2024 Valid ACP-01101 test answers & Autodesk Exam PDF [Q10-Q26
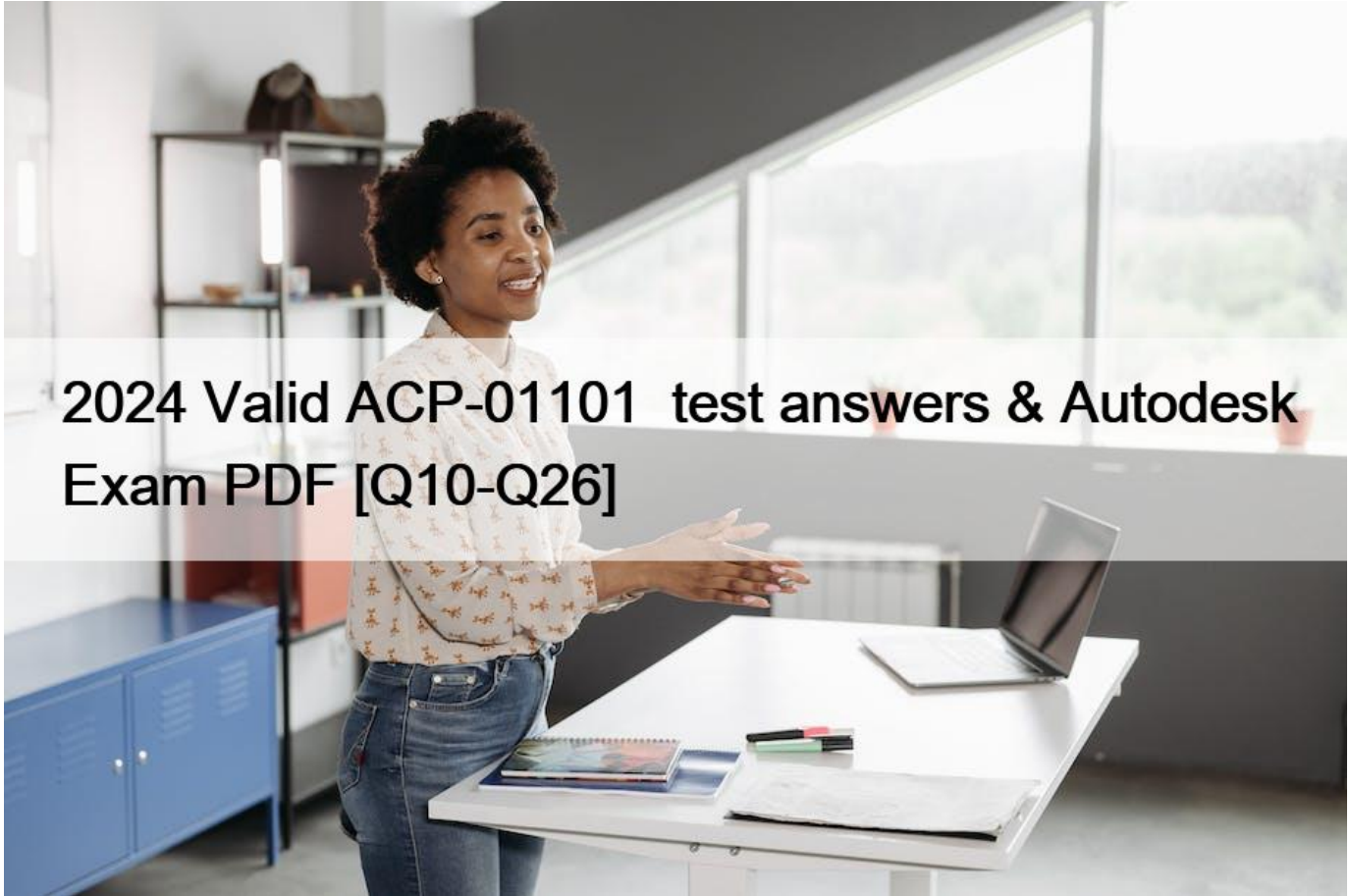

2024 Valid ACP-01101 test answers & Autodesk Exam PDF
Free Autodesk ACP-01101 Exam Questions and Answer from Training Expert ValidExam

Q10. You are modifying an AutoCAD drawing for a customer.

You would like to bring your customers attention to a specific area of the drawing that you modified for review.

Which type of object is conventionally used?
* A viewport
* A construction line
* A solid hatch
* A revision cloud
Explanation

The type of object that is conventionally used for this purpose is D. A revision cloud123. A revision cloud is a way to highlight changes or revisions in a drawing by creating a cloud-shaped polyline around them12. You can create a revision cloud by moving the mouse or by converting existing objects2. You can also change the style, color, layer and properties of the revision cloud13.

Q11. You are using the HATCH command in a drawing. You select the internal point that corresponds with the X as shown in the following exhibit.

Move each setting from the list on the left to the corresponding result on the right.
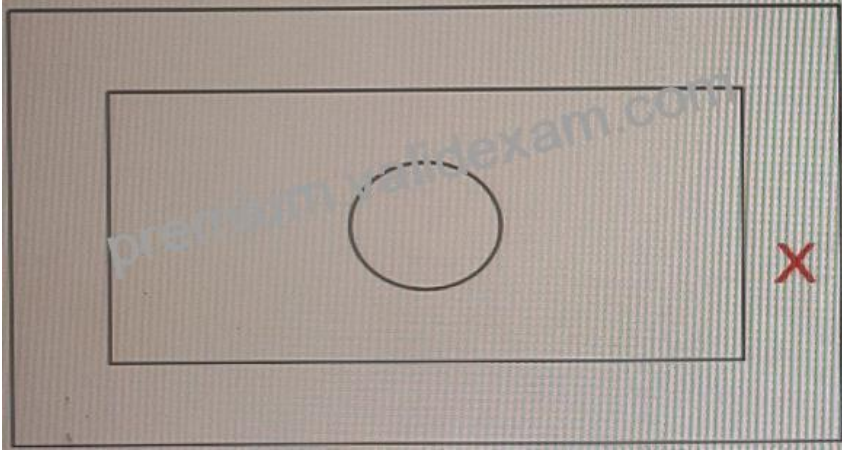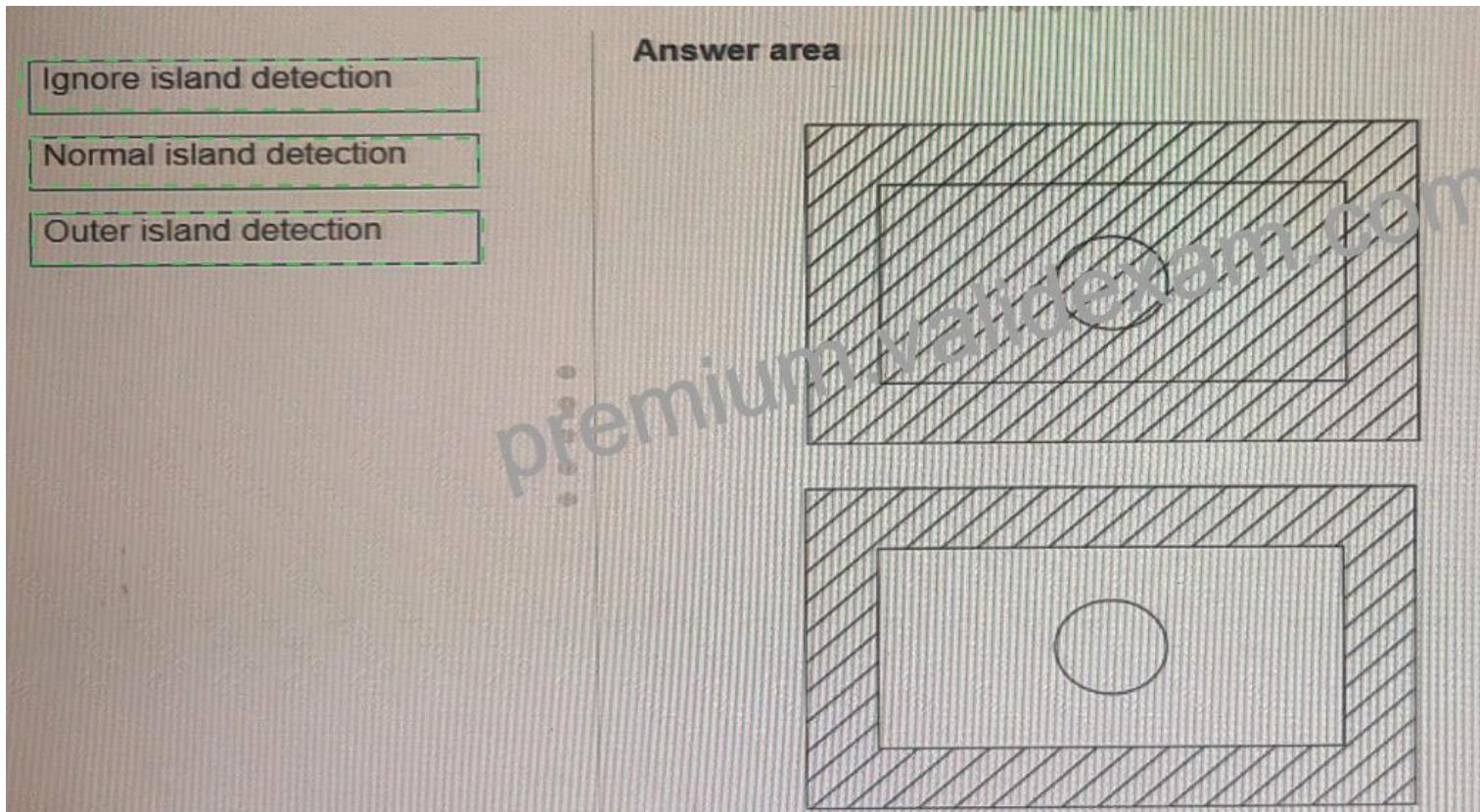
Answer area

Ignore island detection

Normal island detection

Outer island detection

Explanation

Box 1 = ignore island detection

Box 2 = Outer island detection

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

Q12. One of the lines in the drawing is at a 30 degree angle. What’s the quickest way to align the X-axis of the UCS to that line?
* Right-click the UCS icon, click Object, and then select an object in the drawing as the location.
* Start the UCS command and enter a coordinate.
* Start the UCSBASE command and set the new value
* Modify the properties of the UCS icon from Properties palette.

Explanation

UCS is a command that sets a new user coordinate system (UCS) by specifying points or by selecting an object. You can use the Object option to align the UCS with a selected object such as a line.

Based on 3 and 4, UCSBASE is a system variable that sets the name of the current UCS base. PLINEGEN is a system variable that controls how linetype patterns are generated around vertices. LWEIGHT is a command that sets the current lineweight, which is used for new objects and affects objects in the current space only.

https://knowledge.autodesk.com/support/autocad/learn-explore/caas/CloudHelp/cloudhelp/2016/ENU/AutoCAD

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

Q13. 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

Q14. 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.
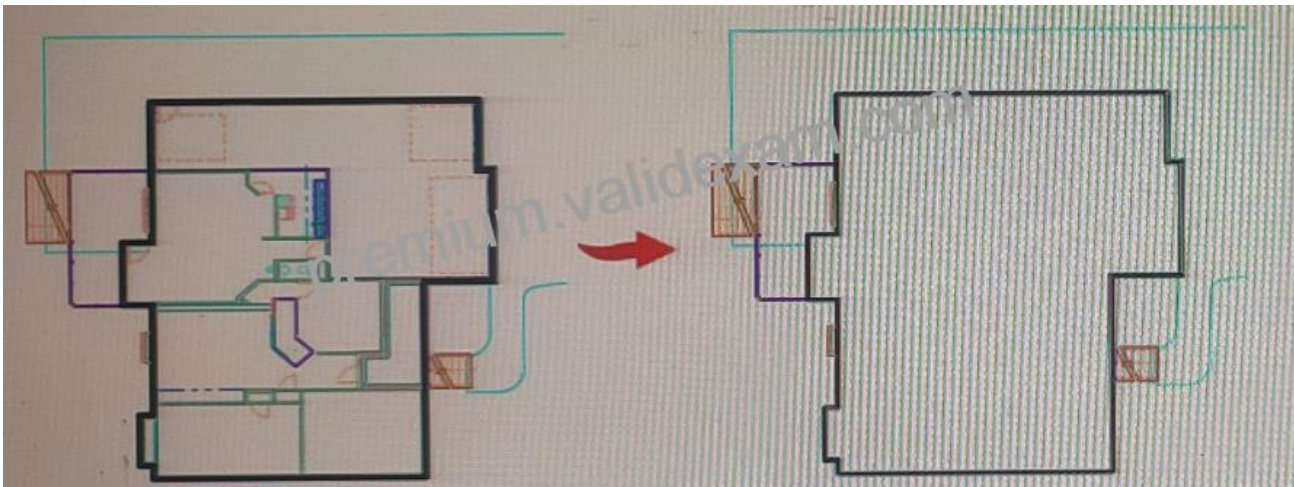
Answer area

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 Poly
WIPEOUT command with the Poly

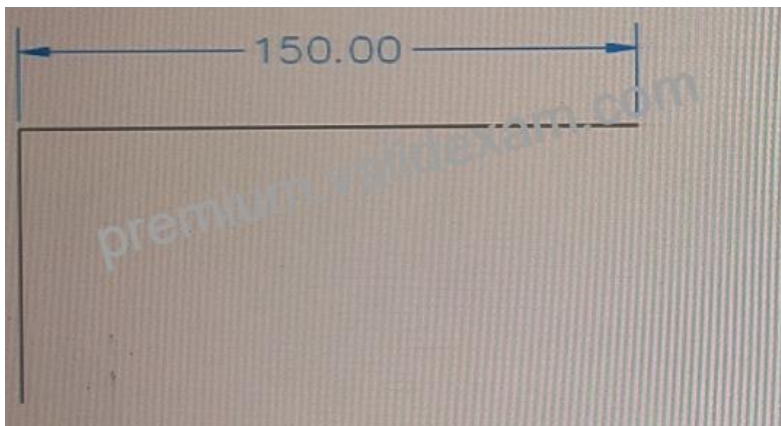To ensure that the polyline remains on top, you should use the

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

Q15.

You have a line segment that is 150 units long as shown in the exhibit.

You need the line to be 200 units long without adding any additional linework. The leftmost end of the line must remain in the current position.

What should you do?
* From the Project option of the Extend command, enter 50 units, and then select the right end of the line.
* From the Delta option of the Lengthen command, enter 50! units and then select the right end of the line
* From the Delta option of the Lengthen command, enter 200 units and then select the right end of the line
* From the Edge Extension option of the Extend command, enter 50 units, and then select the right end of the line
Explanation

From the Delta option of the Lengthen command, enter 50 units and then select the right end of the line. This way, you can increase the length of the line by 50 units from its current endpoint without changing its direction or position2

https://www.learnvern.com/autocad-tutorial-in-hindi/how-to-use-lengthen-command

Q16. You have three multileaders as shown in exhibit 1. (Refer to the exhibit 1 tab.)
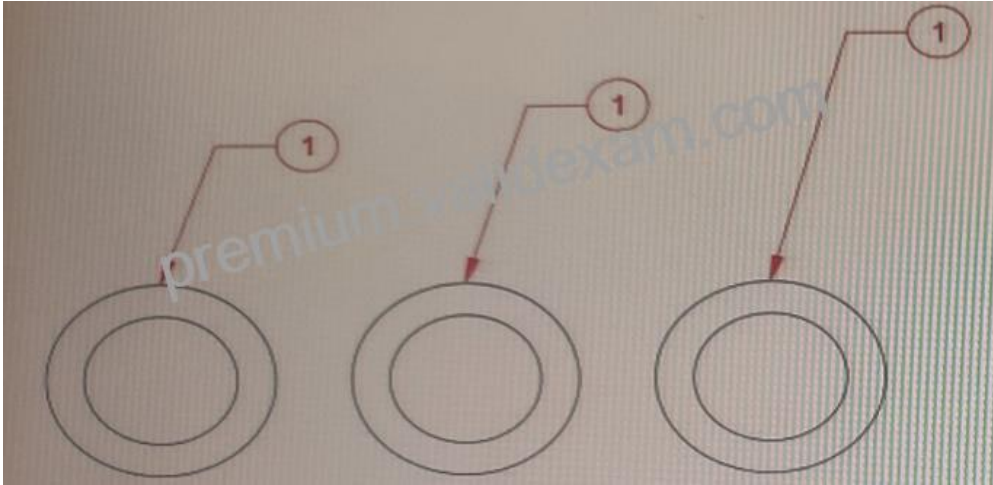
You need to change the display of the multileaders to match the multileaders in exhibit. (Refer to the exhibit 2 tab.)

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.

Q17. You have a simple polyline selected that consists of li Select whether each statement is True or False.

Answer area

By hovering over the center grip of the arc segment you can convert the arc to a line.

You can edit the polyline width by hovering over certain grips.

You can add a new vertex by hovering over certain grips.

Answer area

By hovering over the center grip of the arc segment you can convert the arc to a line.

You can edit the polyline width by hovering over certain grips.

You can add a new vertex by hovering over certain grips.

* FalseYou cannot convert an arc to a line by hovering over the center grip of the arc segment. You need to convert the arc to a polyline first, then use the FLATTEN command, and answer YES to remove hidden lines. Then you can explode the polyline to get line segments.

https://forums.autodesk.com/t5/autocad-forum/autocad-is-it-possible-to-convert-an-arc-to-a-segemented-li

* Falseyou can edit the polyline width by hovering over certain grips if they are vertex grips or midpoint grips. However, if they are arc segment grips, you cannot edit the width by hovering over them1.

Therefore, the statement is false.

https://knowledge.autodesk.com/support/autocad/learn-explore/caas/CloudHelp/cloudhelp/2015/ENU/Auto

* Falseyou can add a new vertex by hovering over certain grips if they are edge grips or midpoint grips. However, if they are vertex grips or arc segment grips, you cannot add a new vertex by hovering over them2. Therefore, the statement is false.

Q18. Refer to the exhibit.
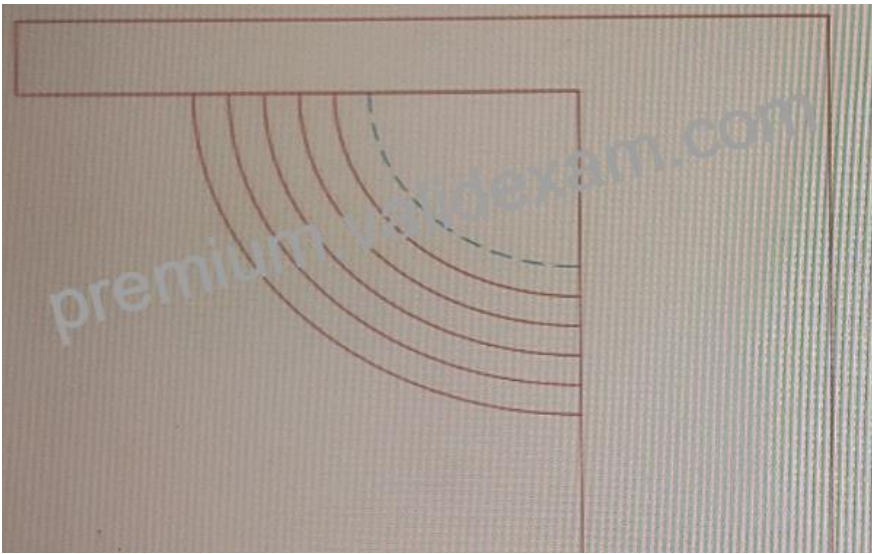
The exhibit shows a set of ornamental garden stairs going up to a walkway as part of a landscaping design.

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
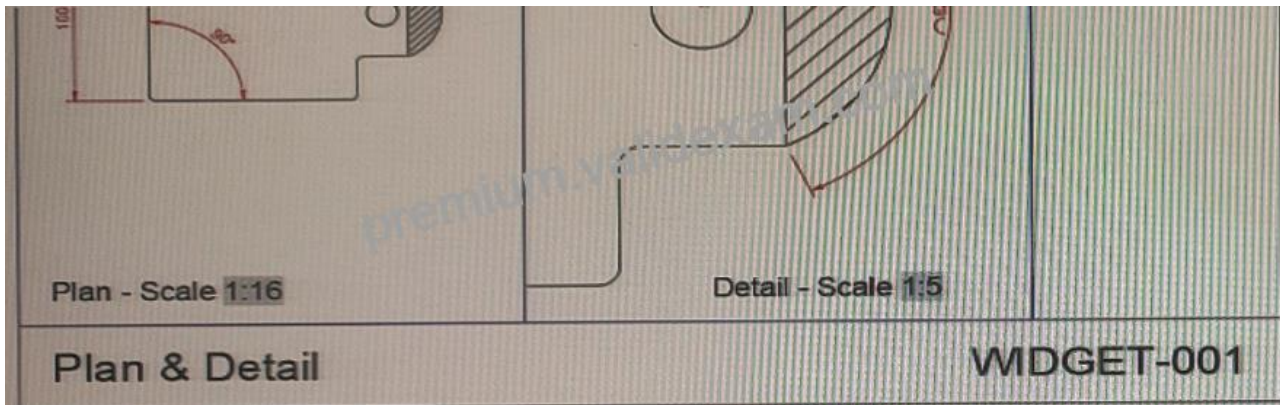* Source
Explanation

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

Q19. 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.

Q20. In model space (MODEL), a room label text height is set to 300 mm in the designated text style.

Which viewport scale should you apply to a layout viewport (PAPER) so that the room label text height will display as 3 mm when plotting the drawing?
* 1:100
* 100:1
* 1:1
* A custom viewport scale
Explanation
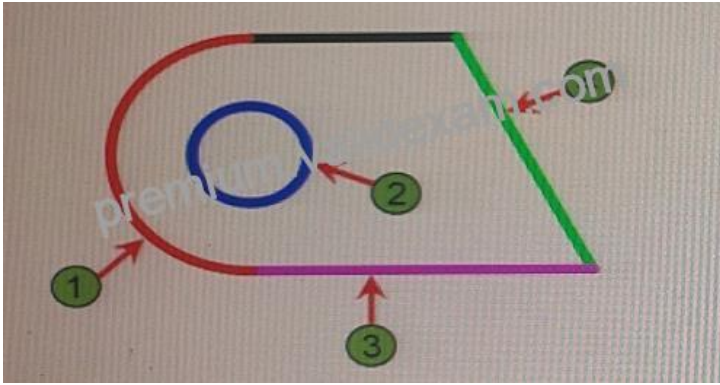
the formula for determining object scaling is:

Object Height x Annotation Scale x Annotation Plot Size = Model Space or Paper Space Height In this case, we want the room label text height to be 3 mm in paper space when plotting the drawing.

Assuming an annotation plot size of 1 mm and an object height of 300 mm, we can solve for the annotation scale as follows:

300 x Annotation Scale x 1 = 3 Annotation Scale = 3 / 300 Annotation Scale = 0.01 Therefore, the viewport scale that corresponds to this annotation scale is B. 100:1.

https://knowledge.autodesk.com/support/autocad-architecture/learn-explore/caas/CloudHelp/cloudhelp/2016/EN

Q21. You have the drawing shown in the exhibit.

You are using the DIM command to preview suitable dimension type objects in the drawing.

Which dimension type will appear when you hover over each object? Select the appropriate dimension type from each of the drop-down lists.

Answer area

① - Arc

Align
Angular
Diameter
Linear
Radial

② - Circle

Align
Angular
Diameter
Linear
Radial

③ - Horizontal line

Align
Angular
Diameter
Linear
Radial

④ - Angled line

Align
Angular
Diameter
Linear
Radial

Answer area

1 - Arc

| Align |
| Angular |
| Diameter |
| Linear |
| Radial |

2 - Circle

| Align |
| Angular |
| Diameter |
| Linear |
| Radial |

3 - Horizontal line

| Align |
| Angular |
| Diameter |
| Linear |
| Radial |

4 - Angled line

| Align |
| Angular |
| Diameter |
| Linear |
| Radial |
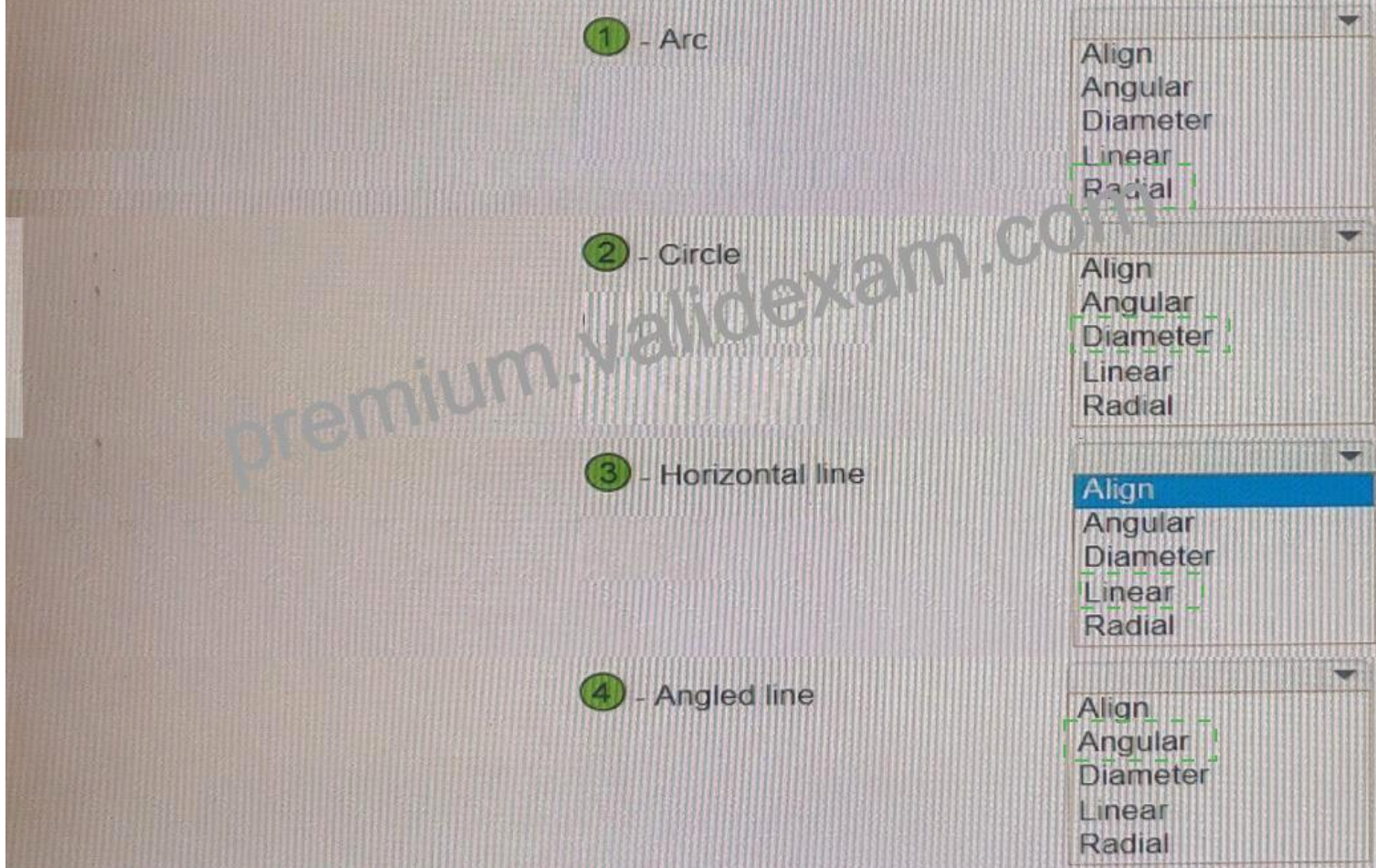
* When you hover over the arc (1), you will see a radius dimension type preview.

* When you hover over the circle (2), you will see a diameter dimension type preview.

* When you hover over the horizontal line (3), you will see a horizontal linear dimension type preview.

* When you hover over the angled line (4), you will see an angular dimension type preview.

Q22. 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.

Q23. What should be typed at the command line in order to start creating a line for drawing reference that starts at a defined point and extends to infinity in the designated direction?
* CURSORSIZE
* MLINE
* XLINE
* RAY
Explanation

According to the AutoCAD for Design and Drafting documents1, the command that should be typed at the command line in order to start creating a line for drawing reference that starts at a defined point and extends to infinity in the designated direction is XLINE2. This command creates a construction line, which is a type of reference line that can help you manage accurate parameters of your drawing2. You can also use point objects as nodes or reference geometry for object snaps and relative offsets3.

Q24. A block entity in the current drawing needs to be converted to a new drawing file of its own.

Which command should be used?
* BEDIT
* BSAVEAS
* BLOCK
* WBLOCK
Explanation

the command that should be used to create a new drawing file from selected objects is WBLOCK. This command allows you to export a group of objects as a drawing, which can be inserted into other drawings as a block2.

Q25. 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.
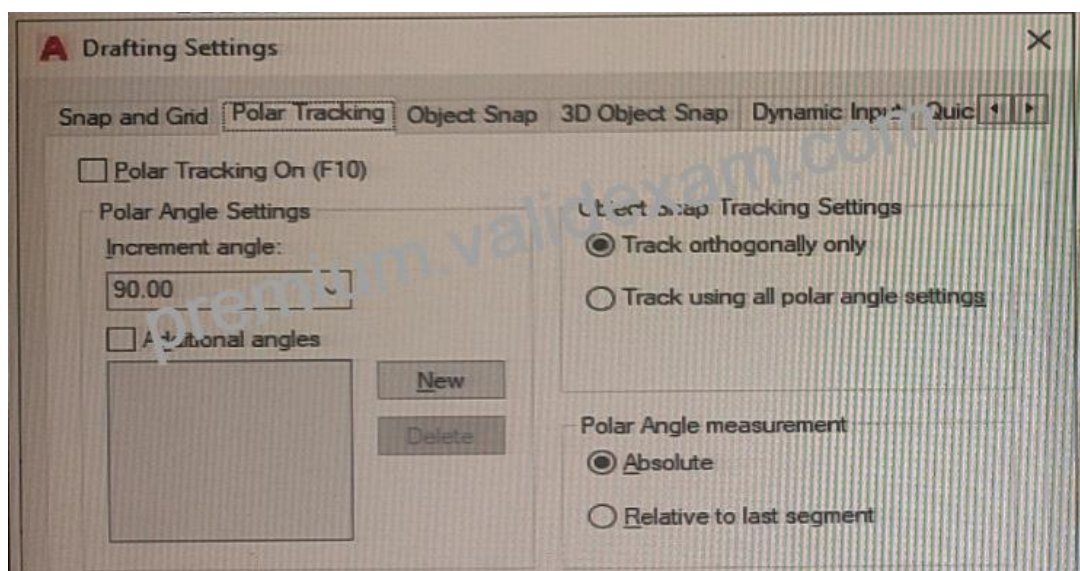
See the answer below:

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.

Q26. Refer to the exhibit.
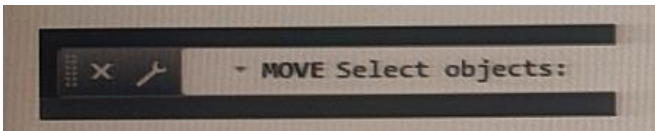
You are working on a drawing. You copy a set of entities and exit the COPY command. You then use the MOVE command on the same set of entities.

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

Top Autodesk ACP-01101 Courses Online: https://www.validexam.com/ACP-01101-latest-dumps.html]