at drawing center
Ctrl+E	Cycles isoplanes	Sets input focus to Edit bar
Ctrl+G	Toggles grid display	Starts editing group content
Ctrl+H	Toggles pick style	Selects hyperlink
Ctrl+K	Displays Hyperlinks dialog	Selects objects by color
Ctrl+L	Toggles ortho mode	Selects all objects on current layer
Ctrl+N	Starts new drawing	Starts new drawing
Ctrl+O	Opens file	Opens file
Ctrl+P	Displays Plot dialog	Prints drawing
Ctrl+Q	Toggles log file	Selects objects by query
Ctrl+R	Cycles to next viewport	Sets focus to status bar
Ctrl+S	Saves drawing	Saves drawing
Ctrl+T	Toggles tablet mode	Selects objects by type
Ctrl+U	Toggles polar tracking	Toggles world coordinate system
Ctrl+V	Pastes from Clipboard	Pastes from Clipboard
Ctrl+X	Cuts to Clipboard	Cuts to Clipboard
Ctrl+Y	Redoes last command	Redoes last command
Ctrl+Z	Undoes last command	Undoes last command
Ctrl+Shift+A	Toggles group mode	...
Ctrl+Shift+C	Copies with basepoint	Toggles Color palette
Ctrl+Shift+S	Displays Save As dialog	Toggles symbols palette
Ctrl+Shift+V	Pastes with insertion point	Sets workplane as active view
Ctrl+1	Toggles Properties palette	...
Ctrl+2	Toggles DesignCenter palette	...
Ctrl+3	Toggles Tools palette	...
Ctrl+4	Toggles Sheetset Manager	...
Ctrl+5	Toggles Info palette	...
Ctrl+6	Toggles dbConnect Manager	...
Ctrl+7	Toggles Markup Set Manager	...
Ctrl+8	Toggles QuickCalc palette	...
Ctrl+9	Toggles command line	...
Ctrl+0	Toggles Cleanscreen mode	...

Shift Keys

AutoCAD Shift Keys	Meaning in AutoCAD	Meaning in TurboCAD
Shift+A	Overrides osnap	Displays absolute coordinates
Shift+C	Overrides CENter osnap	Snaps to center of arc
Shift+D	Overrides snap & tracking	Relocates reference point
Shift+E	Overrides ENDpoint osnap	Relocates reference point
Shift+G	...	Snaps to grid
Shift+H	...	Snaps opposite
Shift+I	...	Snaps to intersections
Shift+L	Overrides snap & tracking	Relocates origin
Shift+M	Overrides MIDpoint osnap	Snaps to midpoint
Shift+N	...	Snaps to nearest
Shift+O	...	Snaps ortho
Shift+P	Overrides ENDpoint osnap	Displays polar coordinates
Shift+Q	...	Snaps to quadrant
Shift+R	...	Displays relative coordinates
Shift+S	Enforces osnap modes	Turns object snap off
Shift+V	Overrides MIDpoint osnap	Snaps to vertex
Shift+W	...	Shows magnetic point
Shift+X	Overrides polar mode	Locks x-coordinate (.xz point filter)
Shift+Y	...	Locks y-coordinate (.yz point filter)
Shift+;	Enforces osnap modes	...
Shift+,	Overrides CENter osnap	...
Shift+'	Overrides osnap	...

Summary of TurboCAD Keyboard Shortcuts

Function Keys

TurboCAD Function Keys	Meaning
F1	Shows online help
F2	Shows Calculator palette
F3	Toggles palette
F4	Selects a rectangular area to create the aerial view
F5	Redraws the active view
F6	Selects the next object in draw order
F7	Selectsnext object in back order
Shift+F1	Displays help for clicked buttons, menus and windows
Shift+F6	Selects the next object in draw order and adds to current selection
Shift+F7	Selects next object in reverse order and adds to current selection
Ctrl+F2	Hides Color palette
Ctrl+F6	Selects the next object in draw order for node editing
Ctrl+F7	Selects the next object in reverse order for node editing

Alt Keys

TurboCAD Alt-keys	Meaning
Alt+G	Shows or hides the grid
Alt+Shift+A	Shows or hides the aperture circle
Alt+Shift+B	Creates a transferable block from the selection
Alt+Shift+C	Showsor hides the crosshair cursor
Alt+Shift+E	Breaks up a graphic into its constituent parts
Alt+Shift+F	Sets workplane by facet
Alt+Shift+G	Groups the selected entities into a single entity
Alt+Shift+I	Shows or hides the isometriccursor
Alt+Shift+R	Creates regions from the selected entities
Alt+Shift+W	Places on workplane
Alt+Shift+X	Defines a file as an external reference for the drawing
Alt+Shift+3	Draws workplane using threepoints

Control Keys

TurboCAD Ctrl-keys	Meaning
Ctrl+A	Selects the entire drawing
Ctrl+C	Copies the selection and puts it on the Clipboard
Ctrl+D	Places camera to the drawing center
Ctrl+E	Sets input focus to editbar
Ctrl+F	Opens the Properties dialog for the selected entity or tool
Ctrl+G	Starts to edit group content
Ctrl+H	Picks hyperlink
Ctrl+I	Selects all entities with a specific info string
Ctrl+K	Selects all entities with a specific color
Ctrl+L	Selects all entities on the active layer
Ctrl+M	Selects all blocks and groups with a specific name
Ctrl+N	Creates new drawings
Ctrl+O	Opens existing drawings
Ctrl+P	Prints active drawings
Ctrl+Q	Selects all entities with aquery
Ctrl+R	Sets input focus to statusbar
Ctrl+S	Saves the active drawing
Ctrl+T	Selectsall entities of specific type
Ctrl+U	Shows or hides the world coordinate system
Ctrl+V	Inserts the Clipboard contents
Ctrl+W	Sets plan view of the World User Coordinate System
Ctrl+X	Cuts the selection and puts it on the Clipboard
Ctrl+Y	Redoes the previously undone action
Ctrl+Z	Undoes the last action
Ctrl+Alt+A	Inserts the result of the Boolean addition of two solid objects
Ctrl+Alt+B	Blends the edges of a 3D entity
Ctrl+Alt+C	Sets camera parameters
Ctrl+Alt+D	Creates a shaded image of 3D wireframe
Ctrl+Alt+G	Sets the WorkPlane to that of an entity
Ctrl+Alt+H	Regenerates a 3D model with hidden lines suppressed
Ctrl+Alt+I	Inserts the result of the Boolean intersection of two solid objects
Ctrl+Alt+L	Slices a 3D entity by a plane
Ctrl+Alt+M	Switches to Model Space
Ctrl+Alt+O	Views/customizes the drawing display settings
Ctrl+Alt+P	Centers the view around defined point
Ctrl+Alt+Q	Creates a realistically shaded image of 3D wireframe
Ctrl+Alt+S	Inserts the result of the Boolean subtraction of two solid objects
Ctrl+Alt+W	Sets WorkPlane by world coordinate system (WCS)
Ctrl+Shift+B	Shows or hides the blocks palette

Ctrl+Shift+C	Shows or hides Color palette
Ctrl+Shift+D	Sets point of interest to the drawing center
Ctrl+Shift+G	Finishes editing the group content
Ctrl+Shift+H	Hatches an object or area
Ctrl+Shift+I	Showsor hides the selection info
Ctrl+Shift+N	Shows or hides Internet palette
Ctrl+Shift+O	Views/customizes the program settings.
Ctrl+Shift+P	Copies properties from one object to another
Ctrl+Shift+R	Relocates the block referencepoint
Ctrl+Shift+S	Shows or hides the symbols palette
Ctrl+Shift+T	Shows or hides the measurement info
Ctrl+Shift+U	Shows or hides the user coordinate system
Ctrl+Shift+Up	Aligns the selected entities to the top of the selection window
Ctrl+Shift+V	Sets workplaneby the active view
Ctrl+Shift+W	Sets plan view of the current User Coordinate System

Numeric Keypad

TuboCAD Numeric Keypad	Meaning
Num 2	Moves cursor one step down
Num 4	Moves cursor one step left
Num 6	Moves cursor one step right
Num 8	Moves cursor one step up
Num -	Zooms out one step about the cursor position
Num *	Sets uniform zoom for all views
Num +	Zooms in one step about the cursor position

Other Keys

TurboCAD Other Keys	Meaning
Space	Selects objects for editing
Tab	Sets input focus to editbar
Delete	Erases the selection
Esc	Cancels the current procedure
Shift+Tab	Setsinput focus to status bar
Shift+Back	Fits the page(s) in thewindow
Alt+Back	Inserts all selected elements as large as possible
Ctrl+Back	Inserts all elements as large as possible
Ctrl+Shift+Back	Selects a rectangular area to fill the window

Keyboard

TurboCAD Keyboard	Meaning
A	Sets absolute coordinates
C	Sets single snap to the center of a planar object
D	Sets single snap to the divide point of a line or arc
E	Sets single snap to the center of a 3D object
G	Sets single snap to the nearest grid point
H	Sets single snap to the opposite point
I	Sets single snap to the intersection of two objects
J	Sets single snap to the projection point
L	Relocates relative/polar origin
M	Sets single snap to the midpoint of a line
N	Sets single snap to the nearest point
O	Relocateslast point by next click
P	Sets polar coordinates
Q	Sets single snap to the quadrant point of an arc
R	Sets relative coordinates
S	Sets no snap at all, get the mouse position (click)
V	Sets single snap to the nearest vertex
W	Assigns a name to workplane or sets a named workplane
X	Locks X position
Y	Locks Yposition
Z	Locks Z position
Shift+A	Sets absolute coordinates
Shift+C	Sets snap to the center of a planar object
Shift+D	Sets snap to the division point of a line or arc
Shift+E	Sets snap to the center of a 3D object
Shift+G	Sets snap to the nearest grid point
Shift+I	Sets snap to the intersection of two objects
Shift+K	Provides snap extension apparent intersection lines
Shift+L	Relocates relative/polar origin
Shift+M	Sets snap to the midpoint of a line
Shift+N	Sets snap to the point on the nearest entity
Shift+O	Sets snap to the ortho angle
Shift+P	Sets polar coordinates
Shift+Q	Sets snap to the quadrant point of an arc
Shift+R	Sets relative coordinates
Shift+S	Sets no snap at all, get the mouse position
Shift+T	Sets snap tangent to an arc and curve
Shift+V	Sets snap to the nearest vertex
Shift+W	Shows the point on which the next snap works
Shift+X	Provides snap extension orthogonal lines

Cursor Pad

TurboCAD Cursor Pad	Meaning
Up	Scrolls view up
Down	Scrolls view down
Left	Scrolls view left
Right	Scrolls view right
Home	Scrolls view home
Shift+Up	Rotates camera left
Shift+Down	Rotates camera right
Shift+Left	Rotates camera left
Shift+Right	Rotates camera right
Shift+Page Up	Brings the selected entity(s) in front of others
Shift+Page Down	Sends the selected entity(s) behind others
Alt+Up	Sets top View
Alt+Down	Sets bottom View
Alt+Left	Sets left View
Alt+Right	Sets right View
Alt+Page Up	Sets SW isometric View
Alt+Page Down	Sets SE isometric View
Alt+Home	Sets front view
Alt+End	Sets back view
Alt+Shift+Page Down	Sets NW isometric view
Alt+Shift+Page Up	Sets NE isometric view
Ctrl+Left	Turns camera left
Ctrl+Right	Turns camera right
Ctrl+Up	Turns camera up
Ctrl+Down	Turns camera down
Ctrl+End	Pans the window to the selected point
Ctrl+Shift+Down	Aligns the selected entities to the bottom of the selection window
Ctrl+Shift+Right	Aligns the selected entities to the right of the selection window
Ctrl+Shift+Left	Aligns the selected entities to the left of the selection window
Ctrl+Shift+Home	Aligns the selected entities to the middle of the selection window
Ctrl+Shift+End	Aligns the selected entities to the center of the selection window

Appendix

TurboCAD-AutoCAD Dictionary

Understanding TurboCAD Jargon

When a TurboCAD user says.....in AutoCAD, it means:	

A

Active	Current
Add	Union
Assemble	Align
Attribute	Similar to extended entity data; TurboCAD can also attach AutoCAD-like attributes to blocks

B

Basepoint	Origin
Blend	Fillet on 3D objects
Block	Same as a block in AutoCAD; TurboCAD also uses symbols
Brush Pattern	The pattern that fills an area; TurboCAD has fills and hatches

C

Camera	View or viewpoint
Construction line	Xline; TurboCAD does not have rays

When a TurboCAD user says.....in AutoCAD, it means:	
D	
Datum	DimOrdinate
Dot	Point
Double Point	2P option, as incircles
E	
Edit bar	Direct distance entry
Editable layer	Unlocked layer
Entity	Object
F	
Format painter	MatchProp
Fit array	Array
H	
Hatch	Vector hatching pattern, as in AutoCAD; creates independent hatch objects
Hemisphere	Dome
I	
Inspector bar	Dynamic input
L	
Line style	Linetype
Local menu	Cursor menu
Locked coordinates	Point filters
M	
Magnetic point	AutoSnap magnet
Major grid	Grid
Minor grid	Subdivision of the grid
Multiline	Multi-segment line, like using Line command repeatedly
N	
Node	Handle
Not visible layer	Off, frozen layer

When a TurboCAD user says.....in AutoCAD, it means:	
O	
Order	Draw order
P	
Paper	Layout
Path Hatching	Hatching applied to an area defined by a series of picked points
Pen	Color, width, and style of lines
Pen style	Linetype
Polyline	Multi-segment line (old usage); polyline (new usage)
Properties toolbar	Object Properties toolbar
R	
Read-only layer	Locked layer
Rubber stamp	Copy multiple
S	
SEKE	Alias
Snap mode	Object snap mode
Split	Break
Style	Predefines properties for objects; whereas in AutoCAD, properties are predefined for a limited number of objects (such as text, multilines, and tables), TurboCAD can create named styles for any type of object
Sky light	Sun light
Symbol	Unlike blocks (which are referenced), symbols are copied
T	
TC Explorer	DesignCenter
Three Point Circle	Circle 3P command
Tools palette	Dashboard
U	
Unblock/Explode	Explode

When a TurboCAD user says.....in AutoCAD, it means:	
V	
Vector pan	Pan
Vector	By distance and angle
Visible layer	On, thawed layer
W	
Worldplan	WCS
Workplane	UCS
Workspace	Model space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