DraftSight®

Getting Started With DraftSight
A Guide For AEC Users

DraftSight.com and Facebook.com/DraftSight

3DS DASSAULT SYSTEMES

Welcome to DraftSight
a valuable tool for any AEC professional!

DraftSight is more than a free, professional-grade 2D CAD product. The product runs on Microsoft® Windows 7®, Windows Vista®, Windows® XP, Mac® and Linux® and takes just a few minutes to download at DraftSight.com. With its layering functionality and the ability to specify units as Architectural, DraftSight is a great way to provide professional-grade construction drawings and is a valuable tool for any AEC professional. Check out DraftSight today!

From this guide, AEC professionals will learn how to:
- Activate DraftSight, open a DWG file, and use that file to explain DraftSight's basic operations like zoom in and zoom out, among others.
- Draw lines and circles using DraftSight. There are various ways of drawing lines, but for the practical purpose, you will learn two methods, one using the coordinates and the other using the "Entity snap".
- Use a special DraftSight feature entitled "Image Layer Functionality".
- Operate commands like Copy, Trim, and Extend that are frequently used to edit drawings. You will also enter texts and dimensions.
- Reference other drawings within a drawing.
- Print a drawing using DraftSight.

Before getting started, please download these sample DWG files that accompany the Guide.

We hope you'll enjoy the DraftSight experience.
For more information about DraftSight or to download the product, please visit DraftSight.com.

Contents

REMINDER: Before getting started with this Guide, please download these sample DWG files that accompany the Guide.

Let's open a DWG file and see what we have

In this section, we will activate DraftSight, open a DWG file, and use that file to explain DraftSight's basic operations like zoom in and zoom out, among others.

1. Activate DraftSight

① Double-click the "DraftSight" icon on the desktop. Or, select "Start" button -> "Dassault Systemes" ->"DraftSight" to start the DraftSight application. Double-click the "DraftSight" icon on the desktop. Or, select "Start" button -> "Dassault Systemes" ->"DraftSight" to start the DraftSight application.
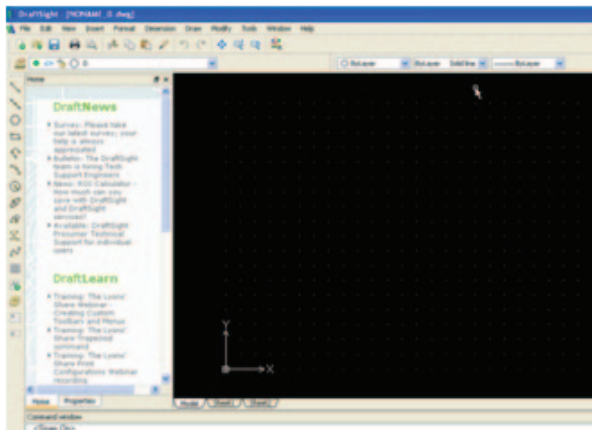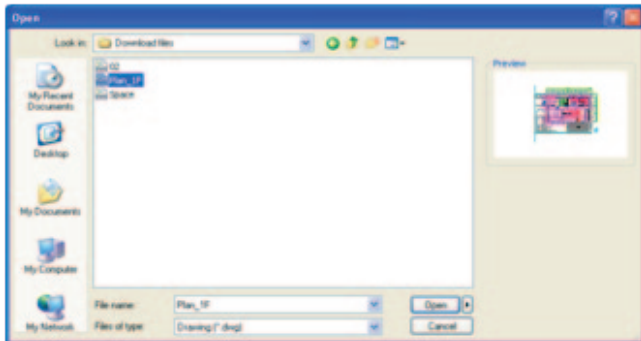
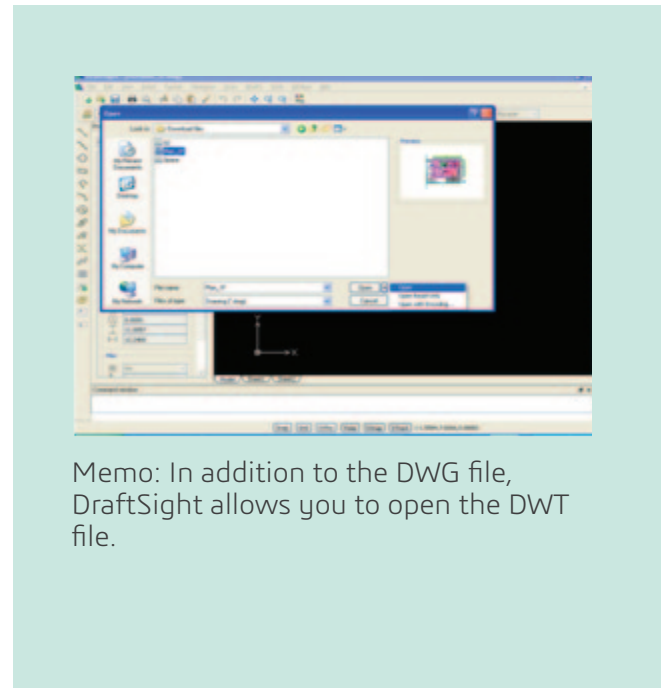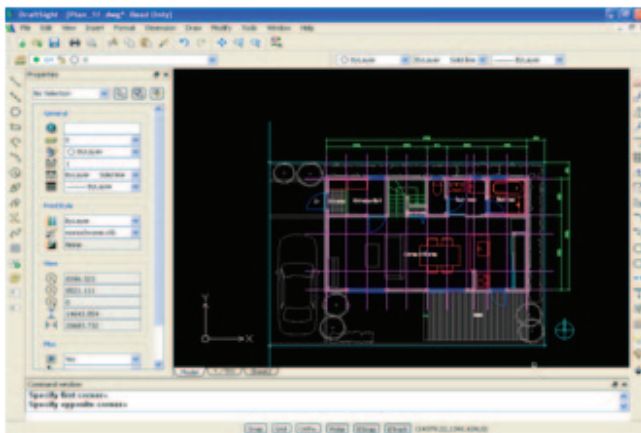DraftSight starts up by opening a new drawing where nothing is drawn.

2. Open the DWG file
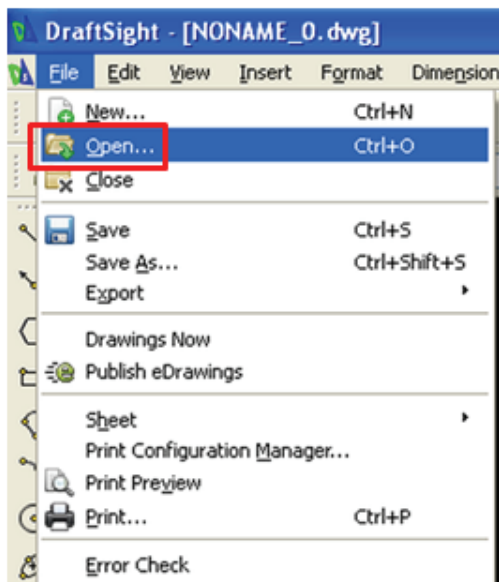
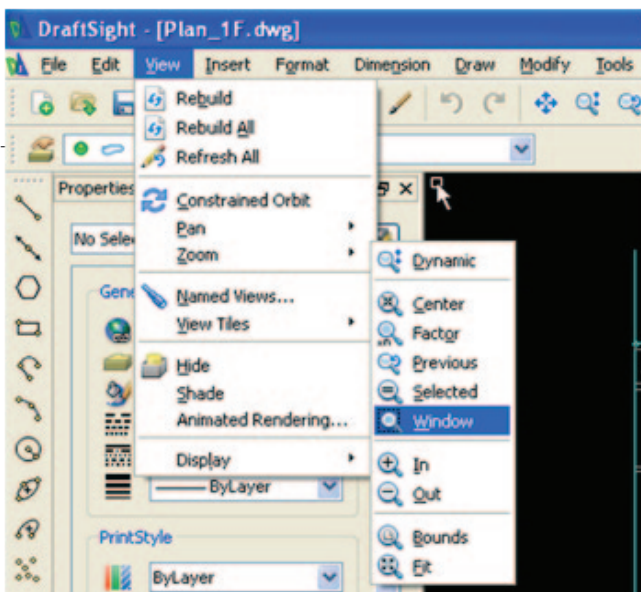① Select "File" -> "Open" from the Menu.

② From the "Open" dialog box that appears, select the downloaded file "Plan_1F.dwg" and click "Open".
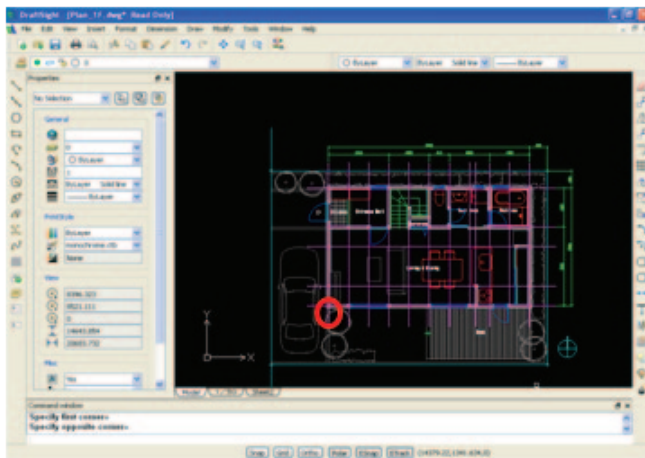

Memo: In addition to the DWG file, DraftSight allows you to open the DWT file.

3. Operate the screen (window zoom)

Window zoom functionality is used to enlarge the specific area of the drawing by surrounding the desired area with the boundary of the window and displaying the area captured within the boundary fully on the window.
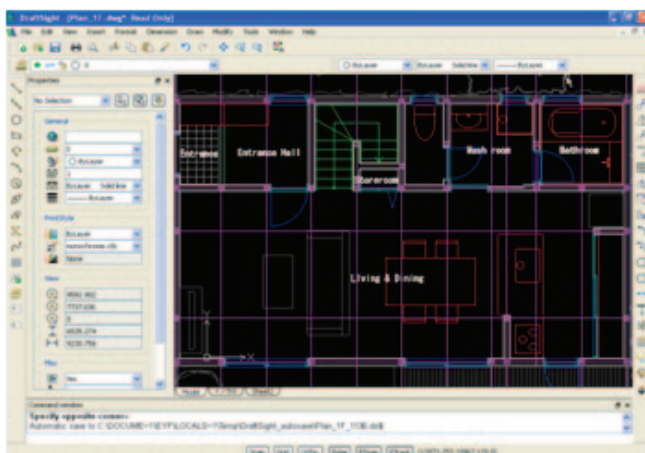
① Select "View" -> "Zoom" -> "Window" from the Menu.

② Click the lower left point of the window (rectangle) for surrounding the area that you want to enlarge.
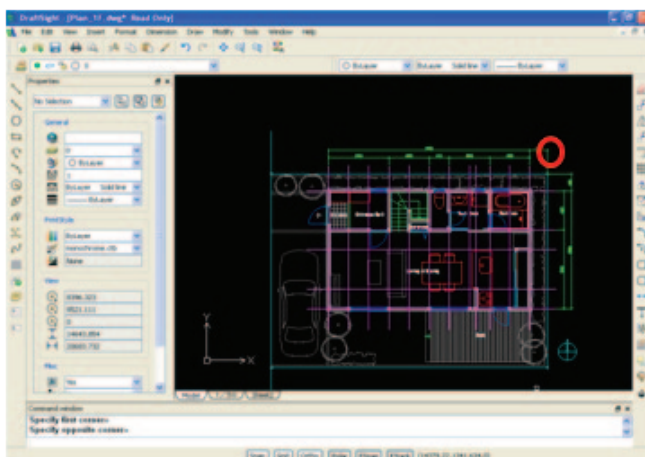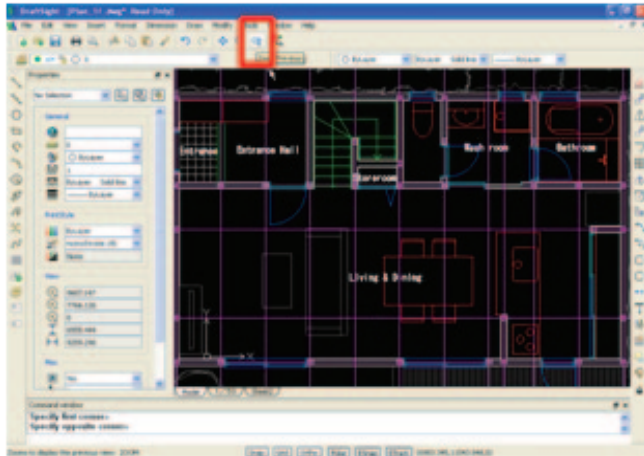

③ Then click the upper right point of this window. (The temporary window will be displayed on the drawing.)
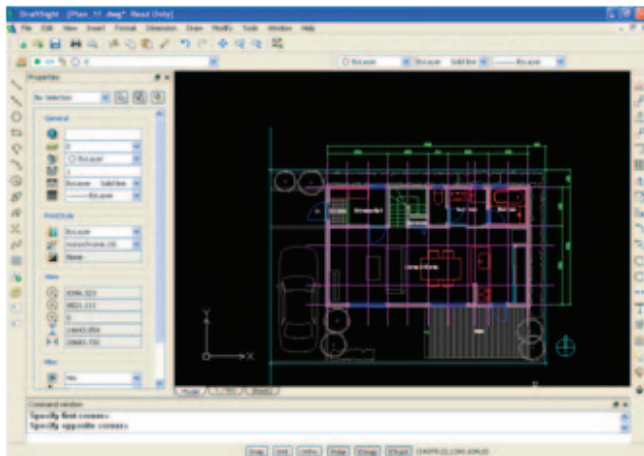

The area specified by the window is enlarged to the extent that it fully occupies the drawing workspace.

4. "Previous screen"

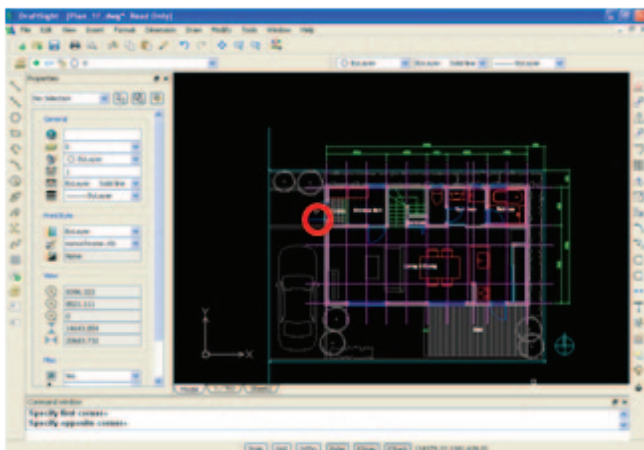

By using the "Previous screen" command, go back one step before the drawing zoomed in.
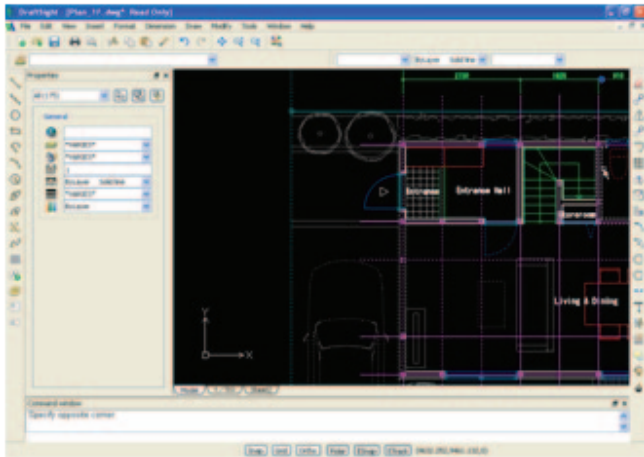① Do so by clicking "Previous screen" on the tool bar.

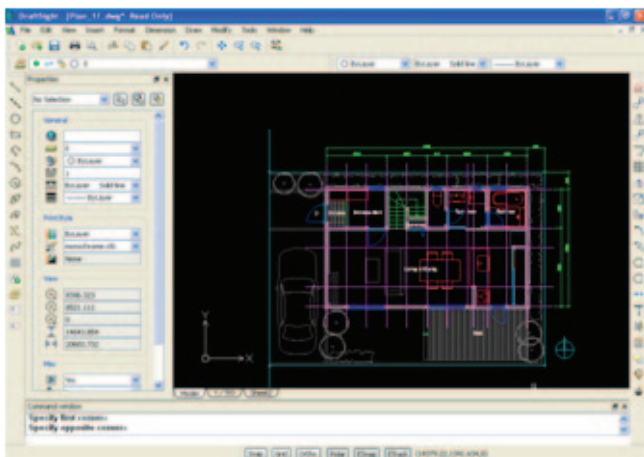The drawing returns to the state shown before operating the window zoom functionality.

5. Use the wheel button (central button) of the mouse to zoom in / out. The drawing can be zoomed in / out by turning the mouse wheel button
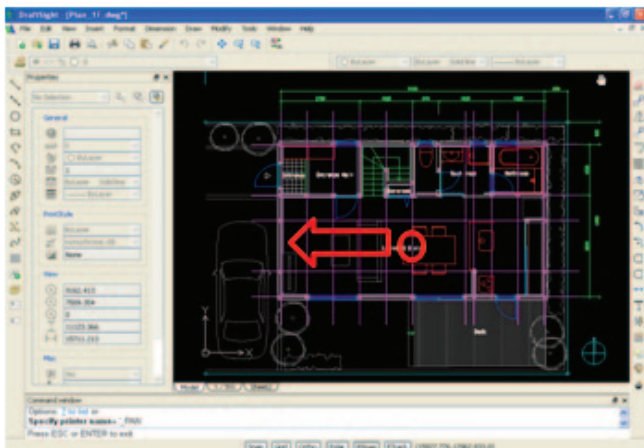

① Turn the mouse wheel button forward.

The drawing zooms in from the center of where the mouse pointer is pointing.

② Turn the mouse wheel button backward.

6. Use the wheel button (central button) of the mouse to move the drawing dynamically.
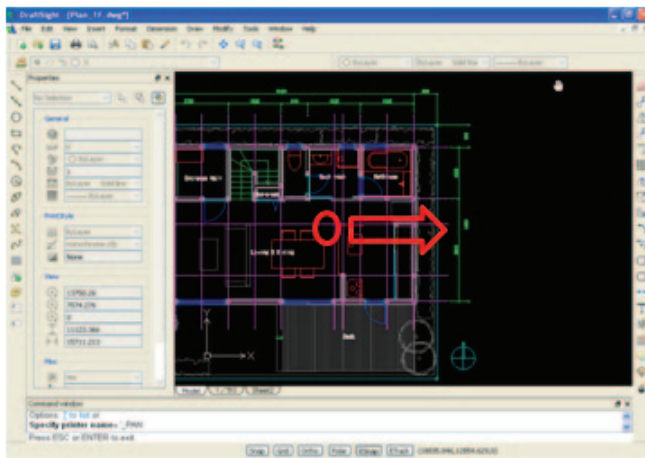

The wheel button (central button) of the mouse can be used to move (drag) the drawing.

① Push the mouse wheel button and see that the cursor changes to hand shape.

② While pressing the button, drag the drawing towards the left and release the button.

The drawing shifts to the left dynamically and only the right side of the drawing is now shown on the screen.

③ Press the mouse wheel button again, and while pressing it, drag the drawing to the right and then release the button.
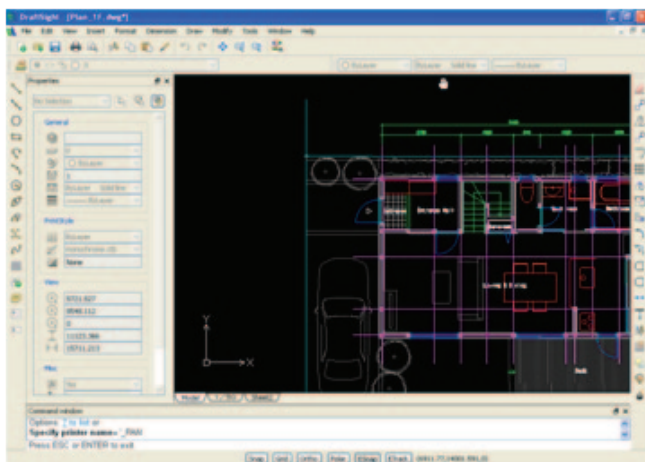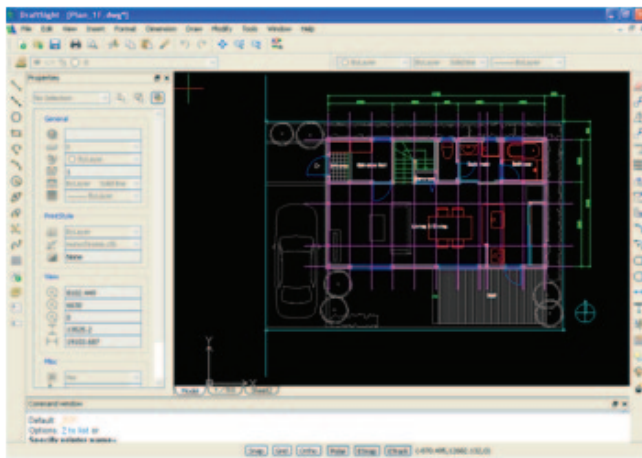

The drawing now shifts to the right dynamically and only the left side of the drawing is shown on the screen.

7. Display the entire drawing.

Use the "Boundary zoom" command to display the entire drawing. By using the "Bounds" command, you can make all the entities of the drawing appear on the drawing workspace.
① Select "View" -> "Zoom" -> "Bounds" from the Menu.

The area where entities exist or the entire drawing within the boundary (whichever is larger) is displayed on the screen. Also include "double-click" of the scroll wheel to zoom bounds.

8. End the operation using DraftSight.
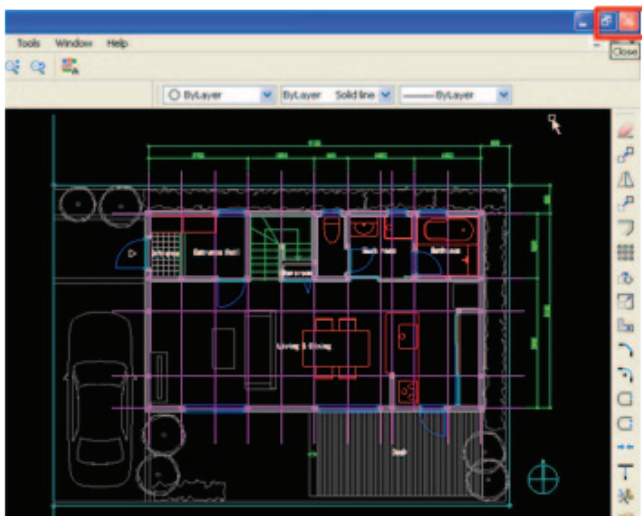
To end the operation using DraftSight, follow the procedure below.

① Click the "X" button on the upper right corner of the window.
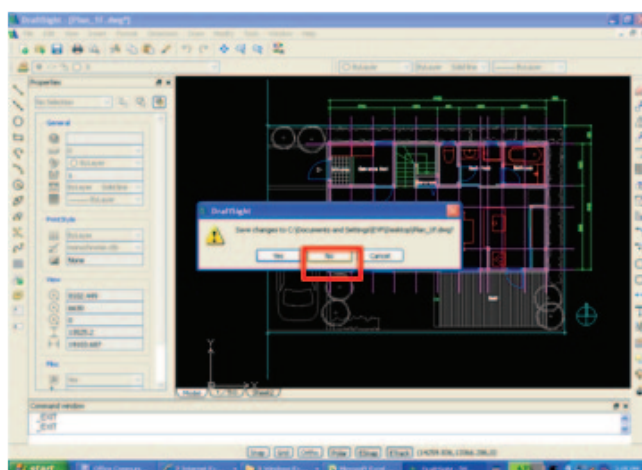
② Since the data has been edited, a message will appear asking "Do you want to save the file?" The editing work you performed on the drawing this time does not require you to save the changes.

As a result, click "No" to close the file that has been opened. The DraftSight application also terminates through this ending operation.

Drawing lines

In this section, we will draw lines and circles using DraftSight. There are various ways of drawing lines, but for the practical purpose, we will focus on two methods, one using the coordinates and the other using the "Entity snap".

1. Activate DraftSight and open the file named "02.dwg".

An empty DWG file is opened. A blank window appears.
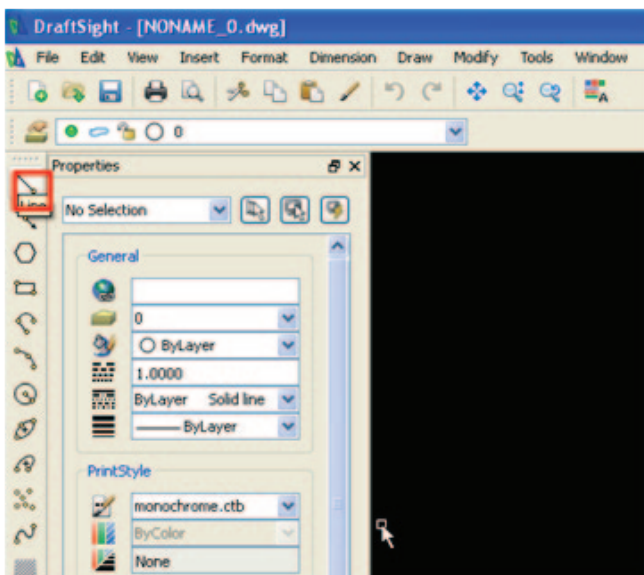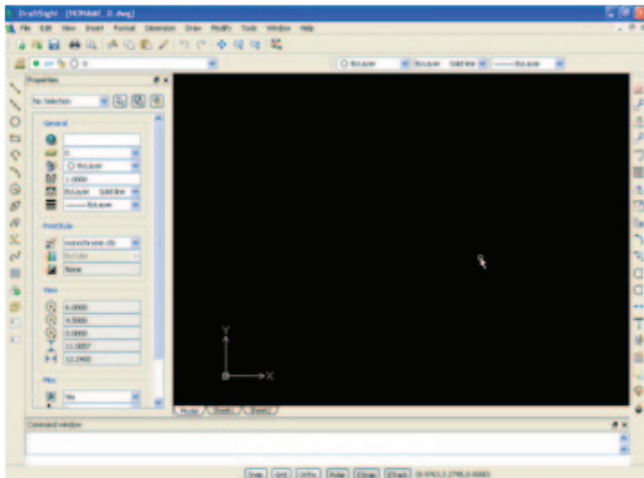
To draw a line by using the coordinate system, follow the procedure below.

① Click "Line segment" from the tool bar.

Memo: The coordinate system is defined by X and Y directions, where X and Y respectively represent the horizontal and vertical directions. Each coordinate value is specified by X or Y. There are two ways of specifying the coordinates to execute a command. You can either select the absolute coordinate method or the relative coordinate method. When absolute coordinate method is selected the coordinate values you enter are based on the point of origin (the lower left point where the arrow of the XY pointer is displayed) that is fixed to 0,0. When the relative coordinate system is selected, the last point you enter becomes the point of origin (0,0) and "@" mark is displayed in front of the coordinate value entered last.
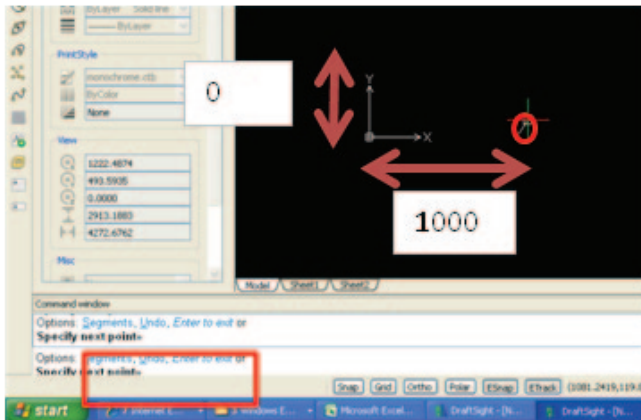
② As the command for specifying the starting point, enter "1000,0" as the absolute coordinates by using the keyboard and press "Enter". Then a line will be drawn to point, 1000 in X direction and point, 0 in Y direction from the point of origin (0,0).
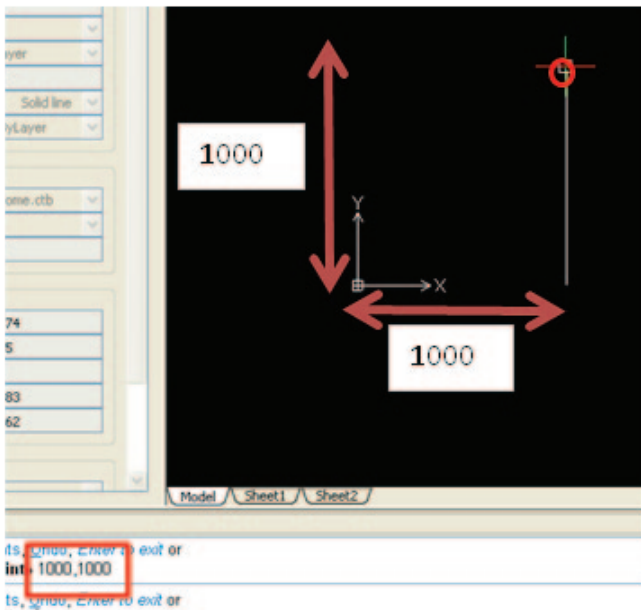
③ Next, specify "1000,1000" as the next point and press "Enter". Then a line segment will be drawn to point 1000 in X direction and point 1000 in Y direction from the point of origin.
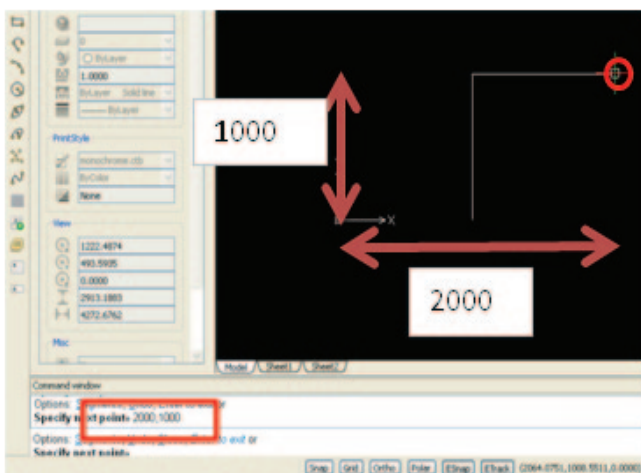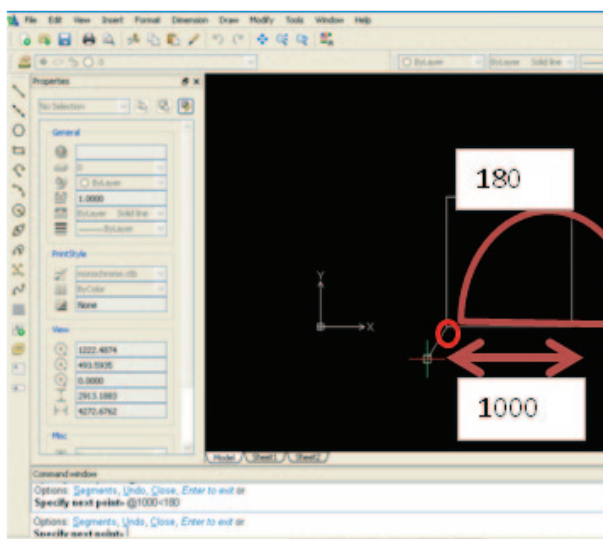
④ Next, specify "2000,1000" as the next point and press "Enter". Then a line segment will be drawn to point 2000 in X direction and point 1000 in Y direction from the point of origin.
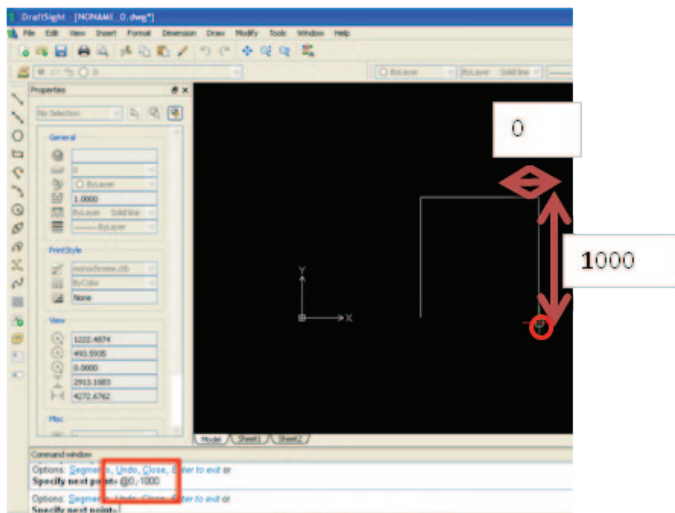
Now, let's draw a line using the relative coordinate method.

⑤ Specify "@0,-1000" as the next point and press "Enter". Then a line segment will be drawn to point, 0 in X direction and point -1000 in Y direction from the specified point.


Next, let's draw a line by specifying the angle using the Polar Coordinate method.

⑥ Specify "@1000<180" as the next point and press "Enter". Then a line stretching for 1000 from the point specified last will be drawn at the angle of 180 degrees.

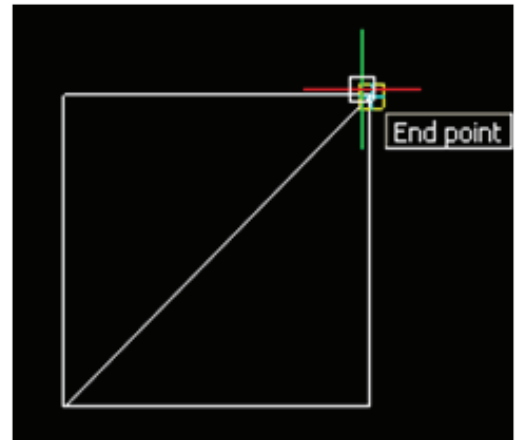⑦ Press "Enter" to end the "Line segment" command. You have finished drawing a square.

## 2 . Draw a line by using the "Entity Snap" functionality.
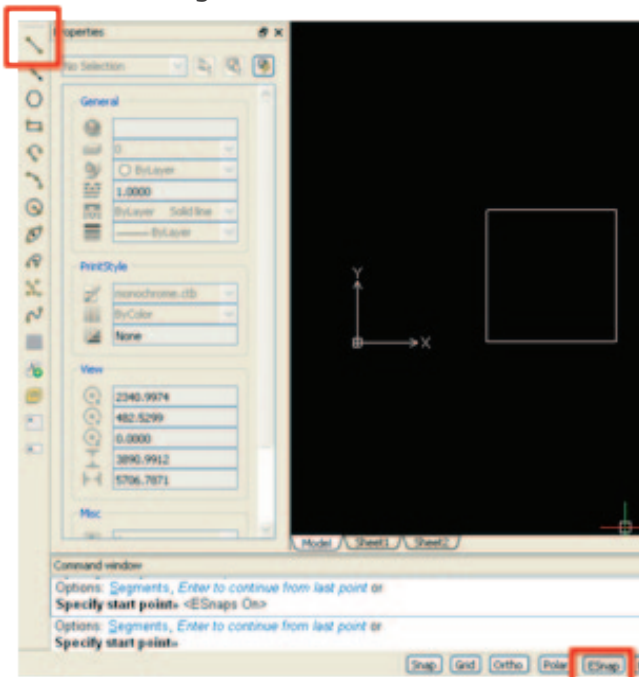
* Right-click on the Esnap button to access the Options dialog.

Let's now draw a diagonal line by using the "Entity Snap" functionality.
① Click "Line command" from the tool bar.

Memo: Entity Snap will enable you to specify the edge points and intersection points accurately.
When you enlarge the lines drawn without specifying the points with Entity Snap, you will notice that they are not very accurate.
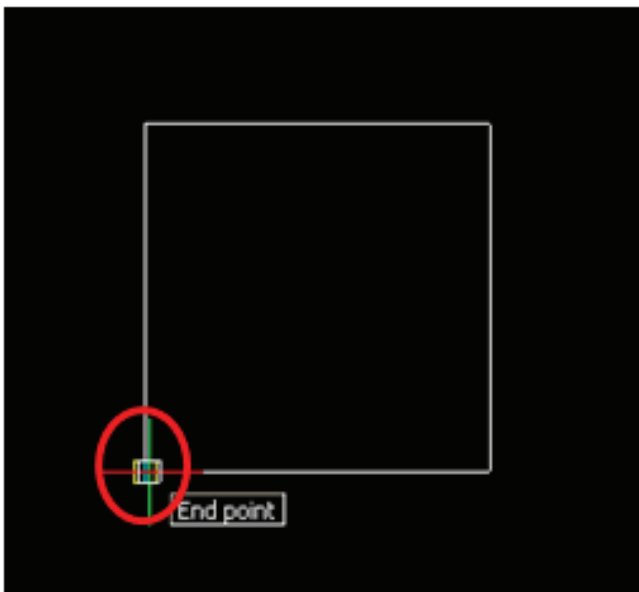

② After confirming that "Entity Snap" in the status bar is active, place the cursor on the lower left vertex of the square.

③ To specify the starting point of the line command, place the cursor at the position where the Endpoint Snap (yellow square shape) appears, and click.
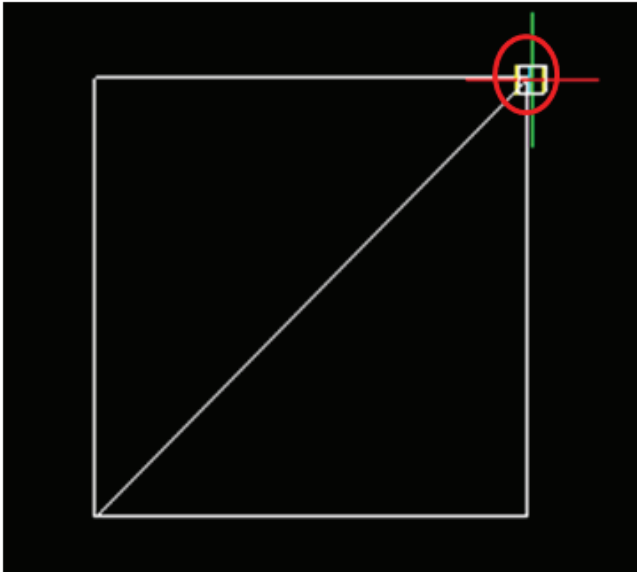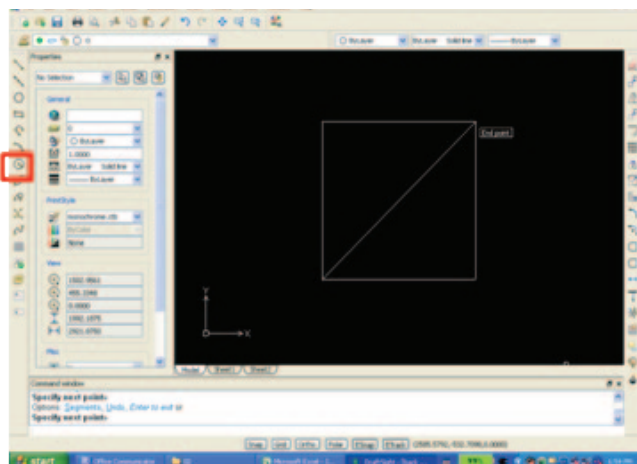
④ Move the cursor to the direction of the diagonal line, and place it on the upper right vertex of the square. Just like in the previous step, place the cursor at the position where the Endpoint Snap appears, and then click to specify the end point of the line segment.

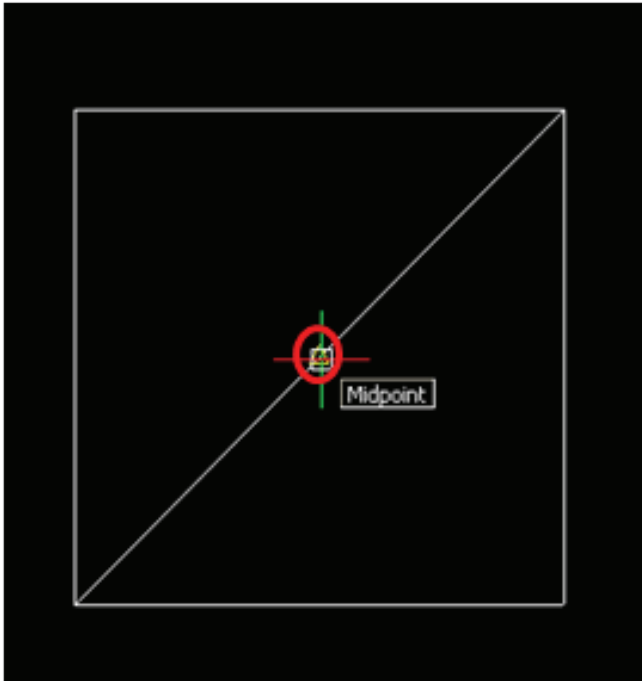⑤ Press "Enter" to end the "Line segment" command.
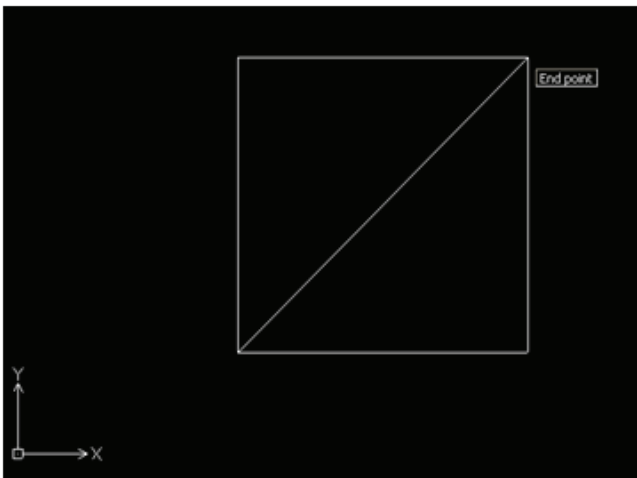
You now have drawn an accurate diagonal line running through a square.

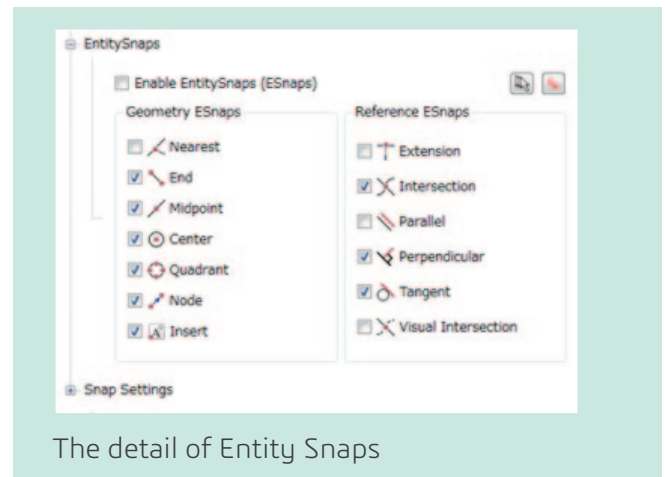## 3. Draw a circle inscribed in the square.

Use the "Circle" command to draw a circle.
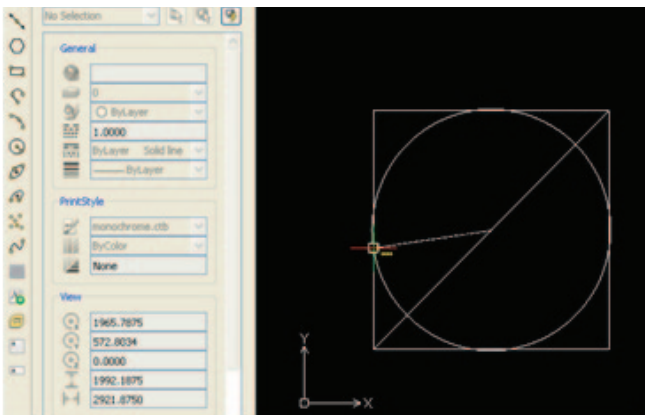① Click "Circle" command from the tool bar.

To specify the central point, use Entity Snap to specify the midpoint of the diagonal line.
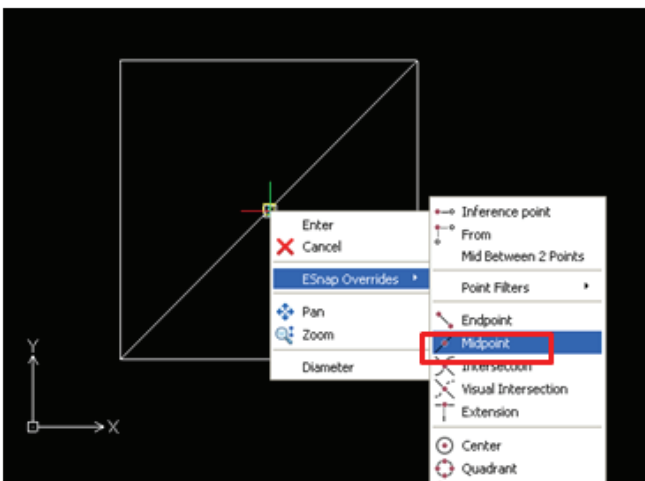
② Move the cursor to the middle of the diagonal line. Place the cursor at the position where the "midpoint" snap (yellow triangle) appears, and click.
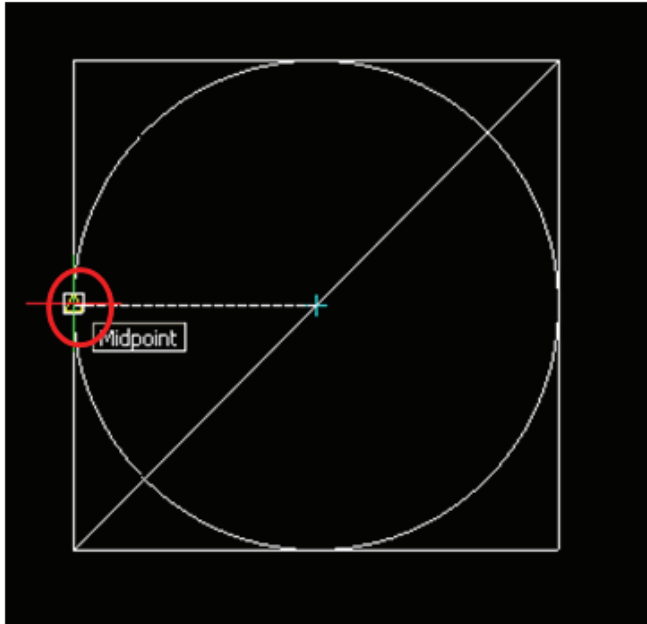
The detail of Entity Snaps

Move the cursor to the left side of the square. The "tangent" snap appears. This snap can specify. But since we want to use the midpoint snap, we will use the Esnap Overrides.
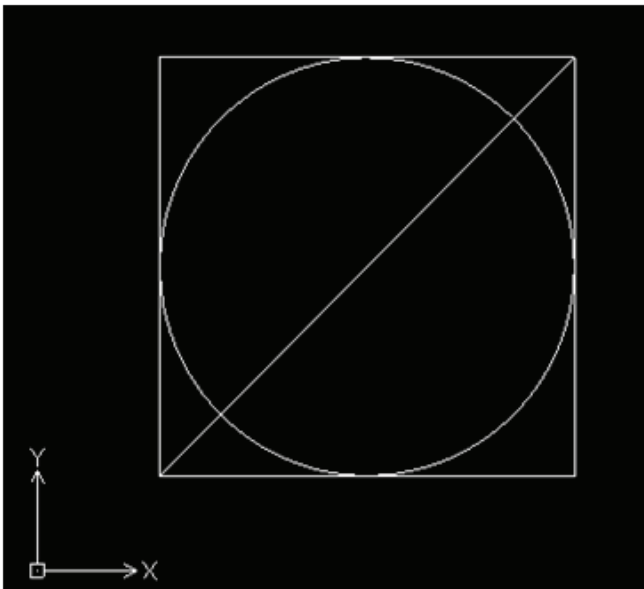
③ Right-click to display the Menu.

④ Select "Esnap Overwrite" -> "Midpoint".

⑤ Move the cursor to the middle of the left side of the square. At this point, only the Midpoint Snap can be used. Place the cursor at the position where the Midpoint Snap appears, and click to specify the midpoint of the left side.


You now have drawn a circle inscribed in a square.

Summary
To create an accurate drawing using DraftSight, there is a need to accurately specify the points to draw lines and other shapes by making use of the coordinate system or "Entity Snap". Beside the type of snap introduced in this section, there are many other types that the users should know so that they can make the best use of what DraftSight can offer.

Operating the Layers

Let's now use one of the special features of DraftSight called the Layer functionality. Layer is a functionality that sorts the different areas and types of information (like "text" and "wall") that constitute a drawing in separate layers. Layers can be easily switched to show or hide mode. When you want to edit a drawing, it is normally easier to work on it by making DraftSight show only the layers that you want to view, and hide the rest. The same applies when you want to print out a drawing that shows only the layers that you want to see. In this section, we will see how the drawing will be displayed when specific layers are switched to show mode or hide mode, and also try changing the graphics placed in specific layers.

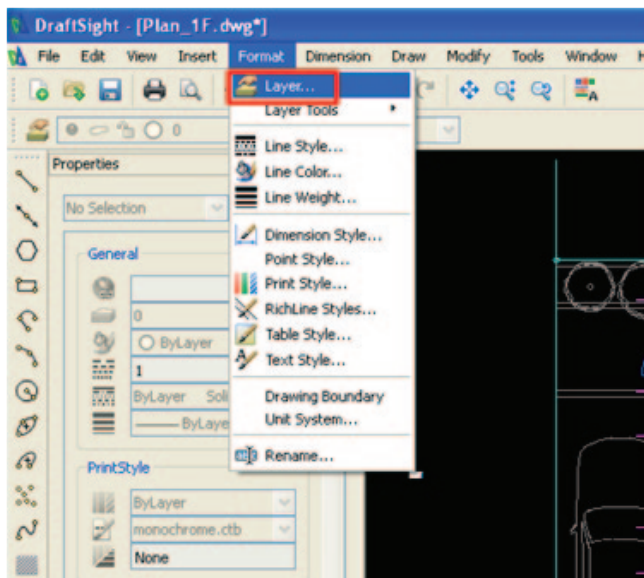1. Activate DraftSight and open the file named "Plan_1F. dwg".
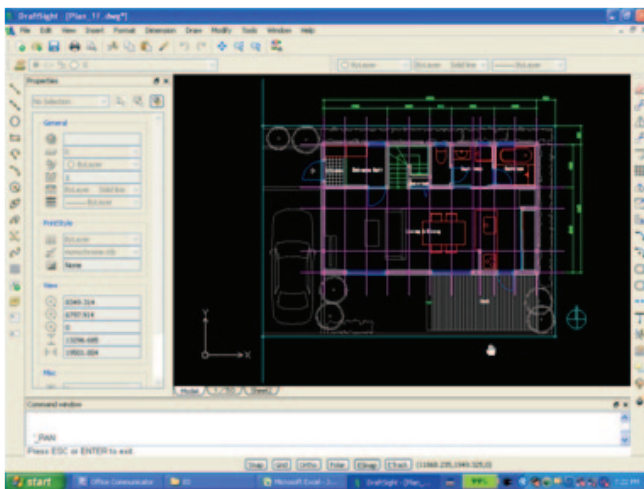
① Activate DraftSight and open the file named "Plan_1F.dwg".

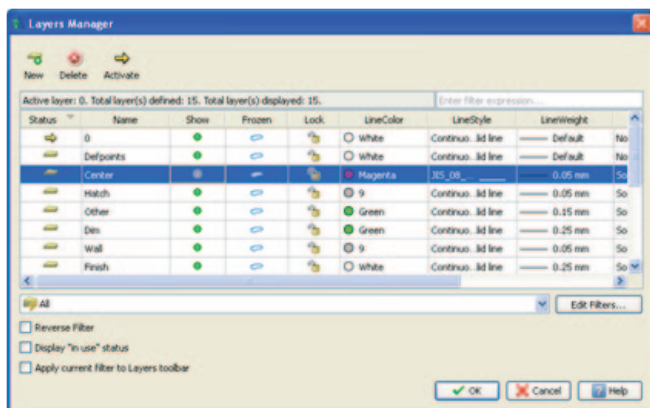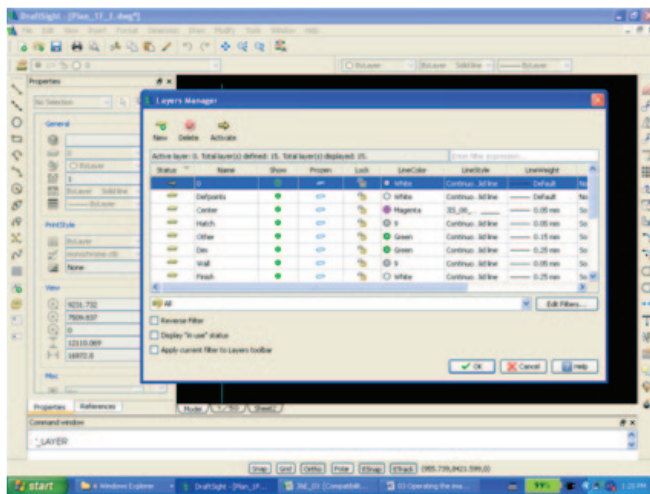The specified DWG file opens, and the drawing stored in this file appears on the screen.

Enlarge the displayed drawing by selecting "View" -> "Zoom" -> "Fit" from the Menu.

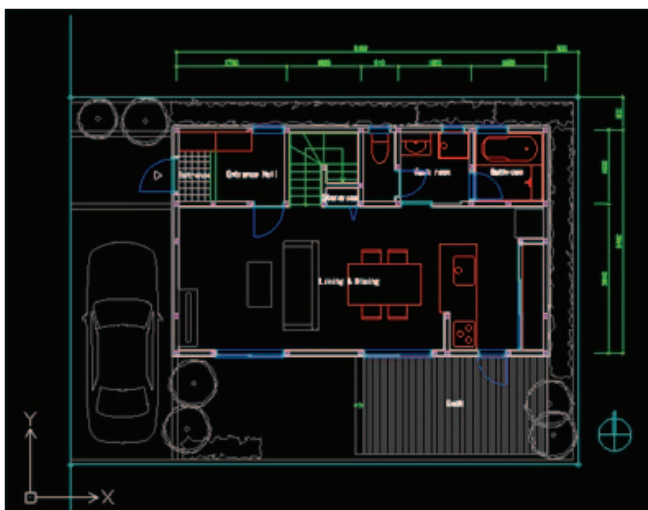2. Show or hide the Layer.

Hide the center line of the wall.

① Select "Format" -> "Layer" from the Menu to display the Layer Manager.

② On the Layers Manager that will be displayed, select the row that says "Center " under the Name column, and click.
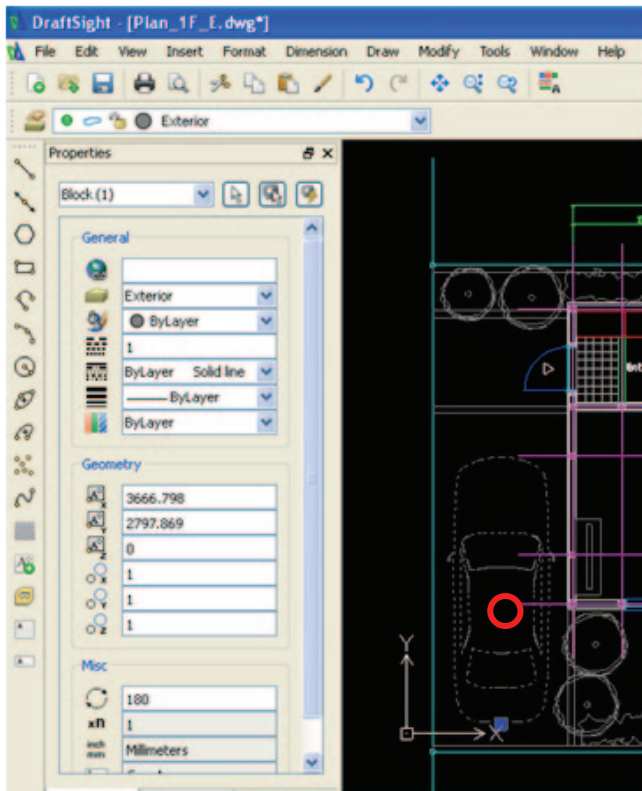
③ Click the green circle in the same row under the "Display" column to change its color to grey.

④ Click the "OK" button located at the lower right to close the Layers Manager.

The center line of the wall is no longer shown in the drawing.

## 3. Find the Image Layer where the graphics are, and hide the graphics.

Let's hide the automobile existing in one of the Layers.

① We don't know in which Layer the graphics for the automobile is sorted.   Let's find out. Place the cursor on the automobile and click it to put it into the selected state. See the Property palette on the left-hand side. In the Layer field, the name of the layer of where the selected graphics is sorted appears. It says "Exterior". To deselect the automobile, press the "ESC" key.

> Memo
> When the Property panel is not displayed, you can make it appear by selecting "Tools" -> "Properties" from the Menu.
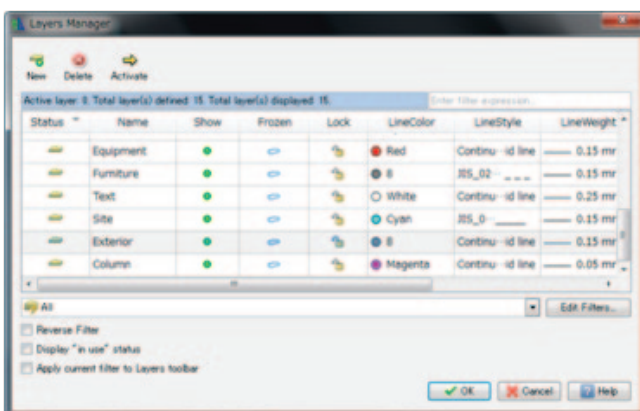

② In the same procedure explained earlier, display the Layer Manager, change the color of the circle in the row "Exterior " under the "Show" column to grey, and click the OK button to close the Layer Manager.
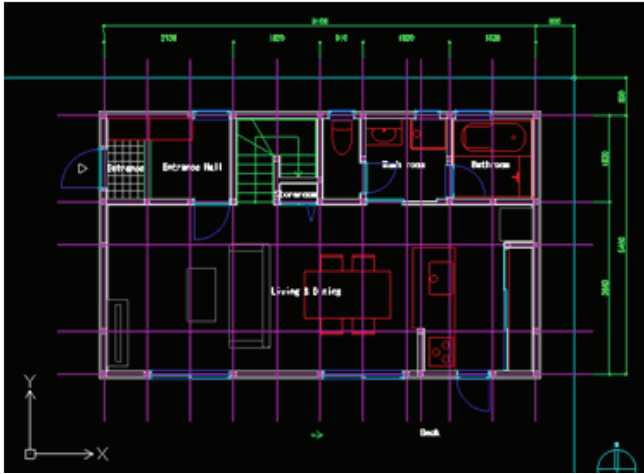
Then the Layers "External Element" will be switched to hide the mode. As a result, the automobile as well as all other objects belonging to the "Exterior" group will no longer appear on the displayed drawing.

4. Show all the image layers
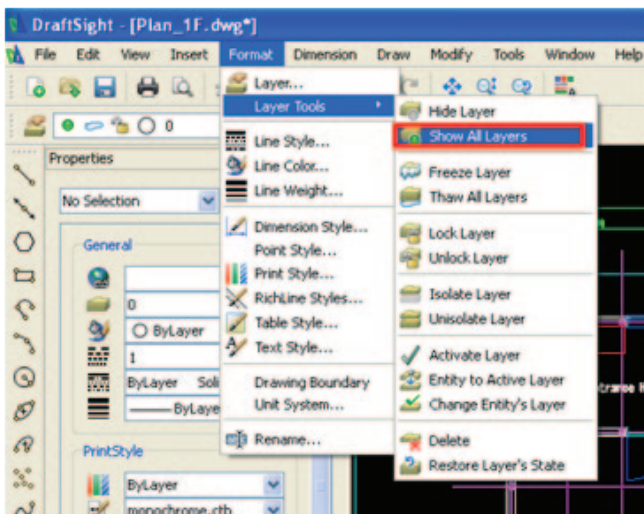
To put the displayed drawing back to the original state, show all the image layers.

① To do so, select Format -> Layer Tool -> Show All Layers from the Menu.
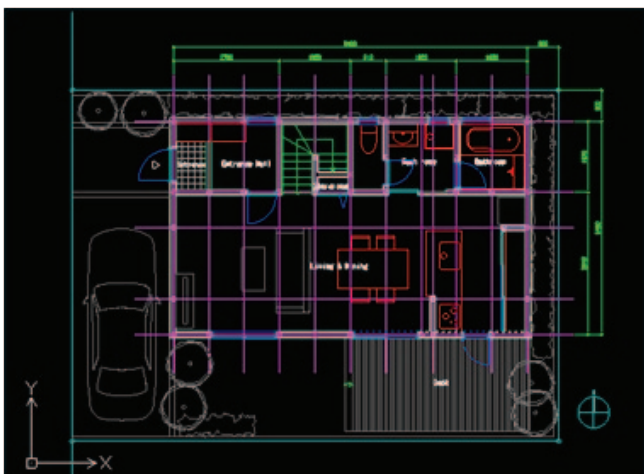
When all the Layers are switched to the Show mode, the Layers that have been switched earlier to the Hide mode will also reappear.

Memo: In addition to switching the Show / Hide mode of the Layers, they can also be set to the "Freeze" or "Lock" mode. The graphics on the Layer set to the Freeze mode will not appear until the Freeze mode is released. While the Freeze mode is active, the users cannot switch the Show / Hide mode of the graphics, nor can they edit them. On the other hand, when the graphics on the Layer is set to the Lock mode, the users can switch the Show / Hide mode of the graphics, but cannot edit them.

## 5. Change the Layers where the graphics are sorted.
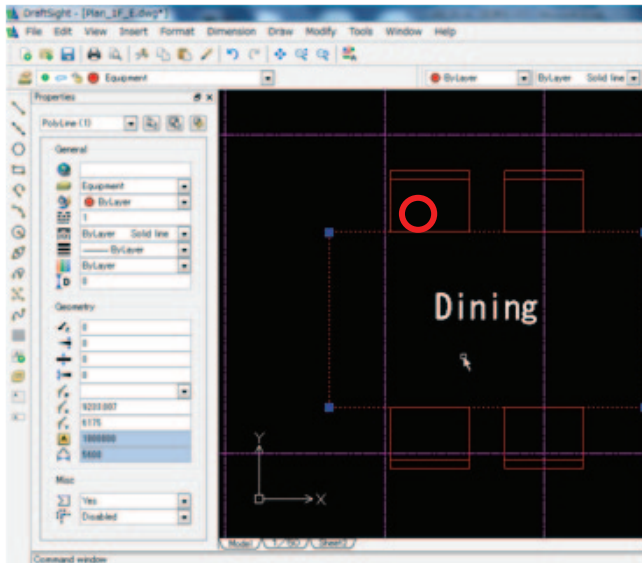
Change the Layers where the table and chairs in the Living / Dining room are sorted to "Furniture". To do so, first change the name of the Layer where the table is sorted, by using the Property Palette.

① Enlarge the area around the table.

② Click the table. You will notice that the name of the Layer containing the table is displayed as "Equipment" in the Property Palette.
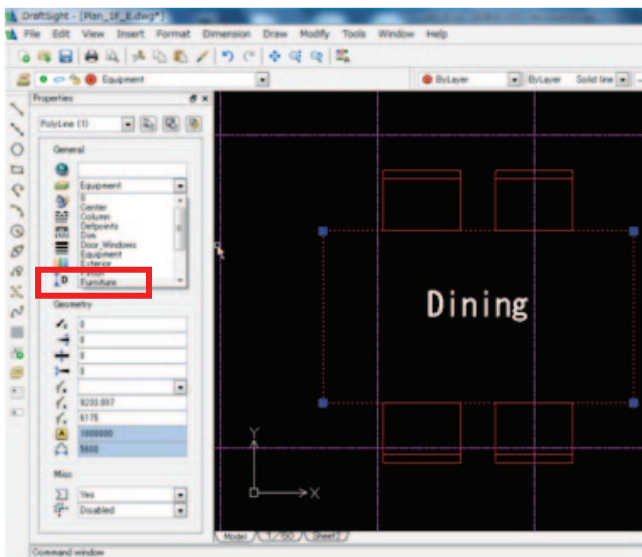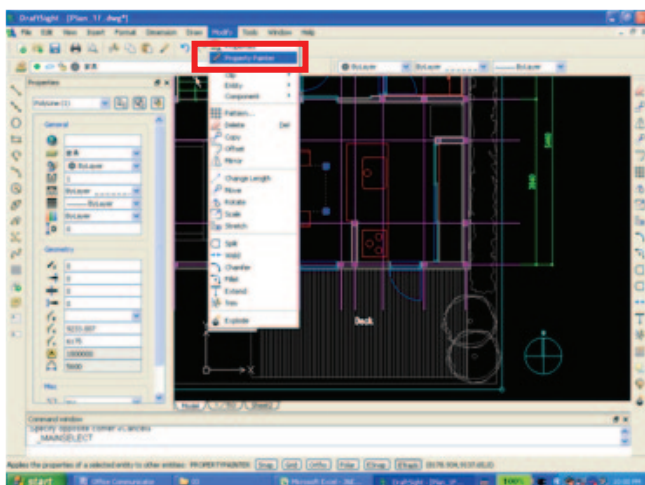

③ Select "Furniture" from the pull-down list of the Layer column in the Property Palette.
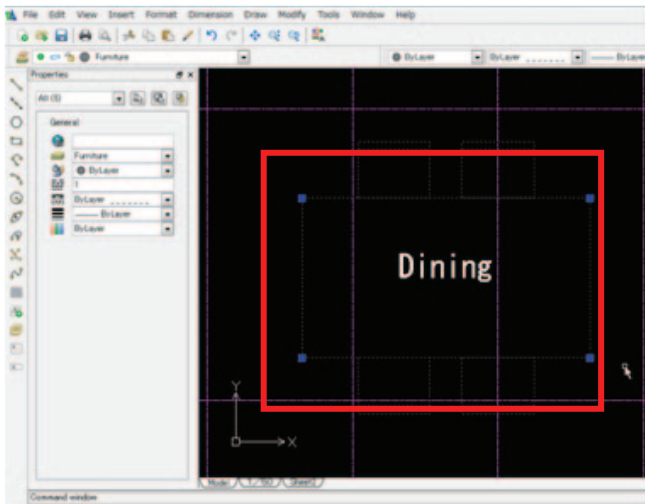
Then the By Layer color of the table in the Layer "Furniture" will become grey.

④ Release the table from the selected state by pressing the "ESC" key.


Now, we need to do the same for the Layer where the chairs are sorted. This time, rather than following the same procedure taken to change the name of the Layer where the table is sorted, we will try another method, which is to copy the Layer property of the table.
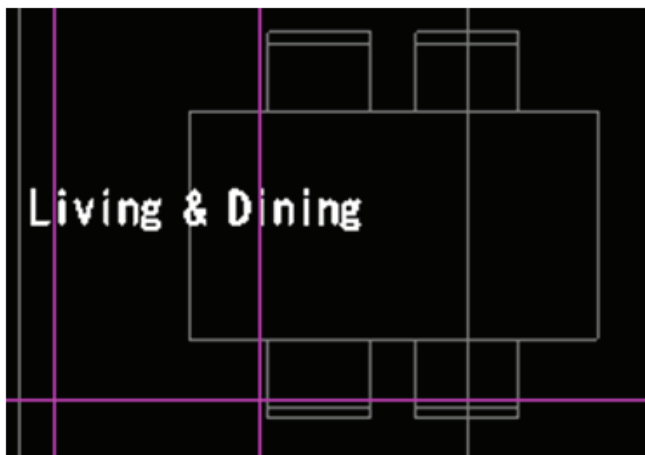
⑤ Select "Modify" -> "Property Painter" from the Menu.
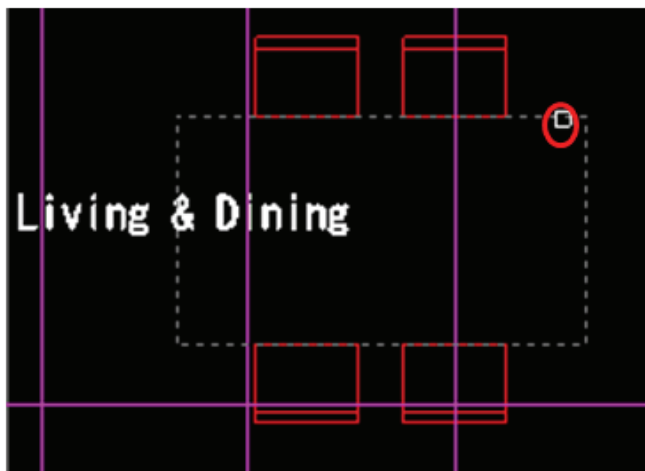
⑥ For "Specify the Source Entity»", click the table to select it as the source entity.
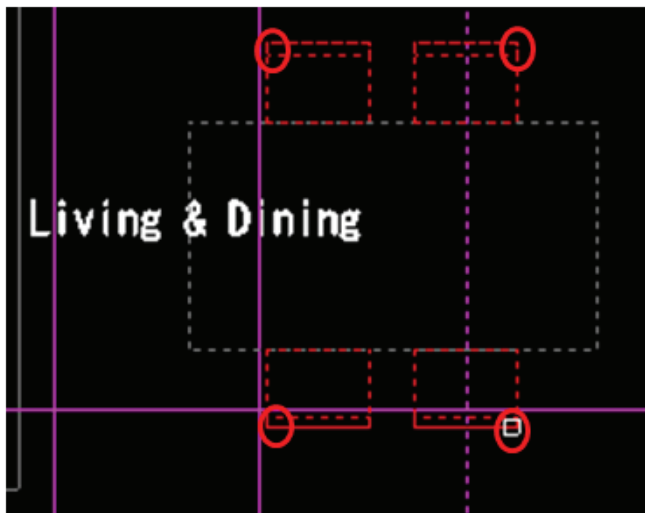
⑦ For "Specify the Target Entity»", click four chairs and put them in the selected state.

Summary
Drawing data that contain well-managed Layers are easy to handle and view. If the data entered in the drawing data are not properly sorted in layers, some graphics contained in the drawing may go unnoticed, serving no purpose. As a result, the reliability of the data itself may be ill-affected. Therefore, it is very important to enter and manage the drawing elements properly in layers to keep the drawing data clean.

Editing the Drawings

In this section, we will master commands like Copy, Trim, and Extend that are frequently used to edit drawings. We will also enter texts and dimensions. As the scenario for this section, we will add a new level to the porch located on the left side of the front gate. The key points of this operation are to execute the commands with an image of the result of the operation in mind, and to align the operation with the pre-defined Image Layer settings as well as with the pre-entered annotations (text, dimensions, etc).

1. Activate DraftSight and open the file named
"Plan_1F.dwg".
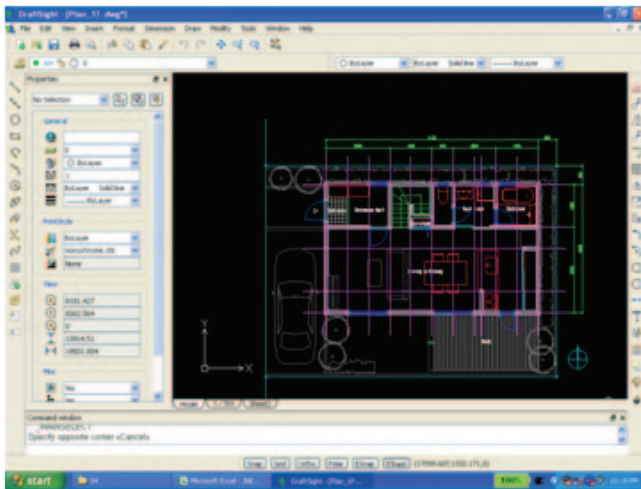
① Activate DraftSight and open the file named "Plan_1F.dwg". The specified DWG file opens, and the drawing stored in this file appears on the screen.Activate DraftSight and open the file named "Plan_1F.dwg". The specified DWG file opens, and the drawing stored in this file appears on the screen.

2. Draw the lines to represent the new level of the porch.
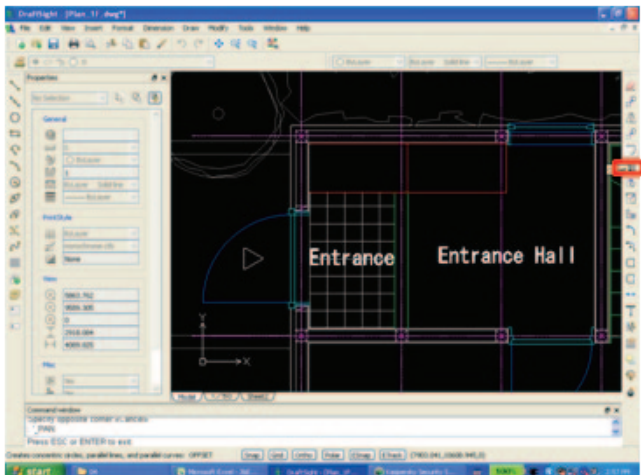
Use the Offset Command to draw the different levels of the porch. Draw a porch that is 900 in width and 300 in elevated level.
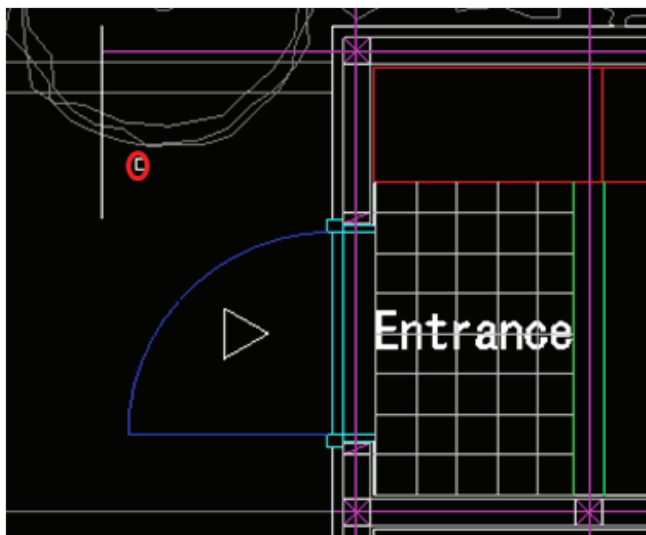
① Enlarge the left side of the front gate.

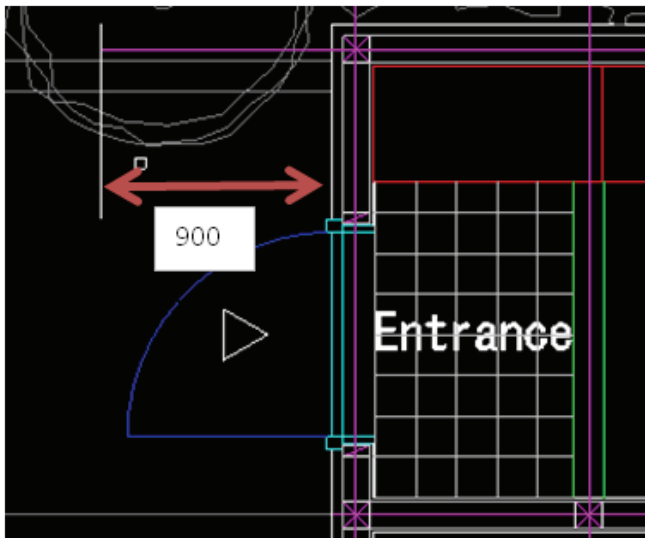② Select "Offset" from the tool bar on the right side, and
    click.
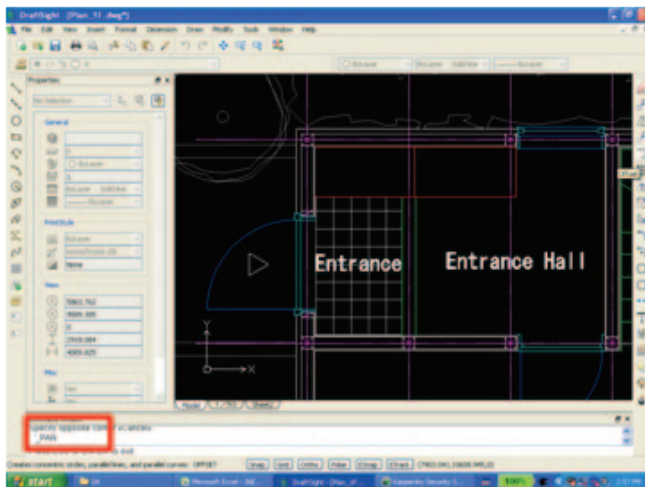
③ You now want to draw a line that ends at a position 900mm away from the outer wall surface Enter "900" by using the keyboard and press "Enter".

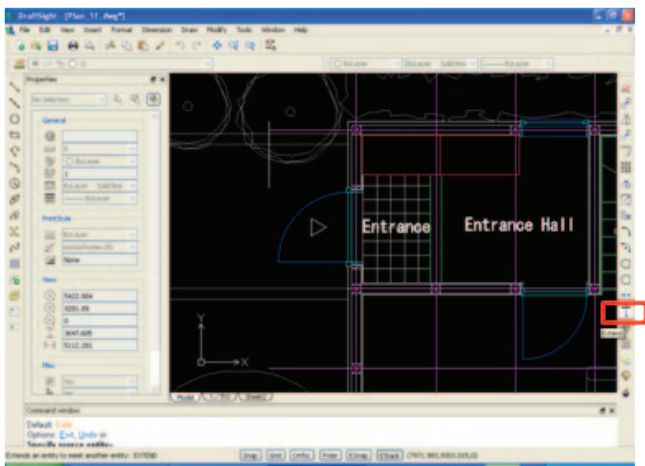④ "Specify the source entity»", click the outer wall line.

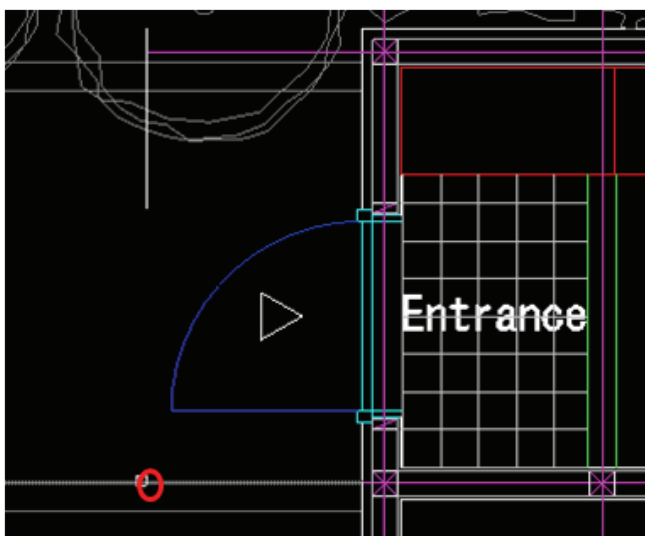⑤ To "Specify the side for destination", click a screen position on the left side.

Then the outer wall line is copied 900mm in parallel to the left.

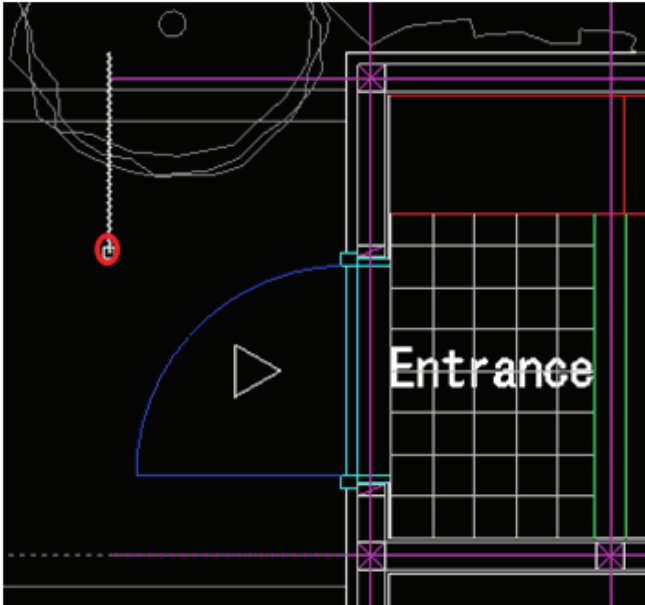**3. Extend the line.**

Extend the line that has been offset.
① Click "Extend" from the tool bar on the right.

Specify the border line to which the line is extended.

② For Specify the boundary edge» click the grey line located on the lower side of the external element area.

③ Press "Enter" to end the operation of selecting the boundary edge.

To put the displayed drawing back to the original state, show all the image layers.

④ To Specify segments to extend >>click the lower side of the offset line. Then the line will be extended to the specified boundary edge.
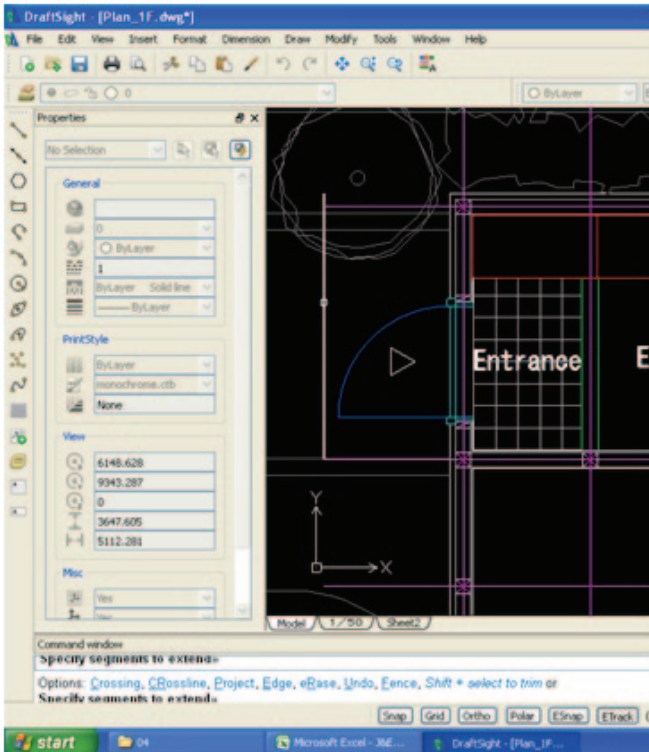
⑤ Press "Enter" to end the operation of extending the specified line.
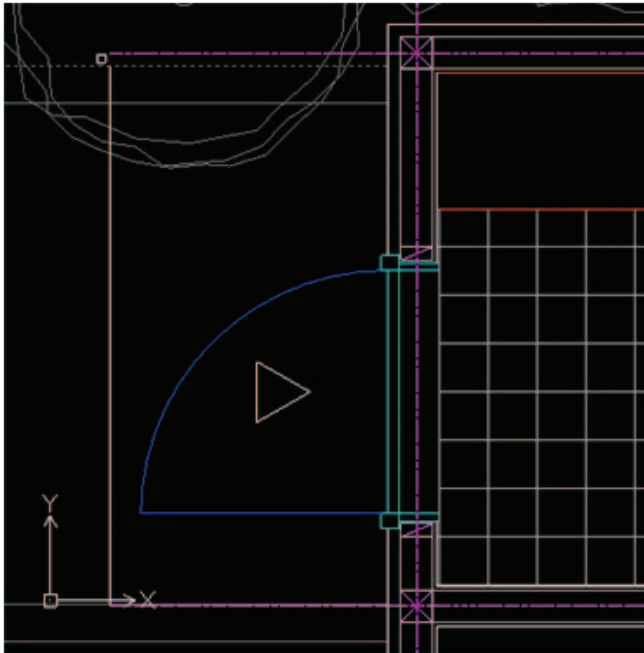
4. Trim (shorten) the line
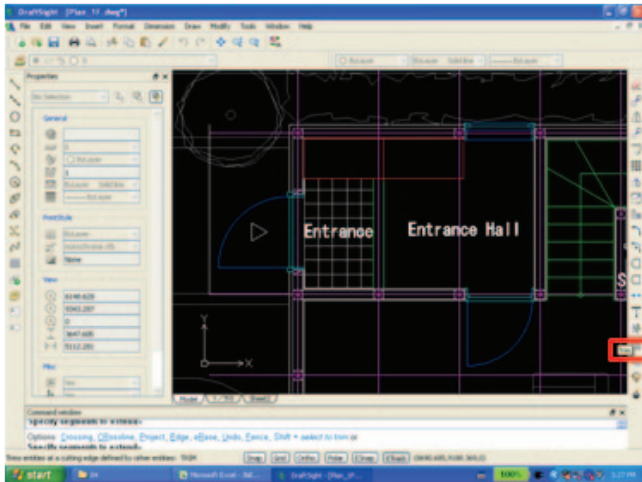
Trim the offset line.
① Click "Trim" from the tool bar on the right.
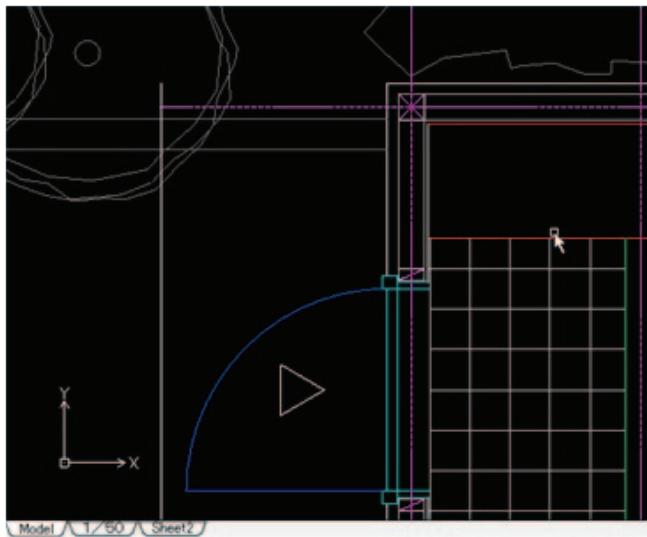
② For Specify cutting edges>> click the grey line located on the lower side of the external element area. Press "Enter" to end the operation of selecting the cutting edges.
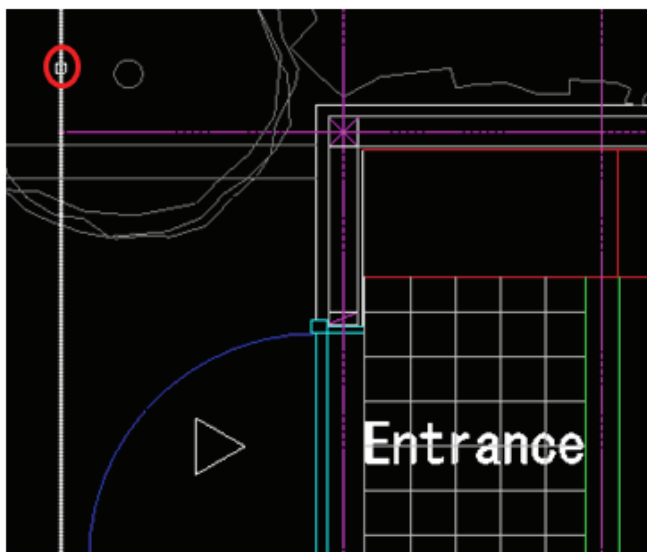
③ For specify segments to remove>>click the area located on the upper side of the border of the offset line.
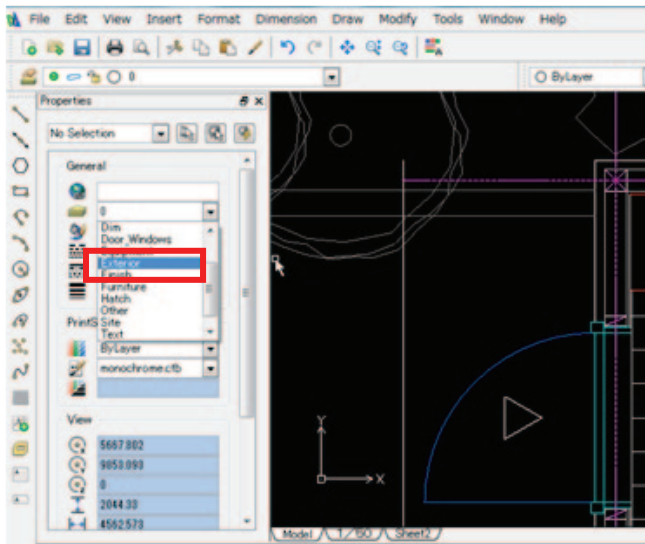
Then the line will be cut at the border line.

④ Press "Enter" to end the Trim command.

5. Change the Image Layer
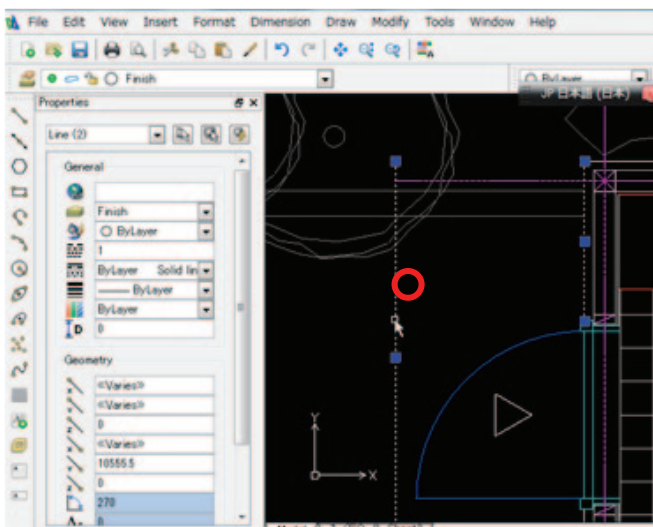
The offset line currently belongs to the Offset Source Property, and is sorted in the Layer called "Finish". We will change the Layer for the Offset Line to "Exterior".
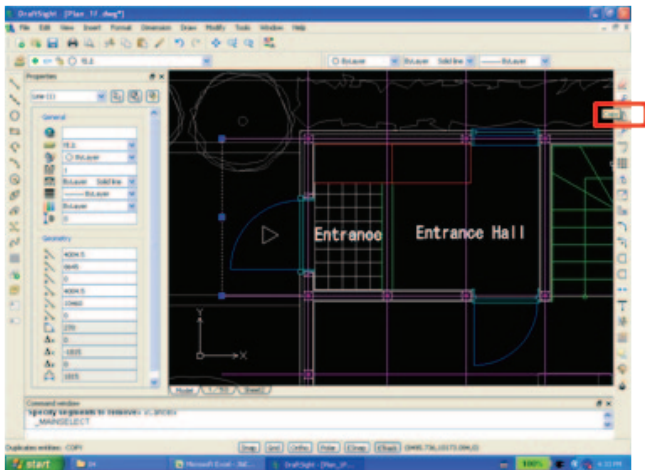
① Click the line that has been offset and edited in order to put it in the selected state.

② From the pull-down menu that lists the Layer items located in the Property Pallet on the left, select "Exterior" to change the Layer for the offset line to this item.

③ Release the offset line from the selected state by pressing the "ESC" key.
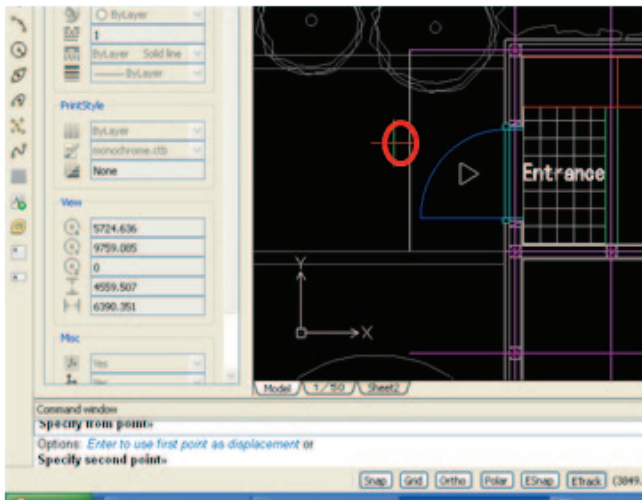
6. Copy the offset line.

Copy the offset line to the position 300mm left of this line to represent the additional level of the porch.
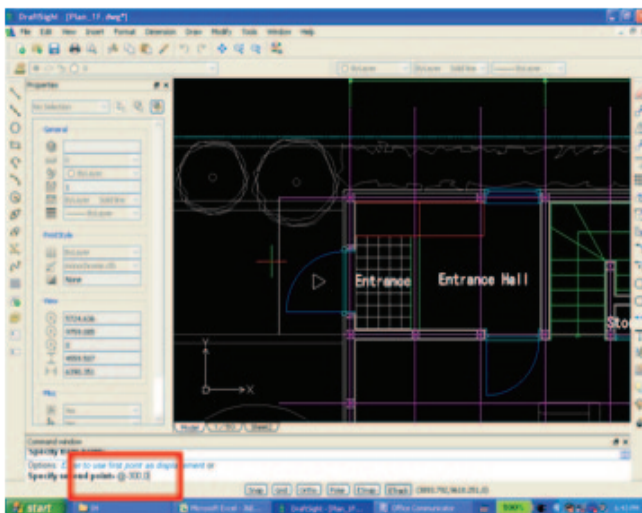① Click "Copy" from the tool bar on the right.
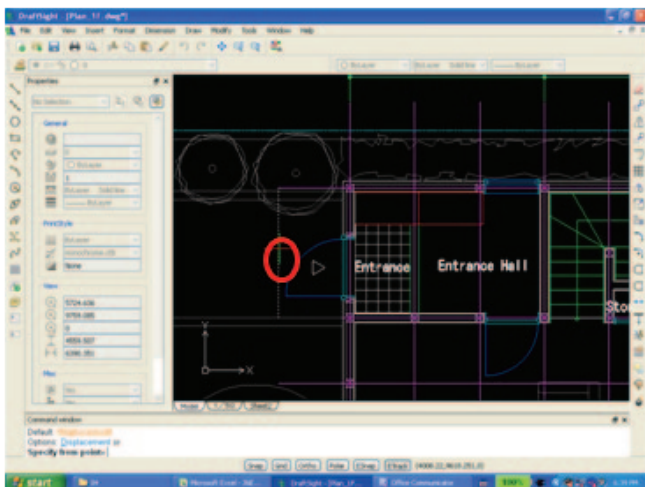
② To specify entities>>click the offset line to put it into the selected state.

③ Press "Enter" to end the operation for selecting the border line.

④ To specify the starting pint point >>click any point randomly
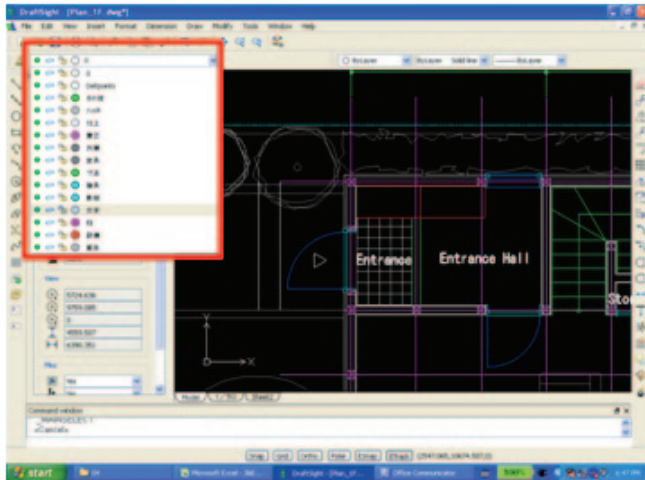

⑤ To "Specify second point»", enter the coordinate values "@-300,0" by using the keyboard, and press "Enter". Then a duplicate line will appear at point -300 in X direction and point 0 in Y direction from the starting point.
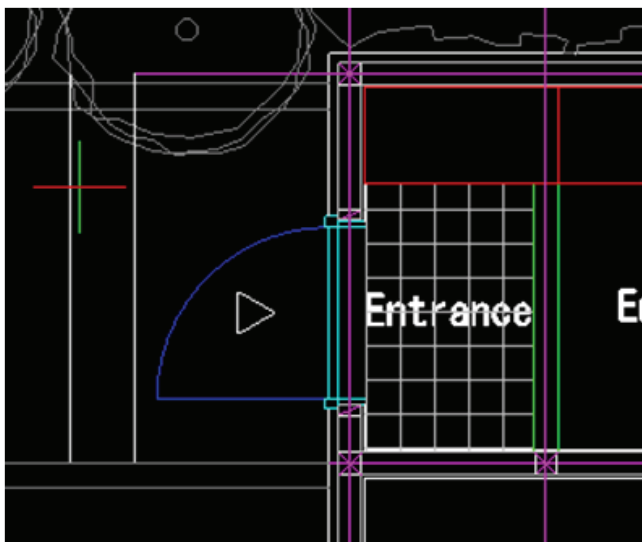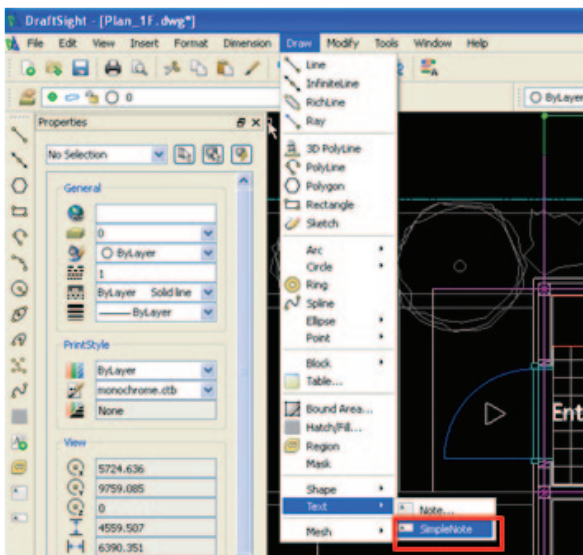

⑥ Press "Enter" to end the Copy command.
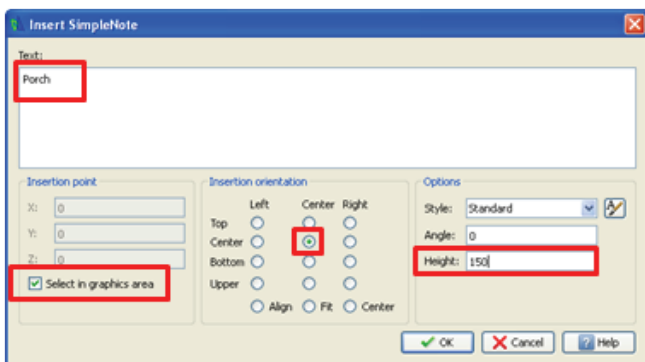
7. Enter the text "Porch".

Enter the text "Porch" anywhere close to the line that has just been added. Since we are going to enter text, we need to change the active layer to "Text".

① From the pull-down menu opened on the upper left side, select "Text" as the active image layer.

② Select Draw > text > Simple Note from the Menu

③ Then the Insert Simple Note dialog box will appear. Enter the following information in the specific fields:

Text: "Porch"

Insertion point: Place a check next to "Select in the graphics area".

Insertion orientation: Select the radio button at Center.

Height: 150

After completing the above entries, click "OK".

Memo: The height of the text is set by considering how the text will appear when the drawing is printed out. If we want to make the text appear 3mm tall when printed in 1/50 size, the calculation will be 50×3mm=150.
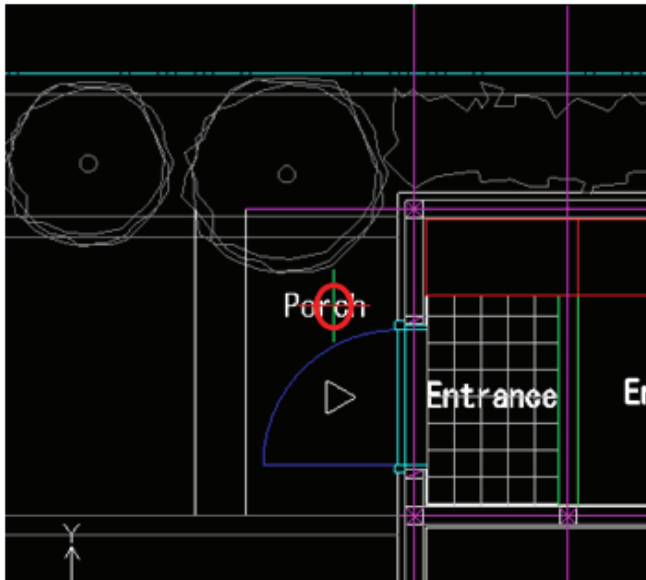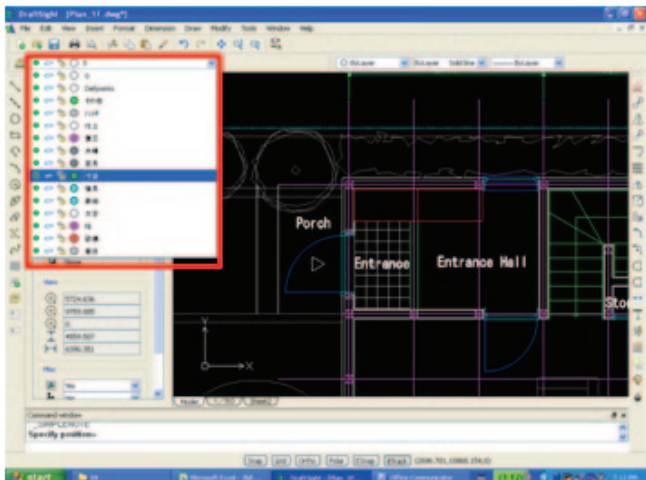
④ Click any desirable position randomly to specify it as the location to enter the text.
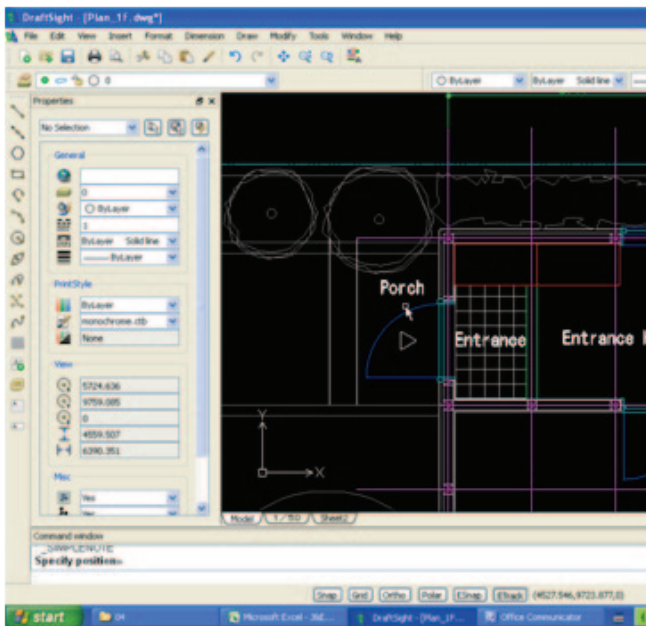
Enter the text.
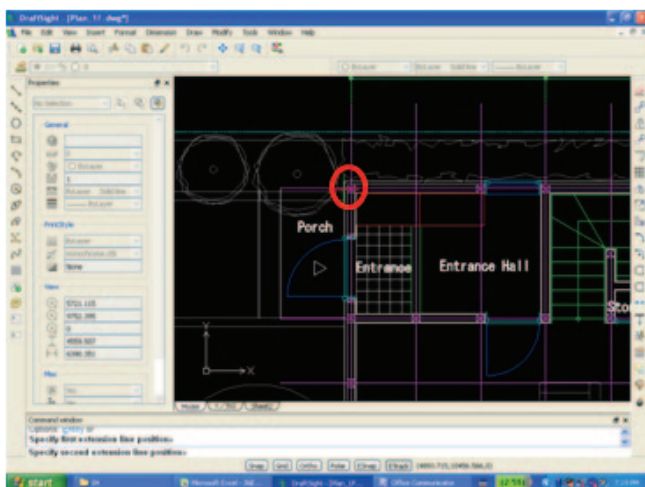
8. Enter the dimensions.

Now, let's enter the dimensions that represent the width and the elevated level of the porch. Since we are going to enter the dimensions, we need to change the active Layer to "Dimensions".
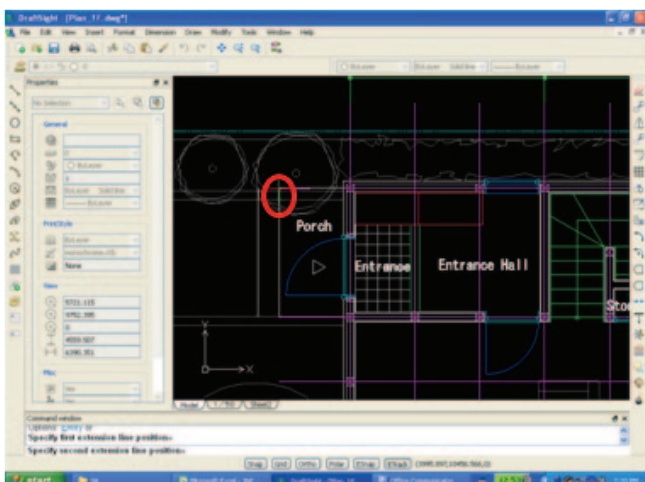
① From the pull-down menu opened on the upper left side, select "Dimensions" as the active Layer.
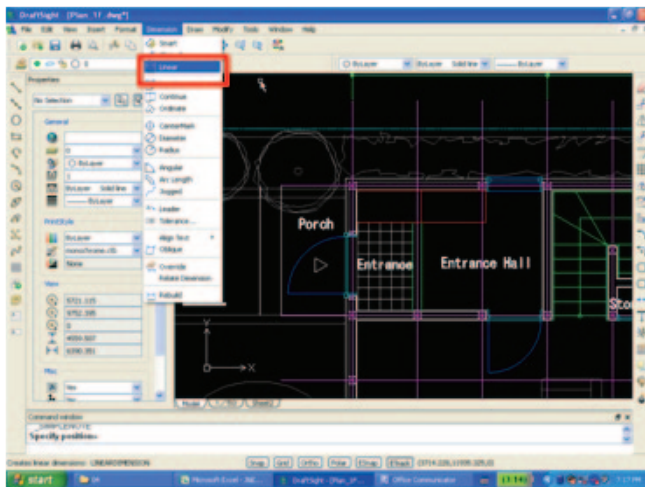
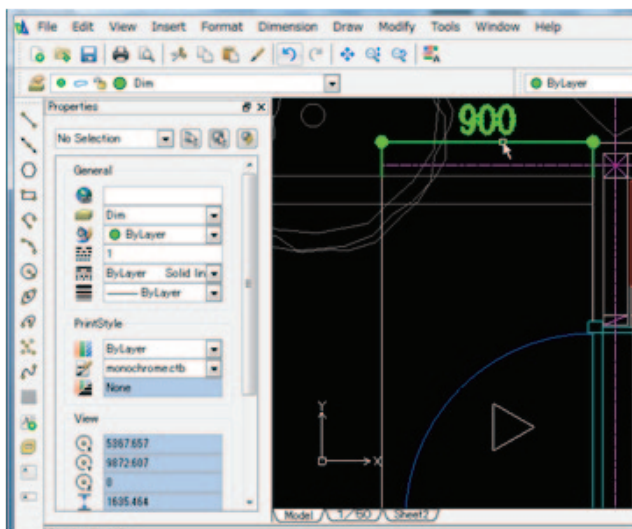② Select Dimension>Linear from the Menu

③ To be capitalized first extension line position>>place the cursor to the point where the grey line of the external element and the outer wall line intersect, and click when the endpoint snap appears.
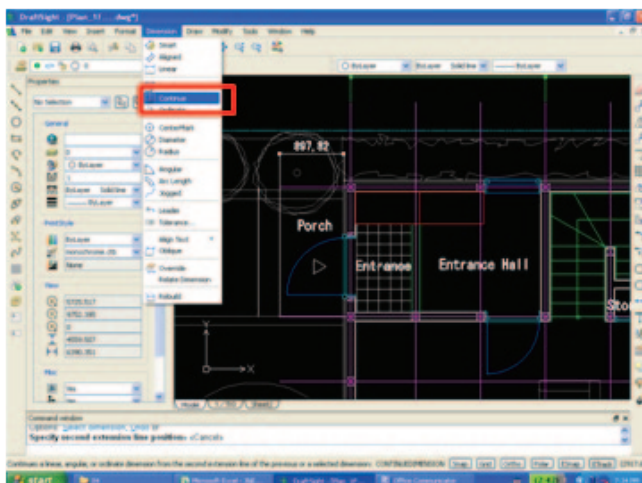

④ To be capitalized second extension line position>>place the cursor to the edge point of the line that has been offset in the beginning, and click when the endpoint snap appears.
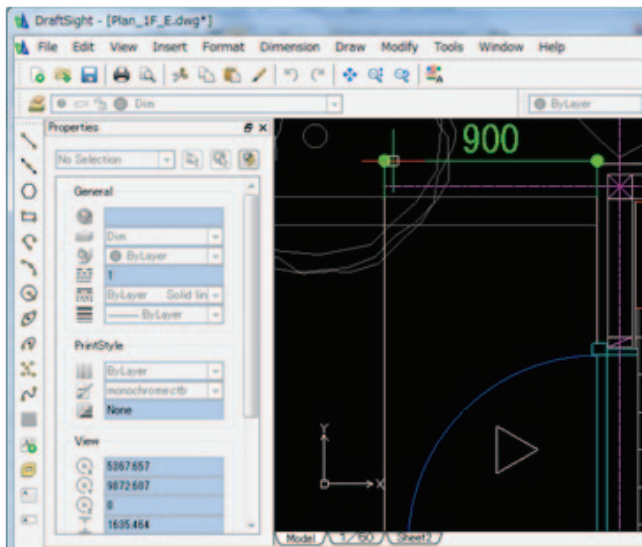
⑤ To Specify dimension line position>>move the cursor upward and click any desirable position randomly.
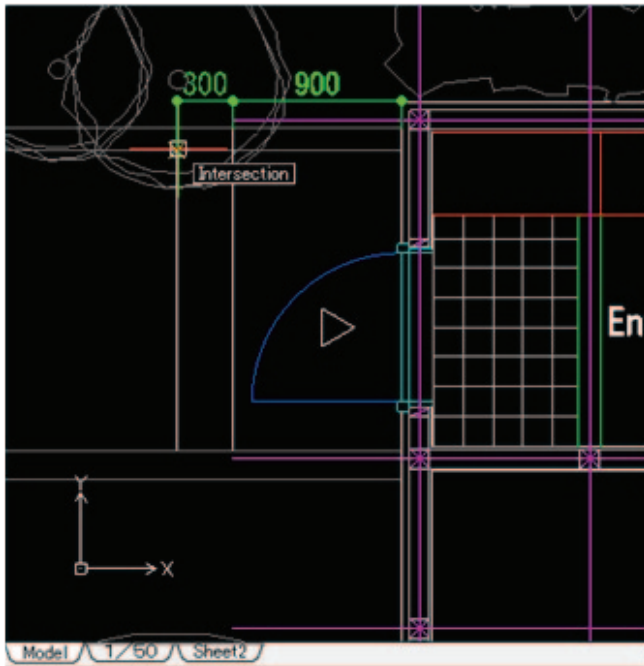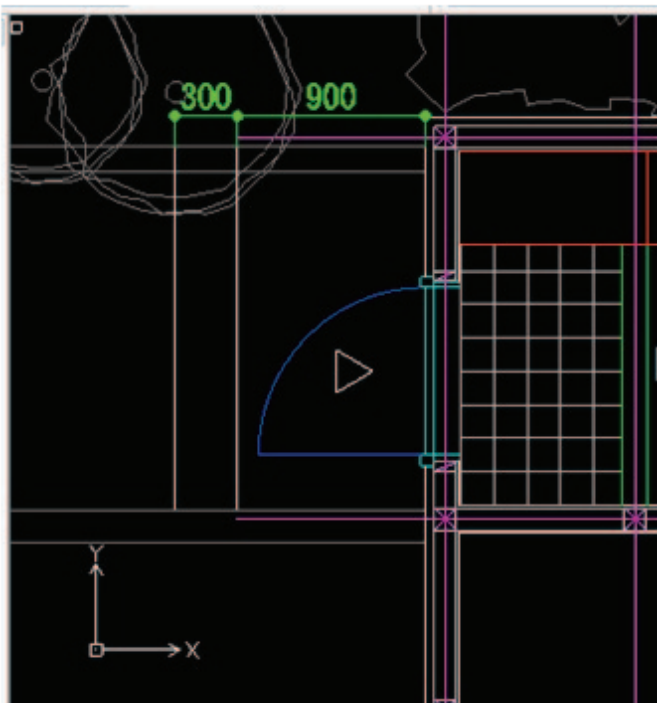

Enter the dimensions.

Enter the dimension of the elevated level of the porch. Here, we will use the Continue command because we are going to enter another dimension in series.

⑥ Select Dimension> Continue from the Menu.

⑦ To specify second extension line position>>place the cursor to the edge point of the elevated level that has been copied, and click when the endpoint snap appears.

End the "Continue" command by pressing the "ESC" key. The operation to enter the dimensions is now completed.

Summary
This section focused on introducing the frequently-used commands like Trim and Copy. But DraftSight has many more commands that are useful. By knowing them and being able to use them effectively in suitable situations, users will be able to improve the efficiency of their edit work considerably. Therefore, please make it a point to use the command references and other sources of information to gain a good knowledge of all the available commands.

# Creating and Inserting Blocks

A block is used to handle as a component like furniture or equipment as one graphical object. By collectively defining the lines and circles and all other graphical elements that constitute the component as one block, these graphical elements can be reused by inserting in block units for the amount required. Moreover, the modifications and changes necessary for these graphical elements belonging to the same block can also be executed at once. In this section, we will look at the procedures for creating, inserting, and editing blocks, as well as the distinctive features of the block.

1. Activate DraftSight and open the file named "Plan_1F.dwg".
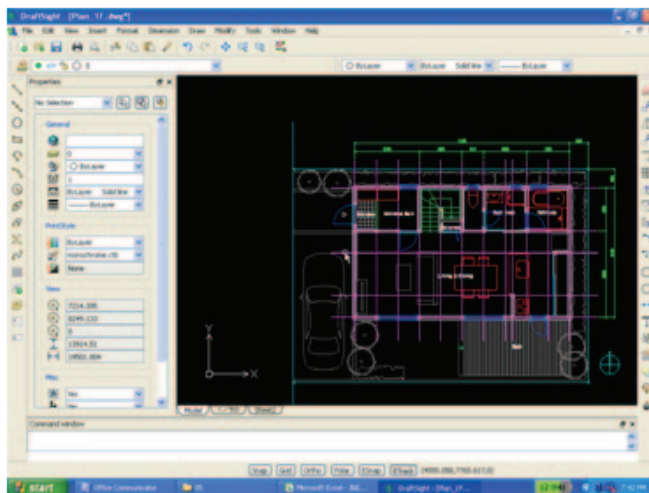
① Activate DraftSight and open the file named "Plan_1F.dwg".The specified DWG file opens, and the drawing stored in this file appears on the screen.

2. Create a block.
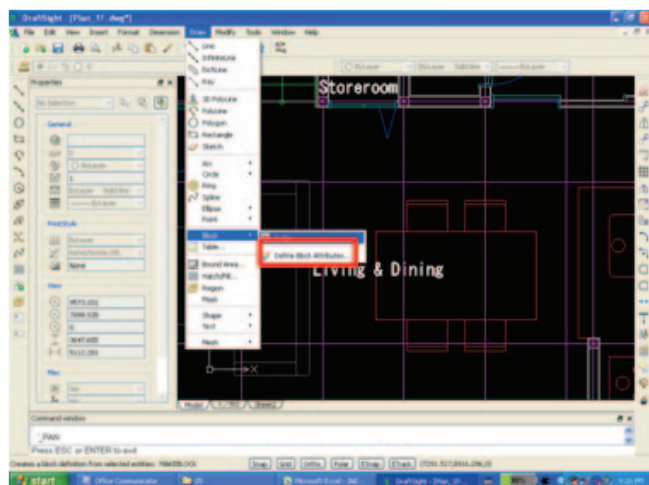
Group multiple graphical elements into a single block. Here, we will group the dining table and four chairs into one block.

① Enlarge the area around the dining table.
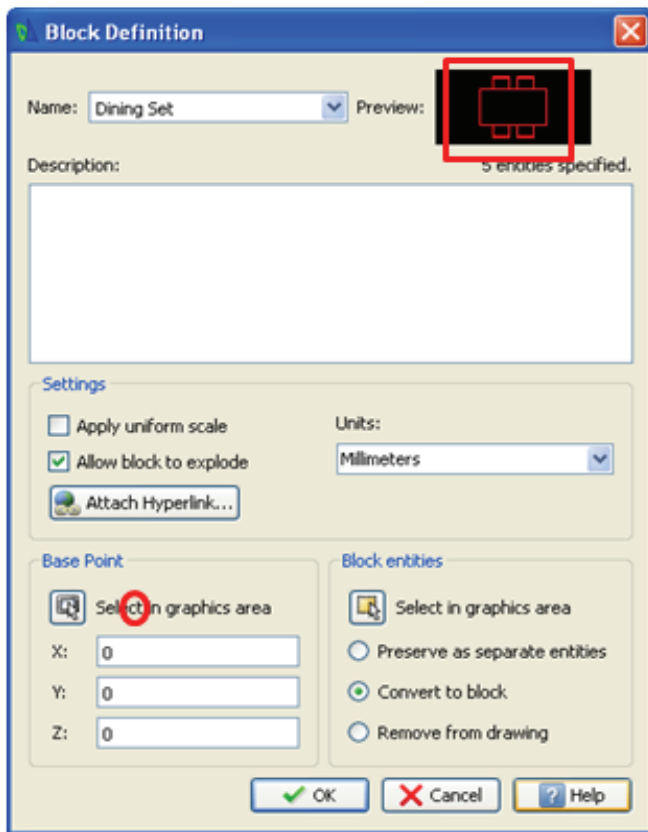
② Select "Draw" -> "Block" -> "Define" from the Menu.
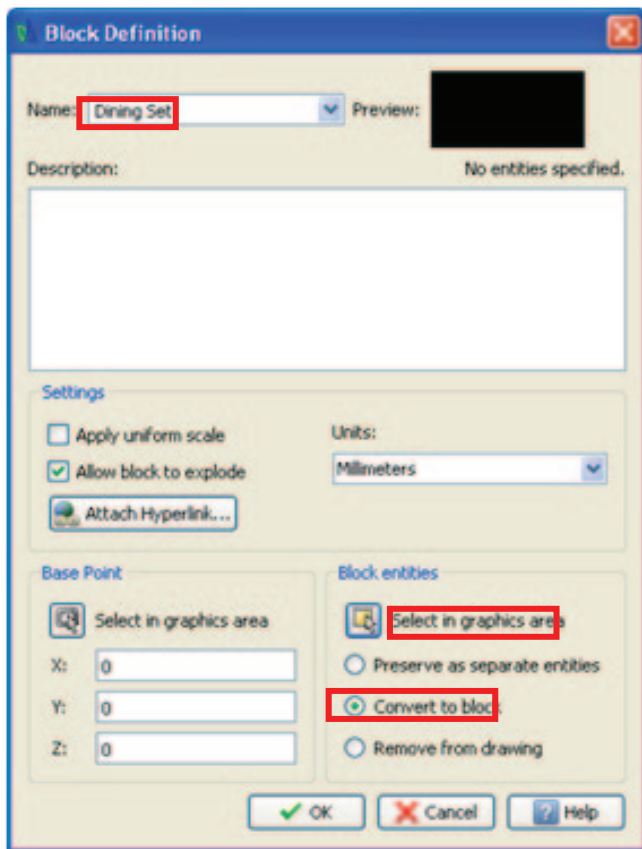
③ "Block Definition" dialog box will appear. Enter "Dining Set" as the name of the block. Place a tick next to the "Convert to block" item under the "Block entities" section. Then click "Select in graphics area".
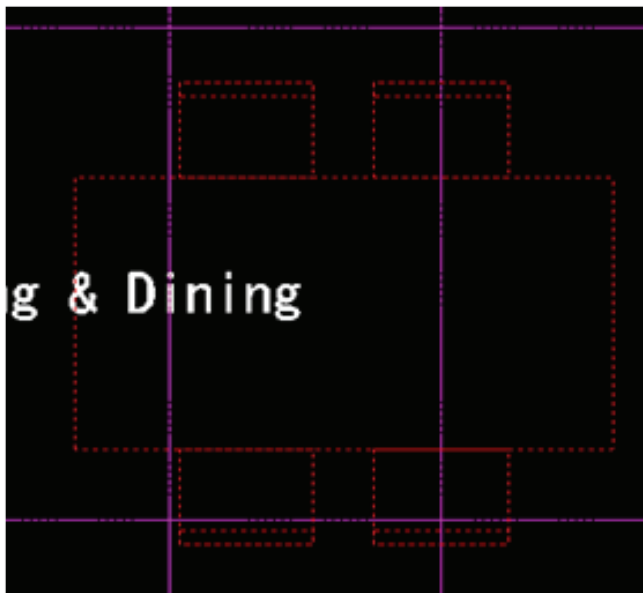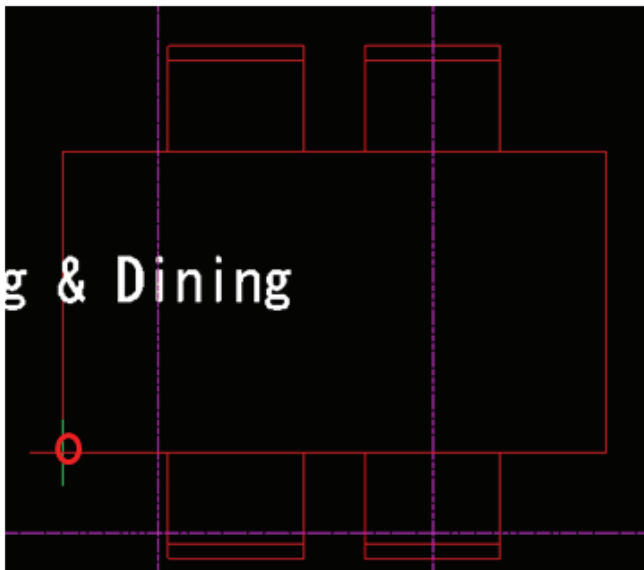

④ To "Specify entities»", click the table and the four chairs to put them in the selected state.
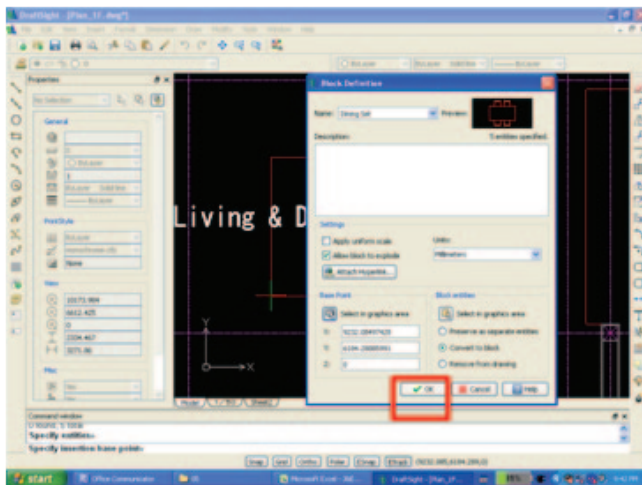
⑤ After selecting them, press "Enter".

The dialog box will reappear.
You will see the selected graphical elements shown on the upper right thumbnail section.
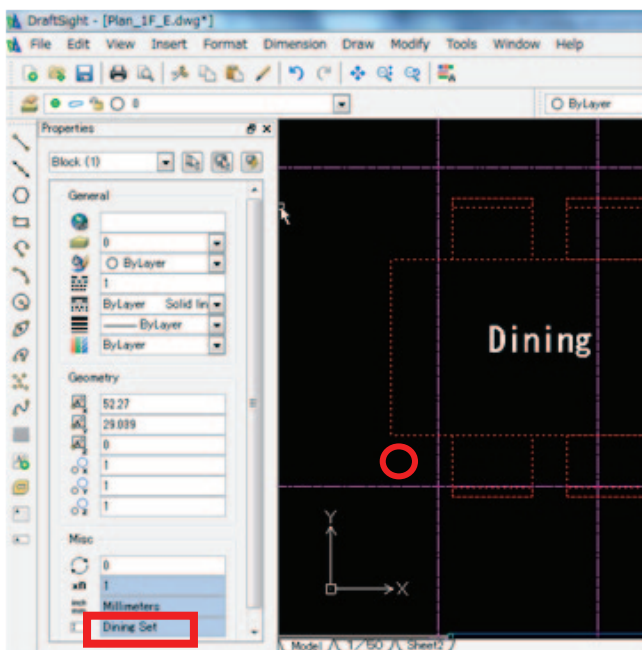
⑥ Then click "Select in graphics area" item under the "Base Point" section.

⑦ To "Specify insertion base point»", place the cursor in the lower left corner of the table, click when the endpoint snap appears.

⑧ The dialog box will reappear again.
Click the "OK" button and close the dialog box.

Confirm that the selected graphical elements are grouped in one block.

⑨ Click one of the lines of the table to put the table into the selected state. You should be able to see that the table as well as the chairs are in the selected state. The bottom field in the Property palette should display "Dining Set". If you can confirm all of them as described, you know that they have been successfully grouped into one block.
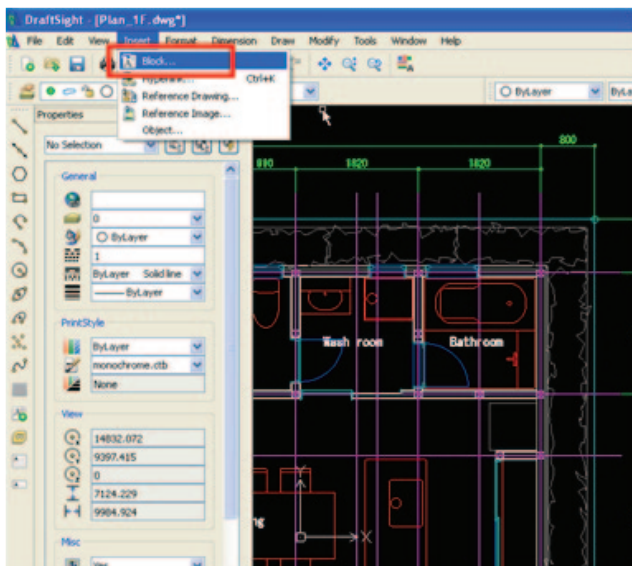
⑩ Release these elements from the selected state by pressing the "ESC" key.
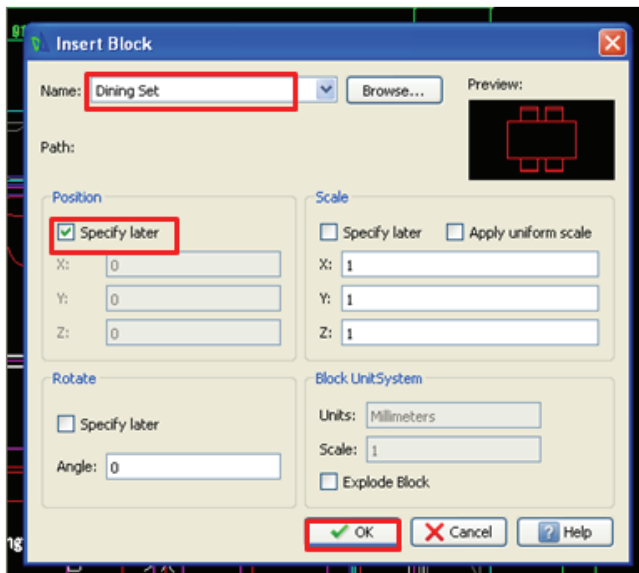
3. Insert the block.
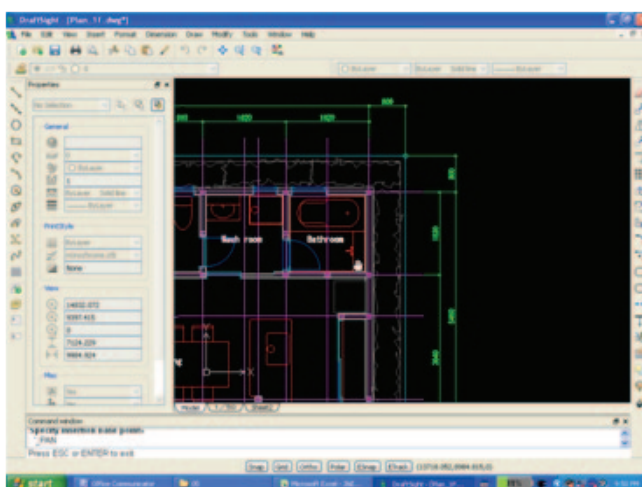
Insert the block called "Dining Set" that you have just created.
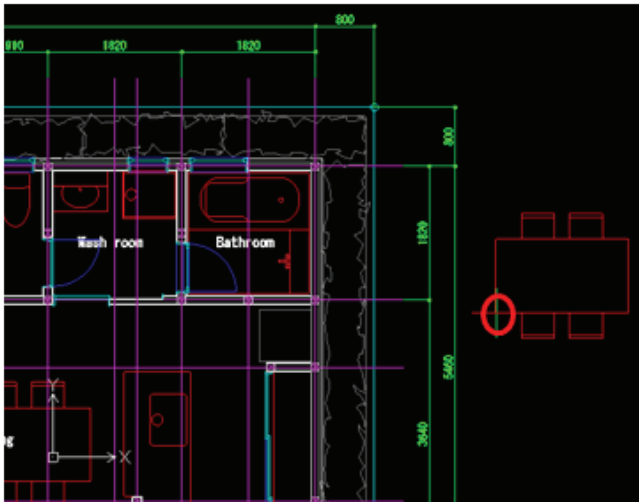
① Move the drawing so that there will be an open space on the right side of the screen.

② Select "Insert" -> "Block" from the Menu

③ "Insert Block" dialog box will appear. Select "Dining Set" from the pull-down list opened at the Name field. Place a check next to "Specify later" under the Position column. Then click "OK".
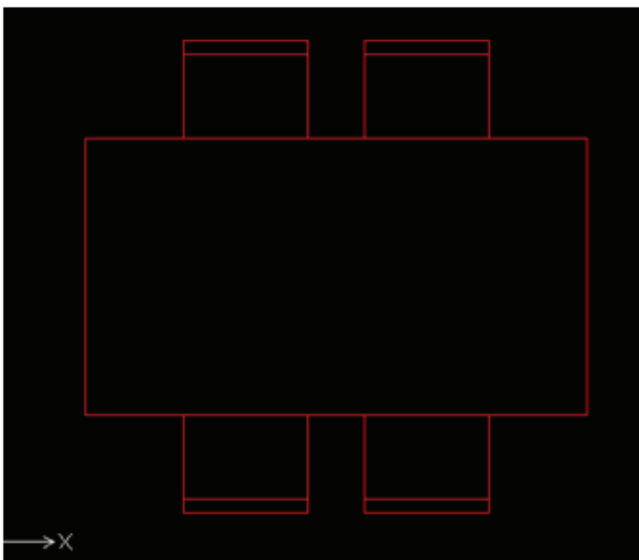
④ The "Dining Set" block will appear on the cursor. To "Specify destination>> select a desirable point randomly somewhere in the open space on the right side of the screen, and click to insert the block.
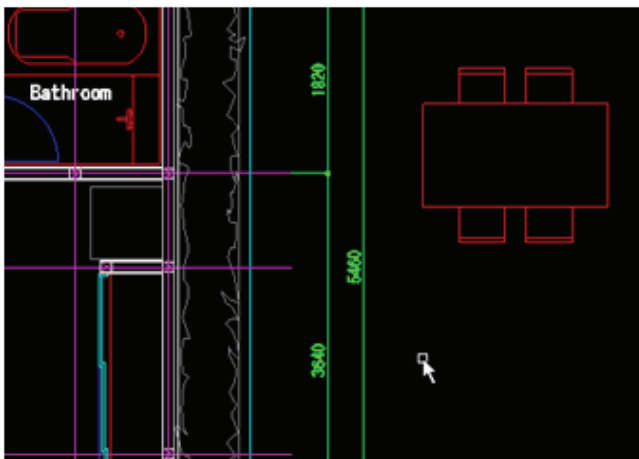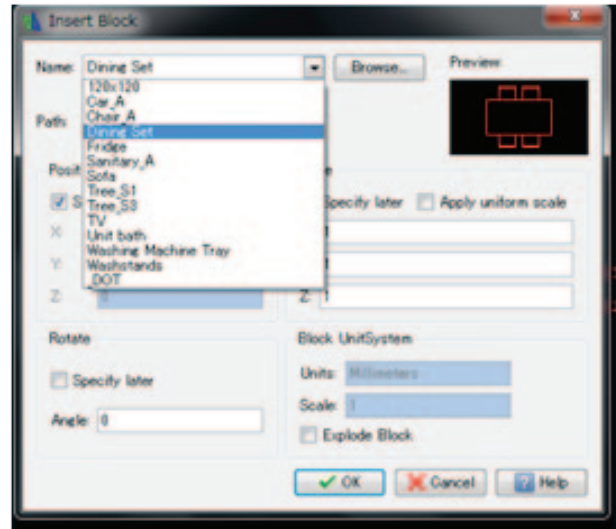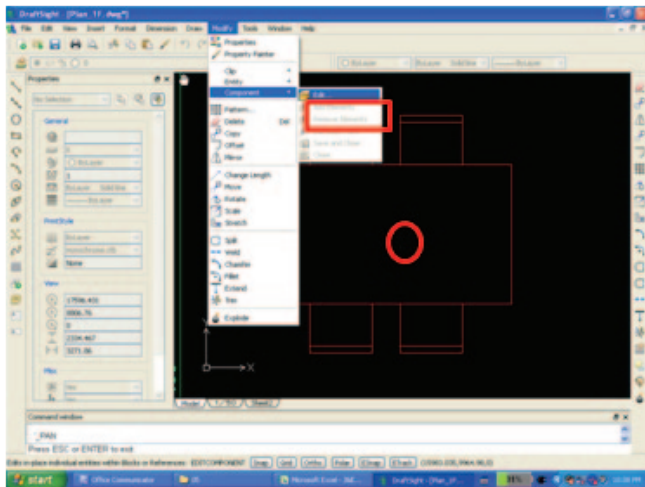

Memo: The pull-down list opened from the "Block Insert" dialog box lists. You can also enter a DWG file as a block.
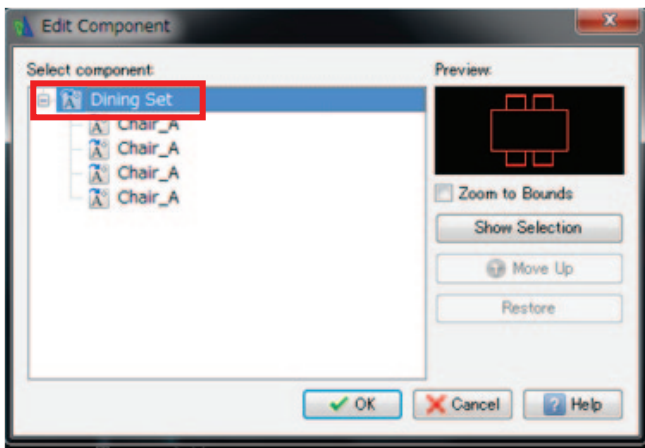
4 . Edit the block

Edit the chairs belonging to the "Dining Set" block to change them to round chairs.

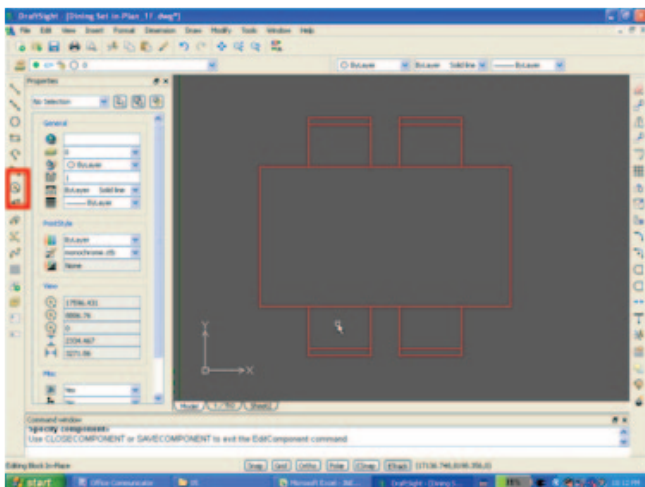① Enlarge the dining Set area that you just inserted.

② Select "Modify" -> "Component" -> "Edit" from the Menu.

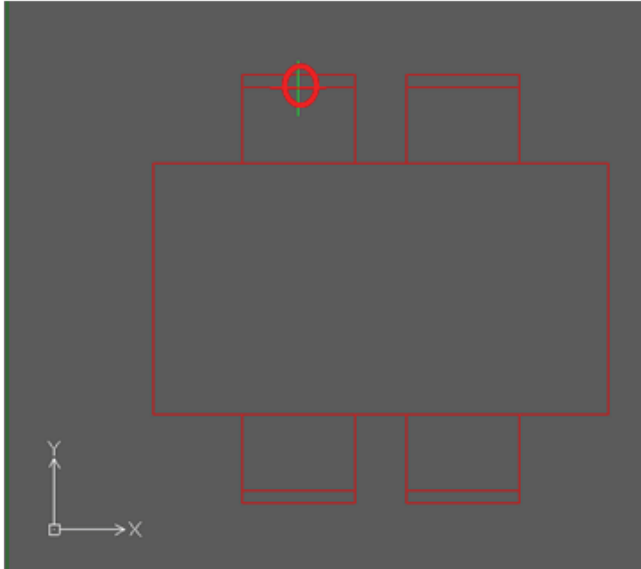③ For "Specify component»", click the dining Set that has been inserted.

④ "Edit Component" dialog box appears.
Confirm that "Dining Set" displayed in the list is highlighted (reversed display). Then click "OK".
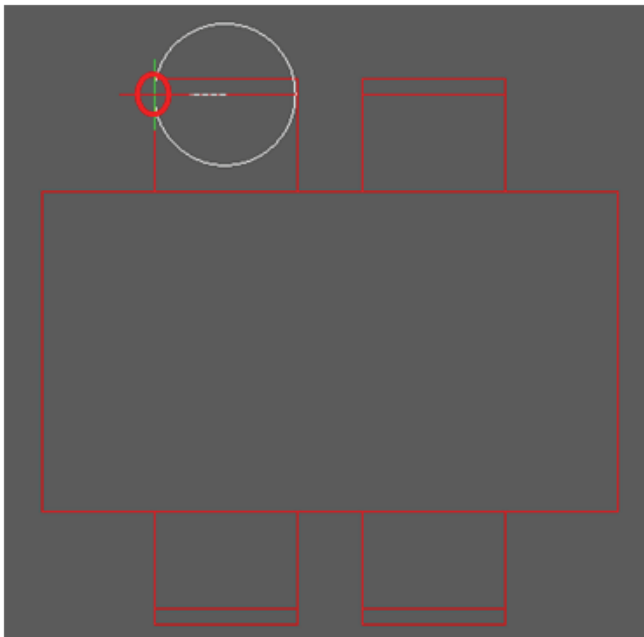

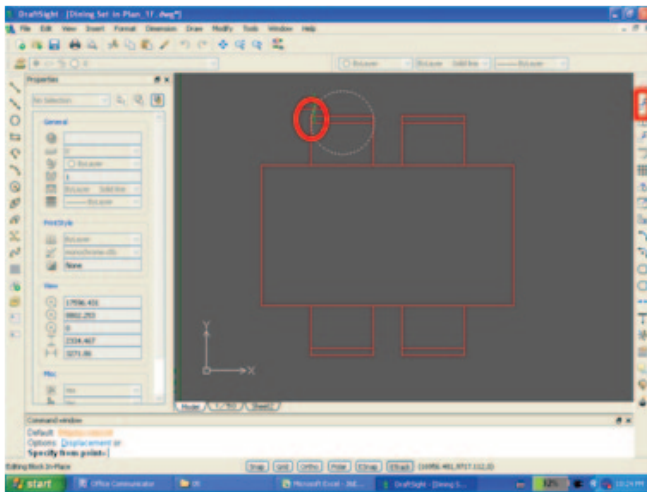The background color of the screen will turn grey.

⑤ Select the "Circle" command to draw a round circle.

⑥ To Specify center point» place the cursor on the midpoint of the horizontal line running across the chair.

⑦ To "Specify radius»", place the cursor on the left edge point of the line, and click it when endpoint snap appears. Then draw a circle.
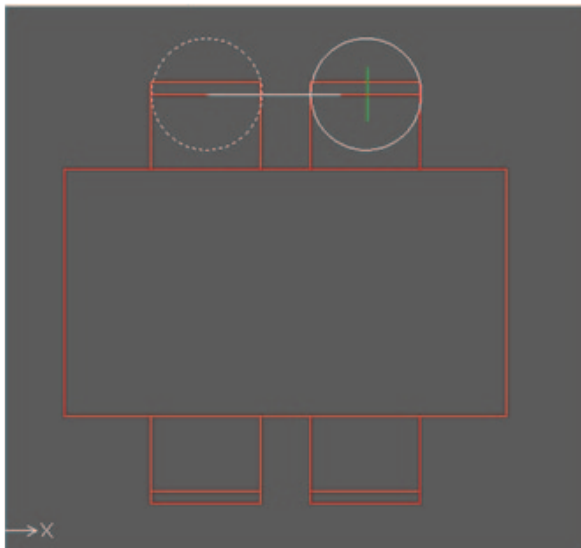
⑧ Since there is a need to copy the drawn circle, select the "Copy" command.

⑨ To "Specify entities»", click the circle and put it into the selected state. Then press "Enter" to finalize the entity to be copied.
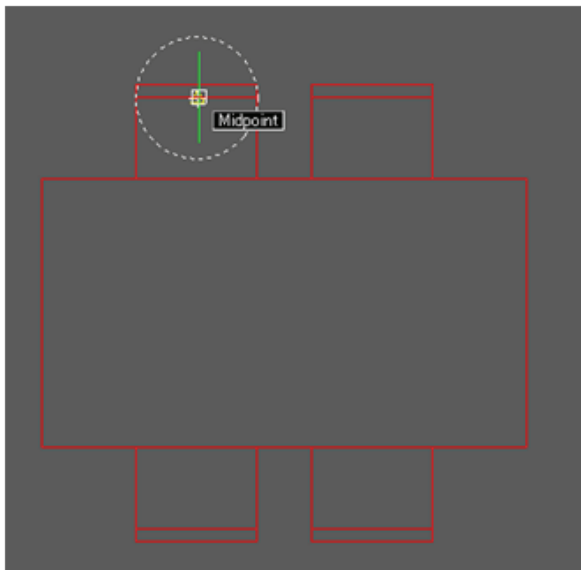
⑩ To "Specify the strating point»", specify the midpoint that is at the same position as the point specified at the center of the circle by using the midpoint snap method.
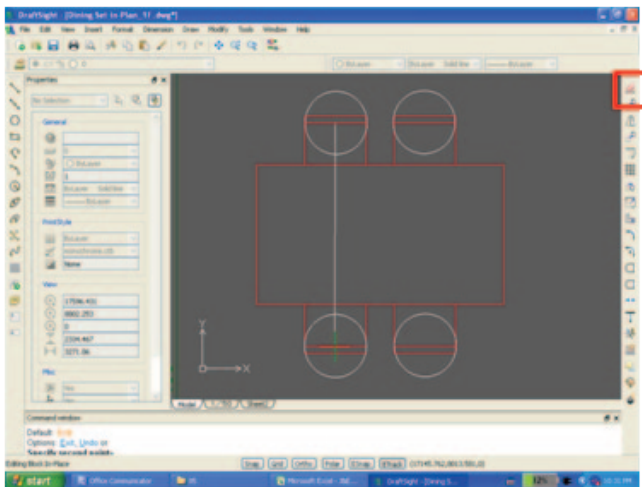

⑪ To "Specify second point»", specify the same midpoint for all three remaining chairs by using the midpoint snap, and copy them.
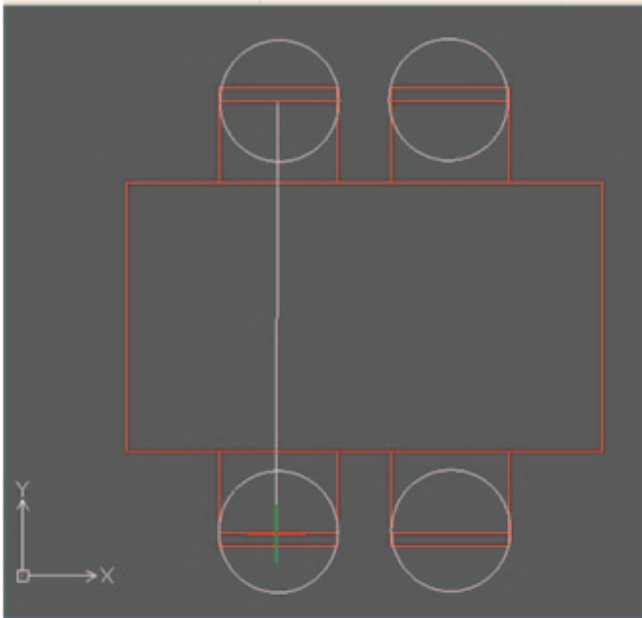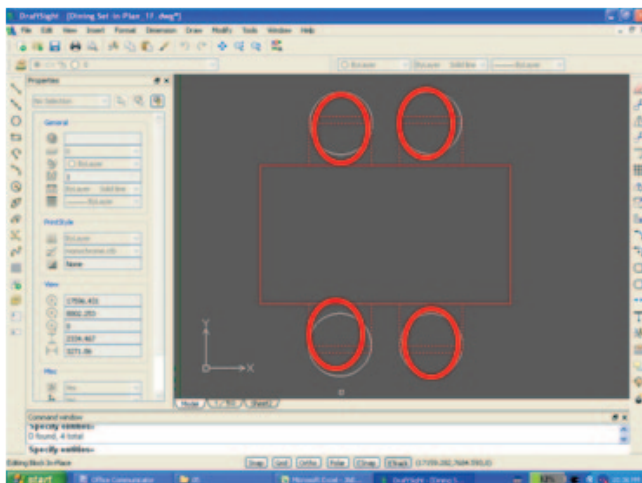
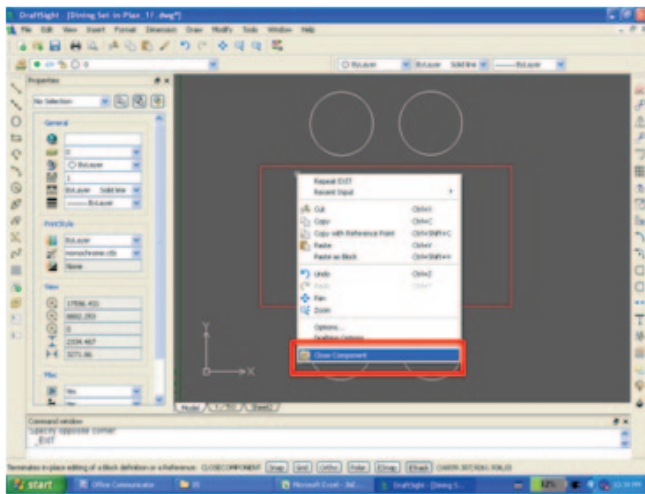A circle is now drawn respectively on all the four chairs.

*Need to press ENTER to exit copy command.

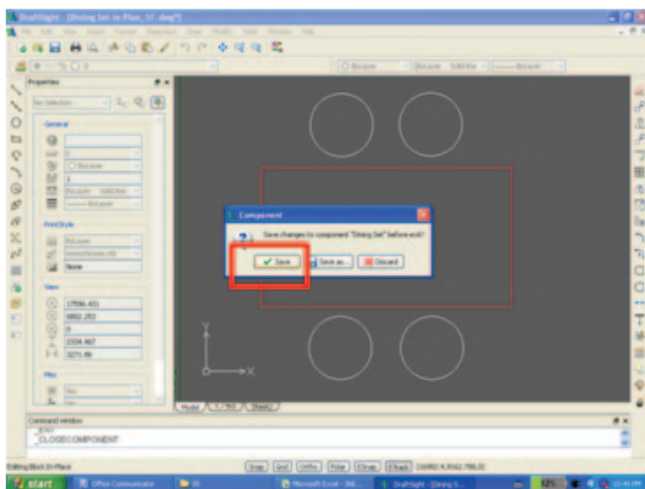⑫ Now, we are going to delete the original square chairs. Click the delete icon

⑬ To "Specify entities»", click the original four square chairs to put them into the selected state, and delete them by pressing "Enter".
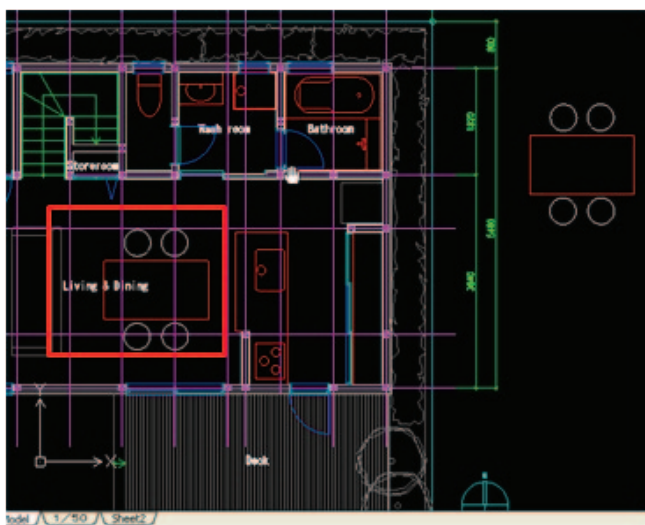
The editing work is over. So we will now end the "Edit Component" dialog.

⑭ Right-click and select "Close Component" from the displayed items.

⑮ A message asking "Do you want to save "Dining Set"?" will appear. Click "Save". Now the block "Dining Set" has been updated to the edited version.

⑯ Zoom out the drawing so that the Living / Dining area will appear on the screen. You should be able to confirm that the block "Dining Set" that you created is now updated, showing the four "round" chairs as edited. Once a block is edited and updated, the edited information is also reflected on the blocks with the same name.

Summary
The blocks are useful and effective in omitting repetition of same entries and in minimizing the edit work. The graphical elements to be grouped into a block should be determined according to the number of components used in the drawing project, and the probability of later editing work.

# Referencing Other Drawings within the Drawing in Work

DraftSight is equipped with a functionality to reference other DWG files and image files. By setting another file or files as the source of reference, the user will be able to view the reference files within his (or her) drawing, and also view the latest version of the reference files just by reloading them. The advantage of using reference data is that, unlike the block, it will not be stored together in the user's DWG file. Therefore, the user will be able to prevent his file from becoming too big. However, the user must not forget to include all the reference data as well in the deliverable when he sends out his finished drawing. In this section, we will reference the premise data to enhance the contents of the drawing in work.

1. Activate DraftSight and open the file named
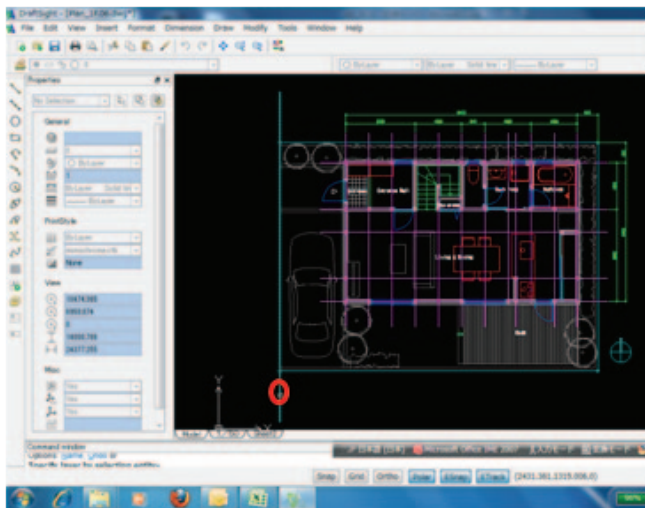   " Plan_1F.dwg ".
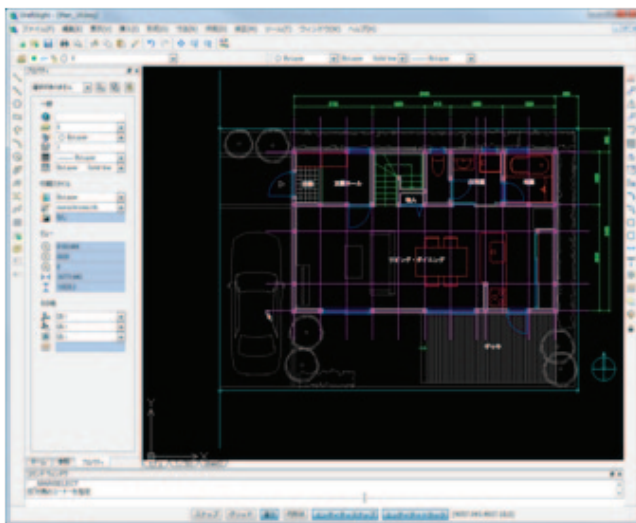
① The specified DWG file opens, and the drawing stored in this file appears on the screen.
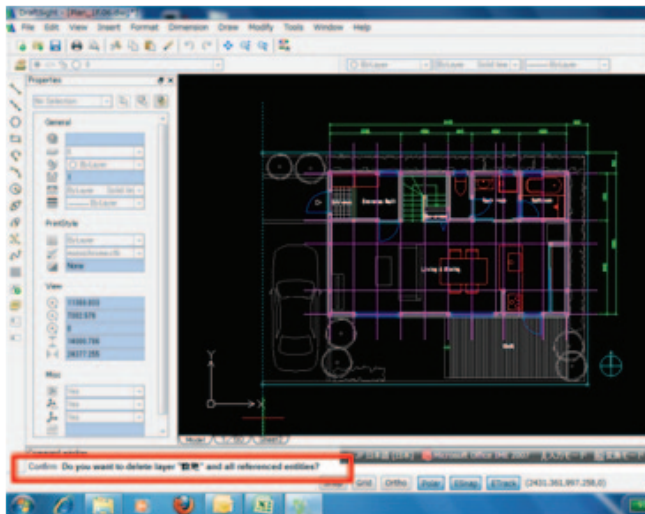
2. Delete the original graphics representing the premise.
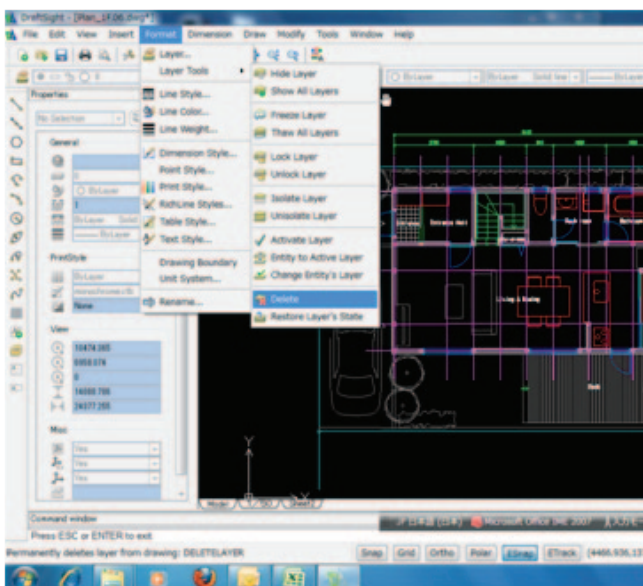
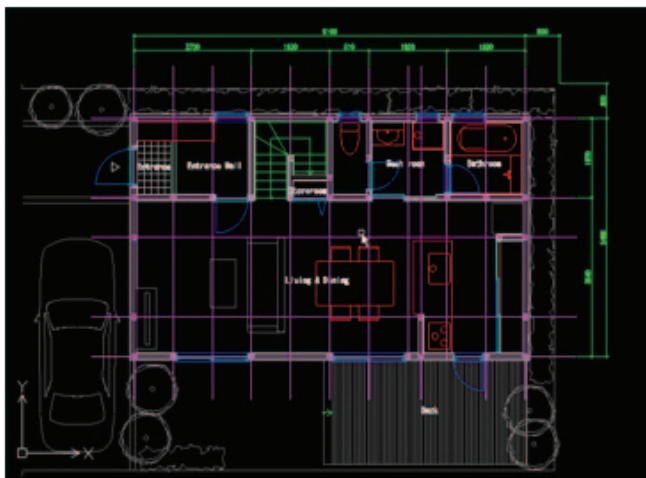Delete the layer called the "Site" and the entire data sorted in this layer.

① Select "Format" -> "Layer Tools" -> "Delete" from the Menu.

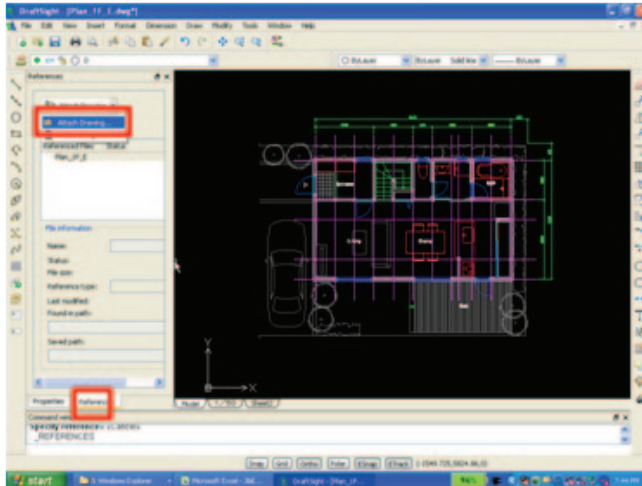② To "Select the entity and specify the Layer»", click any line that is in the Site.

③ "Do you want to delete the Layer "Site" and all the referenced entities? Specify Чes (Ч) or No (N) »", press the "y" key and "Enter".
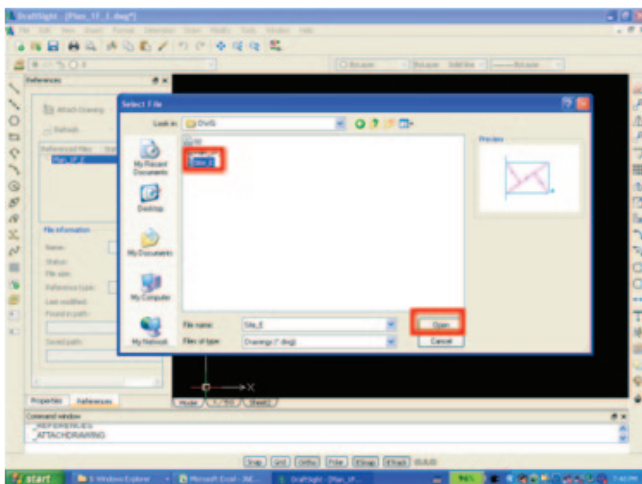

The data sorted in the Layer "Site" are now deleted.

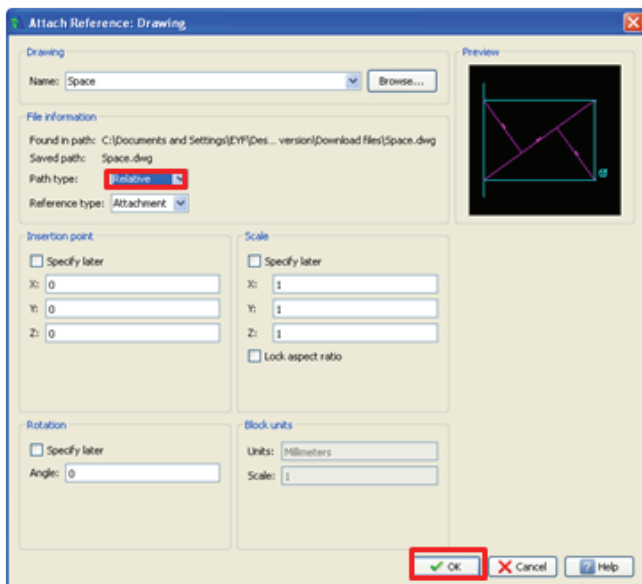## 3. Set other Site drawing file as the reference.

① Select "Reference" from the lower tab in the palette on the left.

② Click "Attach Drawing" located on the upper side of the Reference palette.

③ When the file selection dialog window appears, select "Site.dwg". Click the "Open" button and close the dialog window.


"Attach Reference" dialog box will appear.

④ Open the pull-down menu of the "Path type" field, and change the item to "Relative".

⑤ Click "OK".

Memo: When the Path type is set to "Full", the data saved will be the full-path data from the drive. When it is set to "Relative", the path from the current DWG file will be saved. For sending out the drawing as the deliverable, the data transmission time will be shorter and the receiver of the deliverable will find it easier to view the reference data when the Path type is set to "Relative".
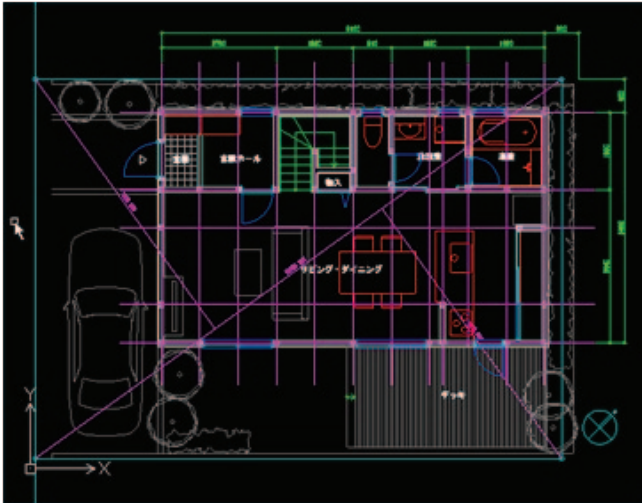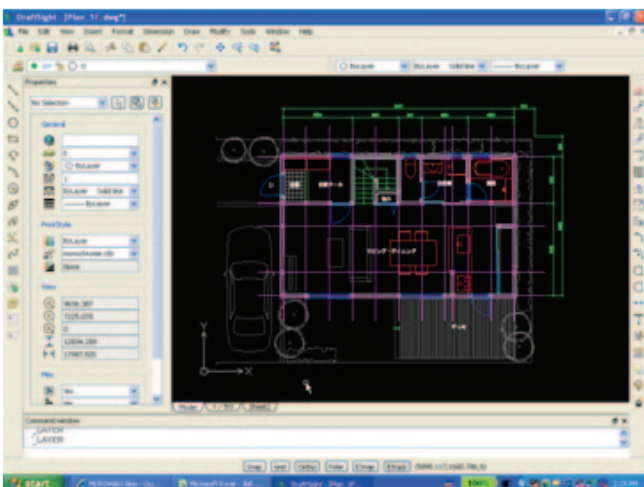
The selected reference data "Site.dwg" will be downloaded.

Hide the data contained in the downloaded site file.
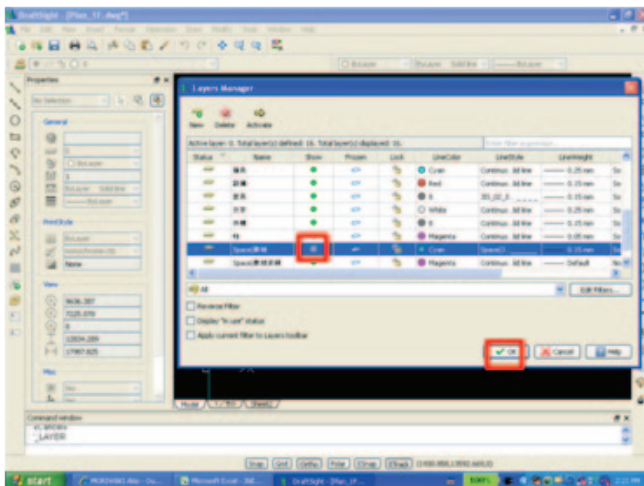
Activate the Layer Manager.
Select the layer named "Site" at the lower side, and click the circle under the "Show" column to change its color to grey. Then click "OK".

Memo: The name of the layer will be displayed as "Reference file name | layer name".

The unnecessary data will not be displayed, and the referenced drawing of the premise will appear clearly.

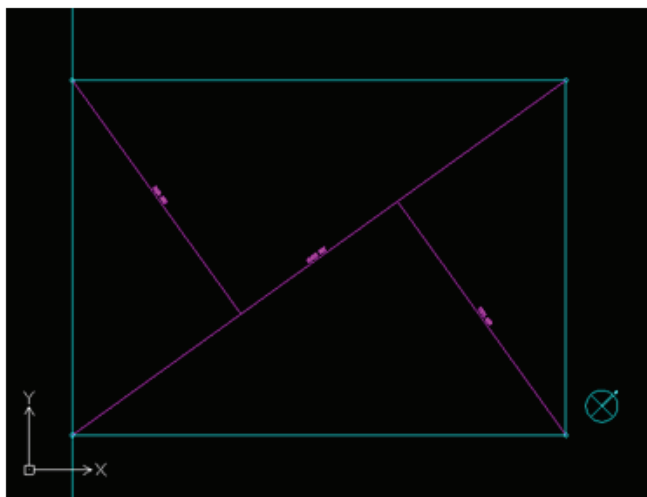## 4. Edit the reference source file, and reflect the changes made.
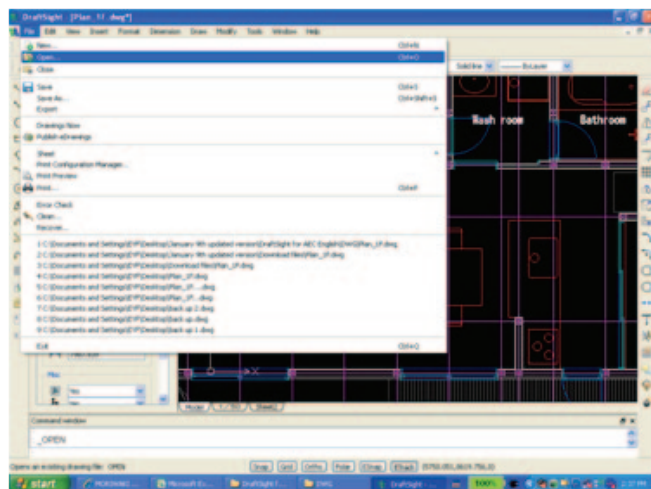
Edit the bearing mark of the reference source file "Site.dwg", and make the changes made in this source file appear on the current drawing. Leave the current drawing as it is, and open "Site.dwg".
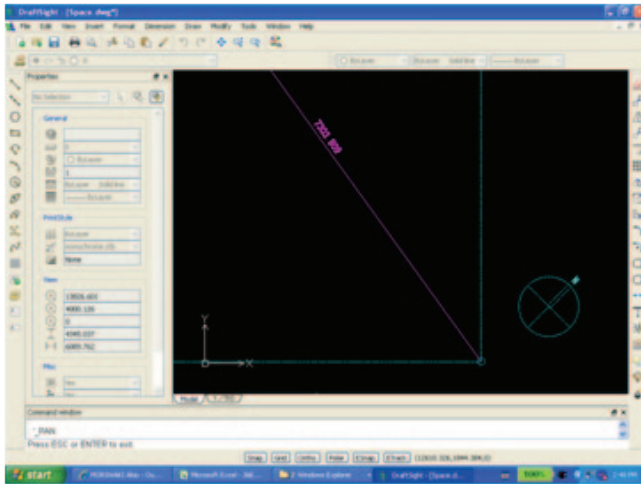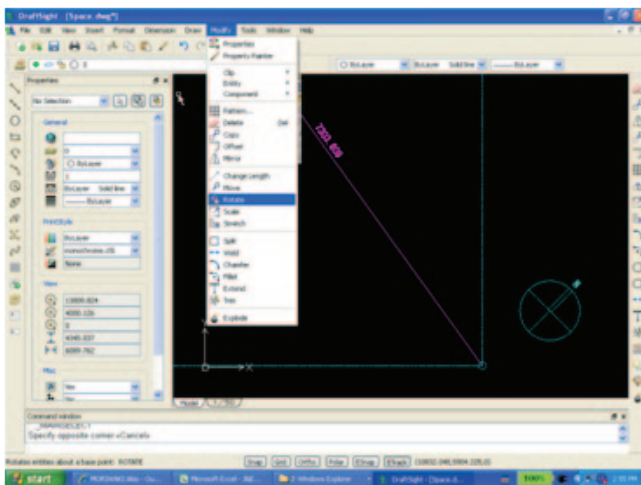
① Select "File" -> "Open" from the Menu.
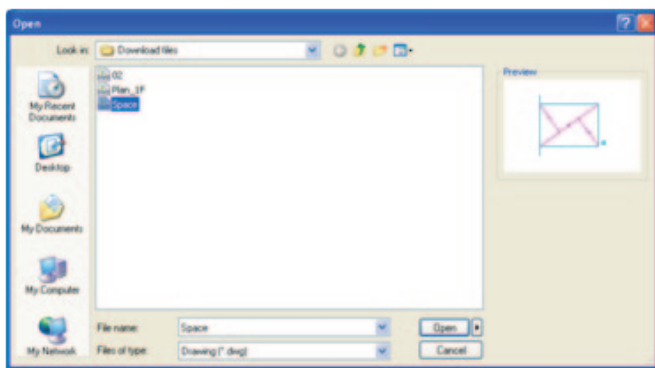
② Select "Site.dwg" and open it.

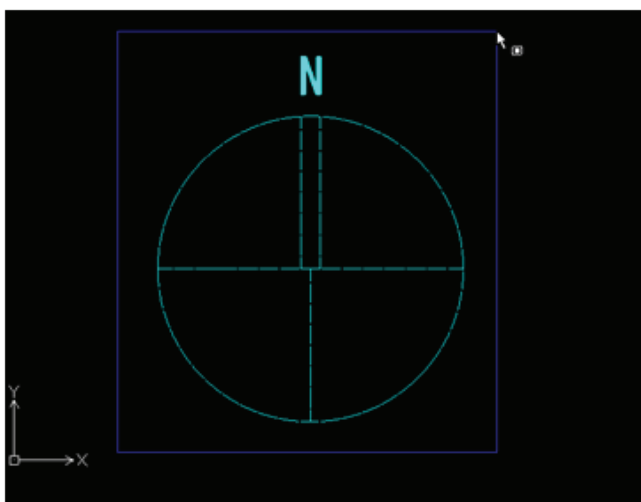The reference source file "Site.dwg" opens.

Enlarge the area around the bearing mark.

Rotate the bearing mark.
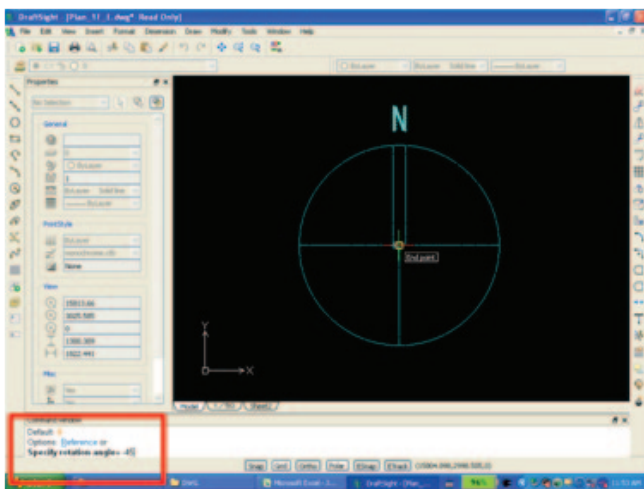③ Select "Modify" -> "Rotate" from the Menu.

To "Specify entities»", click the lower left point of the rectangle surrounding the bearing mark to select it to rotated the bearing mark and display it on the window. A blue rectangle will appear as a temporary rectangle. Click the upper right point of this rectangle. Press "Enter" to complete the selection of the graphics.

Memo: When the two rectangle diagonal corners are selected in the order from left to right, the selection mode will be "Window". When these points are selected in the order from right to left, the selection mode will be "Crossing". In "Window" mode, the entity that is fully surrounded by the rectangle will be selected. In "Crossing" mode, any entity crossing the rectangle will be selected. DraftSight provides various other methods of entity selection. To view the other options, enter "?" when a dialog appears asking you to "Specify entities".
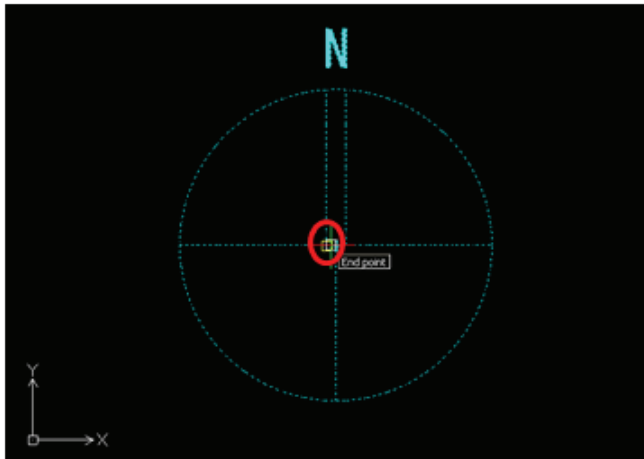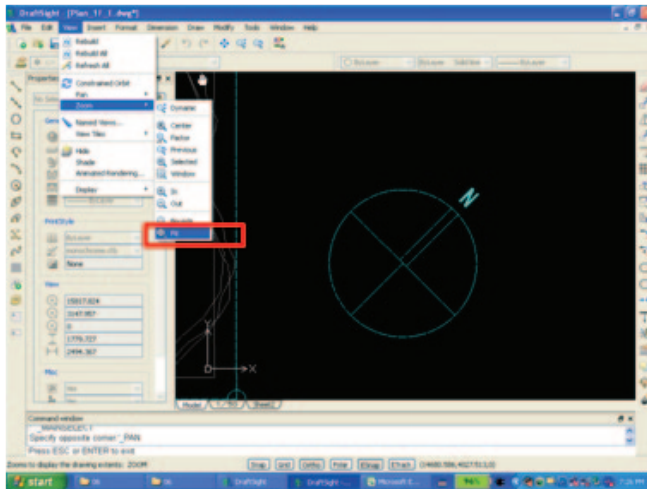
④ To "Specify pivot point»", specify the point that would be the center of rotation.
Place the cursor at the center of the bearing mark, and click when the endpoint snap appears.

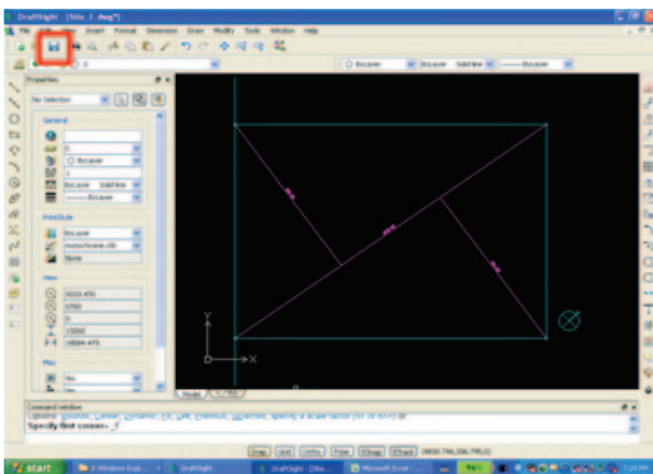⑤ To "Specify rotation angle»", enter "-45" with the keyboard and press "Enter".

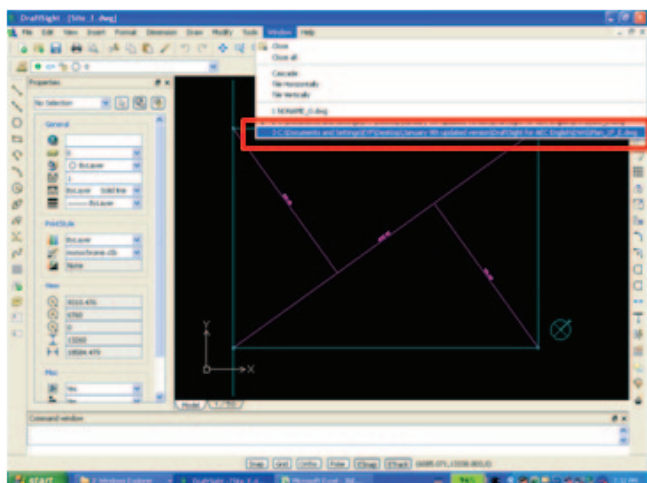The bearing is rotated clockwise to 45 degrees.

Zoom in to the original size.
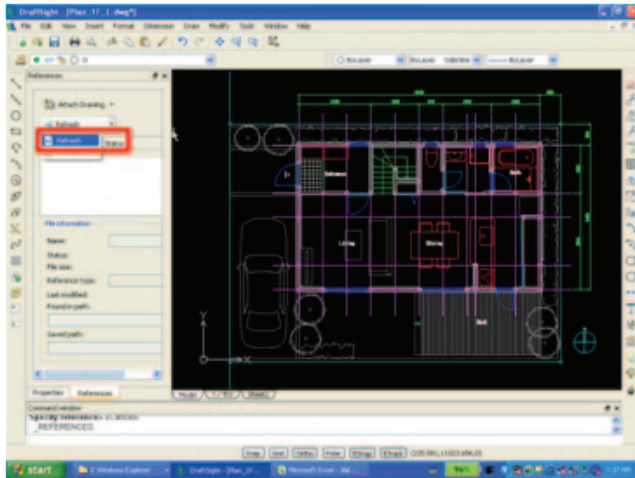⑥ Select "View" -> "Zoom" -> "Fit" from the Menu.

⑦ Save the changes made through editing. Click "Save" from the tool bar.

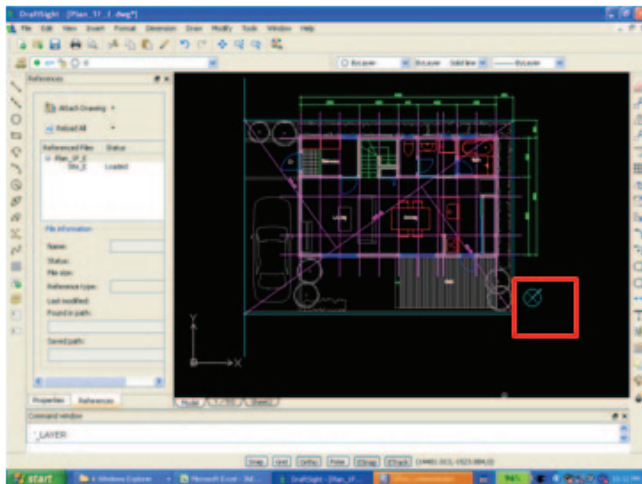**5. Reload the reference drawing.**

Return to the drawing you have been working on.
① Select "Window" -> "Plan_1F.dwg" from the Menu.

② Click "Reload all" on the Reference palette.

Summary

The functionality to reference other drawings is very useful because it will not make the user's DWG file larger and yet will enable the user to display the latest up-to-date data. For example, when the user selects a 2D detailed drawing as the reference data, he will be able to view this detailed drawing by switching the show or hide mode of the layer where the detailed data is sorted and at the same time, keep his drawing in plain 2D plan view. When there is a need to edit his drawing, he will only need to edit the detailed drawing, because whatever he edits will be reflected on his plain 2D plan view. This functionality, as a result, can significantly improve the user's work efficiency.
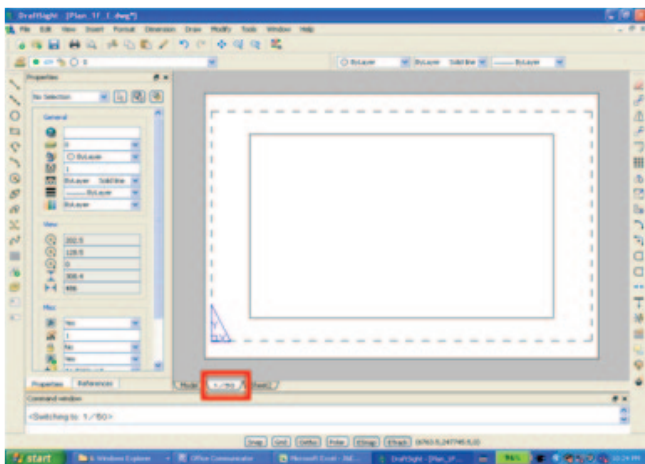
Printing out the Drawing

To print a drawing using DraftSight, two tabs are used. One is the "Model" tab for entering the drawing, and the other is the "Sheet" tab for setting and viewing the drawing's print layout. To print, the user must create a view on the Sheet tab that displays the drawing entered in the Model tab. The user can create multiple sheets of print views with different print settings respectively. In this section, we will work on a scenario requiring us to output a A3-sized, black-and-white drawing on a scale of 1/50 as a PDF file instead of printing it out on paper. To meet the requirements of this scenario, we need to set the scale of the view created on the Sheet tab to 1/50, and output the sheet as 1/1.

1.Activate DraftSight and open the file named "Plan_1F.dwg".
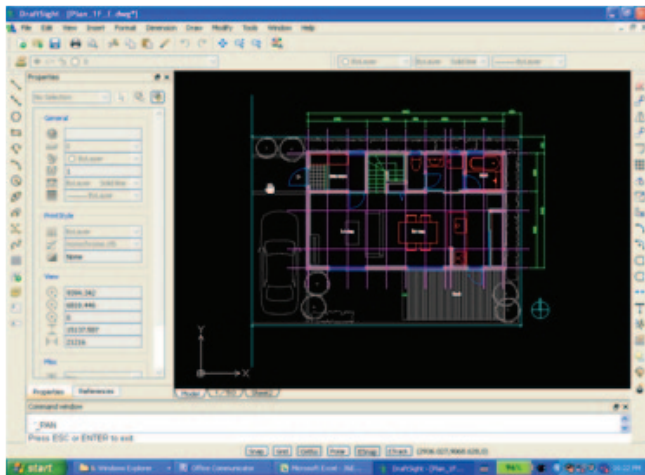
① Activate DraftSight and open the file named "Plan_1F.dwg". The specified DWG file opens, and the drawing stored in this file appears on the screen.
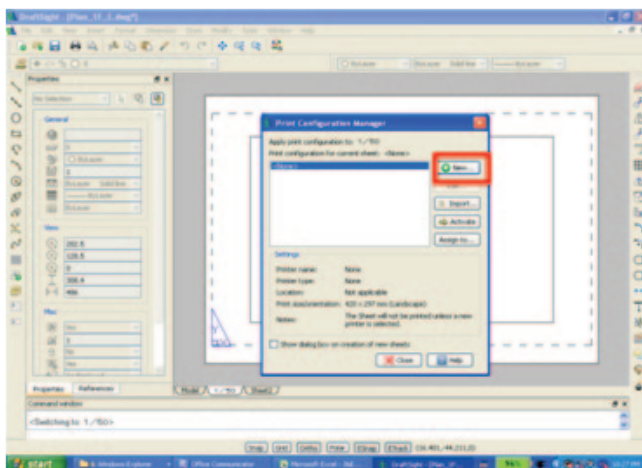
2. Configure the sheet.

Configure the sheet for printing by specifying the printer and paper size using the Printing Configuration functionality.

① Click the sheet that says "1/50", located below the drawing area.
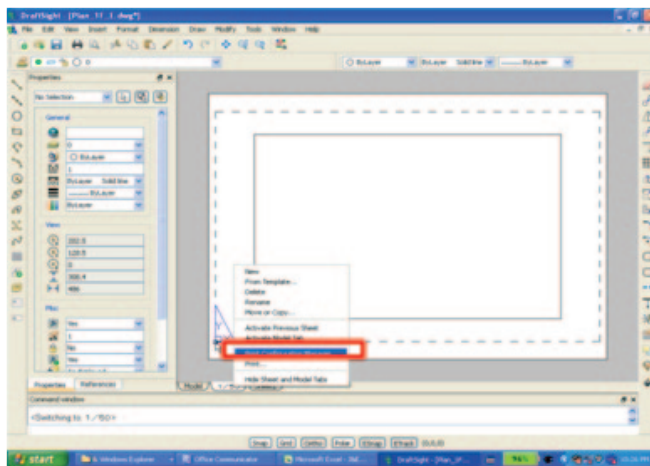
② Place the cursor over the sheet name, and right click. Then select "Printing Configuration Manager...".

③ "Printing Configuration Manager" dialog window appears. Click "New".

Memo
If there is an existing configuration file, the Printing Configuration Manager will allow you to complete the configuration easily by downloading it. Since there is no existing configuration file available for this scenario, we will create a new configuration file.
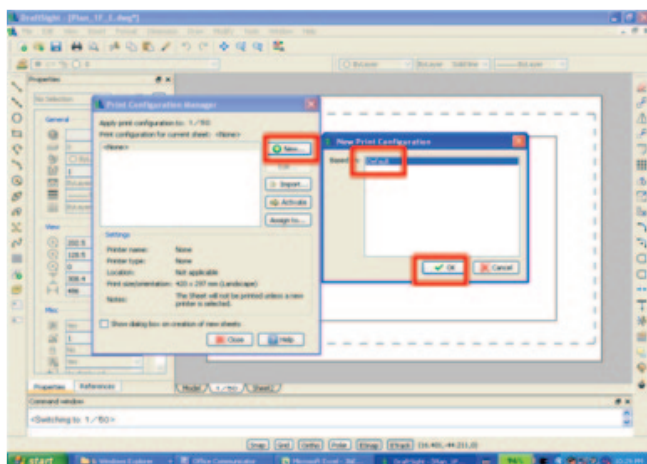

④ Now, we need to select a configuration that will be used as the basis for creating a new configuration. Select "Default" and click "OK".
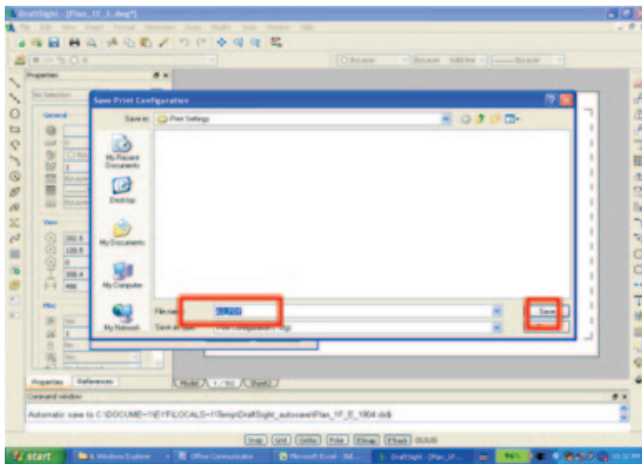
⑤ Enter the name of the configuration that is Enter "A3_PDF" and click "OK".
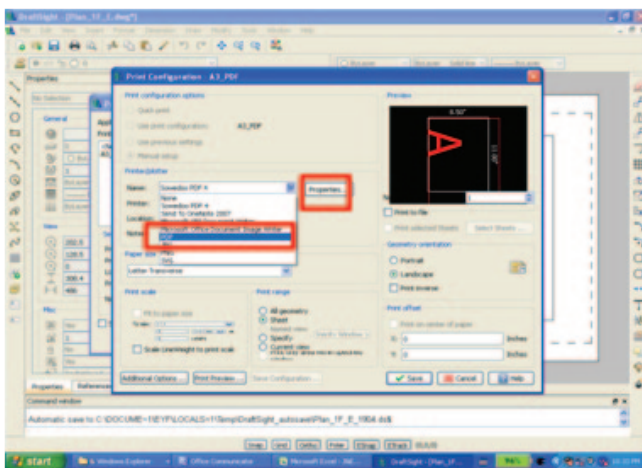
⑥ "Print Configuration" dialog window appears. Select "PDF" from the drop-down list of the field for specifying the printer name. Then click the "Property" button next to this field.
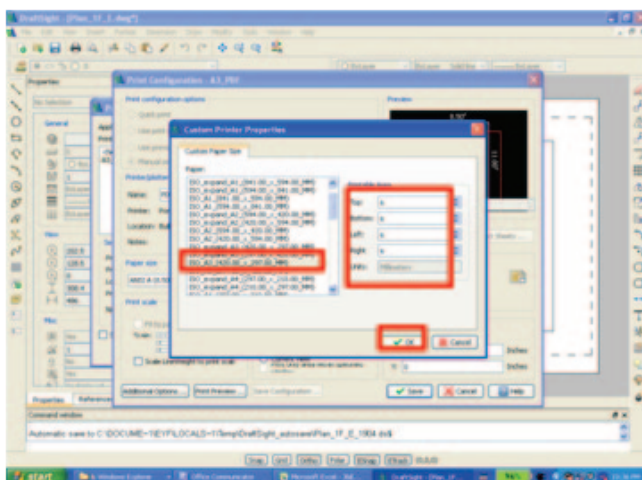
⑦ "Custom Printer Properties" dialog window appears. For the type of "Paper", select "ISO_ A3_(420.00_X_297.00_MM)". Set "6" for all the margins (Upper / Lower / Left / Right) of the "Printable area". Click "OK" after completing the above setting.
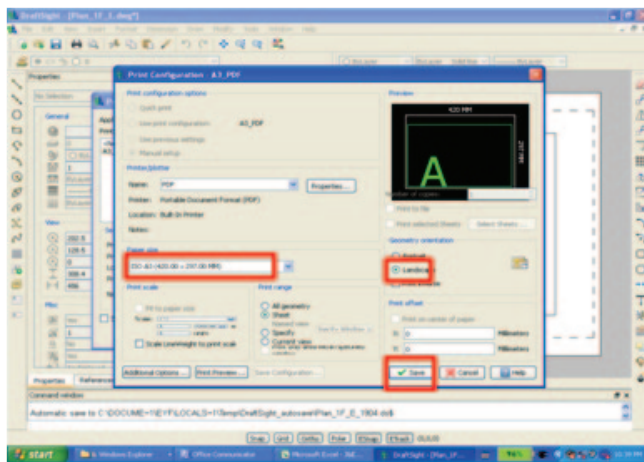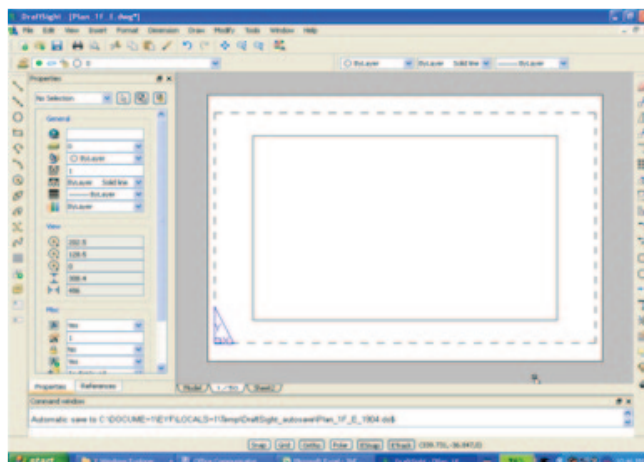
⑧ On the "Printing Configuration" dialog window, select "ISO_A3_(420.00_X_297.00_MM)" as the "Paper size". Select the radio button next to "Horizontal" for the "Geometry orientation". Then select "Save".
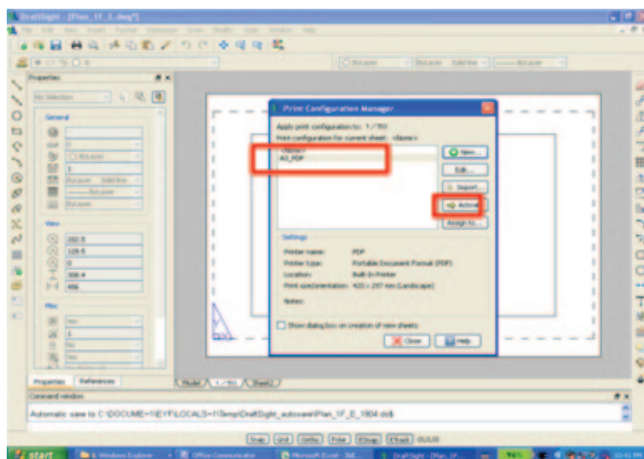

Activate the newly created printing configuration to set it on the selected Sheet tab. "Print Configuration Manager" dialog window reappears. Select the created "A3_PDF" from the list, and click the "Activate" button.

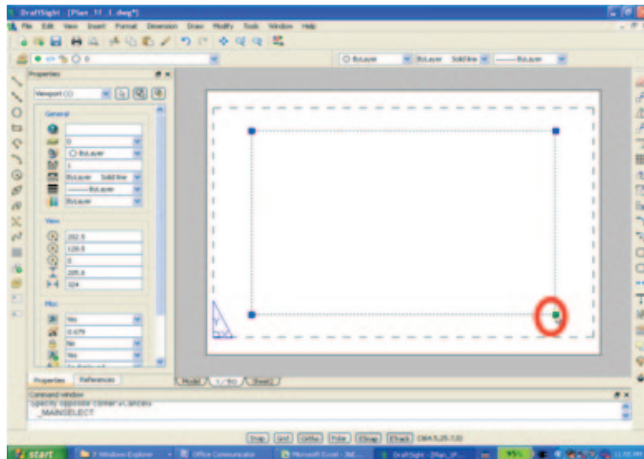Click "Close" from the Print Configuration Manager

Memo
The printing configuration that has been set is saved in the folder selected at the optional file location.
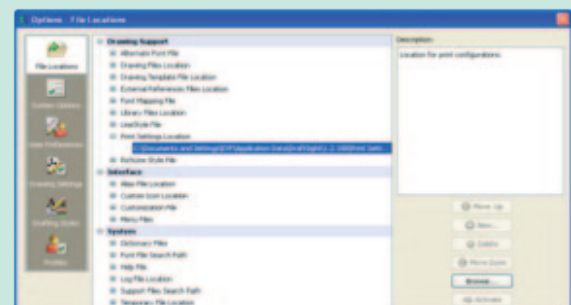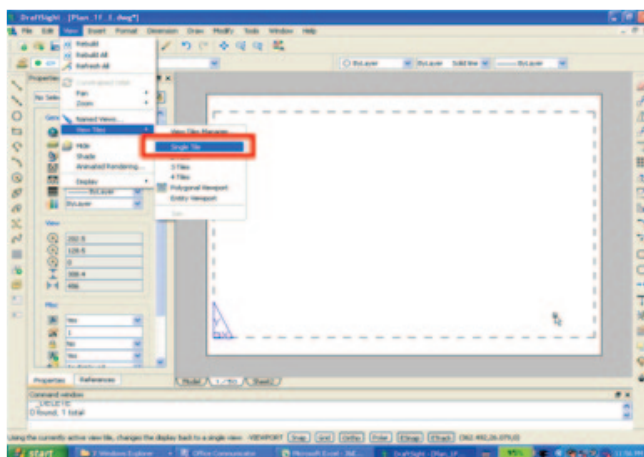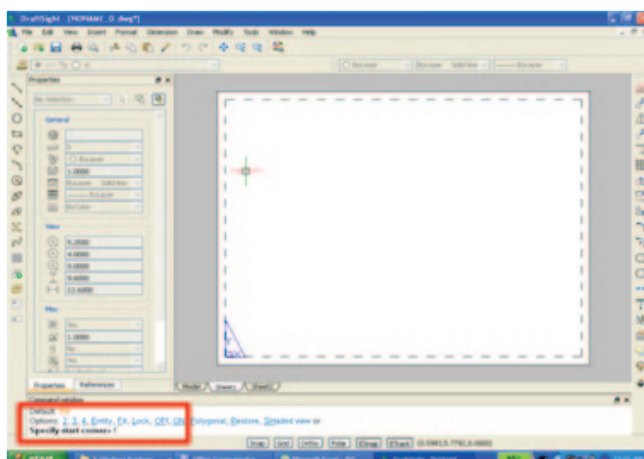
3. Create the view on the sheet.
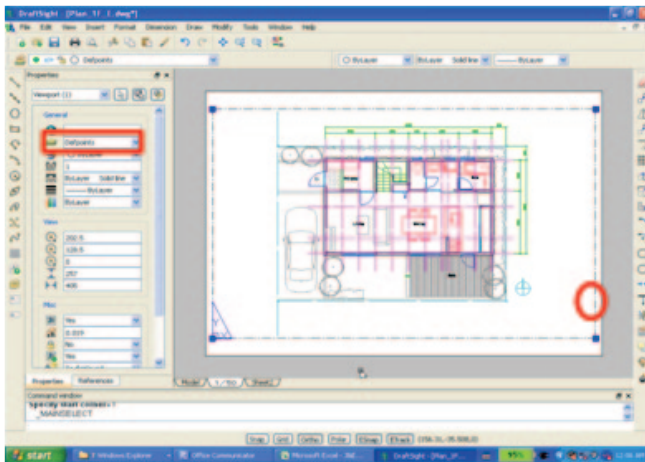

Create the ViewPort to display the model on the sheet.

① If there is a ViewPort already on the sheet, delete it. Click the rectangular frame of the view to put it to the selected state.Press the "Delete" key on the keyboard to delete the selected view.

② Select "View" -> "View Tiles" -> "Single Tile" from the Menu.


③ To "Specify start corner»", enter "f" for fit, and press "Enter".

By using the ViewPort that has been adjusted to be as large as the printing area, create the view for displaying the model.

④ Click the frame of the created view to put it to the selected state. Change the item selected in the Property palette on the left to "Defpoints", which is the item that will prevent the image layer from being printed.

> Memo
> "Defpoints" is the name of a special image layer that is set to be non-printable.
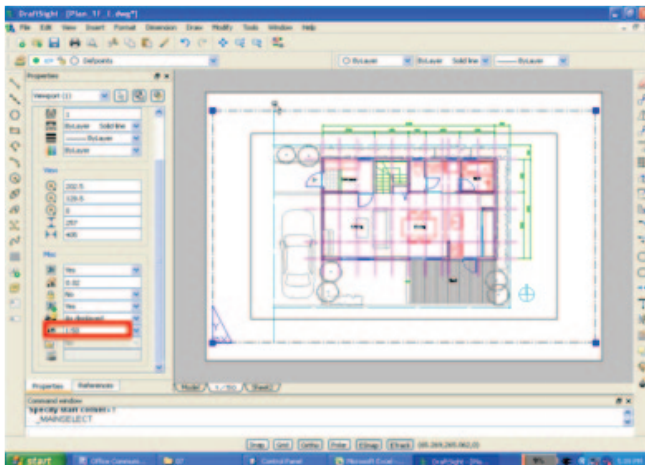

Set the scale of the view to 1/50.

⑤ Among the standard scale options in the Property palette on the left. select "1:50", the fourth item from the bottom. Then the view will show the model set to 1/50.

⑥ Press the "ESC" key to release the view frame from the selected state. This completes the configuration of the Sheet tab for displaying the model set to 1/50 view.
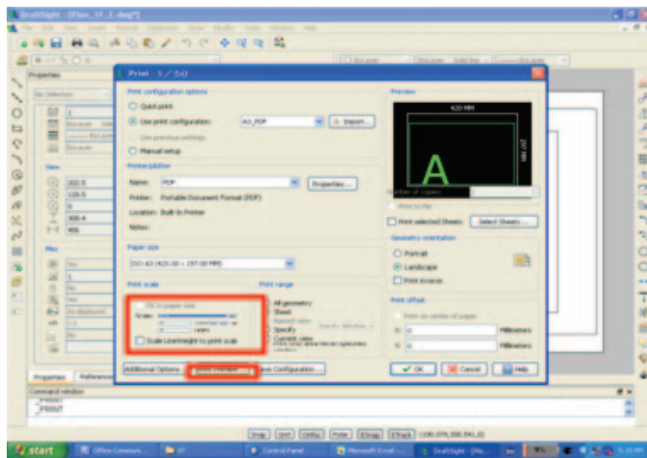
> Memo
> By creating multiple views with a different scale set for each, and specifying 1/1 for the number of sheets to be printed, it is possible to print out multiple drawings of different scales on one sheet.
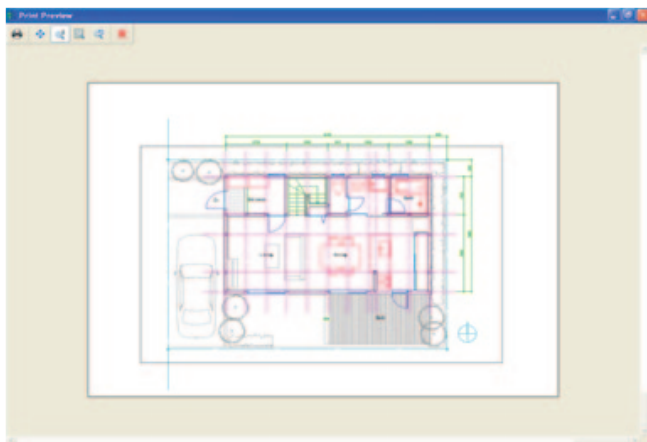
4. Set other printing items.
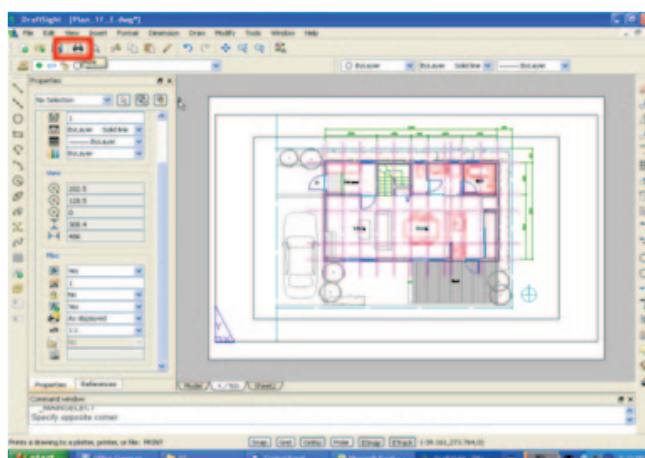
Set other printing items, such as the line color.
① Click the tool bar "Print" button.

② "Print" dialog window appears.
Confirm that the scale is set to "1:1" and click the "Print preview" button.

Print preview appears in color mode. By clicking the "Print" button, the targeted "PDF" file will be created. But to meet the requirement for a black-and-white printout, we need to change from color mode to monochrome mode.
Close the Print preview window by pressing the "X" button on the upper left.
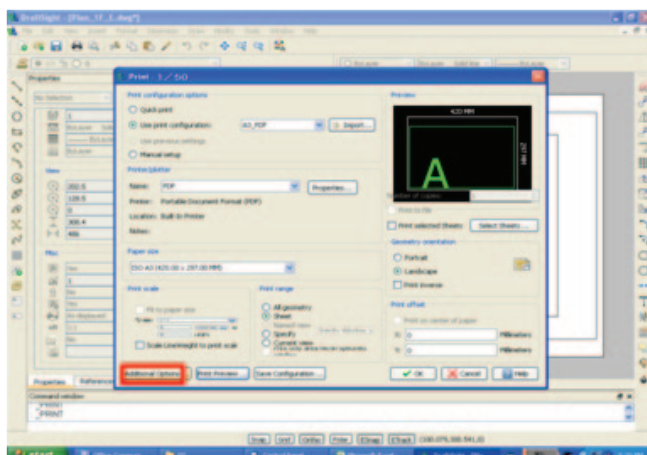

③ Click "Additional options" on the "Print" dialog window.

64
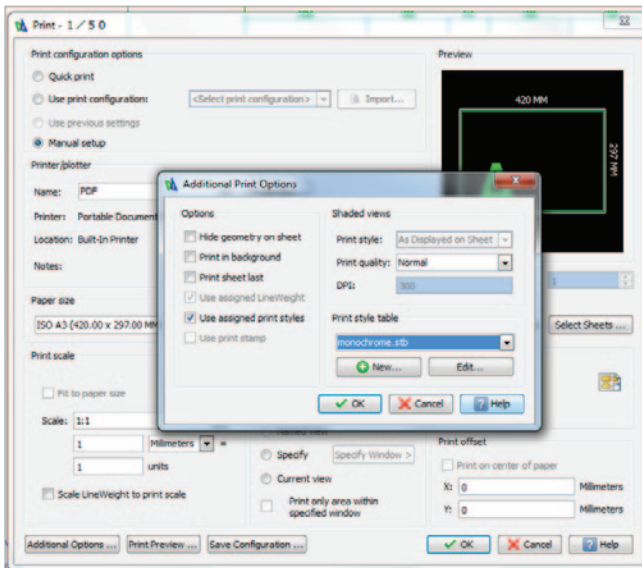
④ "Additional Print Options" dialog window appears. Click "New" under "Print style table".
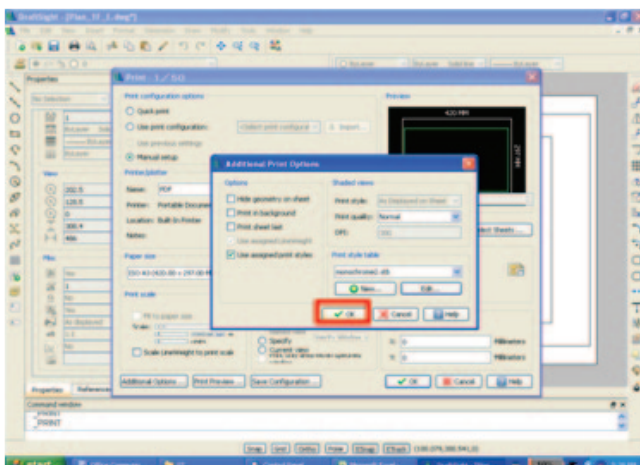
Memo
Two print styles called "ctb" and "stb" are available. With "ctb", the user can specify the line width for every displayed color. With "stb", the user can specify the line width and color for each entity, and the user-defined settings are fixed by specifying a print style name. The user can choose which print style to use only when the user starts to create the drawing.

Memo
The print style can be viewed on Image Layer Manager. When the line type and line width are not specified for each entity, the values pre-defined on that particular image layer will be used.
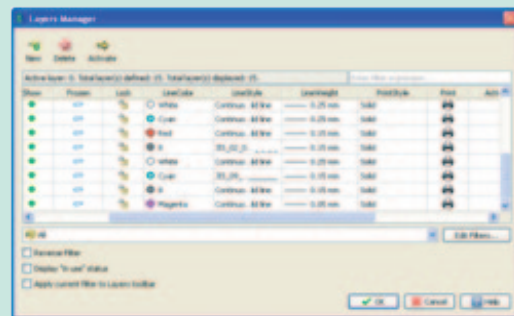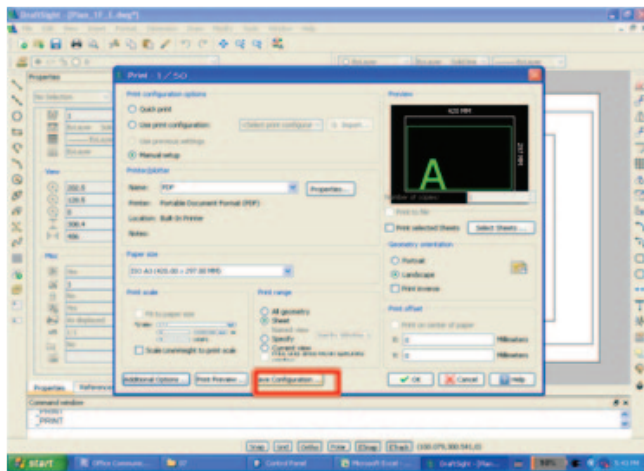

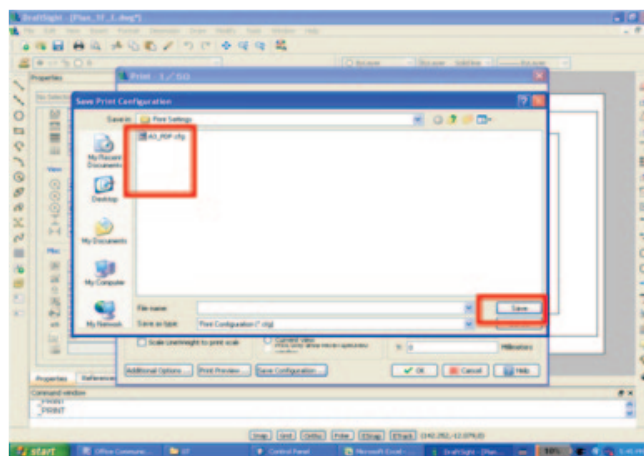⑤Back on the "Additional Print Options" dialog window, press "OK" and close this dialog window.

Memo: The configured print style is saved in the folder specified in "System Option" page.

© 2012 Dassault Systèmes

⑥ Save the configured style details to use them as the sheet definition by selecting "Save the configuration" button.


⑦ Select the sheet "A3_PDF.cfg" that is currently activated. Click "Save". The message asking you to confirm whether you want to overwrite the data or not appears. Click "Чеs" to reflect the edited contents on the printing configuration.
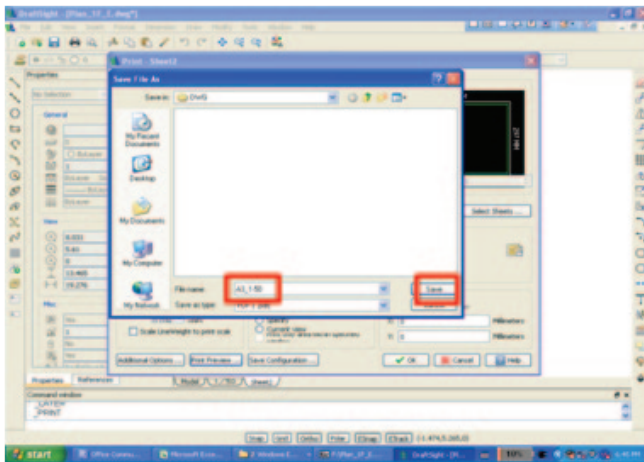
Memo: Unless the active printing configuration is saved, the edited contents of the printing configuration will not be saved on the sheet.
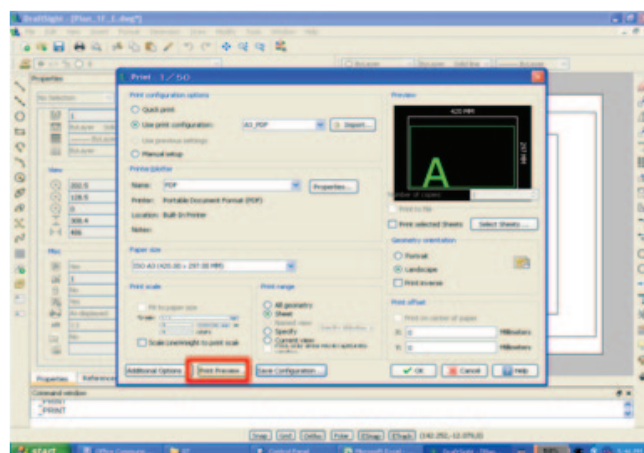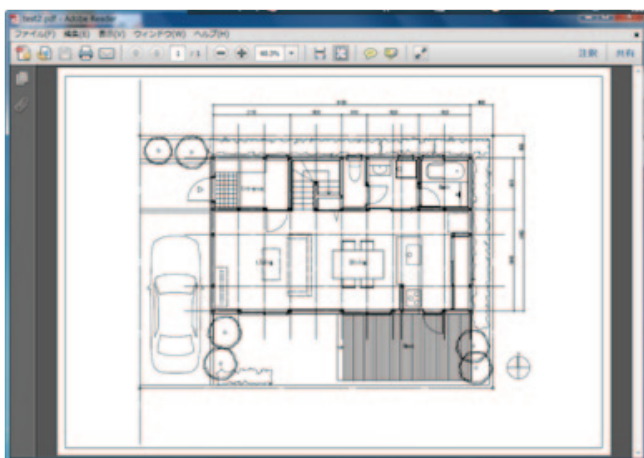

⑧ On the "Print Preview" dialog window, click "Print preview".
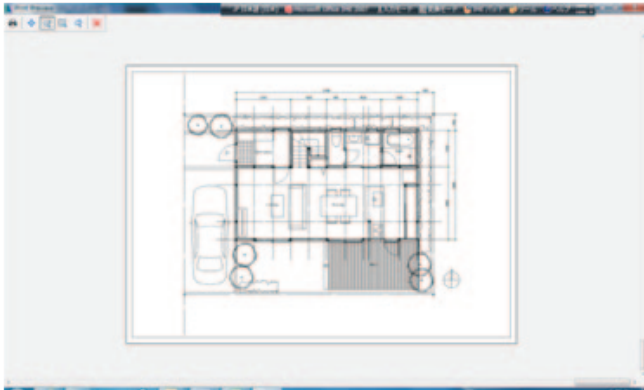
⑨ Confirm that the preview is displayed now in black and white. Then click "Print" icon on the upper left.

⑩ When asked what the name of the PDF file is, enter an appropriate filename. For this particular scenario, enter "A3_1-50", and click "Save".

The intended PDF file is now created.

Summary
Defining the printing configuration is a slightly complicated process. It may be useful for the users to remember: for sheet definition, set the printing configuration; and for defining the line width and type of pen, configure the print style.

© 2012 Dassault Systèmes

DraftSight