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PTC Product Focus

Model Based Definition (MBD) with Wildfire 4.0

Model Based Definition (MBD) has come of age with Wildfire 4.0. Although many of the features and functionality have been available in past releases, the streamlining, flexibility and robustness of many actions now makes it very fast and efficient to place 3D annotations on models. Here, after a brief introduction and background of MBD, we will discuss some of the mechanics of how to create annotations in compliance with the ASME 14.41 standard. Additionally, examples of the new functionality available in Pro/Engineer Wildfire 4.0 will be given.

Model Based Definition, or MBD as it is called, was developed out of an industry desire to compress the product development cycle. In order to reduce time to market and lower the bid model, it was noted that as much as 50% of a design engineer's time is spent creating 2D drawings after the part is completely defined as a 3D model.

Background and ASME Y14.41

Numerous industry players and the government met in 1997 at the Boeing company in Wichita Kansas and then at an ASME meeting to establish a standard for a digital product definition. The crux of the idea was to extend the usage of digital models in the reduction or elimination of 2D drawings. The Standard, ASME Y14.41, Digital Product Definition Data Practices, was approved as an American National Standard on July 7, 2003.

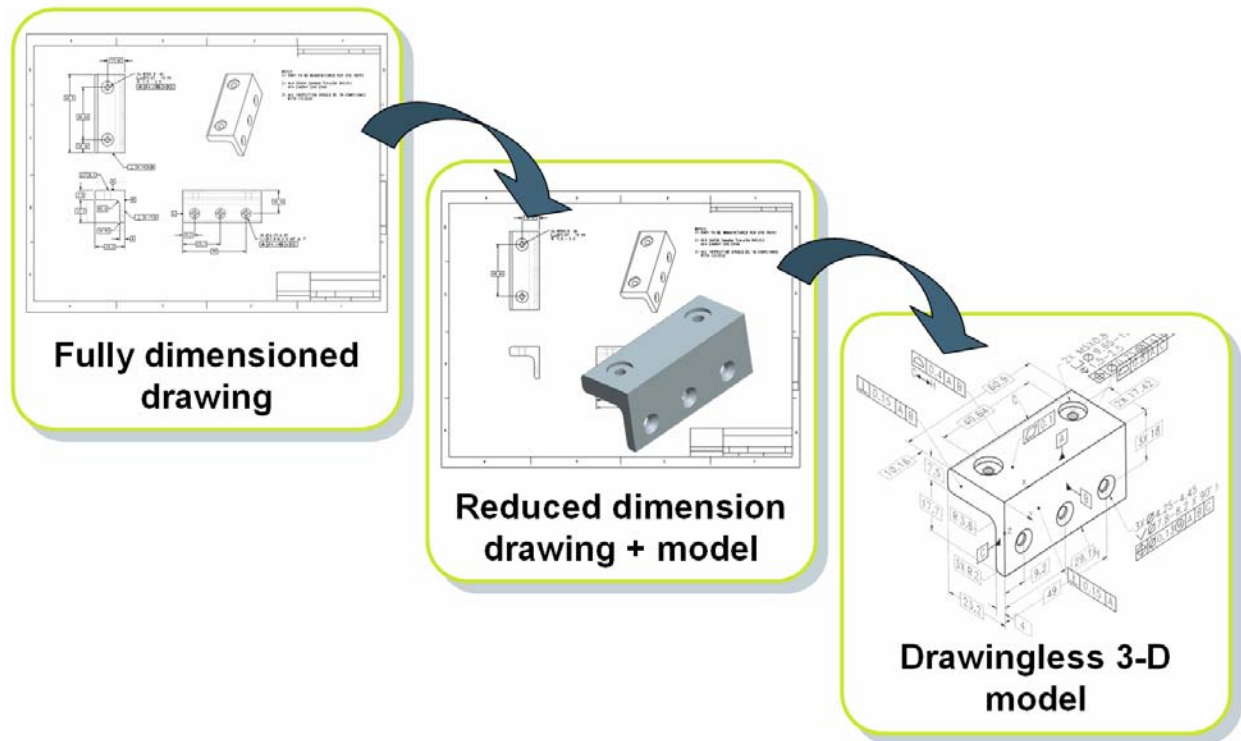
There are also emerging standards from ISO, the International Standards Organization and JAMA, the Japan Automobile Manufacturers Association. These are very closely aligned with the ASME standard.

Model based definition is more than replacing 2-D drawings with 3-D counterparts, but is rather changing the paradigm of how product information is captured and communicated. The record of authority shifts from the drawing to the model. Some companies have extended the thought

process to encompass a complete model based enterprise with electronic communication though all aspects of the product design and life cycle.

Conventional 2D drawings

MBD is a journey which requires significant change for companies. The paradigm change to a drawingless environment is as difficult as it was to move from the drafting board to electronic 2D CAD and then to 3D modeling. Many are trying to move in half steps such as delivering to manufacturing reduced dimension drawings plus a neutral model.

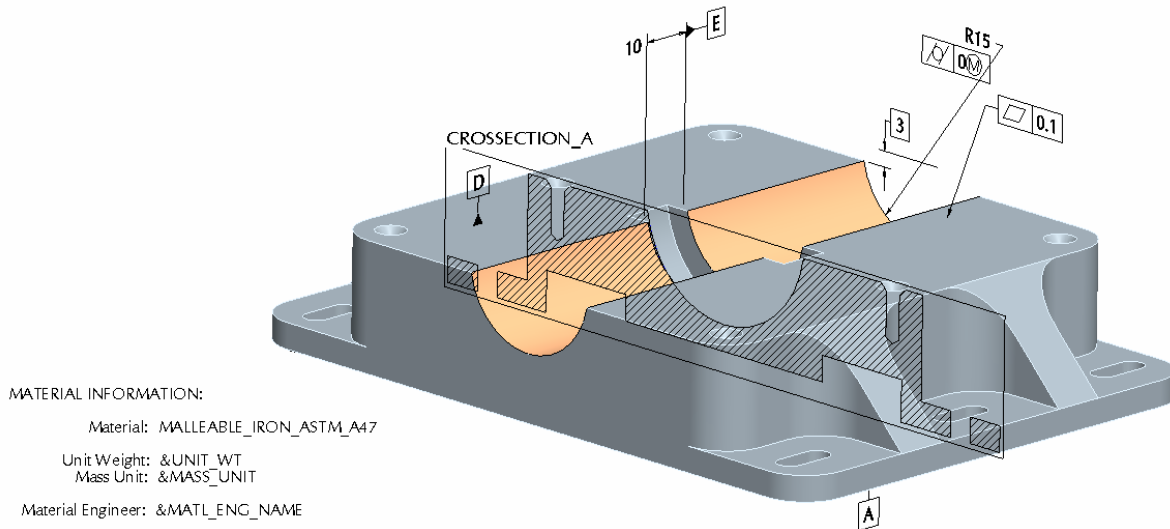


The desire is to create dimensions and annotations in the model that can be used as a stand alone 3D representation of the geometry. In the Pro/Engineer Wildfire world, annotations created in the 3D model can stand alone to fully define the component and, if needed, with very little effort, be transformed into a reduced dimension drawing.

Sample Digital Model

Below is a sample model with dimensions and annotations placed relative to the geometry per the ASME 14.41 standard. We will show how to setup the active annotation orientation which defines the placement of the text relative to the model. Next, dimensions can be created using one of 3 options. First, dimensions can be created directly from the features in Pro/E that are used to define the underlying sketch of the feature – these are known as driving dimensions. Secondly, a dimension can be created by selecting items directly on the geometry, the dimensions will update as the geometry changes and thus these are known as driven dimensions. Thirdly one can create an ordinate dimensioning scheme from a baseline dimension and numerous driven ordinate dimensions. All of these dimensioning schemes are completely associative with the geometry. Various other annotations such as notes, symbols, surface finishes, and set datum's, can also be created.

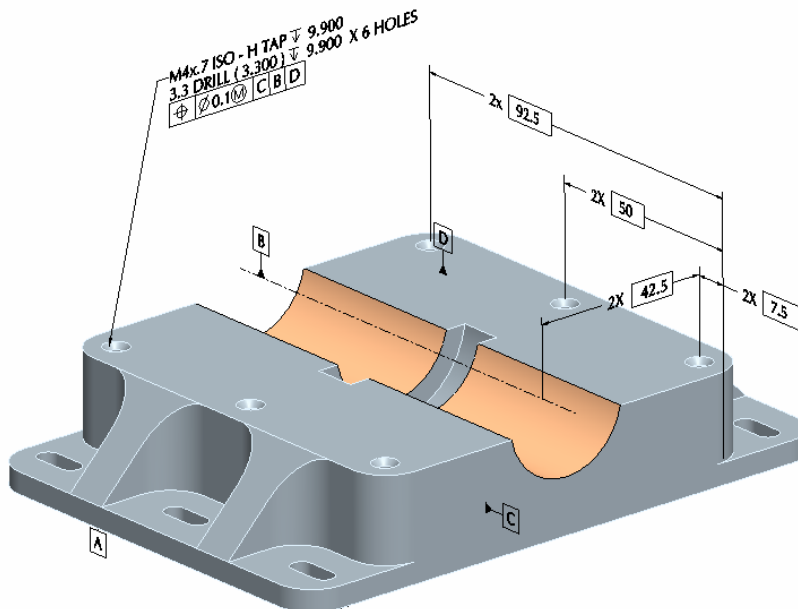
ACME Corporation Proprietary Information



This view of the model uses a flat to screen note with a larger red font to create a proprietary notice or banner. The material information can also be placed flat to screen in a fixed position as shown in the lower left corner. This note references part parameters whose value would appear in place of the parameter names that are preceded by the ampersand. Set datum's, GTOLs, and cross sections are all shown oriented along specific planes.

By using the multiple view states and sorting appropriate annotations, the view manager "All" state can be setup in conjunction with layer display controls to show only the specific information required as in the sample below.

ACME Corporation Proprietary Information

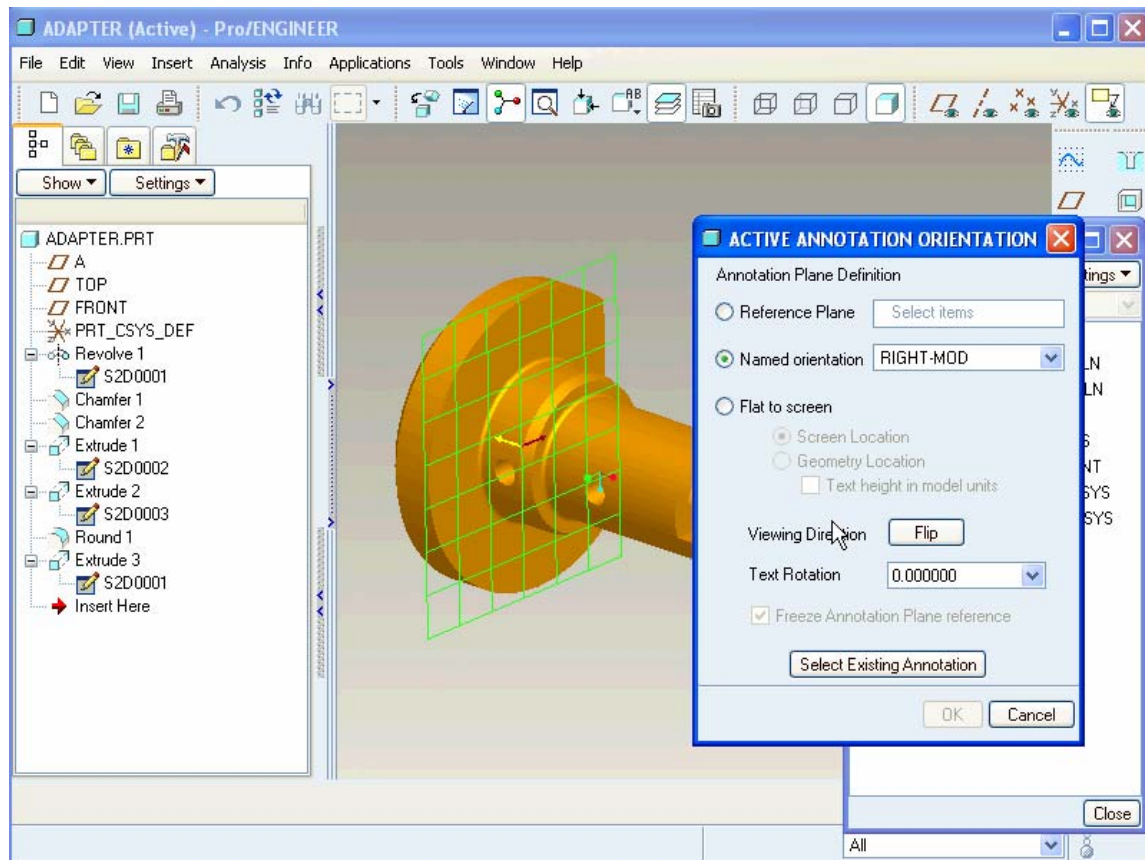


Next we will step through the process and techniques used to create annotations for an MBD machined part.

Definition of the ACTIVE annotation orientation

The key to creating annotations, datum or notes is the orientation and position to either the screen or the model. In Wildfire 4.0 the active annotation orientation is set via a dialog box (shown below >View > Annotation Orientation) which allows the user to define and place annotations by:

- Plane or flat surface
- Named view
- Non rotating plane (either screen based or attached to model)

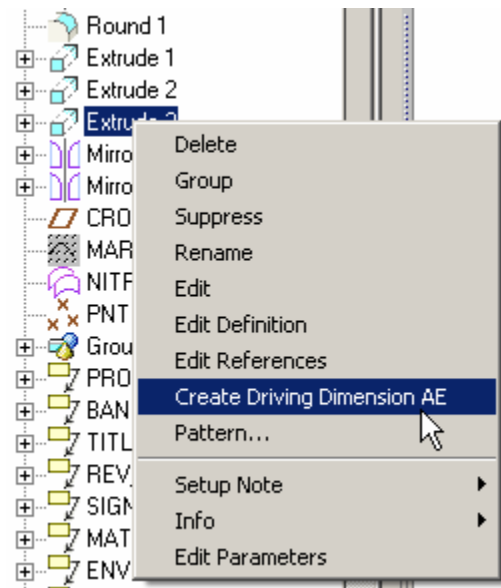


This is a “sticky” dialog – the setting will remain active for subsequent annotation placement in that session or until the user selects a new orientation.

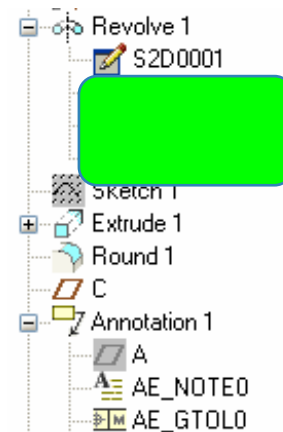
A key new enhancement in Wildfire 4.0 is the option Freeze Annotation Plane reference. If for some reason the reference item used to define the Annotation Orientation is deleted, suppressed or removed the annotation will not fail - but will remain active.

Driving dimension annotations:

The fastest and easiest method to create annotated dimensions is to select a feature either on the screen or from the model tree, press the right mouse button (RMB), and select Create Driving Dimension AE from the pop-up list of options.



All the dimensions for the feature that can be displayed in the active annotation orientation will be displayed on the screen. If desired, these can be easily repositioned to another location. Driving dimension features will be displayed in the model tree and visible when the feature is expanded.



For each sketch dimension, a driving dimension will be created, except for dimensions that can only be shown in a plane orthogonal to the active annotation orientation. To show those dimensions simply select the appropriate active annotation orientation, select the feature again, and RMB to create Driving Dimension AE.

Each driving dim within this feature can be individually manipulated. Users can once again select the annotation, either from the screen or the model tree, then from the screen slide the drag handles to modify the witness line endpoints. Other changes may be made by selecting the properties option in the RMB pop-up menu. See the tips section at the end of this article. These dimensions can be grouped using layers, one for each feature and its corresponding group of dimensions. This will allow for inclusion and display or hide later in an “All” state view to be published as an Annotation Set in ProductView.

Driven Dimensions:


Driven dimensions offer the most flexible method for the creation and display of a dimension. If feature based driving dimensions do not appear in the ideal location for display, then use driven

dimensions. As a best practice, all annotation items except driving dimensions (described above), should be made as an Annotation Elements (AE) of an Annotation Feature (AF). These annotation features behave like any other feature in Pro/Engineer and can be suppressed, hidden, grouped, etc. Driven dimensions are automatically grouped since they are created as elements of an annotation feature.

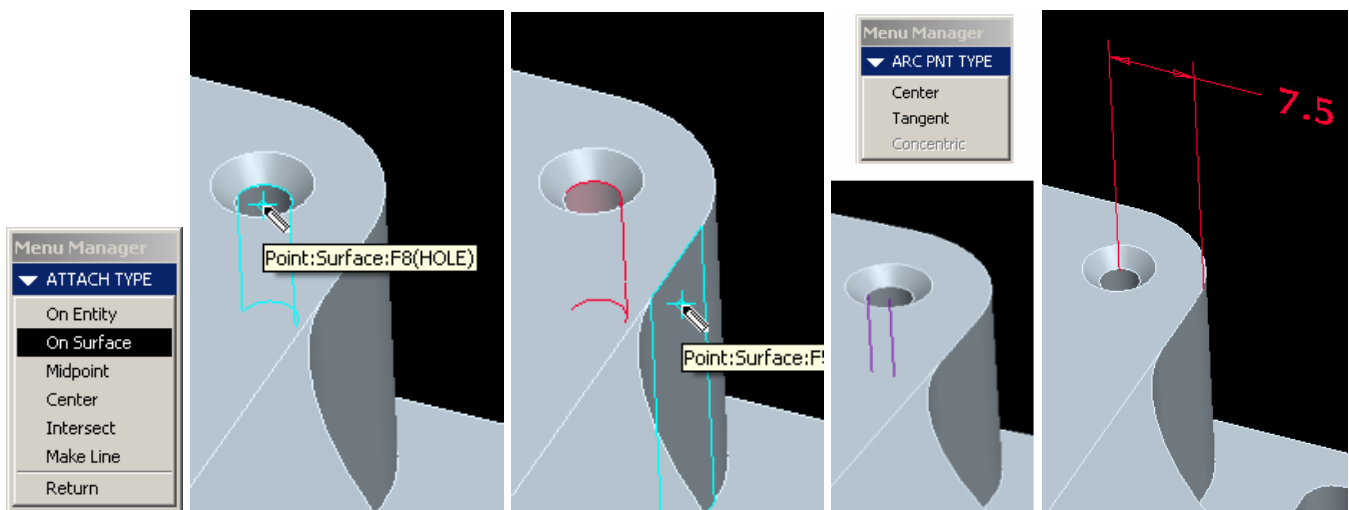
For fully annotated models it is a best practice to create one annotation feature with multiple dimensions for a single geometric feature on the model. If there is a simple additional feature, such as a round or chamfer, or a logical group of dimensions, they should all be created in the same annotation feature.

For limited dimensioned models only the non-standard dimensions need to be annotated. These are dimensions such as those with Geometric Dimensioning and Tolerancing (GD&T) applied or ones that do not lie within the block tolerances.

It takes just a few steps to create a driven dimension.

With the active annotation orientation set, create an annotation feature (pick the  icon) and then select the option for driven dimension.. For the end of the first witness line pick the attach point type; note that the first point selected will define the “z offset” from the active annotation orientation. Pick the next type and location for the second witness line; use MMB to place the location of the dimension text.

For example, using the On Surface reference for the Attach Type, pick cylinder surface of hole, pick the outside surface, select Center, and then MMB to place dimension text.




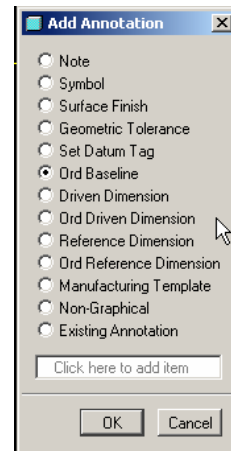
Other selection options are noted in the tips section at the end of this article.

Ordinate dimensions:

Ordinate dimensioning may only be required for fully annotated models where this is the preferred method of displaying the dimensioning scheme.

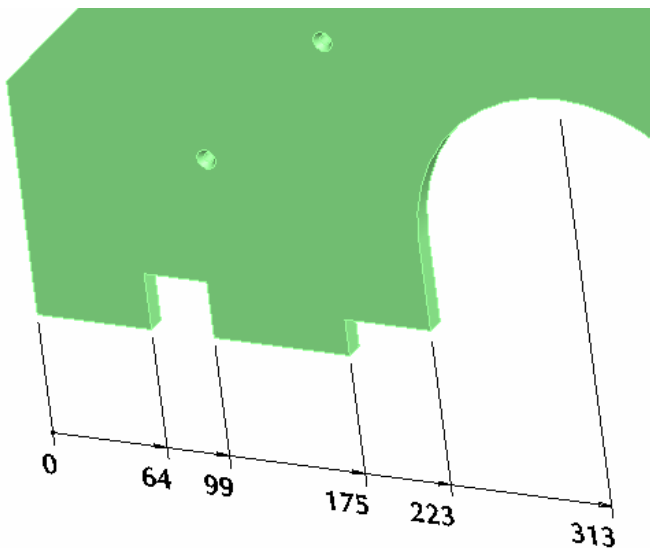
The process is also quite simple:

- Create an Annotation Feature 
- Create an ordinate baseline AE
- Create driven and reference ordinate dimensions
 - Wildfire 4.0 allows for surface references



Once completed Wildfire 4.0 offers many tools to:

- Define the style of ordinate dimensions, arrows and text
- Edit the attachment and/or move the ordinate dimension
- Modify the witness lines, insert jogs and breaks

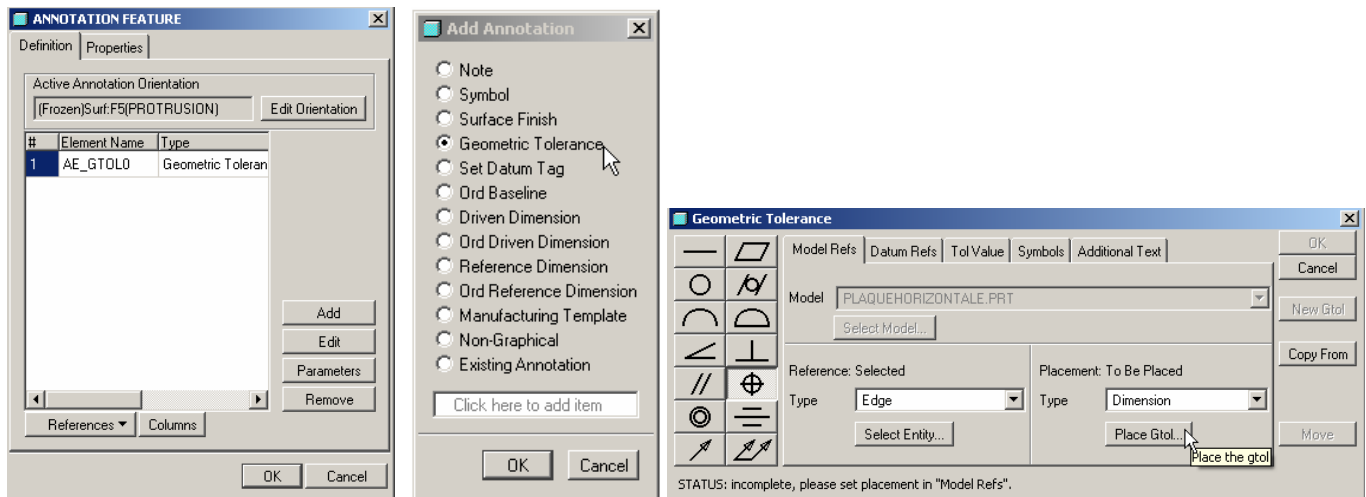


For all driven dimensions: regular, reference or ordinate style, users can once again select the annotation either from the screen or the model tree, then from the screen slide the drag handles to modify the witness line endpoints. Other changes may be made by selecting the properties option in the RMB pop-up menu. See the tips section at the end of this article for other options.

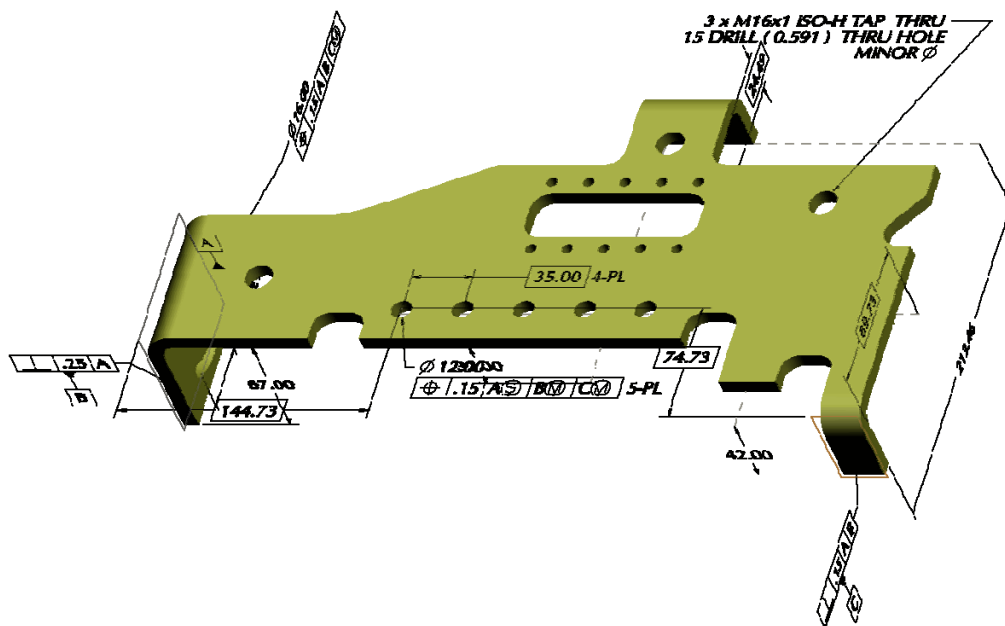
Also remember that the annotation orientation remains active, thus one can create numerous dimensions one after another that are to be displayed parallel to the same plane.

Geometric Tolerances

Geometric Tolerances, GD&T or GTOLS, can be added to any annotation dimension or note. For driven dimensions an annotation feature should be created, then as with all other dimensions, one can simply create a Geometric Tolerance annotation element and attached it to the dimension.



A complete set of associated datum references, tolerance values, symbols and additional text can be created and displayed with the appropriate standard frames.

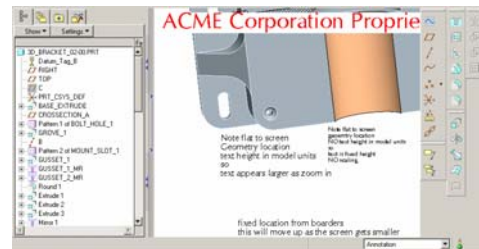
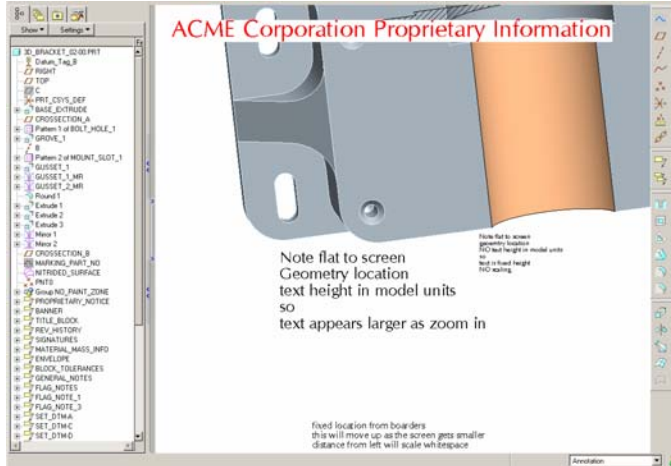


Notes in 3D

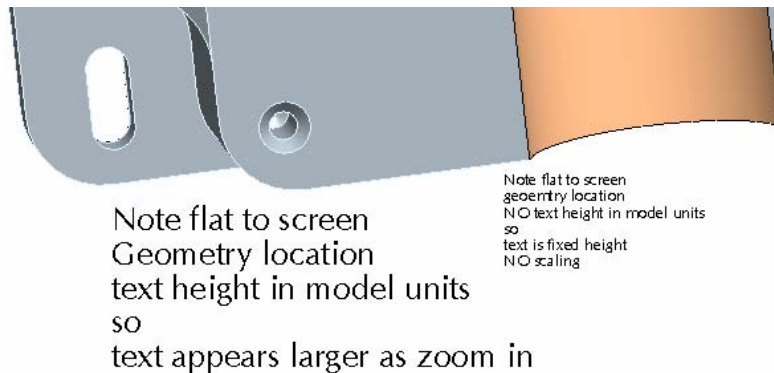
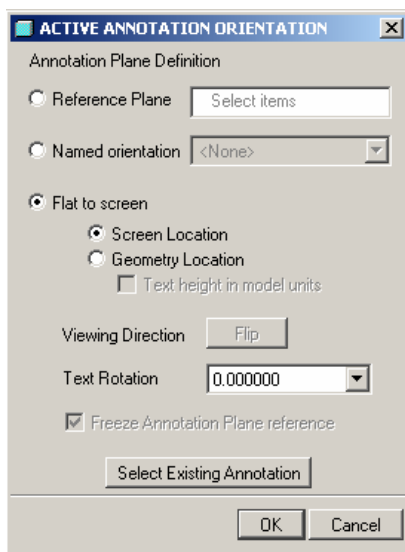
Notes are one of the most versatile annotation elements that may be created. They can contain imbedded parameters (use ¶m_name), GTOL values (use &G#), hyperlinks or URLs to web pages, along with text and symbols.

New in Wildfire 4.0 is the ability to create notes flat-to-screen. The first option is to fix the text height and location on the screen in addition to its color and font and other attributes. The text is

set a specific distance from the borders. Thus as the screen is resized they will always be visible unless the screen is smaller than the text string at its fixed font.

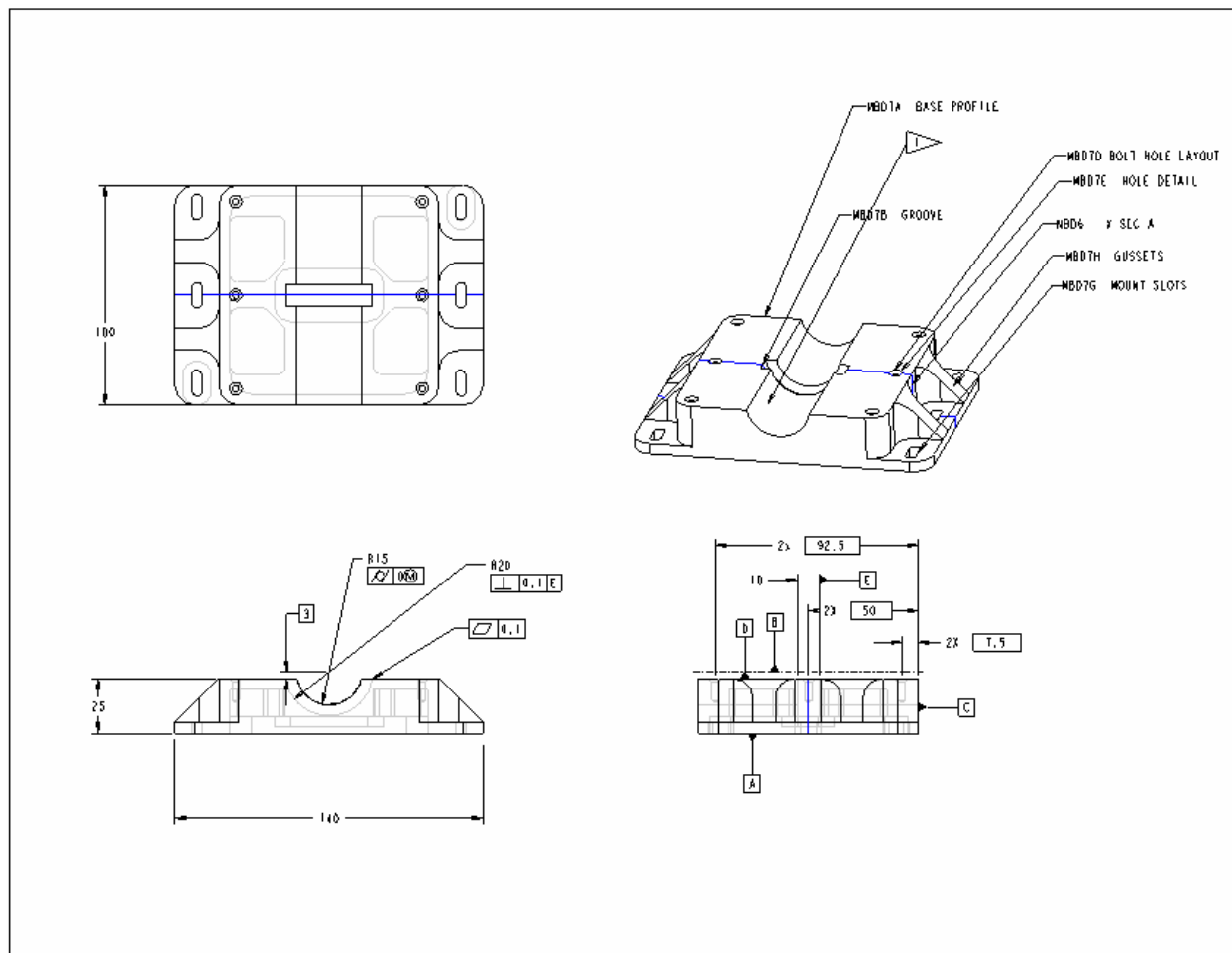


Another option exists for the creation of flat-to-screen notes that are attached to the geometry. These notes can be attached to an edge, a datum point, a coordinate system, a curve, a point on a surface, or a vertex. If the option for “Text height in model units” is selected then the text will appear larger as one zooms in on the geometry.



Combination states and layer status:

The final output for MBD is a list of annotation sets displayed in an easy to use lightweight viewer such as ProductView. An annotation set has a unique view of the model showing the dimensions of a specific feature, an information note, a symbol, or one of the many items typically found on a conventional drawing now displayed in a 3D format. Yet in order to reduce clutter on the single model (also known as the “fur ball effect”) it is best to create a layer for each annotation, a corresponding saved view orientation, a cross section if applicable, and finally a combination or “all” state in the view manager.



This drawing template could also have a complete parameter driven title block to display the required information.

More Information on Model Based Definition

The organization aspects of an MBD machined model have not been covered here. The naming and contents of layers use for display of annotation dimensions, notes, symbols, etc. along with orientation names, and “All” view state names used ultimately to create Annotation Sets in a lightweight viewer such as Productview will be the topic for a follow-on document

PTC in conjunction with industry leaders has developed a set of best practices and standards for the implementation of a model based environment. To learn more details on the process of creating MBD models, organizing the data, and automating the process with PDMLink please contact your PTC representative.

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PTC Product Focus

Windchill 9.0 Role Based UI Simplification – “Profiles”

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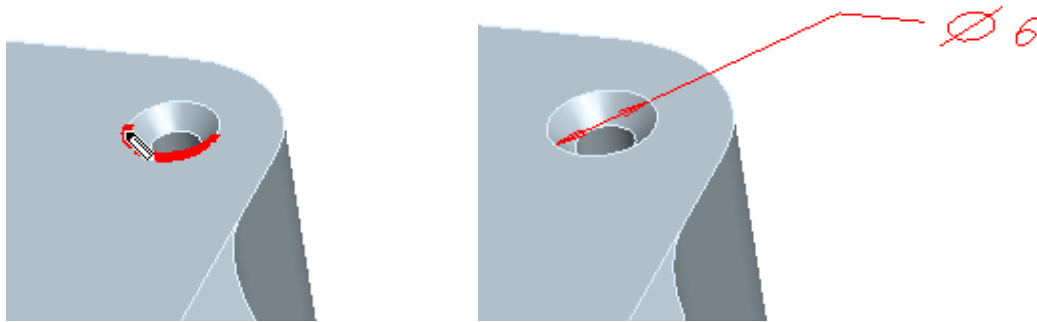
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Tips of the Month

Options for Manipulating Dimension Witness Lines in MBD Models


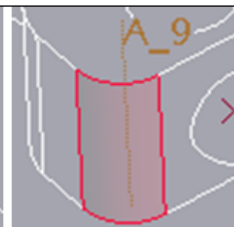
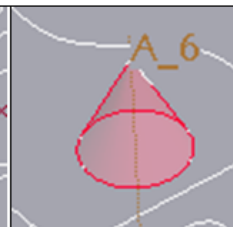

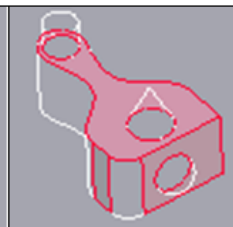

Creating diametral dimensions

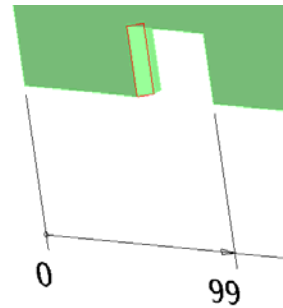
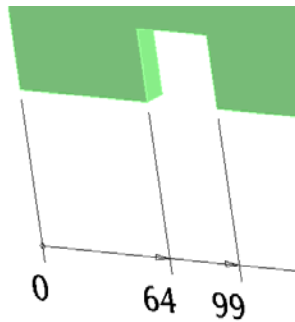
To create a diametral driven dimension which shows the diameter of a cylinder through its axis (must use an annotation orientation normal to the cylinder axis) start by adding a driven dimension annotation element from within an annotation feature. Select “On Entity” as the attach type and select the edge of the cylinder twice with the LMB. Then place the dimension with the MMB.



Use Surface References when creating annotations

Wildfire 4.0 allows the user to select surface references when creating driven dimensions. This is a best practice as surface references are inherently more stable than edge references. Edges tend to disappear or change when rounds and chamfers are added.

					
Cylindrical Surfaces (VALID)	Cylindrical Surfaces (VALID)	Conic Surfaces (VALID)	Spherical Surfaces (VALID)	Planar Surfaces (VALID)	All other Surfaces (INVALID)

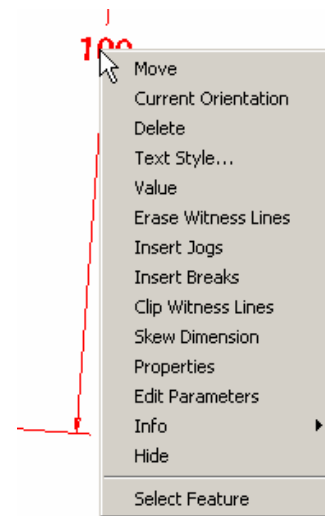


Here a chamfer added after the annotation element will remove its edge reference and the dimension.

Moving and editing dimensions

Most edits to annotations can be made via the Right Mouse Button (RMB). First, use the smart filter in the lower right border of the screen to select annotations (one can create and use the mapkey aa, as it automatically resets to smart). Then place the mouse pointer over the text to get pre-select highlighting (cyan color), use the Left Mouse Button (LMB) to select. (red color).

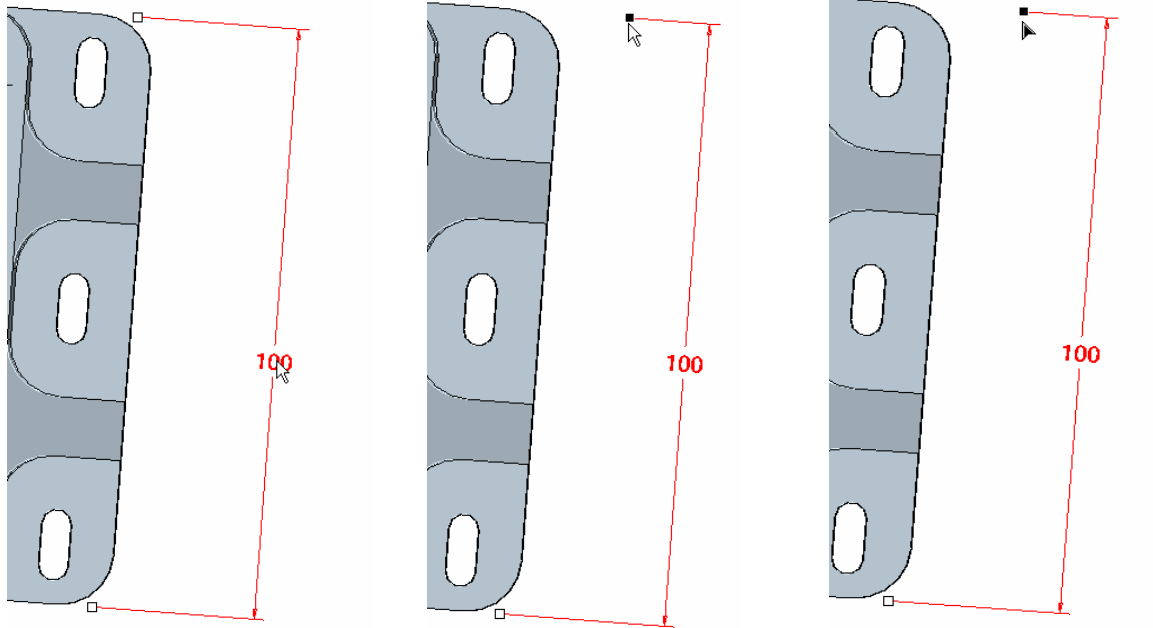
Once selected press the RMB to reveal many options:



This is the fastest way to move the text location. Many other options are explained below.

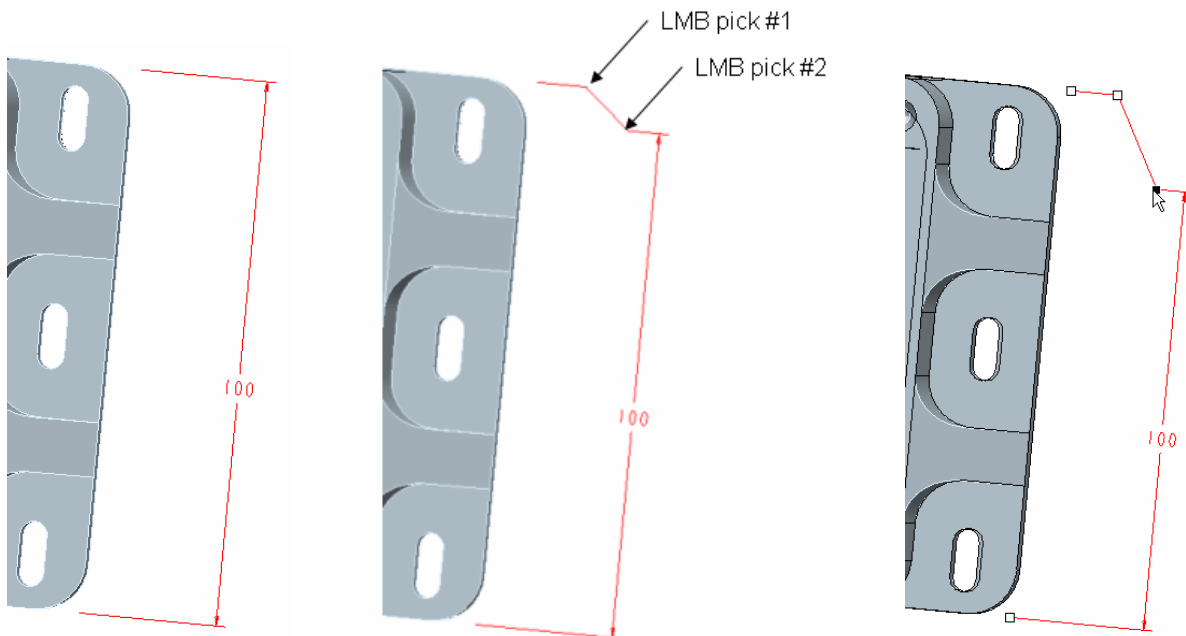
Moving Witness lines

Place the mouse pointer over the text to get pre-select highlighting (cyan color), LMB to select. This will bring up the witness line drag handles (white square boxes). Place the cursor over the white square box until it turns black, then press the LMB and drag to the endpoint to the desired location.



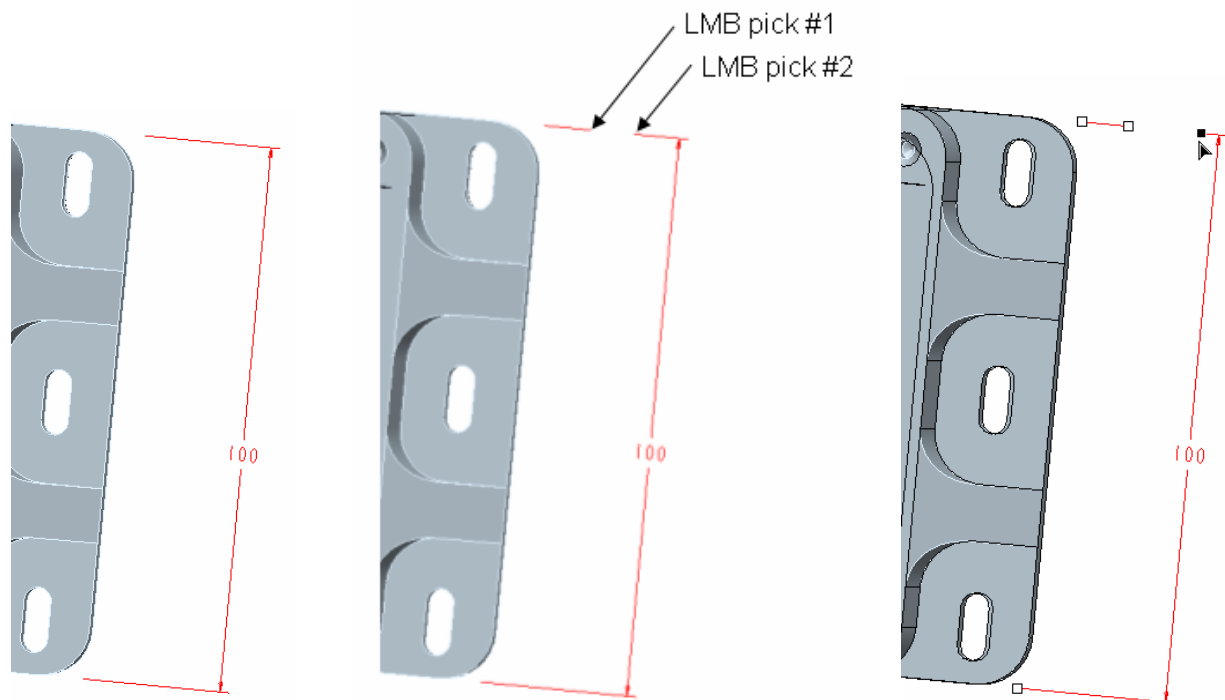
Jogs in witness lines

Select the annotation dimension, RMB & pick “Insert Jog”, Pick the 1st and 2nd jog elbow locations along the witness line. Once created these jog elbow locations will highlight with drag handles for additional manipulation



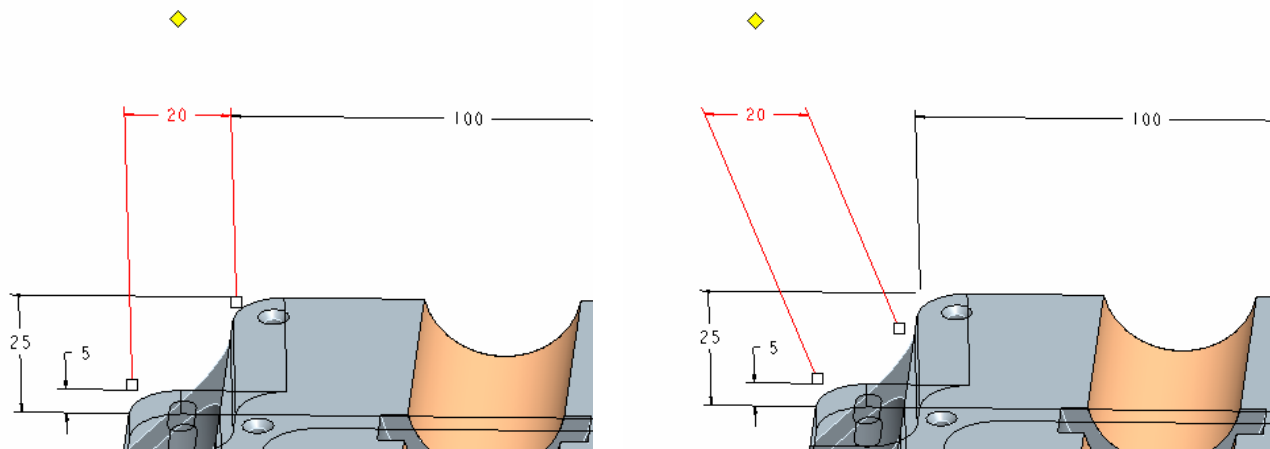
Breaks in witness lines

Select the annotation dimension, RMB & pick “Insert Break”, Pick the 1st and 2nd break points along the witness line. Once created these break point locations will highlight with drag handles for additional manipulation



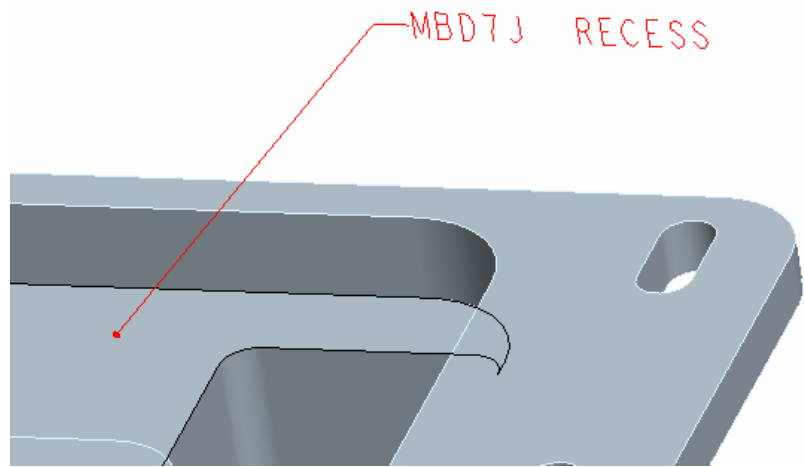
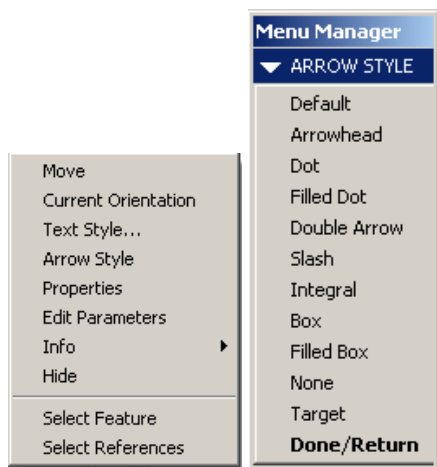
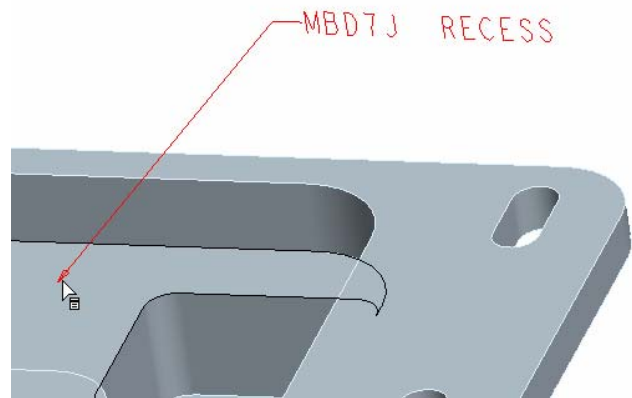
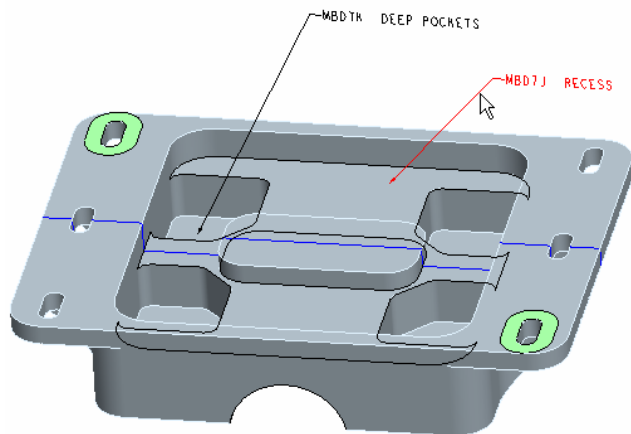
Skew Dimension

Select the annotation dimension, RMB & pick “Skew Dimension”, Pick the yellow drag handle and move. To restore to original location drag handle back to original position.



Arrow head options

Select the annotation dimension, once highlighted in red, move the cursor towards the arrow head until the cursor changes to show an arrow plus a small options list. RMB & pick the desired option from the list.



For more information, tips and recommended best practices are available for creating, organizing and publishing of annotations for use in Model base definition please contact your local PTC representative.

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Tips of the Month

Windchill – Team Management Tips

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Announcements

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provide new useful features not feasible with paper media. e-Profiles will provide you with 24x7, worldwide access to key information previously available exclusively in the print version. "Tips & Tricks," a popular feature pioneered by Pro/USER, has also moved to the web and will be expanded as the site matures.

Please take a few minutes to check out this new web site. We don't think you will be disappointed.

<http://profilesmagazine.com/>

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Upcoming Events & Training Class Schedules

Upcoming, 2008 Your local Pro/Engineer User Groups
<http://www.ptcuser.org/rugs/>

June 1 – 4, 2008 Long Beach, CA USA
PTC/USER World Event
<http://www.ptcuser.org/>

Events

Our seminars and conferences seek to provide you with relevant information regarding product development trends in your industry as well as innovative software learning experiences. Think of them as a constructive day off where you can share experiences and swap ideas with your peers.

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Sessions are performed just like a traditional ILT (including interactive exercises where you and the instructor can work on lab exercises together) and feature some of our most popular ILT courses. These sessions cover the exact same material as the traditional ILT in-center courses. Also look for some of our most frequently requested mini-topics delivered in the same format that are only an hour - two hours in duration.

If you have any questions about these sessions or would like to see getting other courses, not on this list, on the schedule please feel free to contact me for more details. They are a great way to bring training to you without you having to worry about location or being out from work for long stretches.

You can register for these sessions just as you would for any normal ILT class either by:

1. calling order admin at <http://www.ptc.com/services/edserv/training/registra.htm> or
2. you can go to PTC University directly at <http://www.ptc.com/learning> and submit a registration request directly. All you have to do is search the catalog by typing in “virtual” in the search field and you will see a listing.

PTC

Note: This PTC E-Newsletter will continue to be used for the following:

- 1) Inform you on events related to PTC products (user groups, conferences, training schedules, etc.)
- 2) Educate you on solutions that are available at PTC
- 3) Tips & Techniques using PTC Products

Note: These messages are compiled in the local PTC office and will be distributed via e-mail.

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