

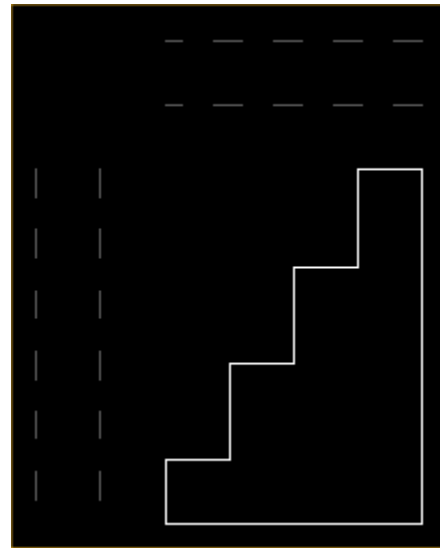
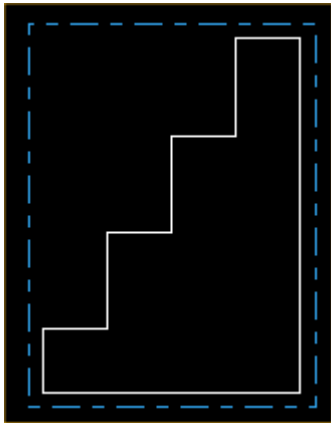
Using Snap Lines to Control Text Locating References

Snap lines are used to locate dimensions, notes, geometric tolerances, symbols, surface finishes, cross section view arrows, projection view arrows and clipped dimensions. Snap lines can be positioned relative to a view outline, a model edge, datum plane, or an existing snap line. Once an entity has been "attached" to a snap line, that entity will move with the snap line if the snap line is moved.

Procedure

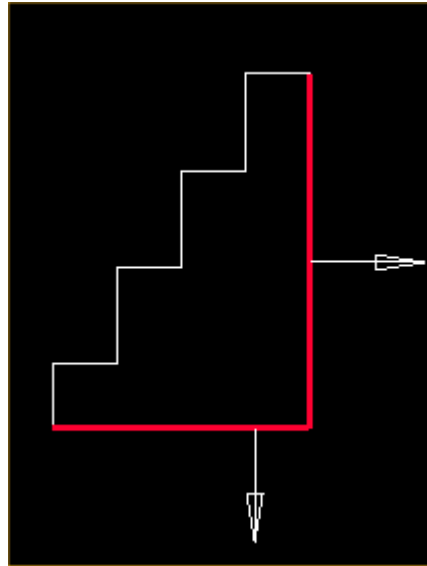
After a drawing is created and a view is placed, create a snap line by selecting **Insert > Snap Line**. The CR SN LINE menu will then appear with the following options: **Offset View** and **Offset Object**.

If the snap line is to be attached to a view border, select **Offset View**, pick one or more of the view borders which are highlighted in blue (see Figure 1) and select **OK** from the SELECT menu. Note: To select multiple items Press CTRL when making selections.

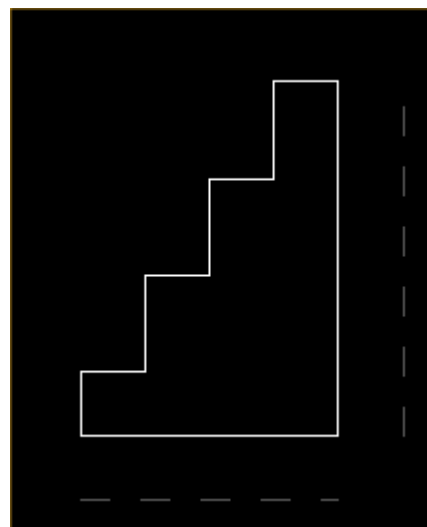


When prompted to "Enter distance of first snap line from boundary (outward is positive) [.5000]", enter in the offset distance from the view boundary to the snap line. When prompted to "Enter the number of snap lines to create [1]", enter in the number of snap lines to create. If more than one snap line is to be created, the prompt "Enter the distance between snap lines [0.3750]:" will appear in the message window. The distance between the snap lines should be entered at this time. Figure 2 shows two snap lines created off of the top and left view borders.

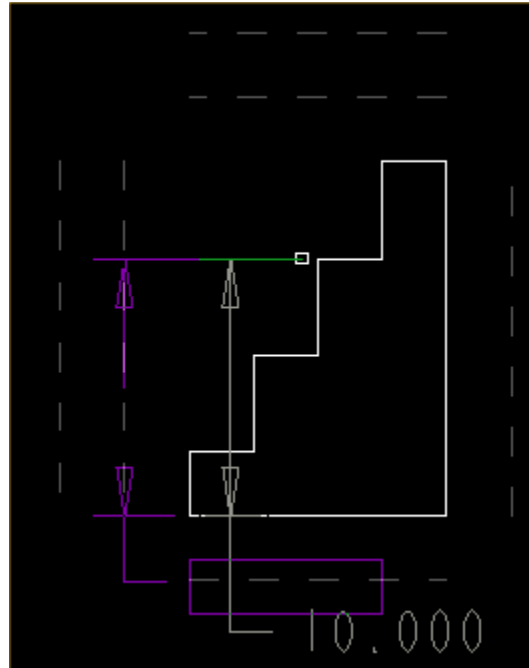
If a snap line is to be attached to a piece of geometry (edge, datum plane, or an existing snap line), select **Offset Object** from the CR SN LINE menu and pick one or more of the entities to create snap lines off of. Select **OK** from the SELECT menu (see Figure 3). Note: To select multiple items Press CTRL when making selections.



When prompted to "Enter distance of first snap line from the reference point [.5000]", enter in the offset distance from the snap line to the geometry it is attached to. White arrows will appear to indicate which direction is considered positive. When prompted to "Enter the number of snap lines to create [1]", enter in the number of snap lines to create. If more than one snap line is to be created, the prompt "Enter the distance between snap lines [0.3750]:" will appear in the message window. The distance between the snap lines should be entered at this time. Figure 4 shows one snap line created off of both the bottom and right sides of the part.



Once the snap lines are in place, dimensions, notes, geometric tolerances, symbols, view arrows and surface finishes can have their position defined relative to the window border or to edges of the model. To place an item on a snap line, move the item onto one or more snap lines. When an item is "snapped" to a snap line, it will display in purple, as seen in Figure 5.



The dimension displayed in Figure 5 has actually been attached to two snap lines. The text of the dimension is attached to the snap line below the drawing view, and the dimension's extension lines have been attached to the snap line to the left of the drawing view. To attach an item to two snap lines, move the item so that it is on top of both snap lines. When either one of these two snap lines is moved, the item will move along with it.

The display of snap lines can be turned off in the Environment dialog box by unselecting **Snap Lines**. Snap line snap can be turned off by unselecting **Snap to Snap Lines** in the Environment dialog box.

To modify the attachment of a snap line select the snap line to modify **Edit > Attachment**. The MOD SNAP ATT menu will then appear with several options:

Keep Position - Maintains the position of the snap line.

Change Orient - Reroutes the snap line, changing its orientation but assigning the offset of the existing reference to the new reference.

Offset View - Reroutes the snap line attachment to a view border.

Offset Object - Reroutes the snap line attachment to other geometry or another snap line.

Select options from the MOD SNAP ATT menu and then select the new reference for the snap line.

Snap lines can be moved in two ways:

Select the snap line that is to be moved and drag-and-drop the selected snap line to the desired location.

Select the snap line once, hold down the right mouse button, and select **Properties** to move the snap line a known distance relative to the entity the snap line is attached to.

Please note miscellaneous functionality regarding snap lines:

If a snap line is deleted, any items that were attached to the snap line will become free, and can be moved as if they were never attached to a snap line.

The end points of snap lines can be moved, however this is a cosmetic action. Entities of one view cannot be added to another view's snap line. In the case of broken views, an item of one segment of a view cannot be snapped to a snap line of another segment.
