



MOVE

- The Move command will cut out the drawing entities and allows you to paste it to a new location.
 - Steps :
 - Type MOVE at the command prompt;
 - Select objects:
 - Specify base point or [Displacement]
 - Specify second point or <use first point as displacement>:



DIPLOMA ENGINEERING IN AIRCRAFT MAINTENANCE
DWM 1203: TECHNICAL DRAWING AND CAD

FOR TRAINING PURPOSE ONLY
JPP/DWM1203/P08/REV00

Preview from Notesale.co.uk
Page 17 of 45



COPY

- The COPY command is used to duplicate one or more existing drawing entities to another location without erasing the original.
- The Move command will cut out the drawing entities and allows you to paste it to a new location.
 - Steps :
 - Type COPY at the command prompt
 - Select objects: Select the objects using the desired selection method
 - Specify base point or displacement, [Multiple]: (M for multiple)
 - Specify base Point: (Pick a point)
 - Second point of displacement: (Pick second point)



DIPLOMA ENGINEERING IN AIRCRAFT MAINTENANCE
DWM 1203: TECHNICAL DRAWING AND CAD

FOR TRAINING PURPOSE ONLY
JPP/DWM1203/P08/REV00

FILLET

- The Fillet command connects two lines, arcs, or circles with a smooth arc of specified radius.
- It adjusts the lengths of the original lines or arcs so they end exactly on the Fillet arc.
- If the Polyline option is used, you can apply fillets to an entire polyline or remove the fillets from a polyline.
 - Steps :
 - Type FILLET at the command prompt
 - Polyline/Radiu3/<select two objects: (enter R to change the fillet radius)
 - Fillet Radius: (enter 1)
 - Polyline/Radiu3/<select two objects>: (select the first object)
 - Select second object: (select the second object)
 - Enter "P" to fillet an entire Polyline. Enter "R" to set the fillet radius. Otherwise select two entities to fillet.

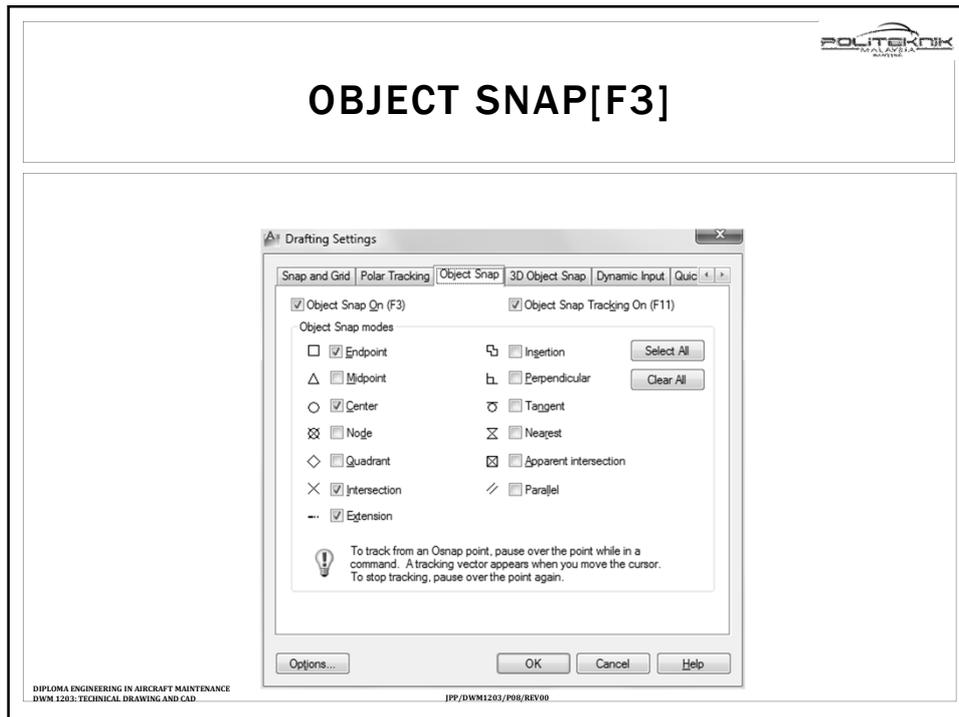


Preview from Notesale.co.uk
Page 21 of 45

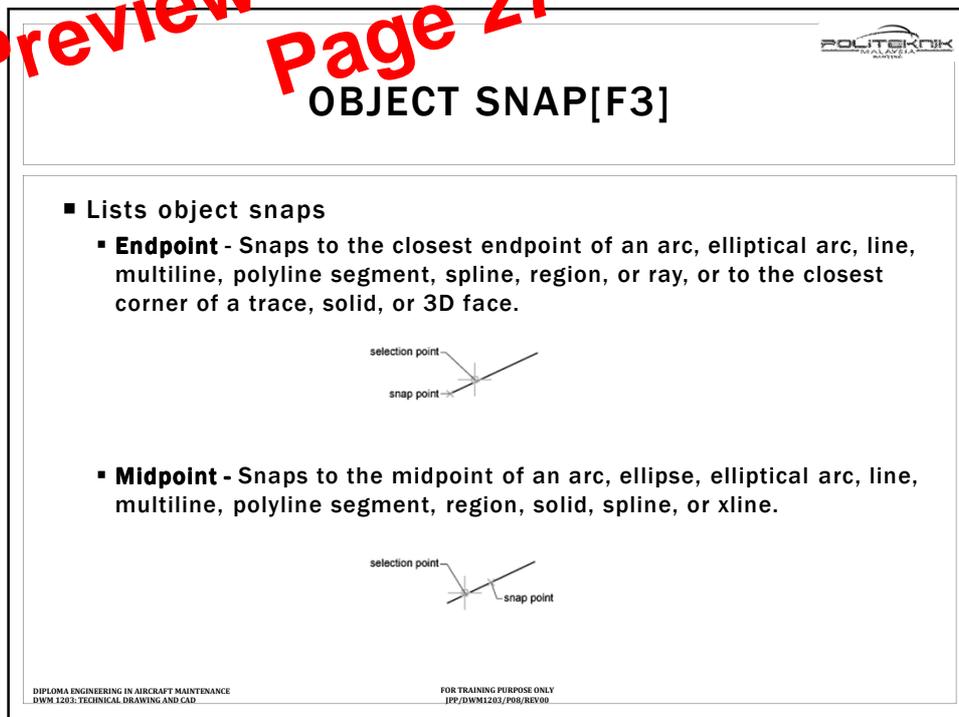
CHAMFER

- This command trims two intersecting lines with a specified distance from the intersection and connects the trimmed ends with a new line segment.
- Chamfering Two Lines - This option trims two intersecting lines.
 - Steps:
 - Type CHAMFER at the command prompt
 - Polyline/Distances/<Select first line>: (type D to select the chamfer distance)
 - Enter first chamfer distance <current>: (enter distance)
 - Enter second chamfer distance <current>: (enter distance)
 - Polyline / Distances / <Select first line>: (select the first line)
 - Select second line: (select the second line)





Preview from Notesale.co.uk
Page 27 of 45





LIST OF LAYERS

- Displays layers and their properties.
- To modify a property, click its icon.
- Names
 - Displays the names of the layers.
- On/Off
 - Turns layers on and off. When a layer is on, it is visible and available for plotting
- Freeze/Thaw in All Viewports
 - Freezes selected layers in all viewports.
 - freeze layers to speed up ZOOM, PAN, and many other operations; improve object selection performance; and reduce regeneration time for complex drawings.
- Lock/Unlock
 - Locks and unlocks the layers. You cannot edit objects on a locked layer.
 - For reference
- Color
 - Changes the color associated with the selected layers.
- Linetype
 - Changes the linetype associated with the selected layers
- Lineweight
 - Changes the lineweight associated with the selected layers
- Plot Style
 - Displays a list of plot styles available to apply to the selected layers.

DIPLOMA ENGINEERING IN AIRCRAFT MAINTENANCE
DWM 1203: TECHNICAL DRAWING AND CAD

FOR TRAINING PURPOSE ONLY
JPP/DWM1203/P09/REV00

Preview from Notesale.co.uk
Page 34 of 45



CREATING LAYERS

- Lines, circles and text are examples of drawing objects.
- In addition to its geometric positions, each object has an associated layer, linetype and colour.
- Normally, the colour and linetype are inherited from the layer on which the object is drawn.
- Therefore, creating of layers is the most important step in organising your drawing.

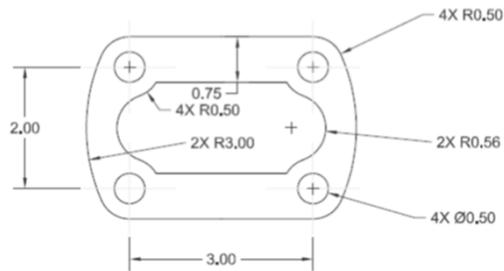
- These four object properties when creating a layers:
 - Colour;
 - Linetype;
 - Line weight; and
 - Plot style.

DIPLOMA ENGINEERING IN AIRCRAFT MAINTENANCE
DWM 1203: TECHNICAL DRAWING AND CAD

FOR TRAINING PURPOSE ONLY
JPP/DWM1203/P09/REV00

EXERCISE7F****

- Draw object using different layer
 - Object (Colour: Black, lineweight : 0.75, linetype : continuous)
 - Dimension line (Colour: red, lineweight : default, linetype : default)
 - Centerline (colour : yellow, lineweight : 0.5, linetype : ISO dash)



DIPLOMA ENGINEERING IN AIRCRAFT MAINTENANCE
DWM 1203: TECHNICAL DRAWING AND CAD

FOR TRAINING PURPOSE ONLY
JPP/DWM1203/P08/REV00

Preview from Notesale.co.uk
Page 38 of 45

OUTCOMES

- Applying drawing tools and setting
- Explain the use of edit commands
- Prepare to make drawings using drawing aids command
- Explain all characteristics about layers
- Explain all characteristics about block

8.5 BLOCK

DIPLOMA ENGINEERING IN AIRCRAFT MAINTENANCE
DWM 1203: TECHNICAL DRAWING AND CAD

FOR TRAINING PURPOSE ONLY
JPP/DWM1203/P08/REV00



TO CREATE A BLOCK DEFINITION

- From the Draw menu, choose Block Make.
- In the Block Definition dialog box, enter a name for the block.
- Under Objects, choose the Select Objects button to use the pointing device to select objects for the block definition.
- The dialog box closes temporarily while you select objects for the block. Press ENTER when you are done selecting blocks.
- The dialog box reopens.
- If you want to create a selection set, use the Quick Select button to create or define a filter for your selection set.
- Under Objects, specify whether to retain, convert to a block, or delete the selected objects.
 - Retain** : Keeps selected objects in the current drawing, in their original state.
 - Convert to Block** : Replaces selected objects with an instance of the block.
 - Delete** : Removes selected objects after the block is defined.

DIPLOMA ENGINEERING IN AIRCRAFT MAINTENANCE
DWM 1203: TECHNICAL DRAWING AND CAD

FOR TRAINING PURPOSE ONLY
JPP/DWM1203/P09/REV00

Preview from Notesale.co.uk
Page 40 of 45



TO CREATE A BLOCK DEFINITION

- Under Base Point, enter the coordinate values for the insertion base point or choose the Specify Insertion Base Point button to use the pointing device.
- Under Description, enter text to help identify the block for easy retrieval.
- Under Icon, specify whether to create an icon from the block definition.
 - Do not include an icon: Omits preview image from block definition.
 - Create icon from block geometry: Saves preview image with the block definition.
- Choose OK.
- The block definition is saved in the current drawing.

DIPLOMA ENGINEERING IN AIRCRAFT MAINTENANCE
DWM 1203: TECHNICAL DRAWING AND CAD

FOR TRAINING PURPOSE ONLY
JPP/DWM1203/P09/REV00

INSERT

- You can insert blocks or entire drawings into the current drawing with INSERT.
- When you insert a block or drawing, you specify the insertion point, scale, and rotation angle.
- When you insert an entire drawing into another drawing, AutoCAD treats the inserted drawing like any other block reference.
- Subsequent insertions reference the block definition (which contains the geometric description of the block) with different position, scale, and rotation settings.
- If you change the original drawing after inserting it, the changes have no effect on the inserted block.
- If you want the inserted block to reflect the changes you made to the original drawing, you can redefine the block by reinserting the original drawing.
- If you want to insert a drawing but you don't want the file name to be the same as the block name, you can insert it using a blockname = filename syntax at the Block Name prompt.

Preview from Notesale.co.uk
Page 42 of 45

INSERT

- From the Insert menu, choose Block.
- In the Insert dialog box, specify the block name, where you want to insert it in the current drawing, and whether you want to explode it after insertion.
- If you have modified the original drawing file for a block, you can redefine the block in the current drawing by choosing File to locate the file for the block.
- Choose OK.
- The block reference is updated in the current drawing.

