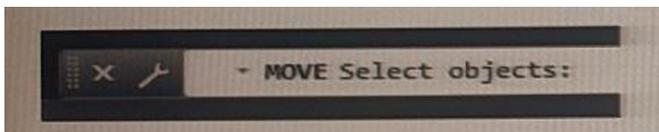# [Q16-Q32 Latest Autodesk ACP-01101 First Attempt, Exam real Dumps Updated [Dec-2023

Latest Autodesk ACP-01101 First Attempt, Exam real Dumps Updated [Dec-2023 Get the superior quality ACP-01101 Dumps Questions from DumpsMaterials. Nobody can stop you from getting to your dreams now. Your bright future is just a click away! Q16. Refer to the exhibit.

You are working on a drawing. You copy a set of entities and exit the COPY command. You then use the MOVE command on the same set of entities.

Which default option should you use when prompted to select entities?

* Spacebar
* P (Previous)
* L (Last)

* A (All)
Explanation

when you use the MOVE command on a set of entities that you have previously copied, you should use B. P (Previous) as the default option when prompted to select entities. This will select the last set of entities that you modified with any command.

https://forums.autodesk.com/t5/autocad-forum/move-previous-command/td-p/3466896

https://www.cadlinecommunity.co.uk/hc/en-us/articles/360002360178-Move-Copy-Rotate-All-in-one-editing-co

Q17. Which two objects support associative dimensions? (Select 2)
* Hatches
* Arcs
* Images
* Polylines
Explanation

associative dimensions adjust to changes in the geometric objects they measure. Dimension associativity defines the relationship between geometric objects and the dimensions that give their distance and angles.

Based on 2 and 3, dimensions in a layout may be associated to objects in model space when the DIMASSOC system variable is set to 2.

Q18. Move three applicable commands from the list on the left to the corresponding function on the right.
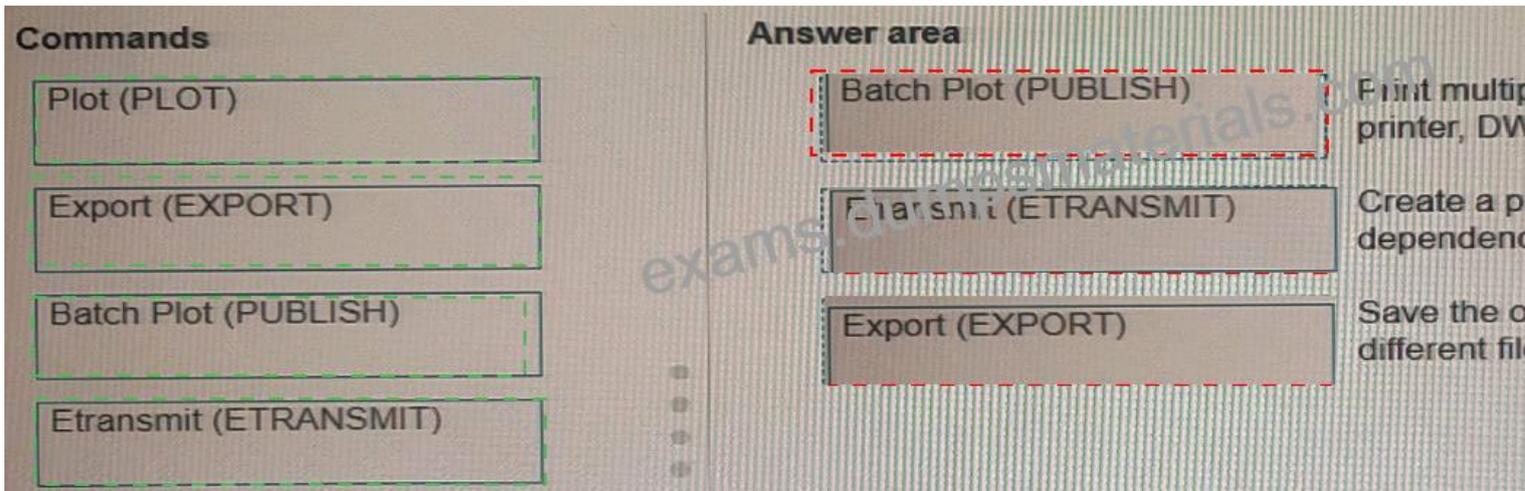
Commands may be used once or not at all (Select 3)

Explanation

Box 1 : Batch plot

One of the topics covered in this course is Plotting Drawings2. In this topic, you can learn how to print multiple sheets or drawings to a plotter, printer, DWF file, or PDF file. The applicable command for this function is Batch plot (PUBLISH)2.

https://www.autodesk.com/certification/learn/course/autocad-drawing-design-drafting-professional Box 2: EtransmitOne of the topics covered in this course is Sharing Drawings2. In this topic, you can learn how to create a package of drawing files and their dependencies. The applicable command for this function is Etransmit (ETRANSMIT)2.

https://www.autodesk.com/certification/learn/course/autocad-drawing-design-drafting-professional Box3: ExportOne of the topics covered in this course is Exporting Drawings2. In this topic, you can learn how to save the objects in a current drawing to a different file format, such as SAT, STL, etc. The applicable command for this function is Export (EXPORT)2.

https://www-int.autodesk.com/certification/learning-pathways/autocad-design-drafting

Q19. Refer to the exhibit.

The exhibit shows a set of ornamental garden stairs going up to a walkway as part of a landscaping design.

You need to make sure that the edge of the top step in shown in a different linetype, as shown by the dashed line in the exhibit.

Which prompt in the OFFSET command should you use?
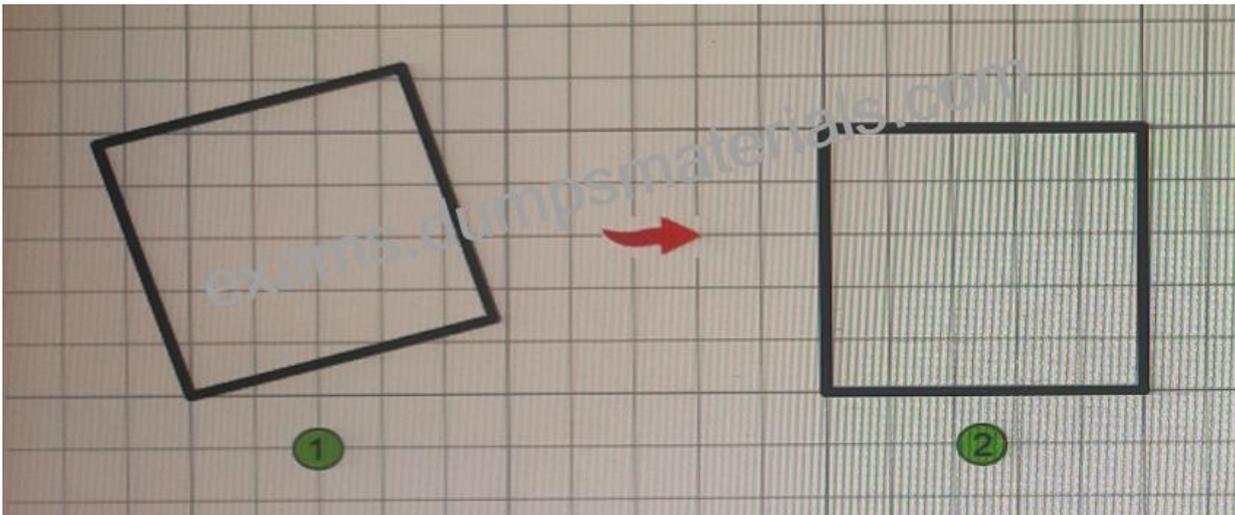* Through
* Current
* Erase
* Source
Explanation

The Current prompt should be used to make sure that the edge of the top step is shown in a different linetype, as shown by the dashed line in the exhibit. This way, you can set a different linetype for your current layer before using the OFFSET command4.

https://knowledge.autodesk.com/support/autocad/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/AutoCAD

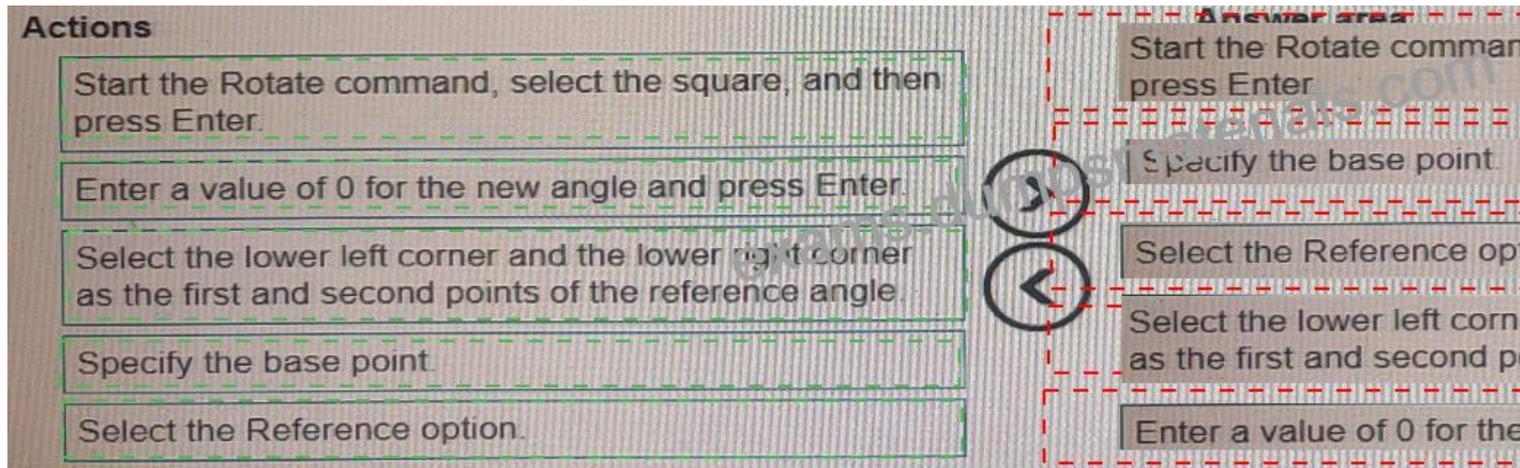**Q20.** In which order should you perform the actions to rotate square (1) to match the angle of square (2).

Move all actions to the answer area and place them in the correct order.

Explanation

rotate is a command that rotates selected objects around a base point to an absolute angle. You can specify the base point and the rotation angle by entering values or selecting points. You can also use the reference option to align objects with other objects.

* 1- Start the Rotate command, select the square, and press Enter

* 4- Specify the base point

* 5- Select the Reference option

* 3- Select the lower left corner and the lower right corner as the first and second points of the reference angle

* 2- Enter a value of 0 for the new angle and press Enter

Q21. A site plan was recently modified to relocate a building to a new location While relocating the building, only the hatch location of the building was moved by mistake as shown in the exhibit.

Evaluate the graphic and answer each question by selecting the correct options from the drop-down lists.
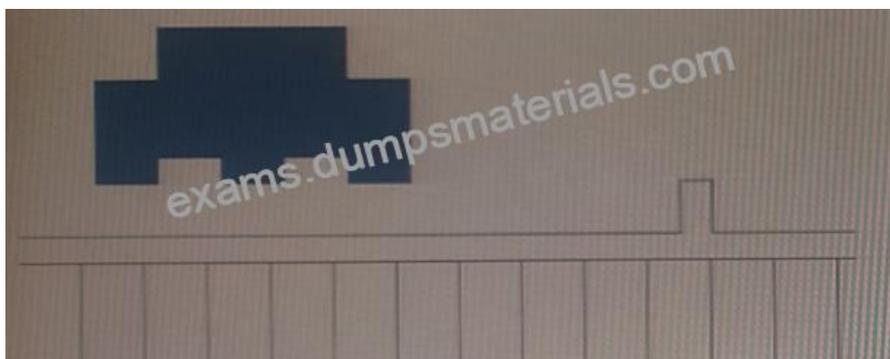
Answer area

What can you do while creating the hatch to prevent the hatch
and the polyline from separating?

Set the hatch origin
Turn on Associative Hatching
Turn on Outer island detection
Select the polyline boundary i

What can you do to obtain the building polyline?

Explode the hatch object
Use the Recreate Boundary o
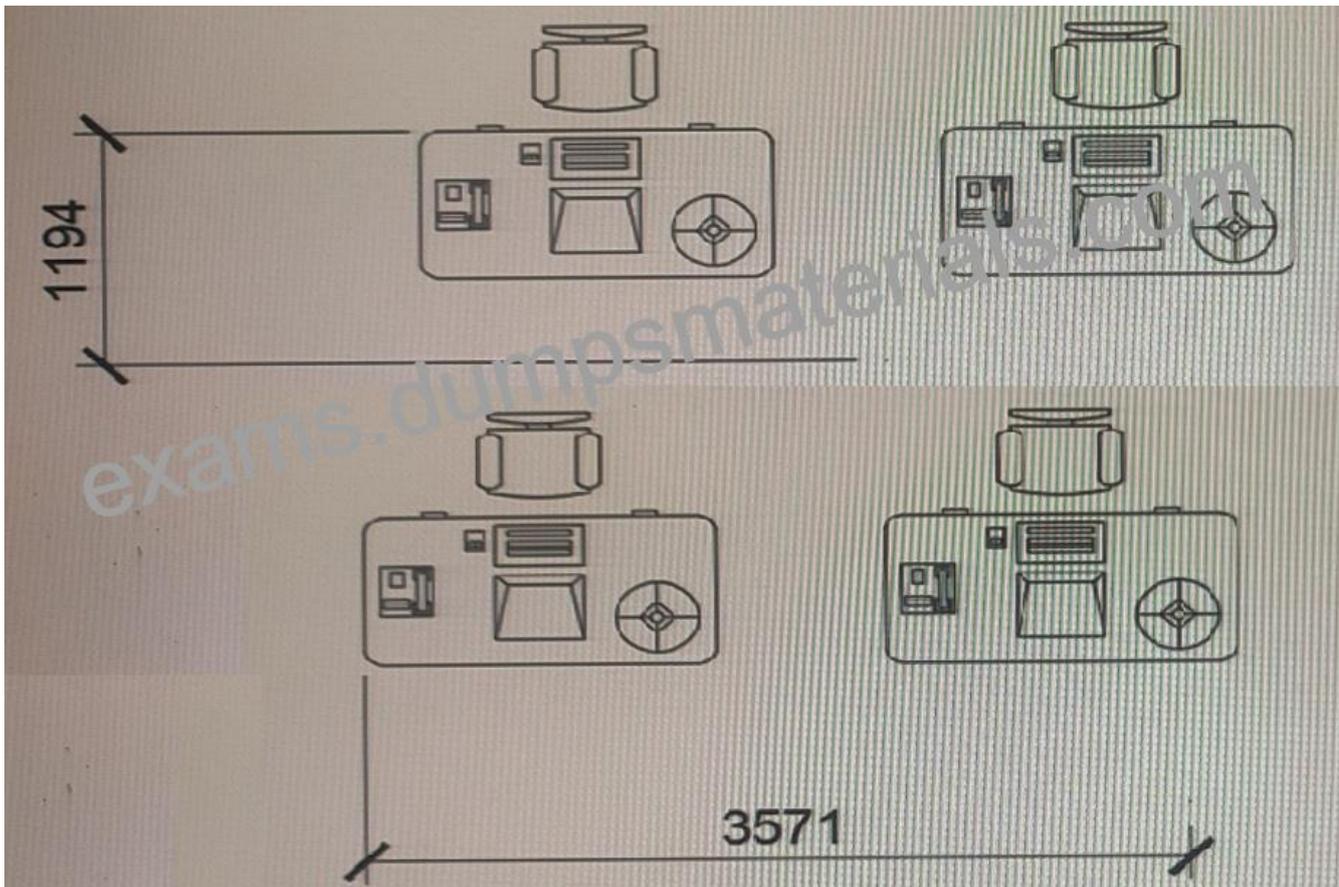Draw a polyline that is snapped

Explanation

Box 1 = Turn on Associative Hatching

Box 2 = use the Recreate Boundary option from Hatch Editor

https://knowledge.autodesk.com/support/autocad/troubleshooting/caas/sfdcarticles/sfdcarticles/Recreating-hatch-

https://knowledge.autodesk.com/support/autocad/learn-explore/caas/CloudHelp/cloudhelp/2018/ENU/AutoCAD

Q22.

The exhibit shows a workstation arrangement, with horizontal and vertical dimensions, representing the row and column spacings.

Both dimensions have been placed incorrectly, with one origin point of each dimension being placed in the wrong position.

What command should be used to ensure that the dimension objects are placed accurately?
* Reassociate (DIMREASSOCIATE)
* Break (D1MBREAK)
* Inspect (DIMINSPECT)
* Adjust Space (DIMSPACE)
Explanation

According to the AutoCAD for Design and Drafting documents1, the command that should be used to ensure that the dimension objects are placed accurately is Reassociate (DIMREASSOCIATE)2. This command associates or reassociates selected dimensions to objects or points on objects3. You can also control whether new dimensions are associative by checking the option Make New Dimensions Associative in the User Preferences tab4.

Q23. You want to print a copy of a floor plan with all the interior objects hidden. You already have a polyline drawn around the footprint of the plan.

Complete the statements by selecting the correct option from the drop-down,lists.
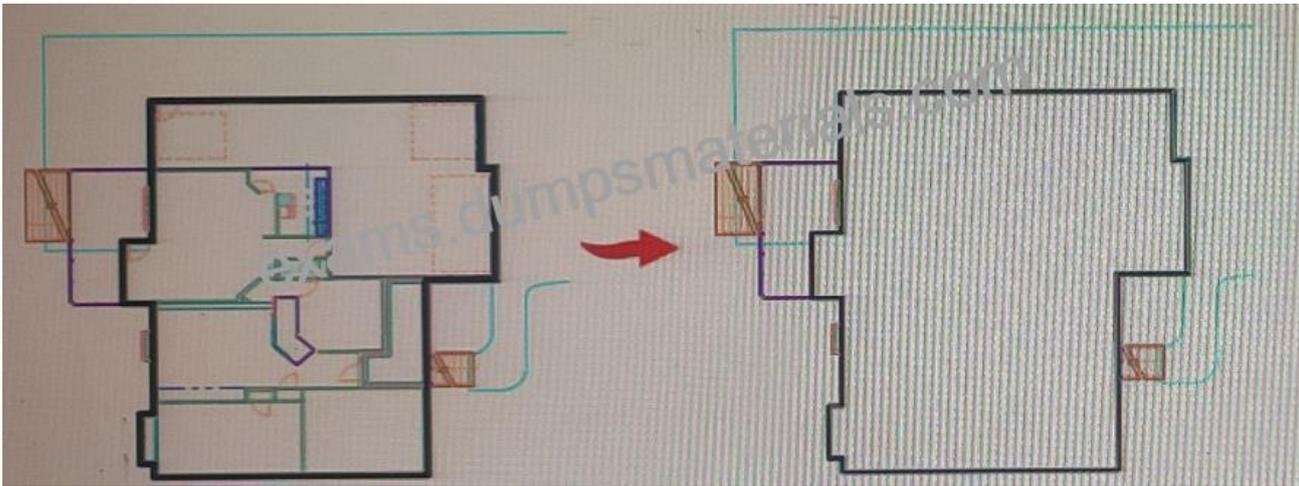
Answer area

To mask the interior objects, you can use the

POLYGON command with the c op
CONVERT command with the Pol
WIPEOUT command with the Poly

To ensure that the polyline remains on top, you should use the

DRAWORDER command.
HATCHTOBACK command.
DRAWORDERCTL system var

Answer area

To mask the interior objects, you can use the

POLYGON command with the c op
CONVERT command with the Pol
WIPEOUT command with the Poly

To ensure that the polyline remains on top, you should use the

DRAWORDER command.
HATCHTOBACK command.
DRAWORDERCTL system var

Explanation

BOX 1 = WIPEOUT command with the Polyline option2.BOX2 = DRAWORDER command23

https://forums.autodesk.com/t5/autocad-forum/polyline-plan-view-boundary-of-point-cloud/td-p/6251860

Q24. Drawings from different sources have been combined and resulted in duplicate background elements that are slightly offset from each other Some of the layer names match and other layers have an E prefix, indicating that they are existing.

To clean up the drawing, the Overkill (OVERKILL) command is used.

Which two settings will resolve the duplicate background and offsetting issues? (Select 2)

* Ignore Layer object property
* Ignore Color object property
* Ignore Transparency
* Maintain associative objects
* Tolerance value

Explanation

You need to ignore layer object property to remove duplicates that have different layer names but are otherwise identical4. You also need to set a tolerance value that is larger than the offset distance between the objects5.

Q25. You are setting up a drawing layout.

You need to rotate the viewport viewing angle by 90 degrees without rotating the viewport.

What should you do next?

* Select the viewport that you want to rotate, and then use the Rotate command.
* Activate the viewport that you want to rotate. Start the Dview command and use the Twist option.
* Select the viewport that you want to rotate, and then use the UCS command with the Z Axis option.
* Activate the viewport that you want to. Start the Dview command and use the Camera option.

Explanation

you need to activate the viewport that you want to rotate and start the Dview command with the Twist option.

This will allow you to rotate the view within a viewport without rotating the viewport itself.

https://knowledge.autodesk.com/support/autocad/learn-explore/caas/CloudHelp/cloudhelp/2020/ENU/AutoCAD

Q26. You plan to draw a line at an angle of 6.25 degrees.

Where in the user interface do you configure polar tracking to work at this angle after selecting Polar Tracking On?

Note: Exhibit shown is the AutoCAD interface in Wind. Which the AutoCAD for the Mac version differs in appearance, the correct answers correlate in the options.
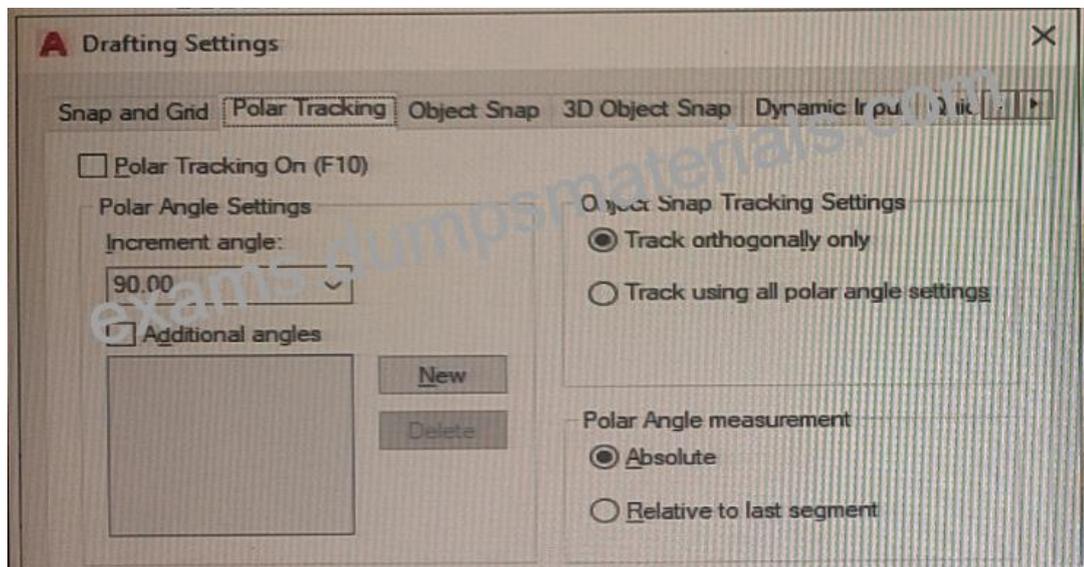
See the answer below:

Explanation

polar tracking is a feature that allows you to draw lines at specific angles. You can configure polar tracking to work at any angle you want by following these steps:

* Click on the arrow next to the Polar Tracking icon in the Status bar and select Settings.

* In the Drafting Settings dialog box, click on the Polar Tracking tab.

* Under Additional Angles, click on New and enter 6.25 as the angle value. Click OK.

* Make sure that 6.25 is checked in the list of additional angles. Click OK.

Now you can draw a line at an angle of 6.25 degrees by using polar tracking.

Q27. Exhibit
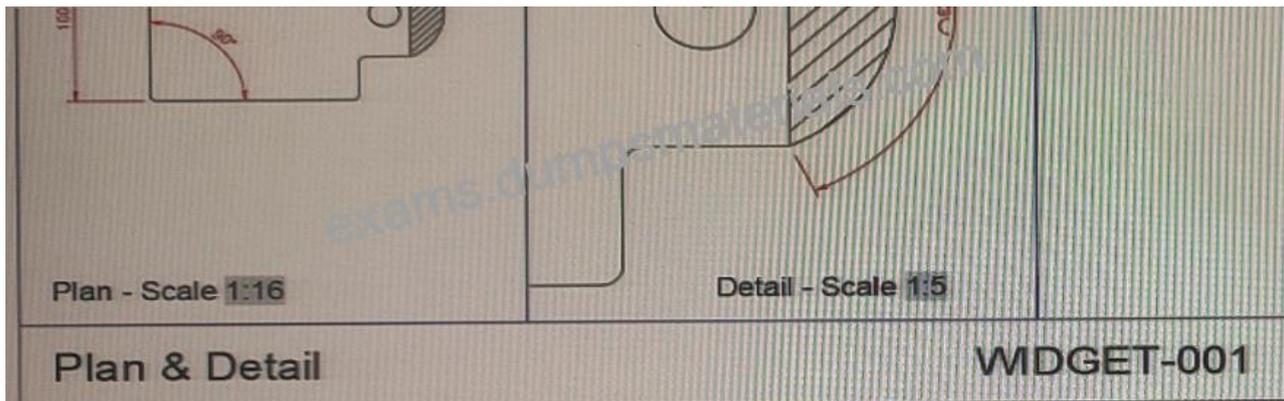
The viewport scales in the title block shown are field value as extracted from the properties of each viewport.

When using the Insert Field command to edit the text value and generate the viewport scale, which object property should you use, if the default scale list is being used?
* Standard scale
* Visual style
* Display locked
* Object name
Explanation

The object property that you should use when using the Insert Field command to generate the viewport scale, if the default scale list is being used, is the “Standard scale”.

The “Standard scale” property is used to specify the standard scale factor for the viewports. It represents a value in the format of 1:n, where n is the scale factor. For example, if you set the standard scale to 1:50, the viewport will display objects at a scale of 50 units per drawing unit.

To use this property in the Insert Field command, you can select “Object” in the “Field category” dropdown and “Standard scale” from the “Field names” dropdown. This will insert a field code that will automatically update the viewport scale based on the properties of the viewport.

Q28. You need to configure the layout viewports shown in the exhibit.

Which three methods can you use to create the viewports in a layout tab? (Select 3)
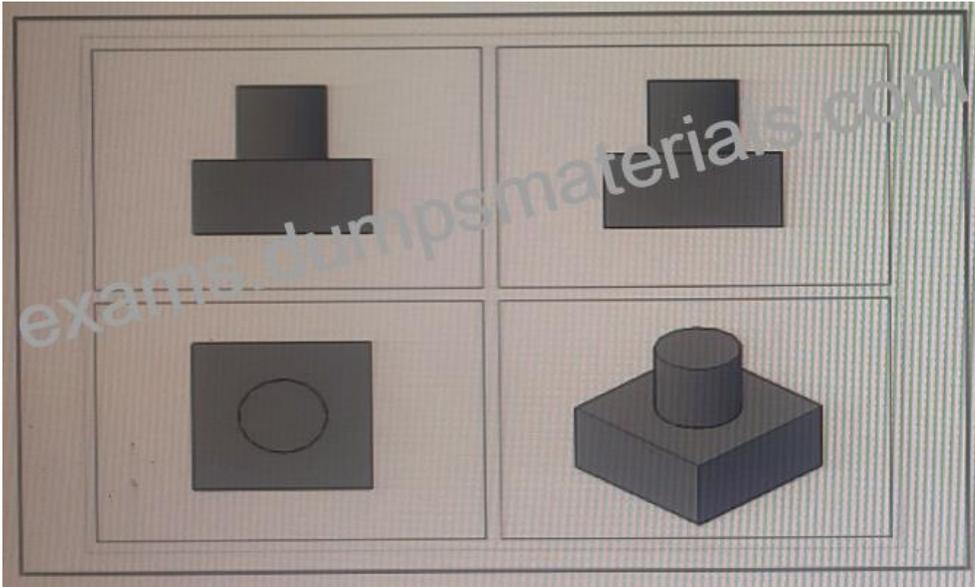
* Clip a viewport to multiple different shapes by using the VPCLIP command.
* From the Viewports dialog box: select a viewport configuration.
* Create individual viewports by using the MVIEW command.
* From the View Manager dialog box, generate named views and add them as layout viewpoints.
* Create a new page setup.
Explanation

a layout viewport is a window that displays a view of model space on a layout tab. You can create multiple layout viewports to display different views of your model at different scales.

To create the viewports in a layout tab, you can use three methods:

* From the Viewports dialog box: select a viewport configuration

* Create individual viewports by using the MVIEW command

* From the View Manager dialog box, generate named views and add them as layout viewpoints These methods will allow you to create and arrange viewports on your layout tab.

https://knowledge.autodesk.com/support/autocad/learn-explore/caas/CloudHelp/cloudhelp/2018/ENU/AutoCAD

Q29. You start the EXTEND command.

You need to be able to trim several objects quickly without exiting the EXTEND command.

What should you do?
* Hold the Shift key while selecting the objects.
* Hold the Alt key while selecting the objects.
* Hold the Ctrl key while selecting the objects.
* Hold the Ctrl+Shift key while selecting the objects.
Explanation

According to the AutoCAD for Design and Drafting documents1, the EXTEND command allows you to extend objects to meet the edges of other objects. To trim objects instead of extending them, you can hold down the Shift key while selecting the objects1. This will invoke the TRIM command within the EXTEND command.

https://autocadprojects.com/trim-and-extend-command/

Q30. You have the drawing shown in the exhibit.
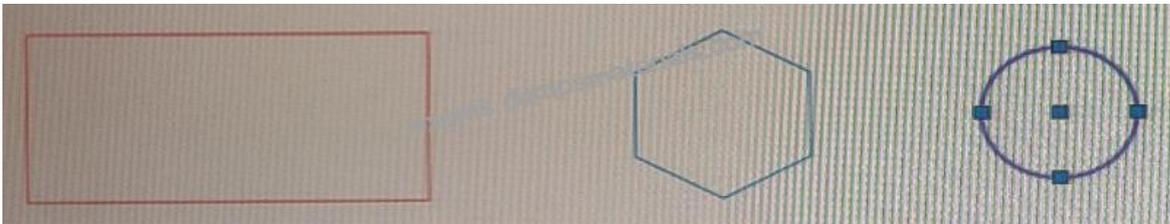
You need to place the polygon and the rectangle on the same layer as the circle.

Which command should you use to complete the task in the least number of steps?
* Match Properties
* Make Current (LAYMCUR)
* ByLayer
* Copy
Explanation

match properties is a command that allows you to apply the properties of one object to another object. You can use this command to change the layer of an object by selecting an object on the desired layer as a source.

To use this command, you can either:

* Click on the Match Properties icon in the Properties panel of the Home tab.

* Type MATCHPROP at the command prompt and press Enter.

Then, you can follow these steps:

* Select an object on the same layer as the circle as a source object and press Enter.

* Select the polygon and the rectangle as destination objects and press Enter.

Now you have placed the polygon and the rectangle on the same layer as the circle.

Q31. You have three multileaders as shown in exhibit 1. (Refer to the exhibit 1 tab.)
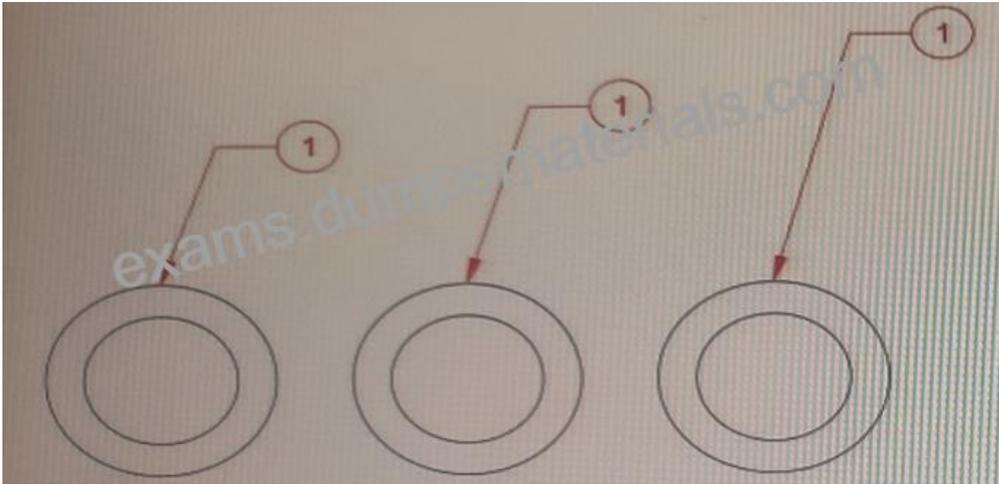
You need to change the display of the multileaders to match the multileaders in exhibit. (Refer to the exhibit 2 tab.)
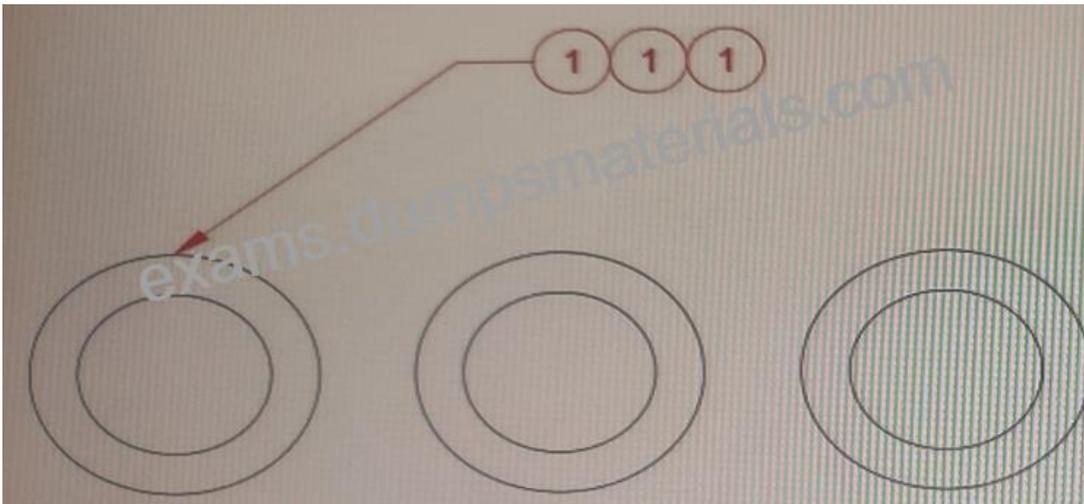
Which command should you use?
* Align Leader
* Collect Leader
* Remove Leader
* Add Leader
Explanation

The command that you should use is B. Collect Leader1. The collect leader option groups multiple leaders into one multileader object with a common landing line1. This option is useful when you want to create a single label or callout for multiple objects2.

To use this option, you need to select the MLEADER command and then select Collect Leader from the Options menu. Then you

can select two or more existing multileaders that have similar properties and press Enter. The selected multileaders will be collected into one multileader object with a common landing line1.

Q32. Refer to the exhibit.
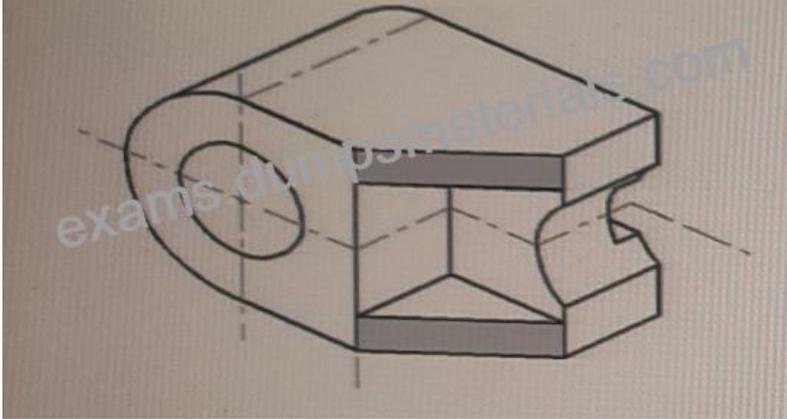
When working on this part created with AutoCAD solids, the dimensions of the shaded planes need to be established.

Which U’CS shows the correct size and shape of the shaded planes?
* Face
* Vertical
* World
* View
Explanation

The UCS is a moveable Cartesian coordinate system that establishes the XY work plane, horizontal and vertical directions, axes of rotation, and other useful geometric references1. You can change the UCS origin and orientation for convenience as you specify points, enter coordinates, and work with drawing aids1.

The face option sets a new UCS based on a selected face of a 3D solid object2. This option will align the XY plane of the UCS with the selected face and show its correct size and shape2.

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