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Autodesk ACP-01101

Version Demo

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Topic Break Down

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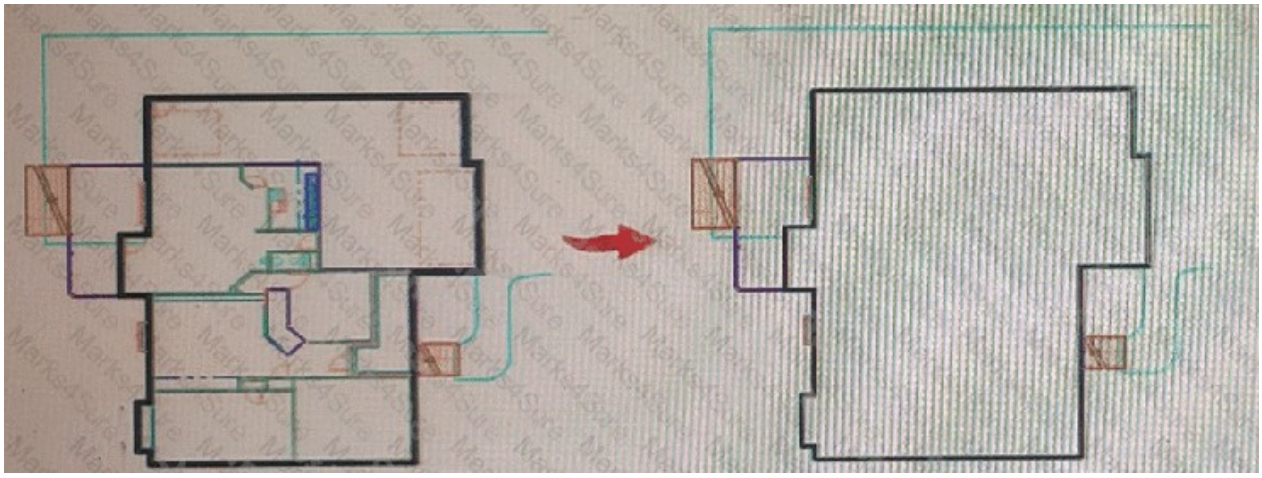
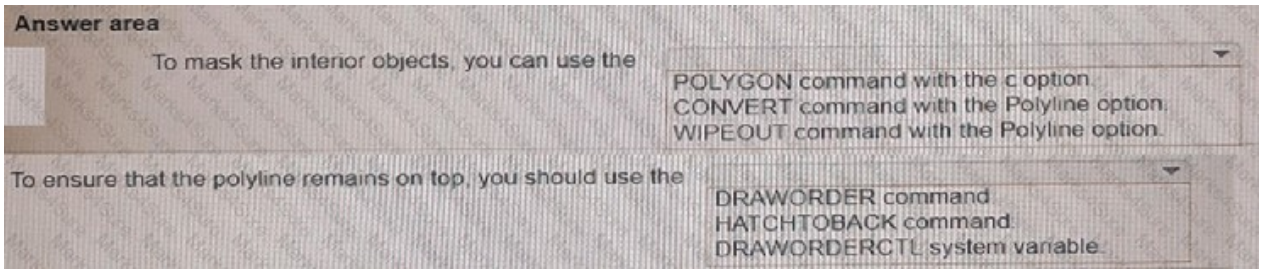
QUESTION NO: 1 - (SIMULATION)

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: See the explanation for the answer

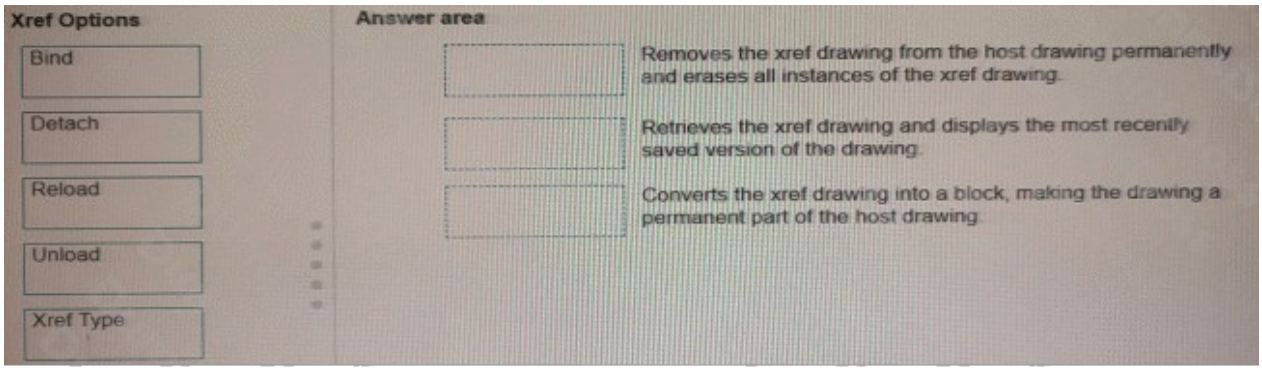
Explanation:

Box 1: Select `WIPEOUT command with the Polyline option`.

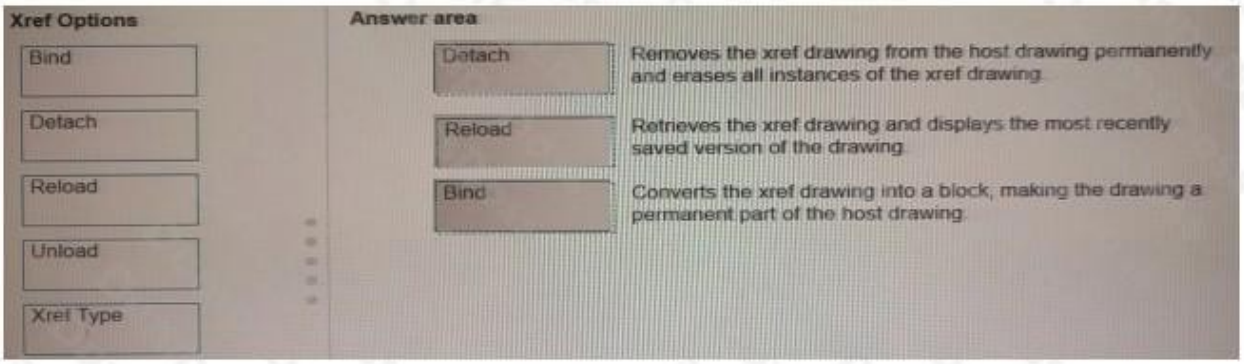
Box 2: Select `DRAWORDER command`.

QUESTION NO: 2 - (DRAG DROP)

Move three applicable xref options from the list on the left to the corresponding functions on the right Xref options may be used once or not at all. (Select 3)



ANSWER:



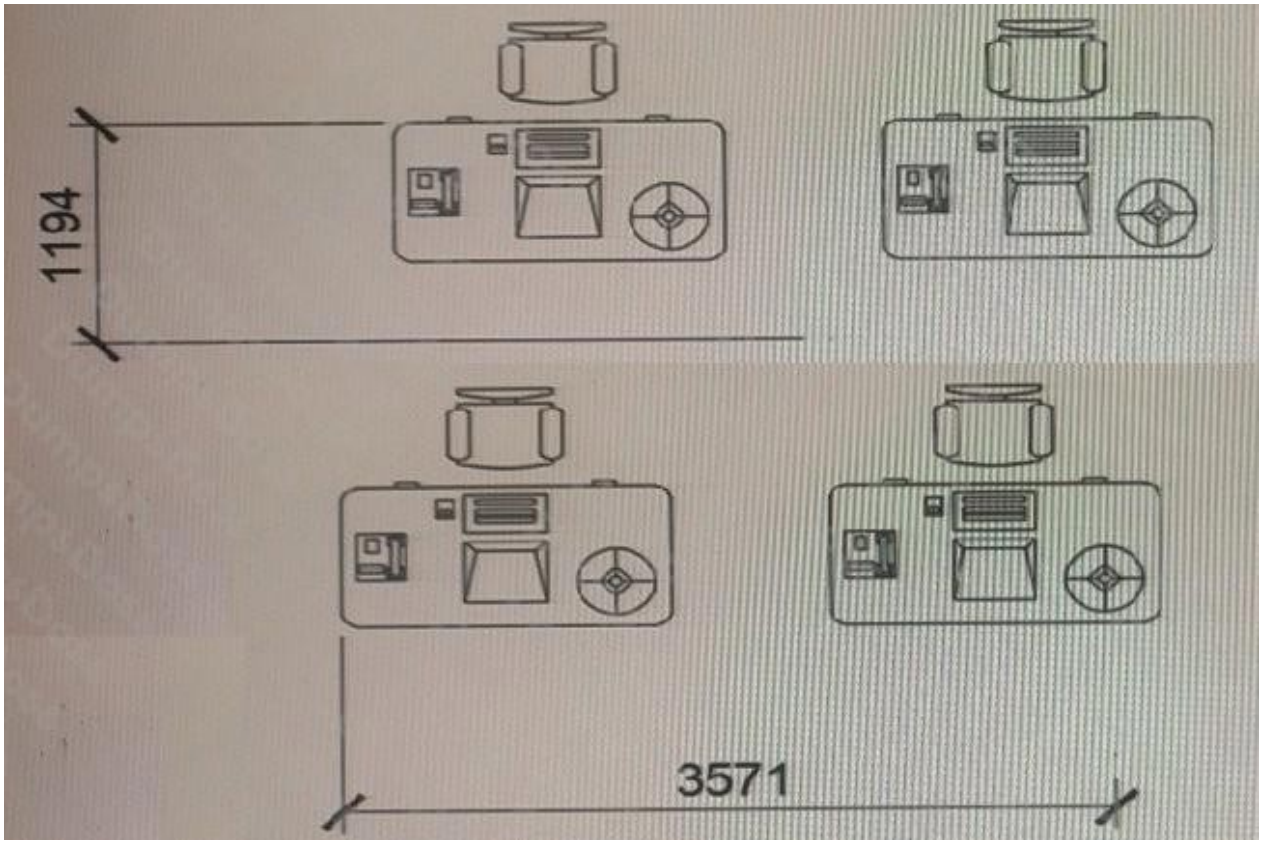
Explanation:

External references let a host drawing display content stored in another drawing file while retaining a connection to that source file. The mapped commands correctly describe three standard ways AutoCAD manages that relationship in the External References palette. **Detach** permanently removes the selected external reference definition from the current host drawing. As part of that removal, every inserted instance of that xref is erased from the host drawing, so the drawing no longer contains or references that external file. This is the appropriate operation when the external drawing is no longer needed in the project file.

Reload updates an existing xref from its source drawing. It retrieves the currently saved source file and refreshes the xref display in the host drawing, allowing edits saved to the referenced drawing to appear without reattaching the reference. This preserves the existing xref relationship, insertion locations, and reference settings while displaying the latest saved source content.

Bind converts the xref into a block definition within the host drawing, making the referenced geometry a permanent part of that drawing rather than maintaining an external-file dependency. Binding is useful when a drawing must be packaged as a self-contained deliverable while preserving the referenced content. Autodesk's documentation on working with external references describes the available reference-management actions, including reloading, detaching, and binding: Manage External References. The visual placements therefore accurately pair each function with the AutoCAD xref command that performs it.

QUESTION NO: 3



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?

A. Reassociate (DIMREASSOCIATE)

B. Break (D1MBREAK)

C. Inspect (DIMINSPECT)

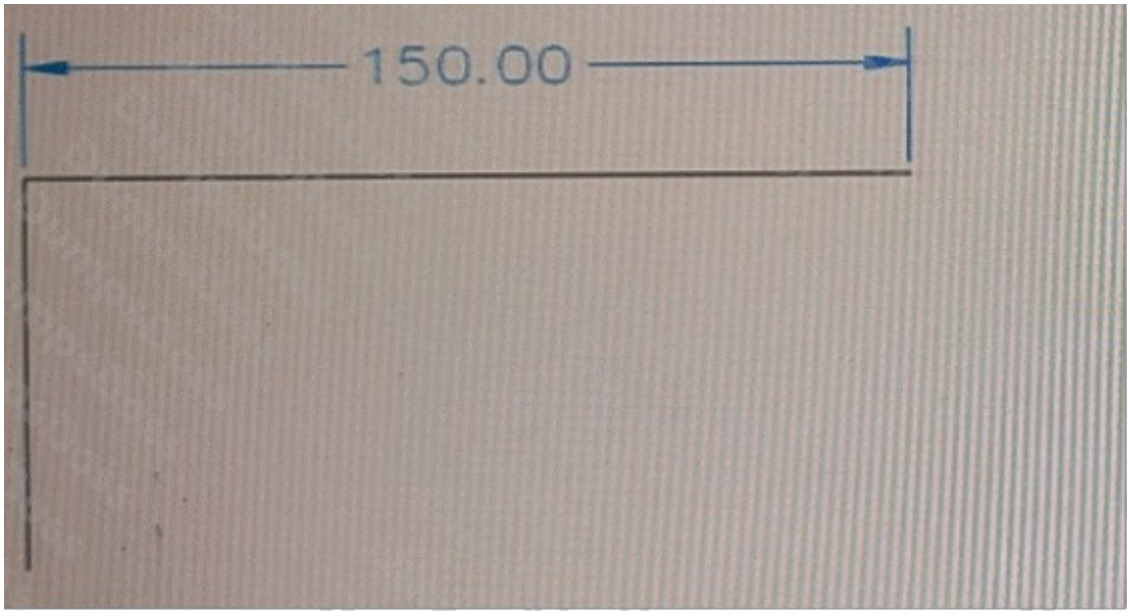
D. Adjust Space (DIMSPACE)

ANSWER: A

Explanation:

Reassociate (DIMREASSOCIATE) is the appropriate command because the issue is specifically that a definition point, or origin point, of each existing dimension was selected at an incorrect location. DIMREASSOCIATE enables the user to select an existing dimension and then re-establish its association with the intended geometry or precise points on that geometry. For the horizontal and vertical spacing dimensions in the exhibit, the command can be used to identify the misplaced extension-line origin and snap it to the correct workstation reference point. This corrects the measured relationship while retaining the existing dimension object, its annotation, and its established dimension style rather than requiring the dimension to be recreated manually. Associative dimensions are intended to maintain their connection to the defining geometry, so reassociating the definition points ensures that the displayed row and column spacing reflects the actual workstation layout and will update appropriately if the associated geometry is later modified. Autodesk describes DIMREASSOCIATE as a tool for associating or reassociating selected dimensions with objects or points on objects. See Autodesk's DIMREASSOCIATE command reference and its guidance on associative dimensions.

QUESTION NO: 4



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?

A. From the Project option of the Extend command, enter 50 units, and then select the right end of the line.

B. From the Delta option of the Lengthen command, enter 50 units and then select the right end of the line.

C. From the Delta option of the Lengthen command, enter 200 units and then select the right end of the line

D. From the Edge Extension option of the Extend command, enter 50 units, and then select the right end of the line

ANSWER: B

Explanation:

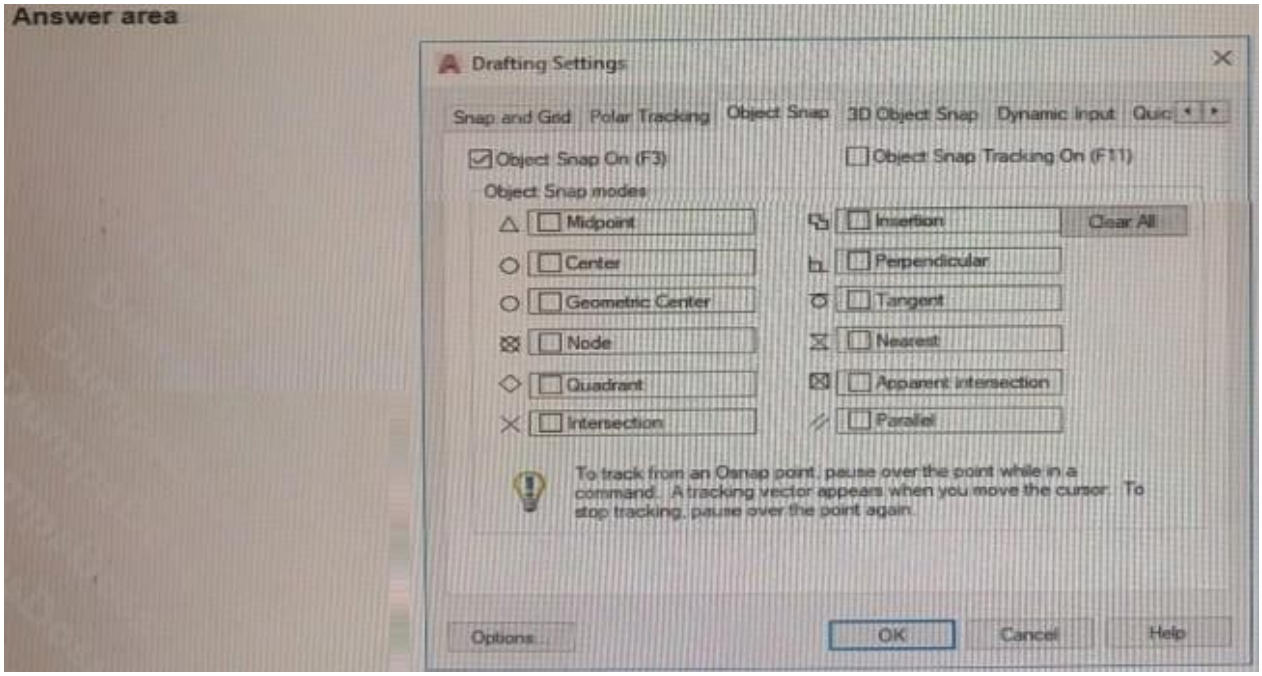
From the Delta option of the Lengthen command, enter 50 units and then select the right end of the line is correct because Delta changes an object's current length by the specified increment rather than setting an absolute final length. The existing line is 150 units long, so increasing it by a delta of 50 units produces the required final length of 200 units. When using LENGTHEN on a line, selecting near the right endpoint identifies that end as the one to be modified. The line is extended along its existing direction from that endpoint, leaving the opposite, leftmost endpoint fixed in its current location. This directly satisfies both requirements: no separate geometry is created, and the original left endpoint does not move. LENGTHEN is intended for precisely this kind of edit, allowing a line, arc, or polyline segment to be adjusted by a delta, percentage, total length, or dynamic point selection. For Autodesk's AutoCAD documentation and command guidance, see the AutoCAD Help and Autodesk's LENGTHEN command support article.

QUESTION NO: 5 - (HOTSPOT)

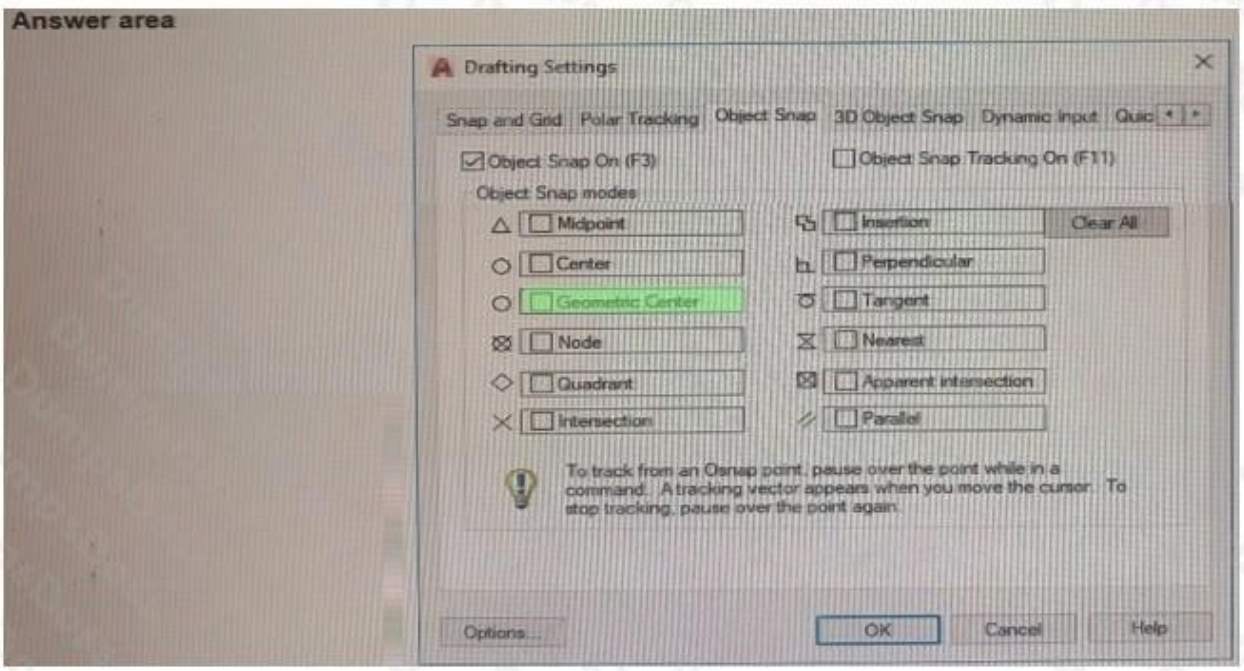
You need to snap to the centroid of a closed polyline.

Which Object Snap mode should you use? Click the correct option in the answer area.

Note: Exhibit shown is the AutoCAD interface in Windows. While the AutoCAD for MAc version differs in appearance, the correct answer correlates in the options.



ANSWER:



Explanation:

Geometric Center is the correct Object Snap mode because it acquires the calculated center point of a closed polyline. When an object is a closed polyline, its boundary encloses an area, and AutoCAD can determine the geometric center of that enclosed shape. This makes Geometric Center the appropriate precision tool when the drawing task specifically requires a snap to the polyline's centroid rather than to a visible vertex, an edge midpoint, or another boundary-related location.

After Geometric Center is enabled as a running object snap, moving the crosshairs over an eligible closed polyline displays the geometric-center Object Snap marker when AutoCAD detects that point. The cursor can then be used to select that exact calculated center as the point for a command, such as placing a block, starting geometry, aligning an object, or creating dimensions and construction geometry from the center of the closed shape. This feature also supports closed splines, provided that AutoCAD can calculate the geometric center for the object.

Object snaps are intended to ensure accurate coordinate acquisition without manually estimating locations. Using Geometric Center is especially valuable where a shape is irregular, because the center may not be obvious from its extents or from the midpoint of any single segment. Autodesk's documentation describes Geometric Center as snapping to the centroid of a closed polyline or spline; see AutoCAD Object Snap reference. Therefore, enabling and selecting Geometric Center provides the requested centroid snap behavior.

QUESTION NO: 6

An attached external reference must become a permanent part of the current drawing.

You use the Bind option in the External References palette.

Which two results occur? (Select 2)

A. The xref becomes a block definition in the current drawing

B. The original referenced drawing file is deleted

C. Dependent named objects are renamed with a prefix based on the xref name

D. The xref continues to reload whenever its source file changes

E. All xref layers are automatically frozen

ANSWER: A C

Explanation:

The xref becomes a block definition in the current drawing and **dependent named objects are renamed with a prefix based on the xref name** are correct. Binding incorporates the externally referenced drawing into the host drawing so that the host drawing no longer depends on the external DWG file. The bound content is retained as a block definition in the current drawing.

When the Bind option is used, AutoCAD preserves named objects such as layers, linetypes, text styles, and dimension styles from the xref by adding a prefix based on the xref name. This prevents conflicts with similarly named objects already present in the host drawing. The Insert option differs because it merges identically named dependent objects into the current drawing without adding the prefix.

See Autodesk's AutoCAD Help documentation for information about binding external references.

QUESTION NO: 7

Exhibit.

The building label shown is a single piece of annotative text with two annotative scales assigned to it (1: 50 and 1100)

The current drawing scale is set to 1:50.

The 1:100 scale label must be moved outside the building and the 1:50 scale label must remain inside the building, as shown on the right in the exhibit

How should the label location be modified?

A. Set the drawing scale to 1:100 and then use the MOVE command

B. Leave the drawing scale set to 1:50 and then grip edit the text label

C. Leave the drawing scale set to 1:50 and then use the MOVE command

D. Set the drawing scale to 1:100 and then grip edit the text label

ANSWER: D

Explanation:

Set the drawing scale to 1:100 and then grip edit the text label. Annotative text can maintain separate scale representations, including representation-specific positioning. To alter the placement that is displayed for only the 1:100 representation, first make 1:100 the current annotation scale. This makes that representation active for editing. Selecting the text and using its grip to relocate it changes the position stored for the active 1:100 representation, allowing it to be placed outside the building while preserving the existing location of the 1:50 representation inside the building.

This distinction is important when an annotative object has more than one assigned scale: the current annotation scale controls which representation is being worked with, and grip editing supports an adjustment to that representation's location. After the grip edit, changing the current annotation scale back to 1:50 displays the unchanged 1:50 placement. Autodesk's AutoCAD Help provides the product documentation for annotation scaling, annotative objects, and their scale representations at AutoCAD Help. Autodesk also provides an overview of annotation tools and workflows at CAD annotations.

QUESTION NO: 8

Refer to exhibit.

[Note: Exhibit shown is the AutoCAD interlace in Windows. While the AutoCAD for Mac version differs in appearance, the correct answers correlate in the options.]

After completing markups for a portion of limit project, a CAD designer must publish new PDFs (01 sheets C200 and C201

The Sheet List (.dsd) Km the project includes all drawing sheets for the project as shown in the exhibit.

Which workflow should be used to publish PDEs for the updated sheets only?

A. Select the C100 and C101 sheets click the Remove Sheets (-) button and click Publish

B. Select the C100 and C101 sheets, change Page Setup to Do Not Plot, and click Publish

C. Select the C200 and C201 sheets from the list, and click Publish.

D. Select the C100 and C101 sheets set Number of copies to 0. and click Publish

ANSWER: A

Explanation:

Select the C100 and C101 sheets click the Remove Sheets (-) button and click Publish is the appropriate workflow because the Sheet List stored in the DSD file contains the full project sheet set, while the required output is limited to the updated C200 and C201 sheets. In AutoCAD's Publish dialog, the sheet list is the set of sheets that will be included in the publish job. Removing the sheets that are not to be issued leaves C200 and C201 as the only entries in the current publish list. Clicking Publish then creates PDFs only for those remaining updated sheets.

This approach is particularly useful when starting from an existing DSD configuration, because it retains the established publishing settings—such as the PDF device, page setups, file naming, and output location—while allowing the designer to temporarily limit the current issue to the revised sheets. Autodesk documents that the Publish dialog supports managing the sheet list and publishing selected drawing sheets to an electronic output format. See Autodesk's guidance for creating PDF output in AutoCAD and the AutoCAD Help documentation for publishing workflows.

QUESTION NO: 9

A drawing contains numerous dimensions that use the same dimension style.

You modify the text height and arrow size in that dimension style.

Which two statements are true? (Select 2)

A. Dimensions that use the modified style update to reflect the new settings

B. New dimensions created with that style use the new settings

C. All dimensions with direct overrides are reset automatically

D. The dimensions are converted to exploded objects

E. The measured model geometry is resized

ANSWER: A B

Explanation:

Dimensions that use the modified style update to reflect the new settings and **new dimensions created with that style use the new settings** are correct. A dimension style stores common formatting settings for dimensions, including text appearance, arrowheads, line properties, units, and fit behavior. When a style is modified, dimensions using that style are updated unless they have direct overrides that control the affected properties.

Direct dimension overrides can cause an individual dimension to retain a property value that differs from its style. Changing the dimension style does not convert dimensions to exploded objects, and it does not alter model-space geometry being measured.

See Autodesk's AutoCAD Help documentation for dimension style management.

QUESTION NO: 10

Which two objects support associative dimensions? (Select 2)

A. Hatches

B. Arcs

C. Images

D. Polylines

ANSWER: B D

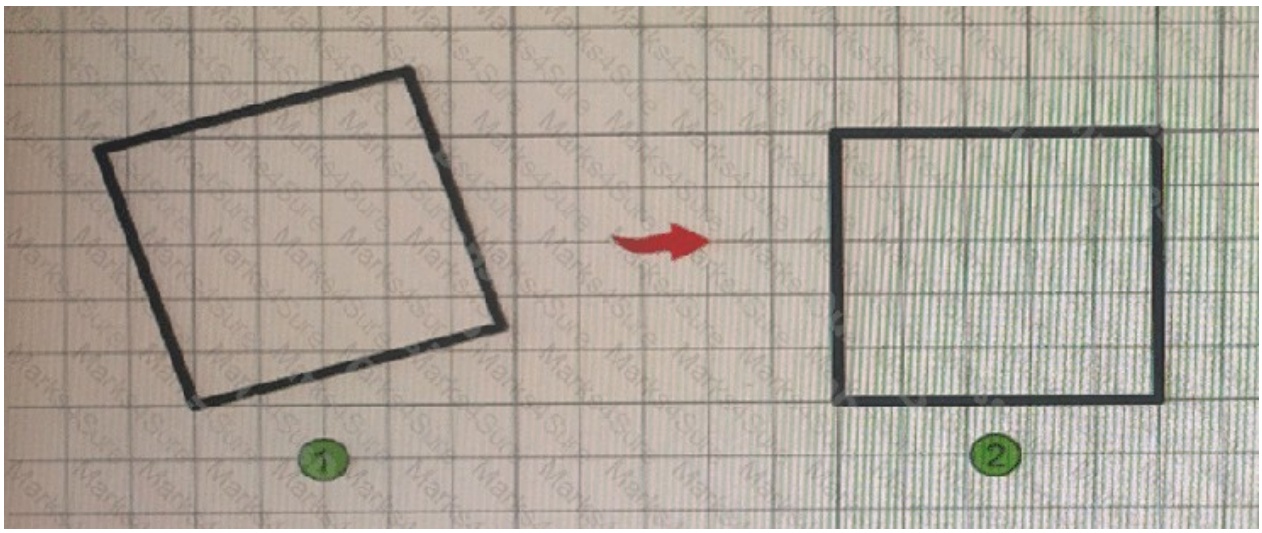
Explanation:

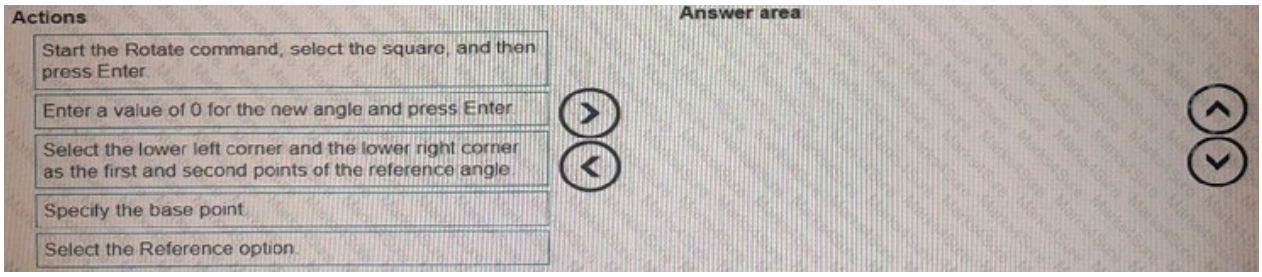
Arcs and **Polylines** support associative dimensions because AutoCAD can maintain a direct association between a dimension's definition points and the geometric features of these objects. With the `DIMASSOC` system variable set to 2, dimensions created from valid object snap points or other geometric locations on an arc or polyline retain that relationship. If the arc's radius, endpoints, or location changes, or if a polyline vertex or segment is edited, AutoCAD recalculates and updates the associated dimensional value and placement as needed. This behavior is particularly important in revision-driven drafting because the annotation continues to reflect the edited geometry rather than requiring manual redimensioning. Arc geometry can be used by appropriate linear, angular, radial, and diameter dimension tools, while polyline vertices and segments provide valid references for dimensions that measure their distances, angles, and locations. Autodesk describes associative dimensions as dimensions that automatically adjust when the geometry they measure is modified, and identifies `DIMASSOC` as the control for dimension associativity. See Autodesk's Dimension Associativity documentation and the DIMASSOC system variable reference.

QUESTION NO: 11 - (SIMULATION)

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.





ANSWER: See the explanation for the answer

Explanation:

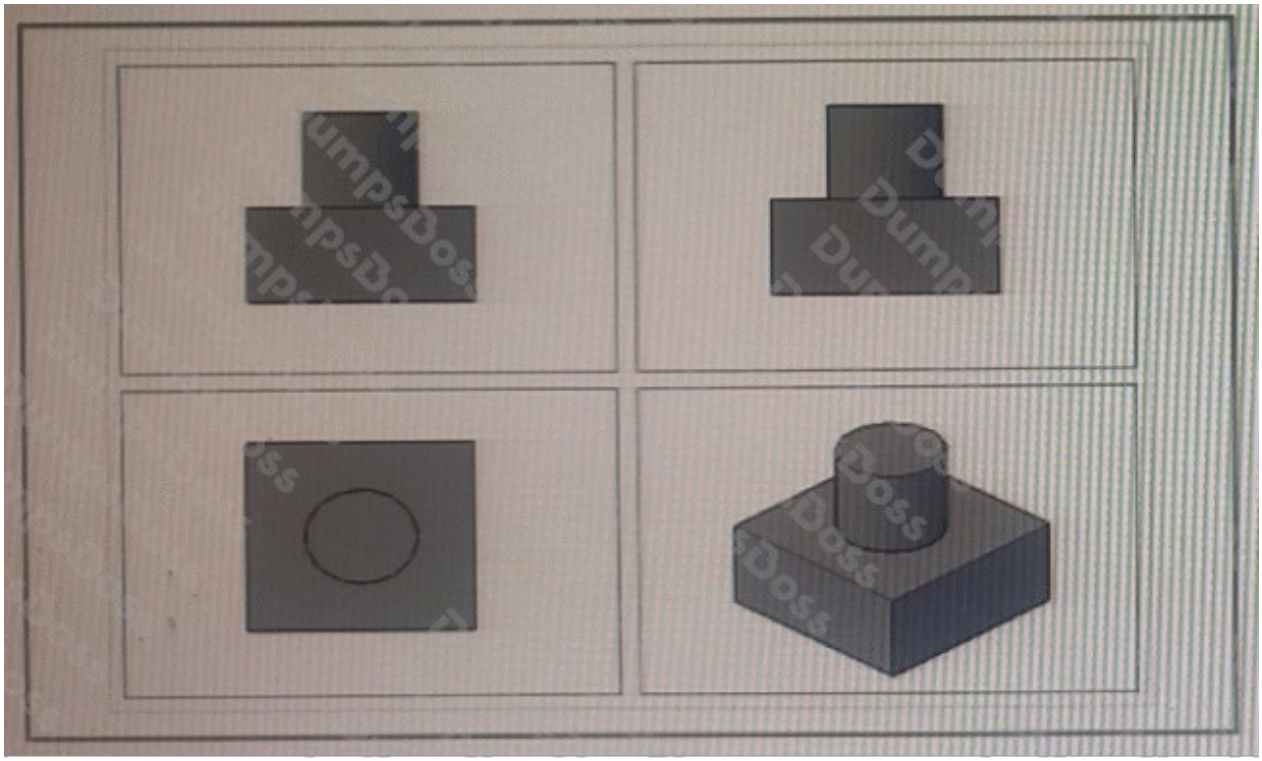
Correct order:

The Reference option measures the existing angle of square (1) from its lower edge. Entering `0` rotates that reference edge to a horizontal, zero-degree angle, matching square (2).

QUESTION NO: 12

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)



A. Clip a viewport to multiple different shapes by using the VPCLIP command.

B. From the Viewports dialog box: select a viewport configuration.

C. Create individual viewports by using the MVIEW command.

D. From the View Manager dialog box, generate named views and add them as layout viewpoints.

E. Create a new page setup.

ANSWER: B C D

Explanation:

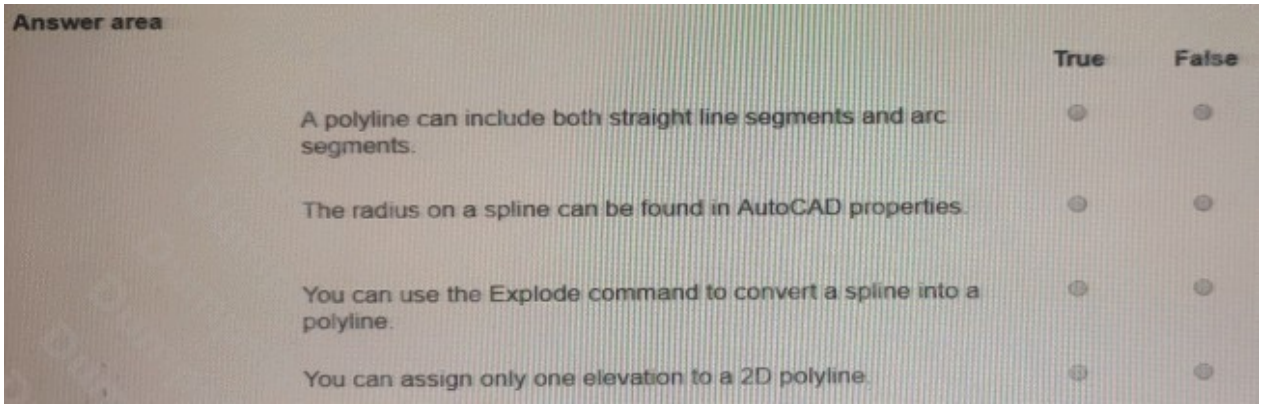
Layout viewports are paper-space objects that provide windows into model space, allowing separate views of the same drawing to be placed on a plotted sheet. Selecting a viewport configuration in the Viewports dialog box is a direct way to create several rectangular layout viewports at once. The selected configuration determines the number and arrangement of the viewport windows, making it useful when a standard multi-view layout is needed.

The MVIEW command is the primary command for creating layout viewports individually. It supports creation of one or more viewport boundaries in a layout and is appropriate when viewport size, location, or shape must be controlled manually. Named views provide another supported workflow: views saved and managed in the View Manager can be restored when creating a layout viewport. This preserves a defined model-space orientation, zoom level, and visual view definition for use on the sheet.

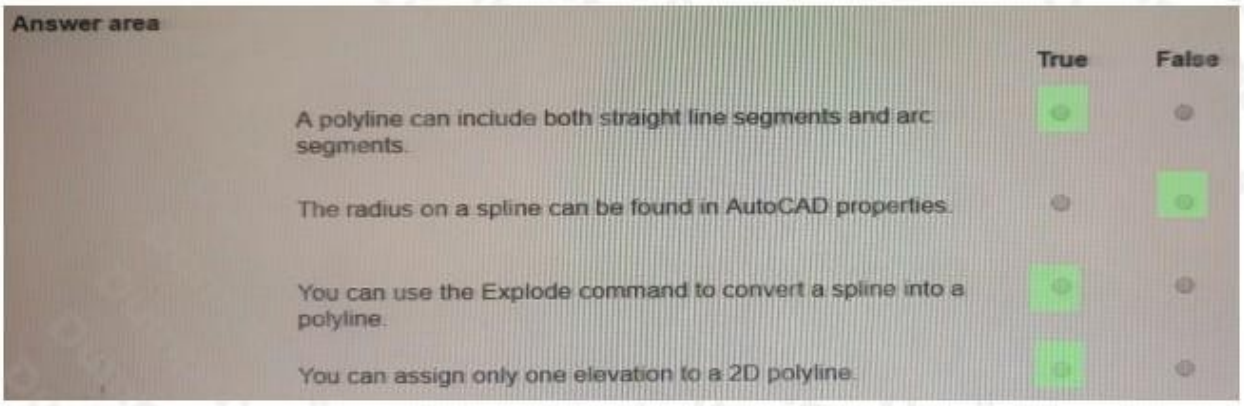
These methods can also be combined: create a configuration for the overall layout, then use MVIEW or named views to establish the specific model views required for each sheet. Autodesk describes layout viewport creation methods and configurations in its AutoCAD layout viewport documentation.

QUESTION NO: 13 - (HOTSPOT)

Select whether each statement is True or False.



ANSWER:



Explanation:

The correct selections reflect the fundamental object definitions and editing behavior of AutoCAD. A polyline is a single connected object that can contain both linear and curved segments. During the PLINE command, users can switch between Line and Arc modes while continuing the same polyline, so a single polyline can combine straight portions with arc portions. Autodesk describes this capability in its Polyline overview.

A spline is a smooth NURBS-based curve defined by fit points or control vertices. It does not have one fixed, object-wide radius like a circle or an arc. Curvature, and therefore the local radius of curvature, may vary continuously along its length. As a result, the standard Properties palette does not provide a single Radius property for a spline. Autodesk's Spline documentation describes splines as smooth curves controlled by fit points or control vertices rather than fixed-radius geometry.

The EXPLODE command can convert a spline into a polyline representation. When exploding a spline, AutoCAD prompts for a precision value; that precision controls how closely the resulting polyline approximates the original spline. This makes EXPLODE an available method when individual polyline segments or vertices are needed for subsequent editing. Autodesk documents EXPLODE and its behavior for compound objects in the EXPLODE command reference.

Finally, a 2D polyline is planar and has one common elevation. Its vertices are located at that same elevation rather than independently varying in Z. Assigning a different elevation to individual vertices requires a 3D polyline or another suitable 3D object workflow. Therefore, only one elevation can be assigned to a 2D polyline.