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MicroStation XM Training Manual 2D Level 2

The first eight pages of Module 4 are shown below. The first two pages are typical for all Modules - they provide the Module title and set out the learning objectives. The suggested time for completion of the Module is given at the end of Page 4-2.

The instruction you see is typical of the combined information and exercise approach used throughout all Modules.

Please note the “Tool Tip” boxes on various pages; these are located throughout the Manual to emphasize a technique or to add specific points of information.

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MICROSTATION V8 XM

2D LEVEL 2

Module 4

ADVANCED DIMENSIONING

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MicroStation V8 XM

Module 4 of 17

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Module Information

Prerequisites:

Module 3 MicroStation - 2D Level 2

Introduction:

This Module covers advanced dimension options not covered in the Level 1 course.

Objective(s):

- 4.1 Insert or remove dimensions within a dimension string.
- 4.2 Associate or unassociate dimensions from elements.
- 4.3 Place dimensions in rotated views.
- 4.4 Understand and apply Dimension Styles.
- 4.5 Saving Dimension Styles.
- 4.6 Placing Geometric Tolerances.
- 4.7 Plotting drawings at multiple scales.
- 4.8 Scalable Place Notes
- 4.9 Dimension Audit tool.
- 4.10 Annotation Scale

Time:

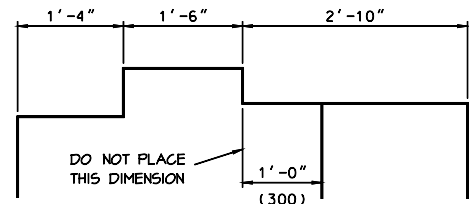
This Module should be completed within 2 hours.

DISCUSSION:

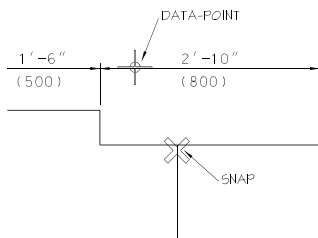
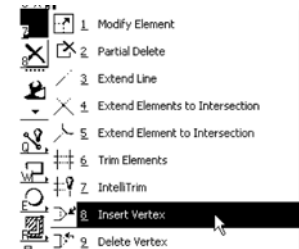
The first dimension option is the ability to *add* a dimension to an existing dimension. This is very useful if your design changes or you have missed dimensioning an element in a dimension string.


4.1 WORKING WITH INSERT VERTEX AND DELETE VERTEX

Draw the shape at the right and place only the *upper* dimensions as shown. Use the *Dimension Size with Arrow* tool to place the dimensions. The vertical lines can be of any size. Make sure that *Association Lock* is OFF in the tool settings window (you will use these dimensions again in the next Section where you will change the dimension associations).

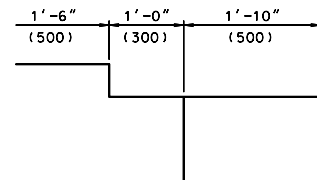


Assume that you missed dimensioning the interior vertical line and would now like to dimension it as part of the existing *upper* dimension string. Instead of deleting the string and starting again, use the *Insert Vertex* tool on the *Modify* tool box.




- Step 1** Start the *Insert Vertex* tool. 
- Step 2** *Data-point* on the *dimension string*.
- Step 3** *Snap* to the *endpoint* of the new line element.

A new extension line is inserted in the dimension string and the dimension line adjusts to accommodate the added dimension.



Try additional examples of this on your own.

It is equally easy to *remove* dimensions from a dimension string. For this operation you use the *Delete Vertex* tool. Using the same drawing:


- Step 1** Start the *Delete Vertex* tool. 
- Step 2** *Data-point* on the *extension line* of a dimension that you want to remove.
- Step 3** *Data-point* to *accept* the dimension removal (a *reset* will cancel the deletion).

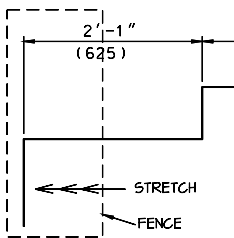
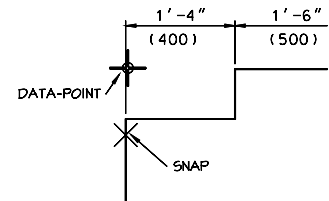
The extension line and dimension text is removed and MicroStation automatically adjusts the existing dimension line.

4.2 DIMENSION ASSOCIATIONS

Normally you should place dimensions with the *Association Lock* turned ON. This ensures that dimensions are “attached” to the elements they are dimensioning. If you forgot to turn Association Lock on when you placed dimensions in a design, you can still associate them to their elements. The tool to use is *Reassociate Dimensions* on the *Dimensioning* tool box.



The dimensions you placed in the previous Section should have been placed with Association Lock off. If not, delete and replace the dimension string for this exercise, making sure that Association Lock is OFF.

- Step 1** Start the *Reassociate Dimensions* tool. 
- Step 2** *Data-point* on the dimension to be associated.
- Step 3** *Data-point* to accept the association.



The end dimension is now associated with the corner of the element. To prove this, do a *stretch* operation on the corner. The first dimension should change in response to the stretched elements.

Dimensions can also be *unassociated* from their elements. There are two tools you can use for this operation.

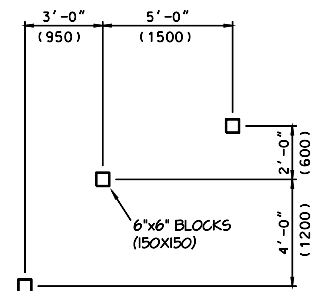
1. On the *Drop* tool box (on the *Tools/Drop* menu) is the *Drop Association* tool. 
2. On the *Groups* tool box (Main tool box) is the *Drop Element* tool. 

Each tool works slightly differently. The *Drop Association* tool simply drops the association and does nothing to the dimension itself. On the other hand, the *Drop Element* tool (with *Dimensions* selected) not only drops the association but also *breaks up* the dimension into *individual* elements.

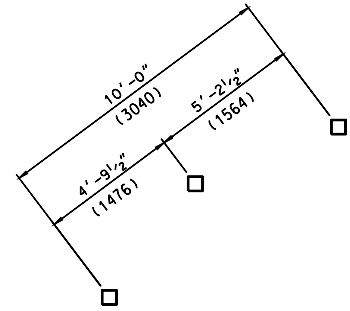
Try each tool for yourself. Simply *data-point* on the dimension and accept the drop operation with another *data point*.

4.3 DIMENSIONS AND ROTATED VIEWS


It is sometime difficult to correctly dimension some elements because of their relative positions. A good example is shown at the right where horizontal and vertical dimensions are simple to add to the three blocks, but a *True* aligned dimension, as shown below, is not. Draw the three blocks yourself, but don't add the dimensions shown at the right.



Now try to dimension the blocks as shown on the right. You will find the dimensions much more difficult to place. Use the *center* snap to locate the dimensions at the center of each block. You will be tempted to use the *True* dimension alignment option, but you will find this ploy doesn't work because of the middle block's offset.



One way to place these dimensions is to *rotate the view* so that the *two outer blocks* are in a *horizontal alignment*. Do the following:

- Step 1** Click on *Rotate View* in the *View Controls* bar (bottom-left of the view window). 
- Step 2** Select *2 Points* as the rotation method in the *Tool Settings* window.

You will now data-point on two points that will define the X-axis of the rotated view.

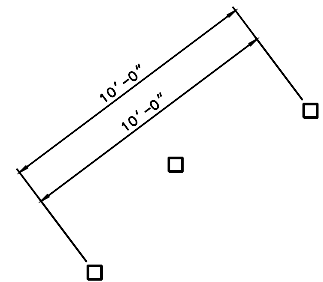
- Step 3** *Snap* to the center of the *lower-left* block.
- Step 4** *Snap* to the center of the *upper-right* block

The view rotates so that the two outer blocks are in horizontal alignment. It is now easy to place the two dimension strings with the standard *View* dimensional alignment. To *return* the view to its *normal* rotation:

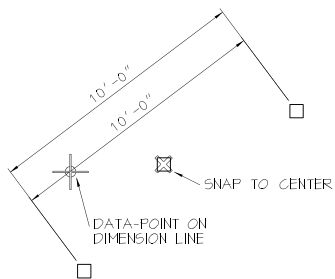
- Step 1** Click on *Rotate View* in the *View Controls* bar (bottom-left of the view window).
- Step 2** Select *Unrotated* as the rotation method in the *Tool Settings* window.

As soon as you select the *Unrotated* option, the view returns to its normal orientation. This is a very useful feature and can be used for all types of drawing operations.

The alternate way to place the same dimensions *without rotating the view*, is to use the *Insert Vertex* tool you used in Section 4.1 above. Do the following:

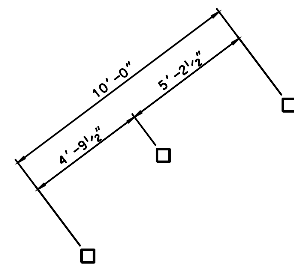


- Step 1** *Undo* the dimension placements from the last exercise. The *view* should be *unrotated*.
- Step 2** *Place* an overall dimension on the two outer blocks using the *True* alignment.



- Step 3** *Repeat* this process and place a second overall dimension as shown at the left.
- Step 4** *Start* the *Insert Vertex* tool.
- Step 5** *Data-point* on the *dimension line* of the lower dimension.
- Step 6** *Snap* to the *center* of the *middle* block.


The *Insert Vertex* tool inserts a new extension line in the lower overall dimension and breaks the dimension into two parts, exactly as needed.



4.5 DIMENSION STYLES

Clearly, it would be very useful to be able to save the attributes of a particular dimension style so that it can be used again both in the *current* design and in *other* design files. As you will see, it is very easy to save a dimension style and to import that style into another design file.

To save your current *active* dimension settings:

- Step 1** *Open the Dimension Settings box with Element/Dimension Styles.*
- Step 2** *Click on the Create Dimension Style icon at the top of the box.* 
- Step 3** *Enter a Name and Description in the Create New Style dialog box.*

The new style name now appears in the style display of the Dimension Styles box. You can make as many styles as you need and you can quickly switch between styles through this box.

If you want to switch styles while *placing dimensions*, you can do so in the *Tool Settings* window when a dimension tool is active. Remember too, that clicking on the *magnifying glass* icon in the Tool Settings window will open the Dimension Styles dialog box.

You can change the dimension style of *all* elements with a particular style to that of another style. To do so, *right-click* on the style you want to change in the *Dimension Settings* box and select the *Remap Elements* item. In the dialog box that opens, select the style you want the elements changed to.

TOOL TIP !

You can change the dimension style of all dimensions with a particular style to that of another style. To do so, right-click on the style you want to change in the *Dimension Styles* settings box and select the *Remap Elements* item. In the dialog box that opens, select the style you want the elements changed to.

IMPORTING DIMENSION STYLES

You can *import* dimension *styles* from other *design files*. This avoids the tedium of creating dimension styles for each new design file.

You can use an existing design file that contains the styles you need, or you can *set up* a design file *specifically* for the purpose of storing various dimension styles for the type of work you do.

The import process is very easy:

- Step 1** *In the Dimension Styles dialog box, select Style/Import.*
- Step 2** *Find and select an existing design file containing the styles you need.*

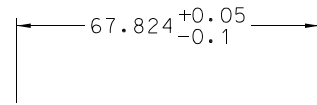
The dimension styles contained in the file are loaded into your design file, and you can select and use the loaded styles.

4.6 GEOMETRIC TOLERANCING

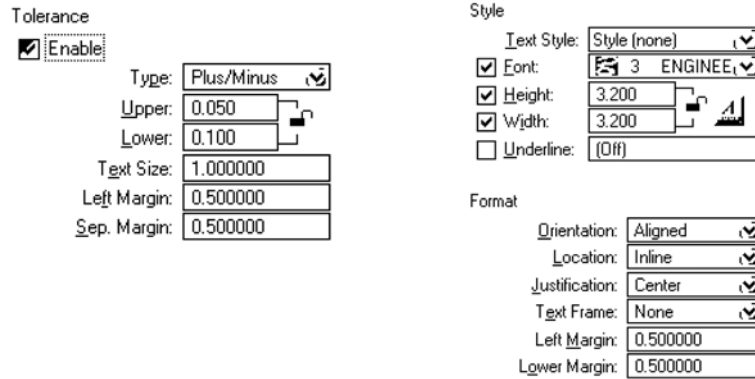
Engineering drawings typically require tolerance dimensions and MicroStation provides two methods for adding geometric tolerancing.

DIMENSION SETTINGS

The tolerance dimension shown at the right was created using a combination of settings in the *Geometry/Tolerance* and *Text/Style* and *Format* tabs. Both are reproduced below.

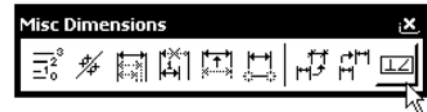


In this case the in-line dimension text is set in the Text settings box, and the tolerance values are set in the Tolerance settings box.

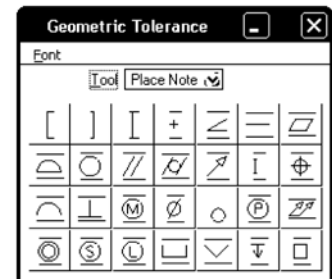


GEOMETRIC TOLERANCE TOOL

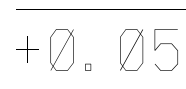
The second method is to build a *Feature Control Frame* in which to place the tolerance information. The *Geometric Tolerance* tool, located on the *Tools/Dimension Tools/Misc Dimensions* tool box provides a selection of standard tolerancing symbols.



On starting this tool, both the *Geometric Tolerance* symbols box and the *Text Editor* boxes will open. The Geometric Tolerance box offers the option of placing a leader and symbol with the Place Note tool, or in plain text with the *Text* tool. If you don't see the symbols shown at the right, click on the *Font* menu and select font *101 Feature Control Symbols*.



Selecting one of the tolerance symbols places that symbol in the *Text Editor* box. You can now *add the tolerance value after the symbol*. Note that the symbol will likely display as a single letter character, not the actual symbol. The example at the right is using the fourth item on the upper row with a value of 0.05.



Additional symbols or text can be added to the text string and, if necessary, you can add vertical lines to close the box.