Chapter 21

Working with Data Exchange & Object Linking and Embedding

Learning Objectives

After completing this chapter, you will be able to:

- *Import and export .dxf files using the Save As and Open tools*
- *Import files*
- *Convert scanned drawings into the drawing editor using the DXB file format*
- *Attach the raster images to the current drawing*
- *Edit raster images*
- *Understand the embedding and linking functions of the OLE feature of Windows*
- *Use DWG Convert and CONTENT EXPLORER*

Key Terms

- *DXF Files*
- *DXB Files*
- *ACIS Files*
- *DWG Convert*
- *PDF Files*
- *3D Studio Files*
- *DGN Files*
- *BMP Files*
- *Attach Image*
- *EPS Files*
- *OLE Objects*
- *Draw Order*

UNDERSTANDING THE CONCEPT OF DATA EXCHANGE IN AutoCAD

Different companies have developed different softwares for applications such as CAD, desktop publishing, and rendering. This non-standardization of software has led to the development of various data exchange formats that enable the transfer (translation) of data from one data processing software to the other. This chapter will cover various data exchange formats provided in AutoCAD. AutoCAD uses the *.dwg* format to store drawing files. This format is not recognized by some other CAD softwares. To address this problem, AutoCAD provides various data exchange formats such as DXF (Data Interchange Format) and DXB (Drawing Interchange Binary).

CREATING DATA INTERCHANGE (DXF) FILES

The DXF file format generates a text file in ASCII code from the original drawing. This allows any computer system to manipulate (read/write) data in a DXF file. Usually, the DXF format is used for CAD packages based on microcomputers. For example, packages like SmartCAM use DXF files. Some desktop publishing packages, such as PageMaker and Ventura Publisher also use DXF files.

Creating a Data Interchange File

Using the **Save** or **Save As** tool, you can create an ASCII file with a *.dxf* extension from an AutoCAD drawing file. Once you invoke any of these tools, the **Save Drawing As** dialog box will be displayed. Next, choose the **Options** option from the **Tools** drop-down to display the **Saveas Options** dialog box, as shown in Figure 21-1. In this dialog box, the **DWG Options** tab is chosen by default. Select the required *.dxf file extension type from the **Save all drawings as** drop-down list. Next, choose the **DXF Options** tab and enter the degree of accuracy for the numeric values. The default value for the degree of accuracy is sixteen decimal places. You can enter a value between 0 and 16 decimal places.

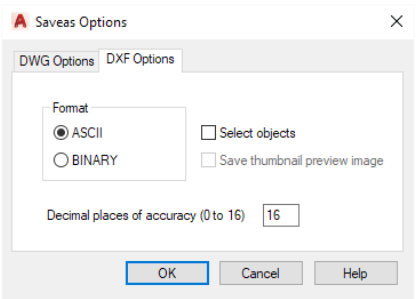

Figure 21-1 *The **Saveas Options** dialog box*

In the **DXF Options** tab, you can also select the **Select objects** check box which allows you to specify the objects to be included in the DXF file. In this case, the definitions of the named objects such as block definitions, text styles, and so on are not exported.

Select the **ASCII** radio button. Choose the **OK** button to return to the **Save Drawing As** dialog box. Choose the **Save** button here. Now, an ASCII file with a *.dxf* extension has been created and this file can be accessed by other CAD systems. This file contains data on the objects specified. By default, DXF files are created in the ASCII format. However, you can also create binary format

files by selecting the **BINARY** radio button in the **Saveas Options** dialog box. Binary DXF files are more efficient and occupy only 75 percent of the ASCII DXF file. You can access a binary format file more quickly than an ASCII format file.



Note

*You can also create a DXF file by using the **-WBLOCK** command. When you execute this command, the **Create Drawing File** dialog box will be displayed, where you can enter the name of the file in the **File name** edit box and select any version of DXF [*dxf] from the **Files of type** drop-down list. By default, the DXF file to be created assumes the name of the drawing file from which it will be created.*

Information in a DXF File

The DXF file contains data of the objects which are specified using the **Select objects** check box in the **Saveas Options** dialog box. You can change the data in this file as per your requirement. To examine the data in this file, load the ASCII file in the word processing software. A DXF file is composed of the following parts:

Header

In this part of the drawing database, all variables in the drawing and their values are displayed.

Classes

This section deals with the internal database.

Tables

All named objects such as linetypes, layers, blocks, text styles, dimension styles, and views are listed in this part.

Blocks

The objects that define blocks and their respective values are displayed in this part.

Entities

All entities in the drawing, such as lines, circles, and so on are listed in this part.

Objects

Objects in the drawing are listed in this part.

Converting DXF Files into Drawing Files

You can import a DXF file into a new AutoCAD drawing file with the **Open** tool. After you invoke the **Open** tool, the **Select File** dialog box is displayed. From the **Files of type** drop-down list, select the **DXF (*.dxf)** option. In the **File name** edit box, enter the name of the file you want to import into AutoCAD or select the file from the list. Choose the **Open** button. Once this is done, the specified DXF file is converted into a standard DWG file, regeneration is carried out, and the file is inserted into the new drawing. Now, you can perform different operations on this file just as with the other drawing files.



Tip

*A data interchange file (DXF) can also be inserted into the current drawing using the **Insert** tool.*

Importing CAD Files

Ribbon: Insert > Import > Import drop-down > Import
Menu Bar: File > Import **Command:** IMPORT



You can import any surface, solid, 2D and 3D wire geometry from the supported file formats to AutoCAD. In AutoCAD, you can import IGES, CATIA, Rhino, Pro/ENGINEER and many other file formats. To import a CAD geometry to AutoCAD, choose the **Import** tool from the **Import** drop-down of the **Import** panel; the **Import File** dialog box will be displayed. From the **Files of type** drop-down list, select the required format. Next, browse to the required file and then choose the **Open** button from the dialog box; the **Import - Processing Background Job** message box will be displayed stating that the import job is processing in the background. Next, close the message box. The details of the progress can be seen on the bottom-right corner of the screen. After the processing gets completed, a link will be displayed as pop-up. Click on the link to import the CAD geometry into AutoCAD; the inserted part geometry will be placed as a block reference.

OTHER DATA EXCHANGE FORMATS

The other formats that can be used to exchange data from one data processing format to the other are discussed next.

DXB File Format

Menu Bar: Insert > Drawing Exchange Binary **Command:** DXBIN



AutoCAD offers another file format for data interchange, DXB. This format is much more compressed than the binary DXF format and is used when you want to translate a large amount of data from one CAD system to another. For example, when you scan a drawing, a DXB file is created. This file has a huge amount of data in it. To import a DXB file, enter the **DXBIN** command at the command prompt; the **Select DXB File** dialog box will be displayed. In the **File name** edit box, enter the name of the file (in DXB format) you want to import into AutoCAD. Choose the **Open** button. Once this is done, the specified DXB file will be converted into a standard DWG file and is inserted into the current drawing.



Note

Before importing a DXB file, you must create a new drawing file. No editing or drawing setup (limits, units, and so on) can be performed on this file. This is because if you import a DXB file to an old drawing, the settings (definitions) of layers, blocks, and so on, of the file being imported are overruled by the settings of the file to which you are importing the DXB file.

Creating and Using an ACIS File

Application Menu: Export > Other Formats **Command:** EXPORT, ACISOUT

Trimmed NURBS, regions, and solids can be exported to an ACIS file with an ASCII format. To do so, choose **Export > Other Formats** from the **Application Menu**; the **Export Data** dialog box will be displayed. Enter a file name in the **File name** edit box. From the **Files of type** drop-down list, select **ACIS (*.sat)** and then choose the **Save** button; you will be prompted to select the objects. Select the solids, regions, or ACIS bodies that you want to add to the file

and press ENTER. AutoCAD appends the file extension *.sat* to the file name. You can also use the **ACISOUT** command to perform this function. It prompts you to select objects before the **Create ACIS File** dialog box is displayed on the screen. Keep in mind that only solids, regions, or ACIS bodies can be selected to add to the *.sat* file.

To import an ACIS file, choose the **Import** tool from the **Import** drop-down of the **Import** panel in the **Insert** tab; the **Import File** dialog box will be displayed. In the **Files of type** drop-down list, select the *.sat* file format. Next, select a file to be imported and choose the **Open** button; the file will be inserted. Alternatively, enter **ACISIN** at the command prompt and import the file using the **Select ACIS File** dialog box.

Importing PDF Files

Ribbon: Insert > Import > Import drop-down > PDF Import **Command:** PDFIMPORT

You can import the geometry, raster images, fills, and true type text objects from a PDF file into the current drawing as AutoCAD objects. To do so, choose the **PDF Import** tool from the **Import** drop-down of the **Import** panel in the **Insert** tab; the **Select PDF File** dialog box will be displayed. Select a PDF file to import and choose the **Open** button; the **Import PDF** dialog box will be displayed, as shown in Figure 21-2.

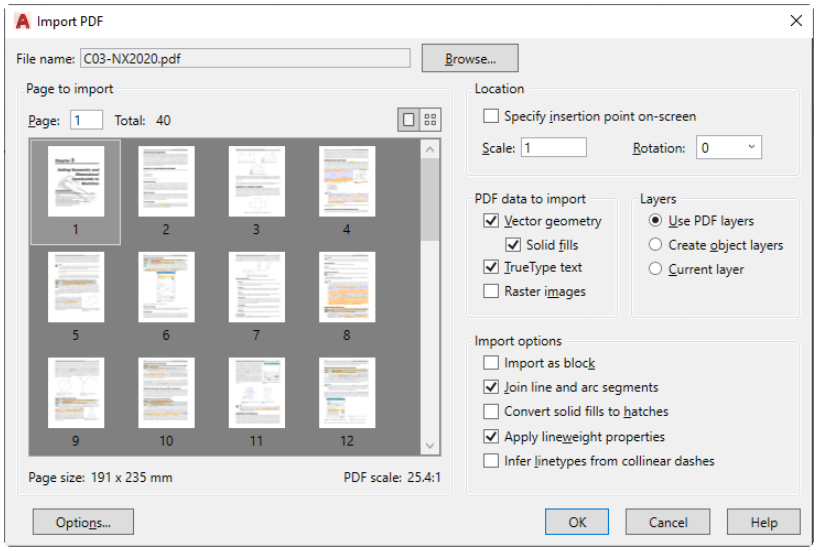

Figure 21-2 *The* ***Import PDF*** *dialog box*

Enter the page number in the **Page** edit box under the **Page to import** area of the **Import PDF** dialog box. To specify the scale, rotation and point of insertion of the imported geometry choose the respective options under the **Location** area. You can also choose the type of data to be imported using the options available in the **PDF data to import** area. To specify the layers in which the PDF data is to be imported, choose the options under the **Layers** area. The options under the **Import options** area are discussed next.

You can join 2D solids with the coincident edges and convert them to hatch objects by choosing the **Convert solid fills to hatches** check box. Choose the **Apply lineweight properties** check box

to apply the lineweights of the PDF to the extracted geometry. You can choose the **Infer linetypes from collinear dashes** check box to collect collinear dash and dot segments and replace them with a single polyline. If you uncheck this box, each segment in a dashed line is imported as a separate polyline. Choose the **Options** button at the bottom of the **Import PDF** dialog box; the **Options** dialog box will be displayed. In the **Options** dialog box, you can specify the file location for image files that are extracted from the PDF and attached to the drawing.

Alternatively, you can import different objects from the previously attached PDF by entering the **PDFIMPORT** command at the command bar. On doing so, you will be prompted to select PDF underlay. Select an attached PDF from the drawing area; you will be prompted to specify a rectangular or polygonal boundary around the objects that you want to import or choose the **All** option from the command prompt to import all the objects in the overlay. Choose the **Settings** option from the command prompt to invoke the **PDF Import Settings** dialog box, as shown in Figure 21-3. The options in this dialog box are same as discussed for the **Import PDF** dialog box. After selecting the boundary you can choose the options in the command prompt to keep, detach, or unload the PDF underlay after the objects are imported into the current drawing.

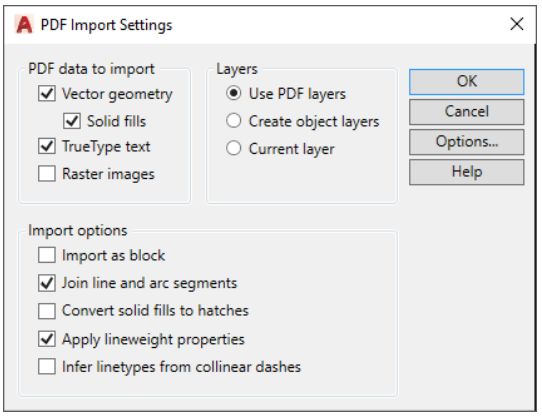

Figure 21-3 *The* ***PDF Import Settings*** *dialog box*

While importing a PDF file, it is possible that some of the text strings are not recognized by AutoCAD. To make this text recognized by AutoCAD, choose the **Recognize SHX Text** tool from the **Import** panel; you will be prompted to select the unrecognized text. Select the text stamps that were previously not recognized by AutoCAD; the selected text stamps will be recognized. This tool recognizes the text by comparing its geometry with some pre-defined text styles. You can also set the list of text styles by using the **PDF Text Recognition Settings** dialog box. To invoke this dialog box, choose the **Recognition Settings** tool from the **Import** panel; the **PDF Text Recognition Settings** dialog box will be displayed, as shown in Figure 21-4. You can select any text style and add or remove more text styles by choosing required options from the **SHX fonts to compare** area of this dialog box.

In the **Create text on** area, you can create the recognized text on the set layer or on the same layer by selecting the **Current layer** or **Same layer as geometry** radio buttons, respectively. Also, you can set the percentage for matching geometry to the defined text style from the **Recognition threshold** slider in the **Conversion settings** area.

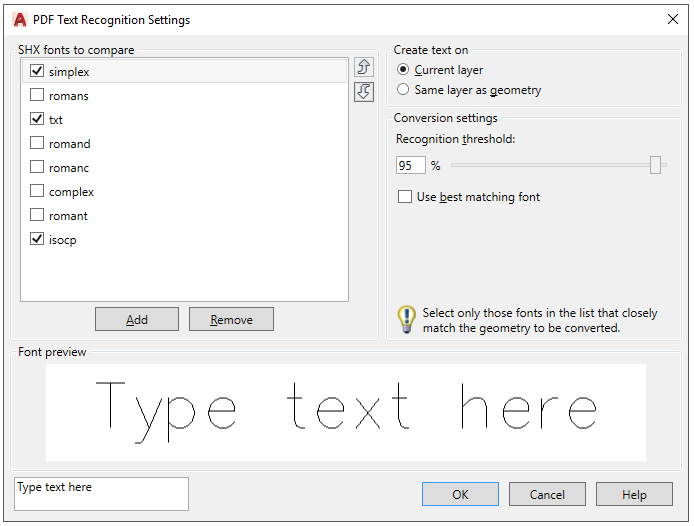

Figure 21-4 *The* ***PDF Text Recognition Settings*** *dialog box*

Tip

You can also access the PDF import options by choosing the ***Import as Objects*** *tool in the* ***PDF Import*** *panel of the* ***PDF Underlay*** *contextual tab which will be displayed on selecting the object.*

Importing 3D Studio Files

Menu Bar: Insert > 3D Studio **Command:** 3DSIN

You can import 3D geometry, views, lights, and objects with surface characteristics created in 3D Studio and saved in *.3ds* format. To do so, choose the **3D Studio** tool from the **Insert** menu; the **3D Studio File Import** dialog box will be displayed. Select the file you wish to import and choose the **Open** button. You can use the **3DSIN** command to import 3D Studio file.

Creating and Using a Windows Metafile

Application Menu: Export > Other Formats **Command:** EXPORT, WMFOUT

The Windows Metafile format (WMF) file contains screen vector and raster graphic formats. In the **Export Data** dialog box or in the **Create WMF File** dialog box, enter the file name. Select **Metafile (*.wmf)** from the **Files of type** drop-down list. The extension *.wmf* is appended to the file name. Next, save the settings and then select the objects you want to save in this file format.

The **WMFIN** command invokes the **Import WMF** dialog box. Select the *.wmf* file you want to import and choose the **Open** button. Specify an insertion point, rotation angle, and scale factor. Specify the scaling by entering a scale factor using the **Corner** option to specify an imaginary box whose dimensions correspond to the scale factor, or by using the **X**, **Y**, or **Z** option to specify 3D scale factors. You can also import a *.wmf* file by using the **Import File** dialog box that will be displayed on choosing the **Import** tool from the **Import** panel in the **Insert** tab.

Creating and Using a V8 DGN File

Application Menu: Export > DGN **Command:** EXPORT, DGNEXPORT

The **V8 DGN** (*.DGN*) file format is used by the Microstation software. A **.dwg* file created in AutoCAD can be exported to Microstation using this format. To do so, choose **Export > DGN** from the **Application Menu**; the **Export DGN File** dialog box will be displayed. Enter a file name in the **File name** edit box. From the **Files of type** drop-down list, select **V8 DGN (*.dgn)** and then choose the **Save** button; the **Export DGN Settings** dialog box will be displayed. In this dialog box, you can specify how the externally referenced files in the current drawing are taken care of while converting the current drawing into the DGN file. You can also specify the seed files (standard template files containing the settings for the units to be used for conversion, working units, resolution of the drawing, and so on) to be used for conversion in this dialog box.

To import a DGN file, choose **Open > DGN** from the **Application Menu**; the **Import DGN File** dialog box will be displayed. Alternatively, you can use the **DGNIMPORT** command to import a DGN file. From the **Files of type** drop-down list, select the **MicroStation DGN (*.dgn)** option, if it is not selected by default. Next, select or enter the name of the design file and choose the **Open** button; the **Import DGN Settings** dialog box will be invoked. You can exchange the data of the drawing files and design files using the **Export/Import DGN Settings** dialog box. For example, you can translate DGN elements to DWG objects, Levels to Layers, Line Styles to Linetypes, Cells to Blocks, Sheet Model to DWG Layouts, etc. You can save and reuse, modify, rename or delete the mapping translations using the **DGNMAPPING** command.

Creating a BMP File

Application Menu: Export > Other Formats **Command:** EXPORT, BMPOUT

This is used to create bitmap images of the objects in your drawing. In the **Export Data** dialog box, enter the name of the file, select **Bitmap (*.bmp)** from the **Files of type** drop-down list and then choose the **Save** button. Select the objects you wish to save as bitmap and press ENTER. Entering **BMPOUT** displays the **Create Raster File** dialog box. Enter the file name and choose the **Save** button. Select the objects to be saved as bitmap. The file extension *.bmp* is appended to the file name.

RASTER IMAGES

A raster image consists of small square-shaped dots known as pixels. In a colored image, the color is determined by the color of pixels. The raster images can be moved, copied, or clipped. They will used as a cutting edge with the **Trim** tool. They can also be modified by using grips. You can also control the image contrast, transparency, and quality of the image. AutoCAD stores images in a special temporary image swap file whose default location is the Windows Temp directory. You can change the location of this file by modifying it under **Temporary Drawing File Location** in the **Files** tab of the **Options** dialog box.

The images can be 8-bit gray, 8-bit color, 24-bit color, or bitonal. When the image transparency is set to On, the image file formats with transparent pixels is recognized by AutoCAD and transparency is allowed. The transparent images can be in color or grayscale. AutoCAD supports many formats and some of them are given next.

Image Type	File Extension	Description
BMP	*.bmp*	Windows and OS/2 Bitmap Format
JFIF or JPEG	*.jpg*, *jpeg*	Joint Photographics Expert group
PCX	*.pcx*	Picture PC Paintbrush Picture
PNG	*.png*	Portable Network Graphic
TARGA	*.tga*	True Vision Raster based Data format
TIFF/LZW	*.tif, .tiff*	Tagged Image File Format

When you store images as Tiled Images, that is, in the Tagged Image File Format (TIFF), you can edit or modify any portion of the image; only the modified portion is regenerated, thus saving time. Tiled images load much faster as compared to the untiled images.

Attaching Raster Images

Ribbon: Insert > Reference > Attach **Tool Palettes:** Draw > Attach Image
Toolbar: Reference > Attach Image or Insert > Attach Image **Command:** IMAGEATTACH



Attaching raster images to a drawing does not make them part of the drawing. To attach an image, choose the **Attach** tool from the **Reference** panel; the **Select Reference File** dialog box will be displayed, as shown in Figure 21-5. Select the image to be attached; a preview of the selected image will be displayed in the **Preview** area of the dialog box. Next, choose the **Open** button; the **Attach Image** dialog box will be displayed, as shown in Figure 21-6. The name of the selected image is displayed in the **Name** drop-down list. You can select another file by using the **Browse** button. The **Name** drop-down list displays the names of all the images in the current drawing. Select the **Specify on-screen** check boxes to specify the **Insertion point**, the **Scale**, and **Rotation** angle on the screen. Alternatively, you can clear these check boxes and enter values in the respective edit boxes. On choosing the **Show Details** button, the **Attach Image** dialog box is expanded and it provides the image information such as the Horizontal and Vertical resolution values, current AutoCAD unit, and Image size in pixels and units. Choose the **OK** button to return to the drawing screen.

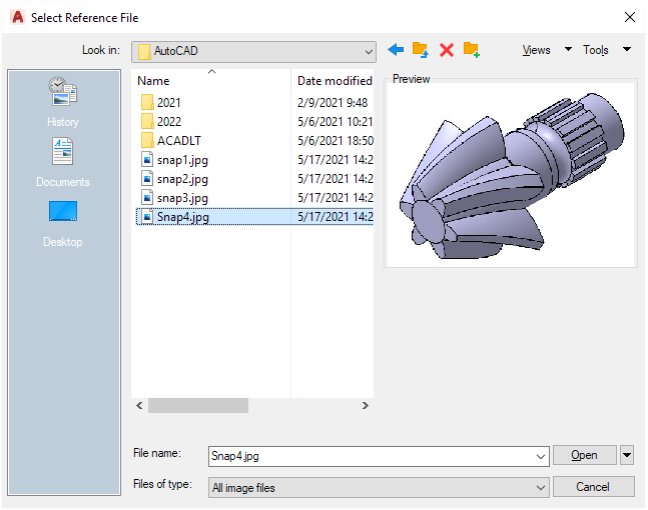

Figure 21-5 *The* ***Select Reference File*** *dialog box*

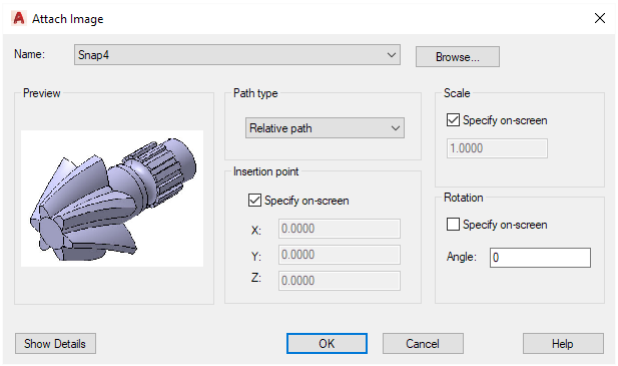

Figure 21-6 *The* ***Attach Image*** *dialog box*

You can also attach and scale an image from the Internet. In the **Select Reference File** dialog box, choose the **Search the Web** button. Once you have located the image file you wish to attach to the current drawing, enter the URL address in the **Look in** drop-down list and the image file name in the **Name or URL** edit box. Choose the **Open** button. Specify the insertion point, scale, and rotation angle in the **Attach Image** dialog box. You can also right-click on the image you wish to attach to the current drawing and choose **Properties** from the shortcut menu to display the **Properties** palette. From here, you can select, cut, and paste the URL address in the **File name** edit box of the **Select Reference File** dialog box.

Managing Raster Images

Ribbon: View > Palettes > External References Palette **Command:** IMAGE
Toolbar: Reference > External References **Menu Bar:** Insert > External References

Choose the **External References Palette** button from the **Palettes** panel; the **EXTERNAL REFERENCES** palette will be displayed, refer to Figure 21-7. If the image has not been inserted earlier, right-click in the **File References** area and choose the **Attach Image** option

from the shortcut menu. You can view the image information either as a list or as a tree view by choosing the respective buttons located at the right corner of the **File References** head of the **EXTERNAL REFERENCES** palette. The **List View** displays the names, loading status, size, date last modified on, and search path of all images in the drawing. The **Tree View** displays images in hierarchal structure. These images show its nesting levels within blocks and Xrefs. The **Tree View** does not display the status, size, or any other information about an image file. You can rename an image file in this dialog box.

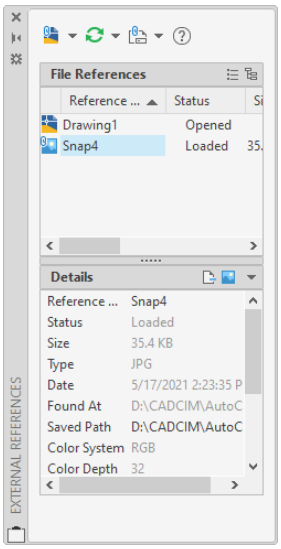

***Figure 21-7** The **EXTERNAL REFERENCES** palette*

Preview

To display the **Preview** area, choose the **Preview** button located at the right corner of the **Details** area. In this area, you can see the preview of the image. This area displays the preview of the image file attached to the current drawing and the file details of the attached image file. To get the file details, choose the **Details** button located adjacent to the **Preview** button. The information about the image file such as file name, saved path, and so on will be listed, as shown in Figure 21-8. Once you have referenced an image file to the current drawing, you can replace it with another image file by using the **Saved Path** edit box. To do so, choose the Browse [...] button in the corner of the edit box to display the **Select Image File** dialog box. Browse and select the image file and then choose the **Open** button. Now, the new path will be displayed in the **Found At** and **Saved Path** edit boxes.

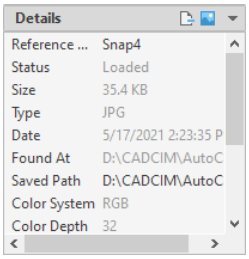

***Figure 21-8** The **Details** area of the **EXTERNAL REFERENCES** palette*



Tip

*You can also use the command prompt for managing the image files using the **-IMAGE** command. Except the **Open** option, all the other options of the **EXTERNAL REFERENCES** palette will be available in the command prompt.*

When you select the name of the attached image and right-click on it, a shortcut menu will be displayed. The options in this shortcut menu are discussed next.

Open
This option enables you to open the attached image in its default viewing software from its current location in the AutoCAD file.

Attach
This option allows you to attach an image file with the current drawing. You can also attach an image file by choosing the **Attach Image** option from the flyout on the upper left corner of the **EXTERNAL REFERENCES** palette; the **Select Reference File** dialog box will be displayed. Select the file that you want to attach to the drawing and the selected image is displayed in the **Preview** box. Choose the **Open** button; the **Attach Image** dialog box is displayed. The name of the file, its path, and extension are displayed in the **Attach Image** dialog box. You can use the **Browse** button to select another file. Choose the **OK** button in the **Attach Image** dialog box and specify a point to insert the image. Next, AutoCAD will prompt you to enter the scale factor. You can enter the scale factor or specify a point on the screen.

Detach
This option detaches the selected and all associated raster images from the current drawing. Once the image is detached, no information about the image is retained in the drawing.

Reload
The **Reload** option simply reloads an image. The changes made to the image since the last insert will be loaded on the screen.

Unload
The **Unload** option unloads an image. An image is unloaded when it is not needed in the current drawing session. When you unload an image, AutoCAD retains information about the location and size of the image in the database of the current file. Unloading a file does not unlink the file from the drawing and the image frame is displayed. If you reload the image, the image will appear at the same point and in the same size as the image was before unloading. Unloading the raster images enhances AutoCAD performance. Also, the unloaded images are not plotted. If multiple images are to be loaded and the memory is insufficient, AutoCAD automatically unloads them.

Change Path Type
The **Change Path Type** option is used to change the path type from where the image can be imported. You can set three types of paths by selecting the desired option from the **Change Path Type** cascading menu invoked by clicking on the **Change Path Type** option in the shortcut menu. The options in this cascading menu are discussed next.

Make Absolute: This option is used to set full path of the attached image which includes drive, url, or network server location.

Make Relative: This option is used to set most flexible path because it does not save drive name. Therefore, by using this option, you can easily move your file in different drawings with same folder structure without breaking the reference.

Remove path: This option is used when you do not want to set any path. On selecting this option, you can move your drawing wherever you want without facing any problem.

Select New Path

The **Select New Path** option is used to change the current path of the attached image whose referenced path has been changed. Also, you can attach a different image by using this option.

Find and Replace

This option is used to simultaneously change the path of all the attached images having same path from one location to another.

EDITING RASTER IMAGE FILES

Raster image files can be easily edited. The different methods to edit a raster image files are discussed next.

Clipping Raster Images

Ribbon: Insert > Reference > Clip **Toolbar:** Reference > Clip Image
Menu Bar: Modify > Clip > Image **Command:** IMAGECLIP



The **Clip** tool is used to clip the boundaries of images and to provide a desired shape to them. To invoke this tool, select the image to be clipped; the **Image** tab will be added to the **Ribbon**. Choose the **Create Clipping Boundary** tool; you will be prompted to specify the boundary. Specify a rectangular or polygonal boundary; the image will be clipped at the specified boundary. Also, an arrow will be displayed pointing inside the clipping boundary. Click on the arrow to flip the clipping direction.

You can also invoke this tool by selecting and right-clicking on an image to be clipped and choose **Image > Clip** from the shortcut menu displayed.

After choosing this tool, AutoCAD will prompt you to select the image to be clipped. Select the raster image by selecting the edge of the image. The image boundary must be visible to select it. The clipping boundary must be specified in the same plane as the image or a plane that is parallel to it. The following prompt sequence is displayed when you choose the **Clip** tool from the **Reference** panel.

Select object to clip: *Select the boundary of an image.*
Enter image clipping option [ON/OFF/Delete/New boundary] <New>:

New boundary

This option is used to define a boundary for clipping the image. When you invoke this option, you are prompted to specify whether or not you want to delete the old boundary, if a boundary already exists. If you enter **No** at the **Delete old boundary? [No/Yes] <Yes>** prompt, this command will end. You can enter **Yes** to specify the new clipping boundary. The boundary can be defined in a rectangular shape or a polygonal shape.

ON
This option is used to turn the image clipping on. The image will be clipped using the last boundary.

OFF
This option is used to turn the image clipping off. If you have a clipped image, you can redisplay the entire image.

Delete
This option is used to delete the existing clipping boundary.

One such clipping is shown in Figures 21-9 and 21-10. Figure 21-9 shows the actual image and Figure 21-10 shows the clipped image where the clipping boundary is created by using the midpoints of all the four sides of the image.

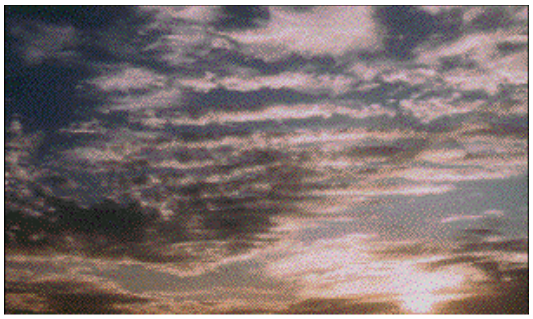

Figure 21-9 *Image before clipping*

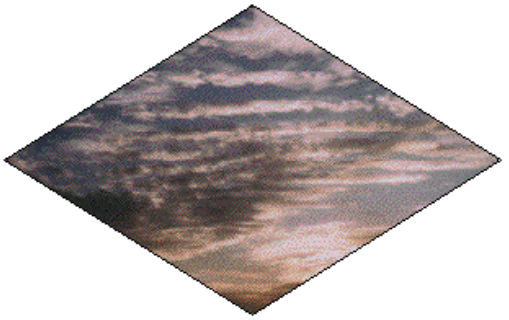

Figure 21-10 *Image after clipping*

Tip

You can improve the performance by turning off the highlighting of an image. This can be done by clearing the ***Highlight raster image frame only*** *check box in the* ***Display*** *tab of the* ***Options*** *dialog box. You can also use the* ***IMAGEHLT*** *system variable. By default,* ***IMAGEHLT*** *is set to 0, that is, only the raster image frame will be highlighted.*

Adjusting Raster Images

Ribbon: Insert > Reference > Adjust
Toolbar: Reference > Adjust Image
Menu Bar: Modify > Object > Image > Adjust
Command: IMAGEADJUST/ADJUST

The **Adjust** tool allows you to adjust the brightness, contrast, and fade of a raster image. On selecting an attached image, the **Image** tab will be added to the **Ribbon**. Adjust the brightness, contrast, and fade using the corresponding option in the **Adjust** panel of the **Image** tab.

Alternatively, choose the **Adjust Image** tool from the **Reference** toolbar; you will be prompted to select the image. Select the attached image; the **Image Adjust** dialog box will be displayed, as shown in Figure 21-11. This dialog box is used to adjust an image. As you adjust the brightness, contrast, or fade, the changes are dynamically made in the image. If you choose the **Reset** button, the values will be returned to the default values (Brightness= 50, Contrast= 50, and Fade= 0).

You can also choose **Properties** in the shortcut menu to display the **PROPERTIES** palette. In this palette, you can enter new values of brightness, contrast, and fade. If you choose the **Adjust** tool from the **Reference** panel, you need to set the values in the command prompt.

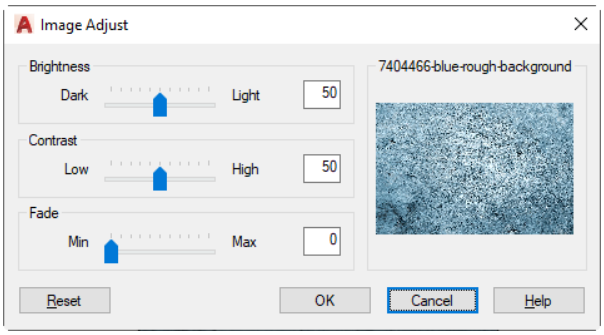

Figure 21-11 *The* ***Image Adjust*** *dialog box*

Modifying the Image Quality

Menu Bar: Modify > Object > Image > Quality **Command:** IMAGEQUALITY
Toolbar: Reference > Image Quality

The **Quality** tool allows you to control the quality of the image that affects the display performance. To control the quality, choose the **Quality** tool from **Modify > Object > Image** in the Menu bar; you will be prompted to specify the quality of image. A high-quality image takes a longer time to display. When you change the quality, the display changes immediately without causing a **REGEN**. The images are always plotted using a high-quality display. Although draft quality images appear grainier, they are displayed more quickly.

Modifying the Transparency of an Image

Toolbar: Reference > Image Transparency **Command:** TRANSPARENCY
Menu Bar: Modify > Object > Image > Transparency

When you attach an image with transparent background to a drawing, the transparent background turns opaque. To control the transparency of the attached image, select it; the **Image** tab will be added to the **Ribbon**. Choose the **Background Transparency** button from the **Image** tab to turn on the transparency of the image; the background of the image will become transparent. The **TRANSPARENCY** command can also be used to control the transparency of an image. Figure 21-12 shows two images one over the other with the transparency of the top image turned off. Figure 21-13 shows the figure with the transparency of the top image turned on. You can also right-click on an image and choose **Properties** from the shortcut menu displayed. In the **PROPERTIES** palette, select an option from the **Background transparency** drop-down list displayed under the **Misc** list.

Note

You have to choose images in such a way that they can give the effect of transparency. Dull and opaque images will not give the effect of transparency.

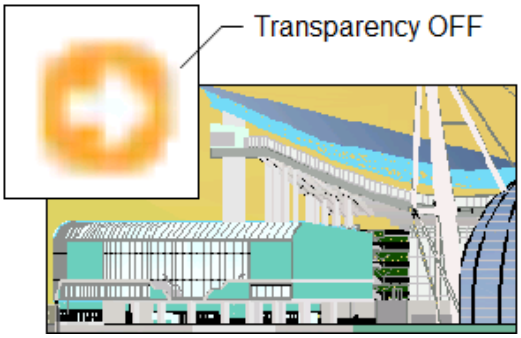

Figure 21-12 Image with the transparency off

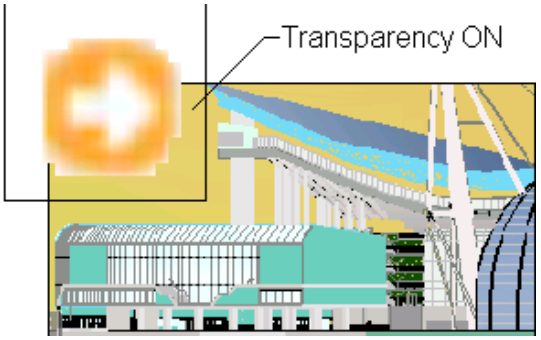

Figure 21-13 Image after turning the transparency on

Controlling the Display of Image Frames

Menu Bar: Modify > Object > Image > Frame **Command:** IMAGEFRAME
Toolbar: Reference > Image Frame

The **FRAME** tool is used to turn the image boundary on or off. This tool can be invoked from **Modify > Object > Image** menu. If the image boundary is off, the image cannot be selected with the pointing device and therefore cannot be accidentally moved or modified.

CHANGING THE DISPLAY ORDER

Ribbon: Home > Modify > Draw Order drop-down
Command: DRAWORDER/DR **Toolbar:** Draw Order

The tools in the **Draw Order** drop-down, refer to Figure 21-14, of the **Modify** panel are used to change the display order of the selected images and other objects.

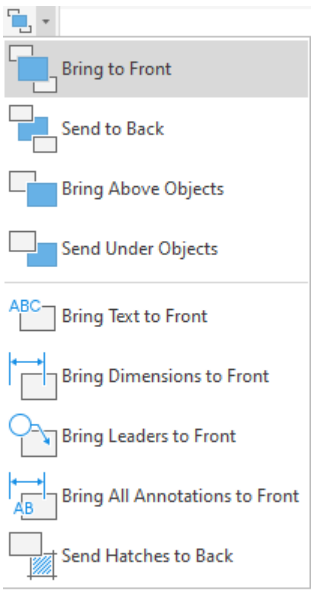

*Figure 21-14 Tools in the **Draw Order** drop-down*

If you choose the **Bring to Front** tool, the selected object will be moved to the front of all entities. If you choose the **Send to Back** tool, the selected object will be moved behind all entities. If you choose the **Bring Above Objects** tool, the selected object will be moved above the referenced objects. If you choose the **Send Under Objects** tool, the selected object will be moved under the referenced objects.

The **Bring Text to Front** tool can be used to bring entire text to the front of all entities. If you choose **Bring Dimensions to Front**, all dimensions will come to front of all the entities. The **Send Hatches to Back** tool is used to send all the hatches to the back of all entities.

Other Editing Operations

You can also perform other editing operations such as copy, move, and stretch to edit a raster image. Remember that

you can also use the image as the trimming edge for trimming objects. However, you cannot trim an image. You can insert the raster image several times or make multiple copies of it. Each copy can have a different clipping boundary. You can also edit the image using grips. You can use the **Properties** palette to change the image layer, boundary linetype, and color, and perform other editing commands such as changing the scale, rotation, width, and height by entering new values in the respective edit boxes. You can also display an image or turn off the display by selecting options from the **Show image** drop-down list in the **Misc** field of the **Properties** palette. If you select **Yes**, the image is displayed and if you select **No**, the display of the image is turned off. You can turn off the display of the image when you do not need it, thus improving the performance.

Scaling Raster Images

The scale of the inserted image is determined by the actual size of the image and the unit of measurement (inches, feet, and so on). For example, if the image is 1" X 1.26" and you insert this image with a scale factor of 1, its size on the screen will be 1 by 1.26 AutoCAD units. If the scale factor is 5, the image will be five times larger. The image that you want to insert must contain the resolution information (DPI). If the image does not contain this information, AutoCAD treats the width of the image as one unit.

DWG Convert

Application Menu: Save As > DWG Convert **Command:** DWGCONVERT
Menu Bar: File > DWG Convert



The **DWG Convert** tool is used to convert a single or batch of the AutoCAD drawing files into any older version of AutoCAD drawing format. You can do the conversion by following the procedure given next.

Invoke the **DWG Convert** dialog box by choosing the **DWG Convert** tool from the **File** menu; the **DWG Convert** dialog box will be displayed, as shown in Figure 21-15. Choose the **Add file** button from the bottom of the **Files Tree** area; the **Select File** dialog box will be displayed. Next, browse to the required location, select the required file/files, and then choose the **Open** button; the selected files will be displayed in tree form in the **Files Tree** area of the **DWG Convert** dialog box. Also, a check box will be displayed on the left of each drawing file. By default, all the check boxes are selected. You can clear the check box corresponding to the files that are not to be converted. You can also view the selected files along with their details such as their path and location, version, size, date, and so on in the form of a table by choosing the **Files Table** tab. If you want to save the current file list, then choose the **Save list** button; the **Save Conversion List** dialog box will be displayed. Save the file list in the desired folder.

You can select the version of output file from the **Select a conversion setup** area; the description of the conversion setup will be displayed in the **Setup description** area. You can also specify the conversion settings, if required, by choosing the **Conversion Setups** button. This is discussed in the next head. Now, choose the **Convert** button located at the lower right corner of the dialog box; the **Conversion Package Creation is in Progress** window will be displayed and in a few moments the files will be converted. Exit the dialog box by choosing the **Close** button.

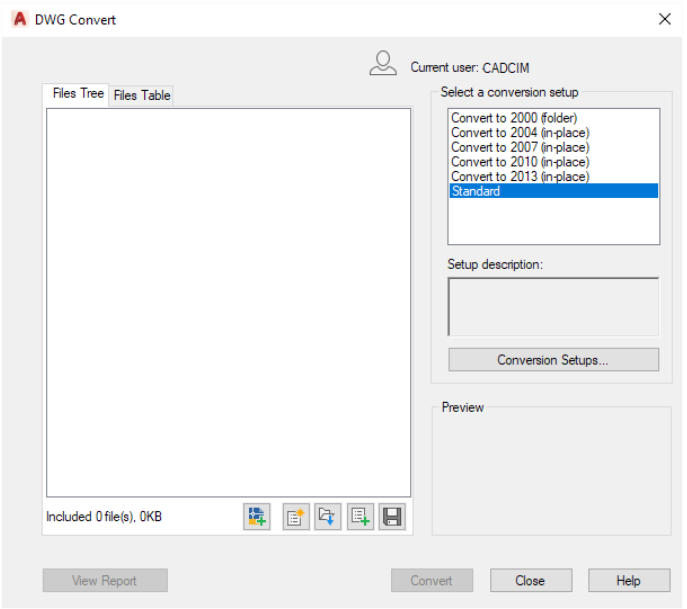

Figure 21-15 *The* ***DWG Convert*** *dialog box*

You can also open any previously saved file list by using the **Open list** button. If you choose this button, the **Open Conversion List** dialog box will be displayed. Browse to the required file and choose the **Open** button; all files in the conversion list will be imported in the **Files Tree** area of the **DWG Convert** dialog box. Later on, you can replace this list with a new list using the **Append list** button.

Conversion Setup Options

The **Conversion Setups** options are used to control the output of the files to be converted. To modify the conversion settings, choose the **Conversion Setups** button from the **DWG Convert** dialog box; the **Conversion Setups** dialog box will be displayed, as shown in Figure 21-16. The options in this dialog box are discussed next.

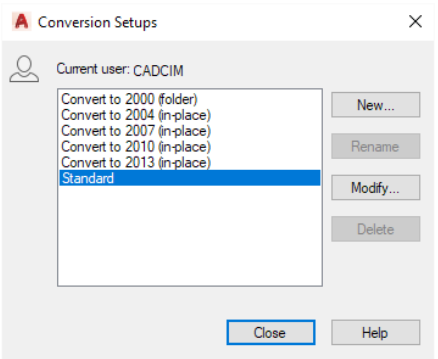

Figure 21-16 *The* ***Conversion Setups*** *dialog box*

New

This button is used to create a new conversion scheme. On choosing this button, the **New Conversion Setup** dialog box will be displayed, as shown in Figure 21-17. You can enter the name of new conversion setup in the **New Conversion setup name** edit box. Next, select an option from the **Based on** drop-down list. Finally, choose the **Continue** button; the **Modify Conversion Setup** dialog box will be displayed. This dialog box is discussed later in this chapter.

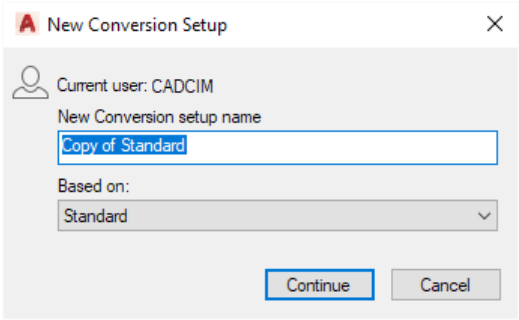

Figure 21-17 *The* ***New Conversion Setup*** *dialog box*

Rename

This button is used to rename the selected conversion setup.

Modify

This button is used to modify the existing conversion settings according to the desired output. On choosing the **Modify** button, the **Modify Conversion Setup** dialog box will be displayed, as shown in Figure 21-18. Using this dialog box, you can modify conversion settings. The options in this dialog box are discussed next.

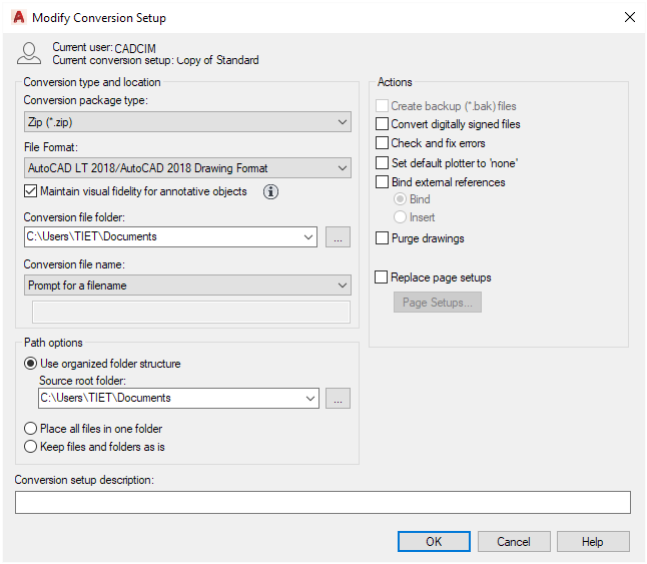

Figure 21-18 *The* ***Modify Conversion Setup*** *dialog box*

Conversion type and location Area

The options under the **Conversion package type** drop-down list in this area are discussed next.

In-place (overwrite files): This option is used if you want the converted files to be in the same directory, in which the source files are present. If you want to overwrite the existing files, this option also allows the software to overwrite the previous files with the converted new files.

Folder (set of files): This option is used if you want the converted files to be saved in the specified directory. On selecting this option, the **Conversion file folder** drop-down list will be activated. Using this drop-down list, you can browse to the desired folder where you want to save the converted files.

Zip (*.zip): This option is used to bind all converted drawing files into a zip file, which can be uncompressed to the specific location later.

Path options Area

The **Path options** area will be enabled when you select the **Folder (set of files)** or **Zip (*.zip)** option from the **Conversion package type** drop-down list. In the **Path options** area, there are three radio buttons that are explained next.

Use organized folder structure: On selecting this radio button, you can store the converted files in the specified folder in same organized way as they are present in the source folder (in **Source root folder** drop-down).

Place all files in one folder: On selecting this radio button, you can save all converted files in one folder which is specified as output folder.

Keep files and folder as is: On selecting this radio button, the files are saved in the way they were originally present. In other word, the hierarchy remains the same as of original file location.

Actions Area

In the **Actions** area, there are 8 check boxes. Description of these check boxes is given next.

Create backup(*.bak) files: This check box will be enabled only when the **In-place (overwrite files)** conversion package type is selected. If you select this check box, then the backup file will also be created with the converted file.

Convert digitally signed files: On selecting this check box, the digitally signed files will be included in the conversion list.

Check and fix errors: If you select this check box, the errors will automatically be fixed during the conversion.

Set default plotter to 'none': If you select this check box, the plotter for all converted files will be set to **None**.

Bind external references: You can bind the external referenced files with the converted drawing files using this check box. This option has two radio buttons, **Bind** and **Insert**. **Bind** is used to bind the external referenced file as external object. **Insert** is used to bind the external reference as internal object of the drawing.

Purge drawings: Select this check box if you want to purge the drawings.

Replace page setups: Select this check box if you want to replace the page setup of the converted drawing files with a pre-stored template. To get a pre-stored template, choose the **Page Setups** button after selecting the check box; the **Page Setups** dialog box will be displayed. Choose the **Browse** button, and then select a pre-stored template.

Conversion setup description Area

This area is located at the bottom of the dialog box. You can enter the description of the conversion setup in the **Conversion setup description** edit box in this area.

Delete

This button is used to delete any of the conversion setup in the list.

WORKING WITH POSTSCRIPT FILES

Application Menu: Export > Other Formats **Menu Bar:** File > Export
Command: EXPORT, PSOUT

PostScript is a page description language developed by Adobe Systems. It is mostly used in DTP (desktop publishing) applications. AutoCAD allows you to work with PostScript files. You can create and export PostScript files from AutoCAD, so that these files can be used for DTP applications. PostScript images have a higher resolution than raster images. The extension for these files is *.eps* (Encapsulated PostScript).

Creating PostScript Files

As just mentioned, any AutoCAD drawing file can be converted into a PostScript file. This can be accomplished by using the **PSOUT** command. When you invoke this command, the **Create PostScript File** dialog box is displayed, as shown in Figure 21-19. You can also use the **Export** tool to display the **Export Data** dialog box. The options are similar in both the dialog boxes.

In the **File name** edit box, enter the name of the PostScript (EPS) file you want to create. Select **Encapsulated PS (*.eps)** from the **Files of type** drop-down list. Next, choose the **Save** button to accept the default setting and create the PostScript file. You can also choose the **Options** option from the **Tools** flyout of the **Create PostScript File** dialog box to change the settings through the **PostScript Out Options** dialog box and then save the file. The **PostScript Out Options** dialog box, as shown in Figure 21-20, provides the options given next.

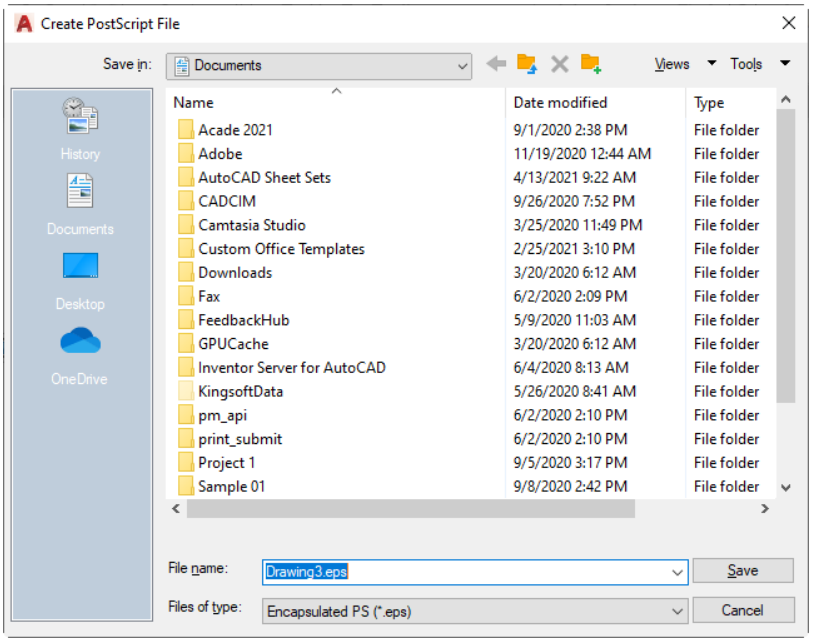

Figure 21-19 *The* ***Create PostScript File*** *dialog box*

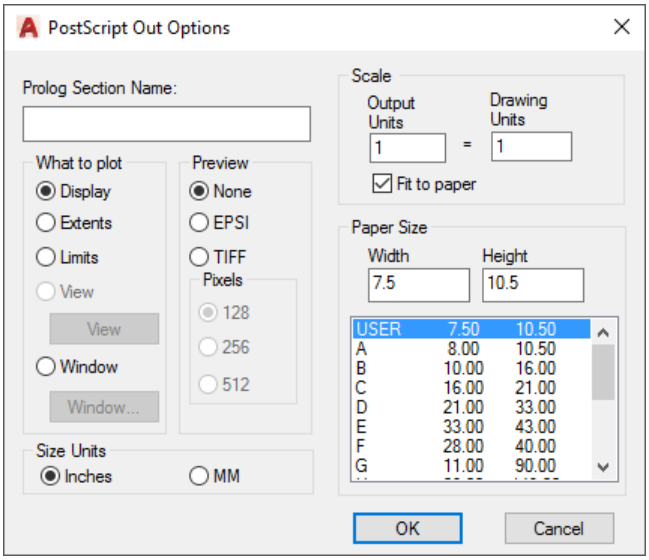

Figure 21-20 *The* ***PostScript Out Options*** *dialog box*

Prolog Section Name

In this edit box, you can assign a name for a prolog section to be read from the *acad.eps* file.

What to plot Area

The **What to plot** area of the dialog box has the following options:

Display: If you select this radio button when you are in the model space, the display in the current view is saved in the specified EPS file. Similarly, if you are in the layouts, the current viewport is saved in the specified EPS file.

Extents: If you use this option, the PostScript file will contain the section of the AutoCAD drawing that currently holds objects. In this way, this option is similar to the **Zoom Extents** option. If you add objects to the drawing, they are also included in the PostScript file to be created because the extents of the drawing are also altered. If you reduce the drawing extents by erasing, moving, or scaling objects, then you must use the **Zoom Extents** or **Zoom All** option. Only then does the **Extents** option of the **PSOUT** command calculate the extents of the drawing to be exported. If you are in the model space, the PostScript file is created in relation to the model space extents; if you are in the paper space, the PostScript file is created in relation to the paper space extents.

Limits: With this option, you can export the whole area specified by the drawing limits. If the current view is not the plan view [viewpoint (0,0,1)], the **Limits** option exports the area just as the **Extents** option would.

View: Any view created with the **VIEW** command can be exported with this radio button. This radio button will be available only when you have created a view using the **VIEW** command. Select the **View** radio button to invoke the **View Name** dialog box from where you can select the view.

Window: In this option, you need to specify the area to be exported with the help of a window. When this radio button is selected, the **Window** button is also available. Choose the **Window** button to display the **Window Selection** dialog box where you can choose the **Pick** button and then specify the two corners of the window on the screen. You can also enter the coordinates of the two corners in the **Window Selection** dialog box.

Preview Area

The **Preview** area of the dialog box has two types of formats to preview images: **EPSI** and **TIFF**. If you want a preview image with no format, select the **None** radio button. If you select **TIFF** or **EPSI**, you are required to enter the pixel resolution of the screen preview in the **Pixels** area. You can select a preview image size of 128 X 128, 256 X 256, or 512 X 512.

Size Units Area

In this area, you can set the paper size units to **Inches** or **MM** by selecting their corresponding radio buttons.

Scale Area

In this area, you can set an explicit scale by specifying how many drawing units are to be output per unit. You can select the **Fit to paper** check box so that the view to be exported is made as large as possible for the specified paper size.

Paper Size Area

You can select a size from the list or enter a new size in the **Width** and **Height** edit boxes to specify a paper size for the exported PostScript image.

OBJECT LINKING AND EMBEDDING (OLE)

With Windows, it is possible to work with different Windows-based applications by transferring information between them. You can edit and modify the information in the original Windows application, and then update this information in other applications. This is made possible by creating links between different applications and then updating those links, which in turn update or modify the information in the corresponding applications. This linking is a function of the OLE feature of Microsoft Windows. The OLE feature can also join together separate pieces of information from different applications into a single document. AutoCAD and other Windows-based applications such as Microsoft Word, Notepad, and Windows WordPad support the Windows OLE feature.

For the OLE feature, you should have a source document where the actual object is created in the form of a drawing or a document. This document is created in an application called a server application. AutoCAD for Windows and Paintbrush can be used as server applications. Now this source document is to be linked to (or embedded in) the compound (destination) document, which is created in a different application, known as the container application. AutoCAD for Windows, Microsoft Word, and Windows WordPad can be used as container applications.

Clipboard

The transfer of a drawing from one Windows application to another is performed by copying the drawing or the document from the server application to the Clipboard. The drawing or document is then pasted in the container application from the Clipboard. Hence, a Clipboard is used as a medium for storing the documents while transferring them from one Windows application to another. The drawing or the document on the Clipboard stays there until you copy a new drawing, which overwrites the previous one, or until you exit Windows. You can save the information present on the Clipboard with the *.clp* extension.

Object Embedding

You can use the embedding function of the OLE feature when you want to ensure that there is no effect on the source document even if the destination document has been changed through the server application. Once a document is embedded, it has no connection with the source. Although editing is always done in the server application, the source document remains unchanged. Embedding can be accomplished by means of the following steps. In this example, AutoCAD for Windows is the server application and MS Word is the container application.

1. Create a drawing in AutoCAD (the server application).

2. Open MS Word (container application).

3. It is preferable to arrange both the container and server windows so that both are visible, as shown in Figure 21-21.

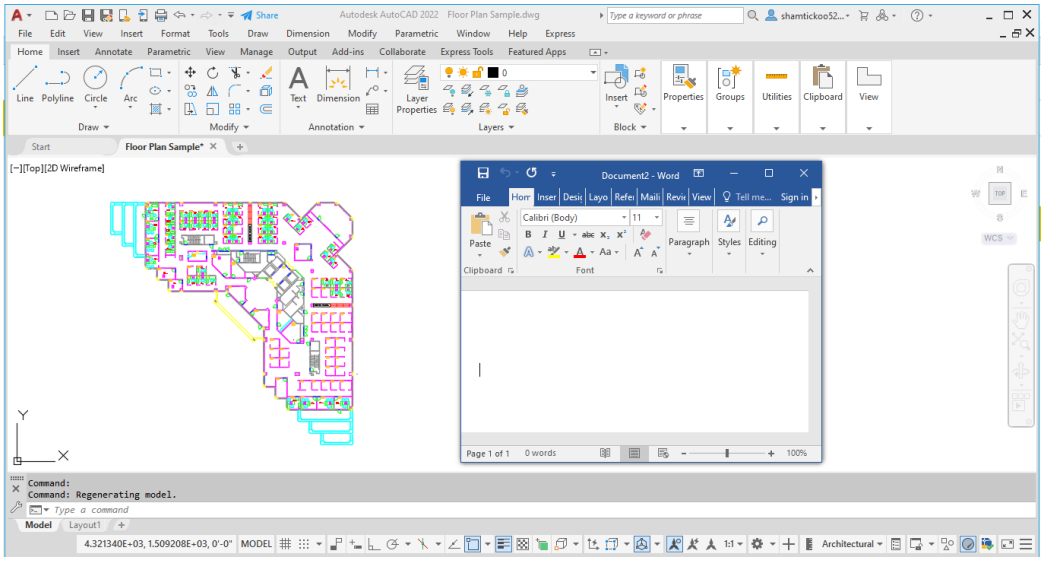

Figure 21-21 *AutoCAD graphics screen with the MS Word window*

4. In AutoCAD, copy the drawing using the CTRL+C keys; the **copyclip** tool is invoked. This tool can be used to embed drawings. You can also invoke this tool from the **Edit** menu, or by entering **COPYCLIP** at the command prompt. The next prompt, **Select objects**, allows you to select the entities you want to transfer. You can either select the full drawing by entering **ALL** or select some of the entities by selecting them. You can use any of the object selection methods for selecting objects. With this command, the selected objects are automatically copied to the Windows Clipboard.

5. After the objects are copied to the Clipboard, make the MS Word window active. To get the drawing from the Clipboard to the MS Word application (client), you need to paste it into the Word application. Choose the **Paste Special** tool from **Home > Clipboard > Paste** drop-down, refer to Figure 21-22; the **Paste Special** dialog box will be displayed, as shown in Figure 21-23. In this dialog box, select the **Paste** radio button (default) for embedding, and then choose the **OK** button. The drawing is now embedded in the MS Word window.

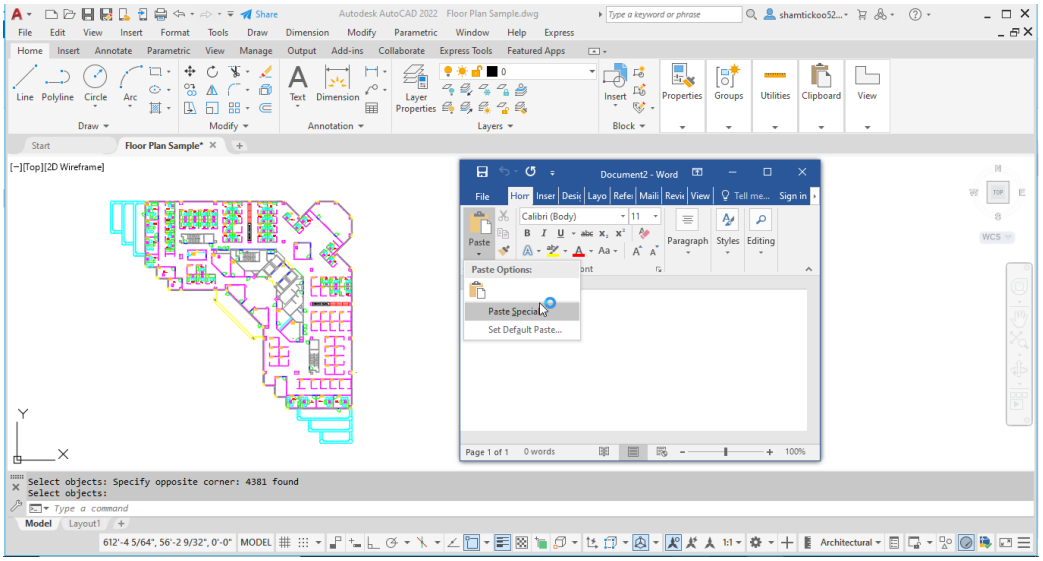

Figure 21-22 *Pasting a drawing into the MS Word application*

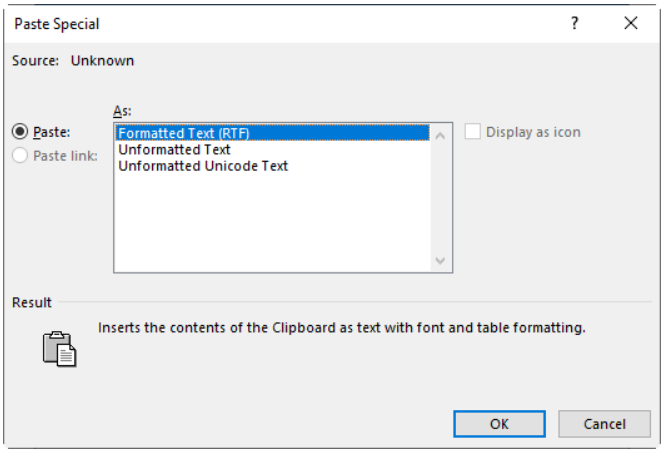

Figure 21-23 *The* ***Paste Special*** *dialog box*

6. Your drawing is now displayed in the MS Word window, but it may not be displayed at the proper position. You can get the drawing displayed in the current viewport by moving the scroll bar up or down in the MS Word window. You can also save your embedded drawing by choosing the **Save** button in MS Word. On doing so, the **Save As** dialog box is displayed where you can enter a file name. Save the file and then exit AutoCAD.

7. Now, you need to edit the embedded drawing. Editing is performed in the server application, which in this case is AutoCAD. You can get the embedded drawing into the server application (AutoCAD) directly from the container application (MS Word) by double-clicking on the drawing in MS Word. The other method of editing an embedded drawing is by right-clicking on it and then choosing **AutoCAD Drawing Object > Edit** from the shortcut menu.

8. Now, you are in AutoCAD, with your embedded drawing displayed on the screen, but as a temporary file with a file name such as [Drawing in Document]. Here, you can edit the drawing by changing the color and linetype or by adding and deleting the text, entities, and so on. In this example, the plate have been hatched.

9. After you have finished modifying your drawing, choose the **Update Microsoft Word** option from the **File** menu in the server (AutoCAD), refer to Figure 21-24. To display the menu bar, click on the down arrow in the **Quick Access Toolbar** and select **Show Menu Bar**. AutoCAD automatically updates the drawing in MS Word (container application). Now, you can exit this temporary file in AutoCAD.

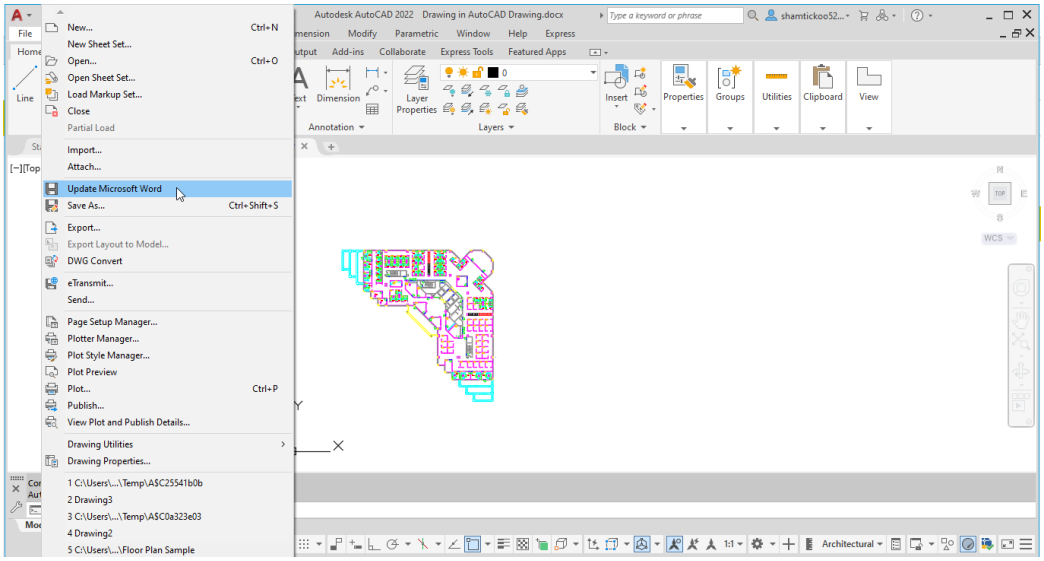

Figure 21-24 *Updating the AutoCAD drawing in Microsoft Word*



Note

Do not zoom or pan the drawing in the temporary file. If you do so, this will be included in the updating and the new file will display only that portion of the drawing that lies inside the original area.

10. This completes the embedding function. Now, you can exit the container application. While exiting it, a message box asking whether or not to save changes in MS Word is displayed.

Linking Objects

The linking function of OLE is similar to the embedding function. The only difference is that here a link is created between the source document and the destination document. If you edit the source, you can simply update the link, which automatically updates the client. This allows you to place the same document in a number of applications, and if you make a change in the source document, the clients will also change by simply updating the corresponding links. Consider AutoCAD for Windows to be the server application and MS Word to be the container application. Linking can be performed by means discussed next.

1. Open a drawing in the server application (AutoCAD). If you have created a new drawing, then you must save it before you can link it with the container application. Next, open MS Word (container application).

2. It is preferable to arrange both the container and the server windows so that both are visible.

3. In AutoCAD, choose the **Copy Link** option from the **Edit** menu of the menu bar to invoke the **COPYLINK** command. This command can be used for linking the drawing. This command copies all objects that are displayed in the current viewport directly to the Clipboard. Here, you cannot select objects for linking. If you want only a portion of the drawing to be linked, you can zoom into that view so that it is displayed in the current viewport prior to invoking the **COPYLINK** command. This command also creates a new view of the drawing having the name OLE1.

4. Make the MS Word window active. To get the drawing from the Clipboard to the MS Word (container) application, choose the **Paste Special** tool from **Home > Clipboard > Paste** drop-down; the **Paste Special** dialog box will be displayed. In this dialog box, select the **Paste link** radio button for linking. Choose the **OK** button. Note that the **Paste link** radio button in the **Paste Special** dialog box will not be available if you have not saved the drawing.

5. The drawing is now displayed in the MS Word window and is linked to the original drawing. You can also save your linked drawing by choosing the **Save** button from the **Quick Access Toolbar** in MS Word. It displays the **Save As** dialog box where you can enter a file name.

6. You can now edit your original drawing. Editing can be performed in the server application, which, in this case, is AutoCAD for Windows. You can edit the drawing by changing the color and linetype or by adding and deleting text, entities, and so on. Next, save your drawing in AutoCAD by using the **Save** tool. You can now exit AutoCAD.

7. You will notice that the drawing is automatically updated, and the changes made in the source drawing are present in the destination drawing also. If the drawing is not updated automatically, right-click on the drawing and then choose the **Update Link** option from the shortcut menu displayed. Automatic updating is possible if the **Automatic** radio button is selected in the **Links** dialog box, refer to Figure 21-25. The **Links** dialog box can be invoked by choosing **Linked AutoCAD Drawing Object > Links** from the shortcut menu displayed on right clicking on the drawing in the Word application.

8. Exit the container application after saving the updated file.

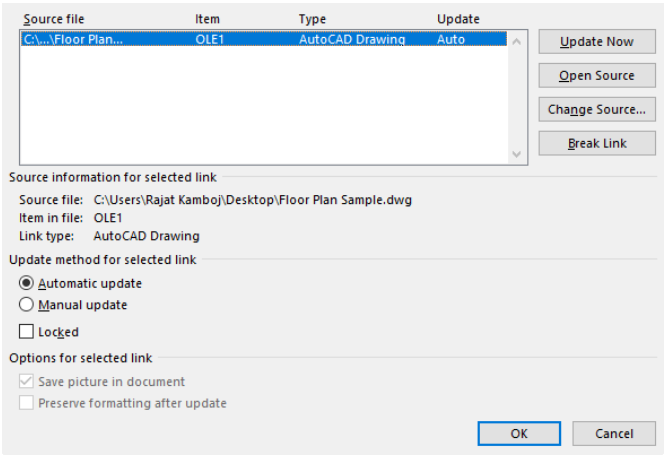

Figure 21-25 *The* ***Links*** *dialog box*

Note

The dialog box may be different in your case as it depends on the version of MS Word being used currently.

Linking Information to AutoCAD

You can also embed and link information from a server application into an AutoCAD drawing. You can also drag selected OLE objects from another application into AutoCAD, provided this application supports Microsoft ActiveX and the application is running and visible on the screen. Dragging and dropping is like cutting and pasting. If you press the CTRL key while you drag the object, it is copied to AutoCAD. Dragging and dropping an OLE object into AutoCAD embeds it into AutoCAD.

Linking Objects to AutoCAD

Start any server application such as MS Word and open a document in it. Select the information you wish to use in AutoCAD with your pointing device and choose the **Copy** tool to copy this data to the Clipboard. Open the AutoCAD drawing to which you want to link this data. Choose the **Paste Special** tool from **Home > Clipboard > Paste** drop-down or use the **PASTESPEC** command; the **Paste Special** dialog box will be displayed, as shown in Figure 21-26. In the **As** list box, select the data format you wish to use. Select the **Paste Link** radio button to paste the contents of the Clipboard into the current drawing. If you select the **Paste** radio button, the data will be embedded and not linked. Choose the **OK** button to exit the dialog box. Specify the insertion point; the data will be displayed in the drawing.

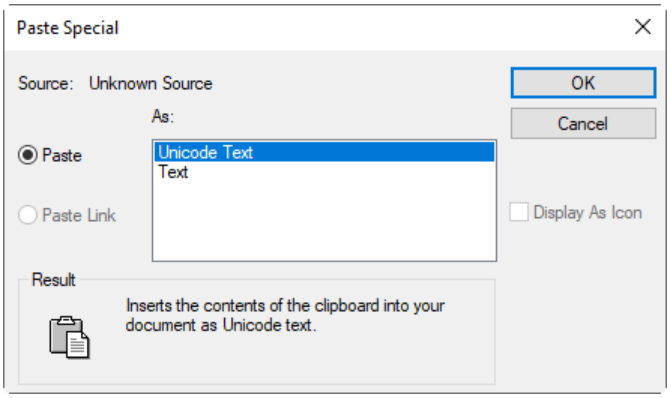

Figure 21-26 *The* ***Paste Special*** *dialog box*

You can insert objects in the drawing by choosing the **OLE Object** tool from the **Data** panel of the **Insert** tab, or by choosing the **OLE Object** tool in the **Insert** toolbar. This command links an entire file to a drawing from within AutoCAD. On invoking this command, the **Insert Object** dialog box is displayed, as shown in Figure 21-27.

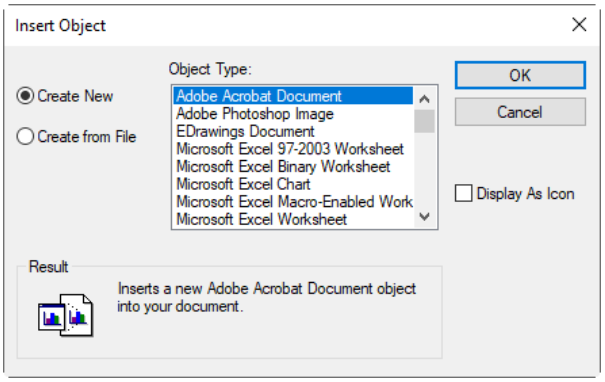

Figure 21-27 *The* ***Insert Object*** *dialog box*

Select the **Create from File** radio button. Also, select the **Link** check box. Choosing the **Browse** button displays the **Browse** dialog box. Select a file you want to link from the list box or enter a name in the **File name** edit box and choose the **Open** button. The path of the file is displayed in the **File** edit box of the **Insert Object** dialog box. If you select the **Display As Icon** check box, an icon is displayed in the dialog box as well as in the drawing. Choose the **OK** button to exit the dialog box; the selected file is linked to the AutoCAD drawing.

Whenever the server document changes, AutoCAD updates links automatically. But, you can use the **Links** dialog box to change these settings, refer to Figure 21-28. This dialog box can be displayed by choosing the **OLE Links** tool from the **Edit** menu. In the **Links** dialog box, select the link you want to update and then choose the **Update Now** button. Next, choose the **Close** button. If the server file location changes or if it is renamed, you can choose the **Change Source** button in the **Links** dialog box to display the **Change Source** dialog box. In this dialog box, locate the server file name and location and choose the **Open** button. You can also choose

the **Break Link** button in the **Links** dialog box to disconnect the inserted information from the server application. This is done when the linking between the two is no longer required.

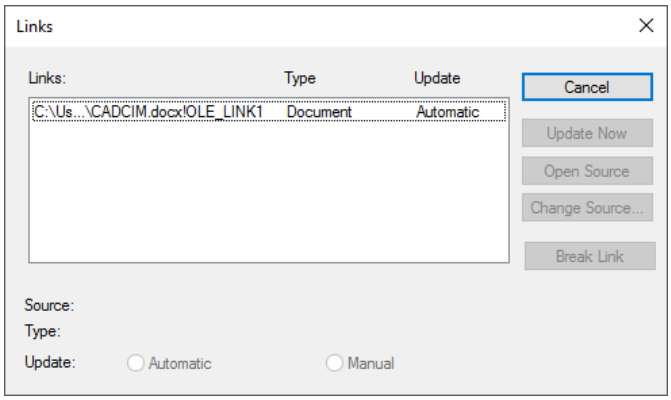

Figure 21-28 *The* ***Links*** *dialog box*

Embedding Objects into AutoCAD

Open the server application and select the data you want to embed into the AutoCAD drawing. Copy this data to the Clipboard by choosing the **Copy** tool in the toolbar or **Copy** from the **Edit** menu. Open the AutoCAD drawing and choose **Paste** from the AutoCAD **Edit** menu. You can also use the **PASTECLIP** command. Specify the insertion point; the selected information is embedded into the AutoCAD drawing.

You can also create and embed an object into an AutoCAD drawing. To do so, choose the **OLE Object** tool from the **Data** panel of the **Insert** tab; the **Insert Object** dialog box will be invoked. In this dialog box, select the **Create New** radio button and select the application you wish to use from the **Object Type** list box. Choose the **OK** button, the selected application opens. Now, you can create the information you wish to insert in the AutoCAD drawing and save it before closing the application. You can edit information embedded in the AutoCAD drawing by double-clicking on the inserted OLE object; the server application will open. The modifications made in the server application get automatically reflected in the AutoCAD drawing.

Working with OLE Objects

Select an OLE object and right-click to display a shortcut menu. Next, choose the **Properties** option from the shortcut menu; the **PROPERTIES** palette will be displayed. Specify a new height value and a new width value in the **Height** and **Width** edit boxes in the **Geometry** area. Enter a value in percentage of the current values in the **Scale height %** and **Scale width %** edit boxes. If you select the **Yes** option in the **Lock aspect** field, whenever you change the height or the width from the **Scale height %** and **Scale width %** edit boxes, the respective width or height changes automatically to maintain the aspect ratio. If you want to change only the height or the width, select **No** for this field.

The pointing device can be used to modify and scale an OLE object in the AutoCAD drawing. Selecting the object displays the object frame and handles. These handles allows you to move and stretch a frame. Stretching the object's frame does not scale objects proportionately.

Select an OLE object and right-click to display a shortcut menu. You can choose **Clipboard > Cut** to remove an object from a drawing and paste it on the Clipboard. The **Copy** option is used to place a copy of the selected object on the Clipboard. The **Erase** option is used to remove an object from a drawing but it does not place the object on the Clipboard. Choose the **OLE** option from the shortcut menu; a cascading menu will be displayed containing the **Open**, **Reset**, **Text Size**, and **Convert** options. Choose the **Convert** option to display the **Convert** dialog box where you can convert objects from one type to another. Choose the **Open** option to open an object in the Server application where you can edit and update it in the current drawing. Choose the **Reset** option to restore the selected OLE objects to their original size, that is, the size they were in when inserted. Choose the **Text Size** option to resize the text. You can set the draw order of the object by first invoking a shortcut menu and then choosing the corresponding option from the **Draw Order** cascading menu.

If you want to change the layer of an OLE object, select the object, and then select the required layer from the **Layer** drop-down list in the **Layers** panel.

The **OLEHIDE** system variable controls the display of OLE objects in AutoCAD. The default value is 0, which makes all OLE objects visible. The different values and their effects are as follows:

0 All OLE objects are visible
1 OLE objects are visible in paper space only
2 OLE objects are visible in model space only
3 No OLE objects are visible

The **OLEHIDE** system variable affects both screen display and printing.

SHARING DRAWING VIEWS

Ribbon: Collaborate > Share > Shared Views **Application Menu:** Publish > Share View
Command: SHAREDVIEWS



The **Shared Views** tool is used to share designs with internal and external stakeholders without releasing the original drawing files. With the introduction of this tool, it is no more required to publish DWF or PDF format files or email them whenever you need to share the data. Using this tool, you can extract views and data from the drawing and store them in the cloud, generate a link. You can share this link with others to view the design on a browser and leave comments from any web-enabled devices such as desktop, tablet, or mobile.

After the link is generated and shared, the recipients can view, review, measure, comment on, and mark up the design right from the Autodesk Viewer in their web browser, and can share their feedback. You can access that feedback from within AutoCAD. Note that, shared views automatically expire after 30 days, but you can extend or terminate the link at any time.

Choose the **Shared Views** tool from the **Share** panel of the **Collaborate** tab in the **Ribbon**; the **SHARED VIEWS** palette will be displayed, refer to Figure 21-29.

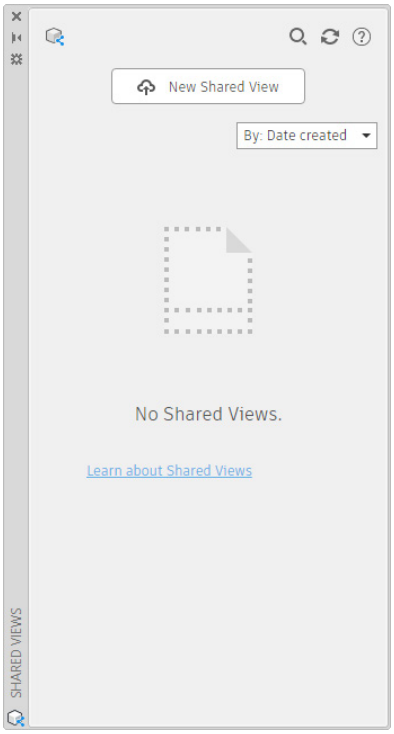

Figure 21-29 *The* ***SHARED VIEWS*** *palette*

Now, choose the **New Shared View** button from the **SHARED VIEWS** palette; the **Share View** dialog box will be displayed, refer to Figure 21-30.

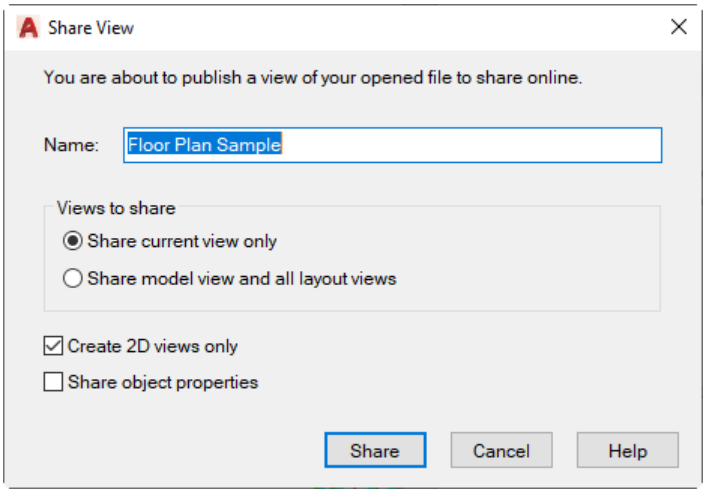

Figure 21-30 *The* ***Share View*** *dialog box*

The options in this dialog box are discussed next.

Name

This edit box is used to specify the name of the shared view. The name can have up to 255 characters and can include letters, numbers, blank spaces, and any special character which is not being used by the operating system or the program for other purposes.

Views to share

The options in this area allow you to publish a representation of the current space or the entire drawing for online viewing and sharing. This facilitates communication and collaboration with stakeholders without sharing the source DWG files. The options in this area are discussed next.

Share current view only

This radio button is selected by default and allows you to publish the unclipped extents of model space or the current layout.

Share model view and all layout views

If you select this radio button, the unclipped extents of model space and all layouts will be published.

Create 2D views only

This check box is selected by default and limits the view creation to 2D objects only.

Share object properties

If you select this check box, the object properties will also be extracted along with the view creation.

After selecting the required options in the **Share View** dialog box, choose the **Share** button; the **Share View - Processing Ready to Start** message box will be displayed, refer to Figure 21-31.

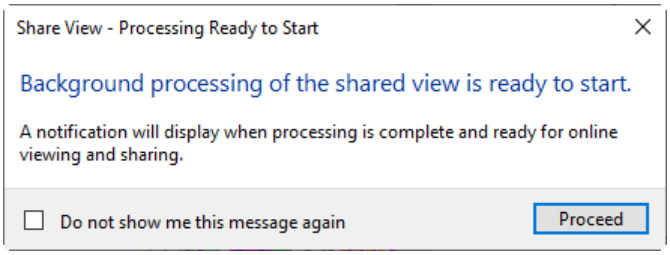

Figure 21-31 *The* ***Share View - Processing Ready to Start*** *message box*

Choose the **Proceed** button from this message box. Once the sharing process gets completed, the **Share View Upload Complete** notification bubble appears, refer to Figure 21-32.

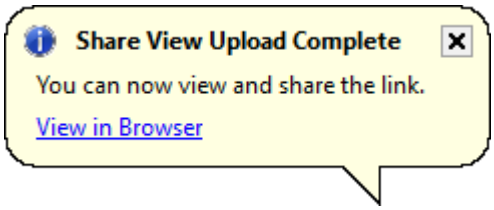

Figure 21-32 *The* ***Share View Upload Complete*** *notification bubble*

Click on the **View in Browser** link in this notification bubble; the **Autodesk VIEWER** window will be displayed, refer to Figure 21-33.



Figure 21-33 *The* ***Autodesk VIEWER*** *window*

Choose the **Share** button available at the right in this window; the **Sign in to share** palette will be displayed. Sign in to your Autodesk account; the **Share** palette will be displayed along with the shareable link of the file, refer to Figure 21-34.

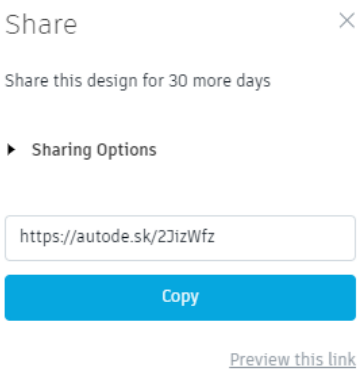

Figure 21-34 *The* ***Share*** *palette along with the link*

Now, choose the **Copy** button from the **Share** palette; the link will be copied to the clipboard and can be shared with anyone. Once the sharing process gets completed, the shared file is displayed in the **SHARED VIEWS** palette. If the shared file is not displayed automatically, then choose the **Refresh** button available at the top in the **SHARED VIEWS** palette.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. You can import a DXB file into an AutoCAD drawing file by using the _____ command.

2. The _____ system variable controls the display of OLE objects in AutoCAD.

3. If you import a DXB file into an old drawing, the settings of layers, blocks, and so on of the file being imported are _____ by the settings of the file into which you are importing the DXB file.

4. The _____ is used as a medium of storing documents while transferring them from one Windows application to the other.

5. You can get an embedded drawing into the Server application directly from the container application by _____ on the drawing.

6. The **Copy** tool copies a drawing from the _____ to the Clipboard.

7. The DXF file format generates a text file in ASCII code. (T/F)

8. You can directly open a DXF file in AutoCAD by using the **Open** tool. (T/F)

9. An image attached to AutoCAD cannot be detached. (T/F)

10. A file in the container application is automatically modified only when a link is maintained between the server application and the container application. (T/F)

Review Questions

Answer the following questions:

1. Which of the following tools can be used to open a DXF format file?

   (a) **Open** (b) **New**
   (c) **Save** (d) None of these

2. Which of the following tools is used to add an image to the drawing as an external reference?

   (a) **Attach** (b) **Insert**
   (c) **Adjust** (d) **Create Block**

3. In which of the following panels, the options are used to modify the brightness of an image?

   (a) **Options** (b) **Visual Styles**
   (c) **Clipping** (d) **Adjust**

4. The _____ tool is used to modify frames by retaining only the desired portion of the images.

5. The _____ tool is used to create an ASCII format file with the *.dxf* extension from the AutoCAD drawing files.

6. Binary DXF files are _____ efficient and occupy only 75 percent of the ASCII DXF file.

7. File access for files in binary format is _____ than for the same file in ASCII format.

8. If the **OLEHIDE** system variable value is _____, the OLE objects are visible in the paper space only.

9. When you select an image file to attach, the _____ dialog box is displayed where you can define the insertion point, scale factor, and rotation angle for the image.

10. The clipping boundary for the raster images can be of _____ or the _____ shape.

11. In AutoCAD, you can clip the boundary of an image and give it the desired shape. (T/F)

12. The scanned files can be imported into the current session of AutoCAD using the **Attach** tool. (T/F)

13. You can export an AutoCAD drawing into 3D Studio MAX. (T/F)

14. Raster images can be attached using the **Attach** tool. (T/F)

Exercise 1

In this exercise, you will create a cup and a plate, refer to Figure 21-35 for dimensions. Assume the missing dimensions. Below the cup and plate, enter the following text in MS Word: **These objects are drawn in AutoCAD**. Then using OLE, paste the text into the current drawing.

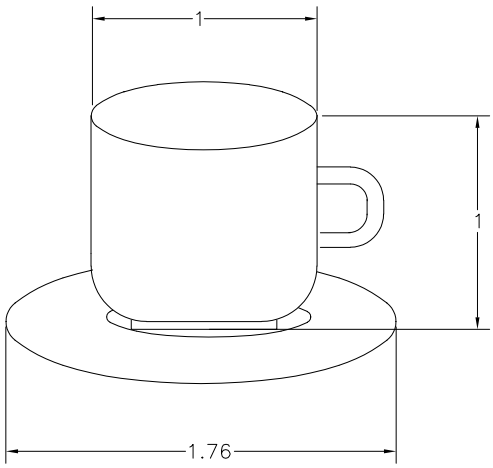

Figure 21-35 *Dimensions for Exercise 1*

Answers to Self-Evaluation Test

1. DXB**IN**, **2. OLEHIDE**, **3.** overruled, **4.** Clipboard, **5.** double-clicking, **6.** current display, **7.** T, **8.** T, **9.** F, **10.** T