MBD

Model Based Definition

Organizational Schema Standards for Pro/Engineer

Prepared by: John Hentges

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Change History

REV	DATE	DESCRIPTION OF CHANGE
A	10/14/09	Initial Release
B	11/10/09	Incorporated User Feedback, Define D6 as assembly views, D8 as Weld views, Added list of recommended parameters, Added to the glossary

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1. Introduction

This is the Organizational Schema Standard for Model Base Definition (MBD). This schema document was developed to provide standards and guidelines for the CAD user. It is intended to be the foundation for design development efforts.

Previously, 3D CAD models had an accompanying 2D drawing. Now, with advances in CAD architecture, all product definition previously shown on a drawing is now defined and displayed directly in the 3D CAD model (See Figure 1). This is accomplished by utilizing a combination of annotations and naming conventions to organize the 3D solid model. This makes the 3D model a single master source for obtaining product definition data and eliminates the need for a 2D drawing.

Figure 1: The Documentation Journey

2. Purpose

This document defines requirements for the application and display management of the product definition data. This standard is necessary to establish a common method to facilitate access to the digital product definition data by downstream users.

The Schema document is designed to be compliant with ASME Y14.41-2003 (See Figure 2, Digital Product Definition Data Practices). It is defined to work with Pro/ENGINEER®, Product View and Adobe Reader.

The goal of this schema is to define a common practice to improve design productivity and to deliver consistent data to the customer.

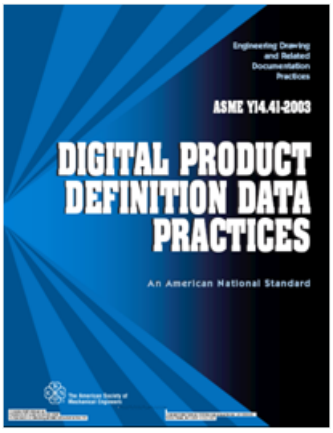

Figure 2: ASME Y14.41-2003

3. Schema Architecture Overview

Pro/Engineer has several tools for organizing product data. This schema uses the organizational elements listed below to format the product data for ease of viewing by downstream users.

Utilizing these tools in a consistent manner provides the foundation for consistent data sharing by downstream users. Defining data in a consistent manner is necessary for successful extraction to light weight viewers used for viewing shared data.

- **Combination Views** – are used to combine multiple **View Manager** display states and layer display per the use of the “all-states” tab on the Pro/E View Manager dialog box. Multiple combination views may be saved to quickly navigate through all the required product definition data.
- **View States** - are used to manage the orientation position of the product that best presents the product definition details. View States include the following display options: Orientation; X-Sections; Style; Simplified Reps and Explode States.
- **Layers** - are used to manage the display of annotation data and support definition for combination views.
- **Annotations** – are a note-type entity created to store model information without geometry. They may include both dimensional types and note types. Annotations are used to provide complete product definition.
- **Notes** - are used to capture supplemental data related to the model. This includes General Notes and other notes associated to the model. Additional notes will be created to support data previously displayed on the 2D drawing.
- **Meta Data** - is data that supports the definition, administrative or supplemental data package. Metadata includes all relations, parameters and system information used in a model. This data resides at the model and feature level.

Throughout this document standard naming conventions will be used for the various organizational elements. This will allow for future expansion of other model based venues.

4. Solid Modeling Philosophy

The goal of a solid model is to create and provide all design and detailed information, including design intent, to downstream users. This means all the required engineering information is communicated to everyone from one source, the Solid Model. In order for this communication to be effective, consistency is needed in the creation and presentation of each product.

Models are created using “start models” to ensure that “start model information” is present. Using a start part ensures a common beginning for all models, facilitates model specific information extraction, aids navigating thru model data, and sets you up for efficient MBD creation. Maintaining the integrity of model information is the responsibility of all engineers/designers and anyone else who may add or change the model during its creation and revision.

All start part models shall contain the following organizational elements to accommodate MBD:

- Standard Combination Views
- Supporting View States
- Supporting Layers
- Required Notes
- Required Metadata

5. Annotation Framework

One of the first decisions that must be made when annotating a model is what level of detail is required. If the documentation is intended to be laser cut or as a prototype then only a minimum level of annotations may be needed. If the design has secondary operations and the model is used for primary operations then only a partial set of annotations may needed to define the unique qualities of the model. If the design is to be built by outside suppliers or delivered to a customer then full annotation may be required.

These different types of annotations translated to the following levels of annotations. The levels are depicted in the examples below.

5.1 Level 1 – Minimal Annotation

This level is depicted in Figure 3 and contains the following:

- Envelope Dimensions (overall boundary dimensions of the part)
- "Block" or Profile tolerance as applied to the entire part
- Material and Finish Requirements
- Title Block Information

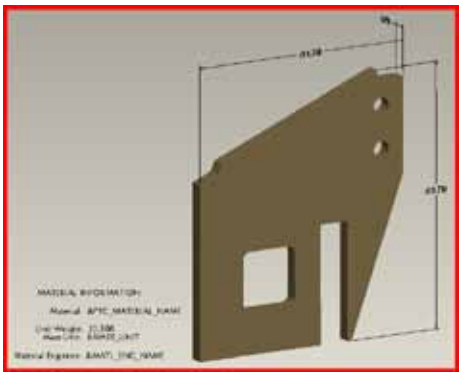

Figure 3: Level 1

5.2 Level 2 – Partial Annotation

This level is depicted in Figure 4 and contains the following:

- Envelope Dimensions
- "Block" or Profile Tolerance
- Material and Finish Requirements
- Title Block Information
- Non-Standard Dimensions
- Site Map
- Critical Notes

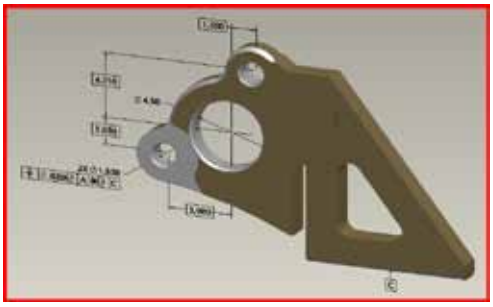

Figure 4: Level 2

5.3 Level 3 – Full Annotation

This level is depicted in Figure 5 and contains the following:

- Envelope Dimensions
- "Block" or Profile Tolerance
- Material and Finish Requirements
- Title Block Information
- Full Dimensions
- Site Map
- Full Notes
- Auxiliary Views

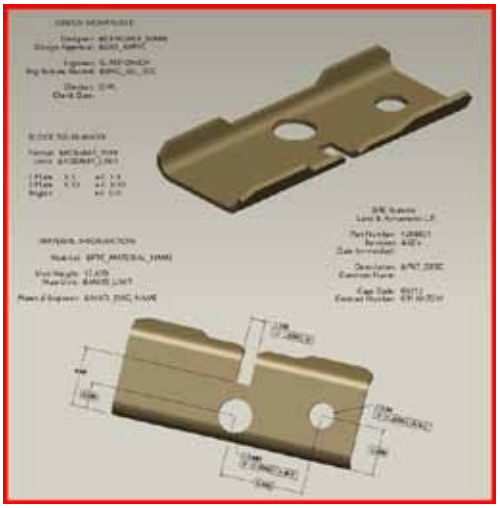

Figure 5: Level 3

6. Standard Organizational Requirements

For the annotation framework to be functional it must meet a standard minimum set of organizational requirements. These requirements include any information typically displayed on a 2D drawing (See Figure 6) including:

- Basic title Block and legal information are populated on pre-defined views
- Basic revision history information is populated on pre-defined views
- Dimensional and tolerance information are documented using annotation elements and features.
- The annotations are grouped in appropriate layers as defined by their purpose
- Detail views orientations are saved to best display the appropriate annotations
- Annotation, view and layer information are grouped in combination views for ease of access and publication
- Some textual information may be written to a neutral third party format as an alternative to a “flat-to-screen” display.
- The annotated model is published to a light weight viewer for downstream use by non-CAD users

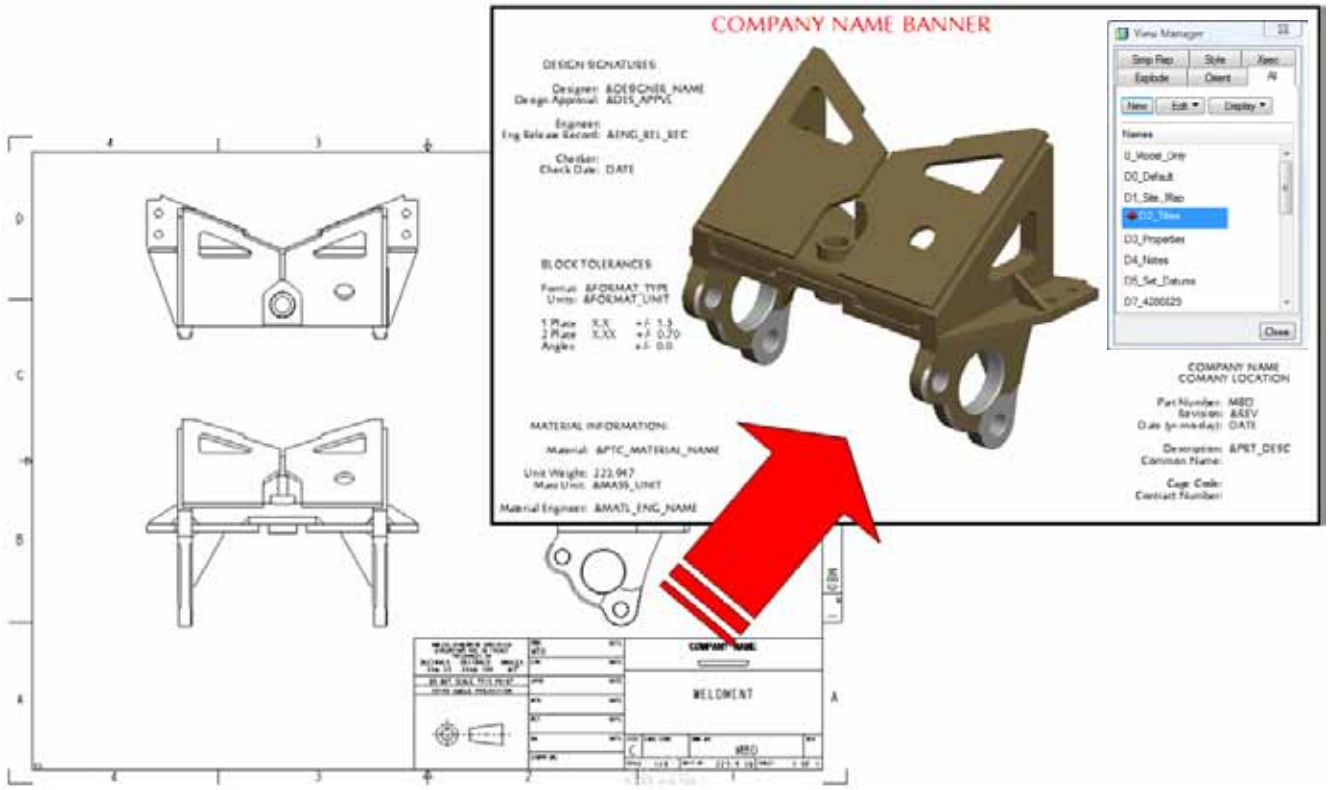

Figure 6: Annotation Framework

7. Defining a model using Model Based Definition

The actual instructions for detailing a model are defined in the Part and Assembly Work Instructions. What follows is the basic methodology and list of the basic tools and guidelines used to create and organize product data in the 3D model. The guidelines establish a common practice to organize and structure model data for viewing by the downstream user (See Figure 7). The downstream user may be a non-CAD user and may view the data in a lightweight 3D viewer as apposed to viewing data from a 2D drawing.

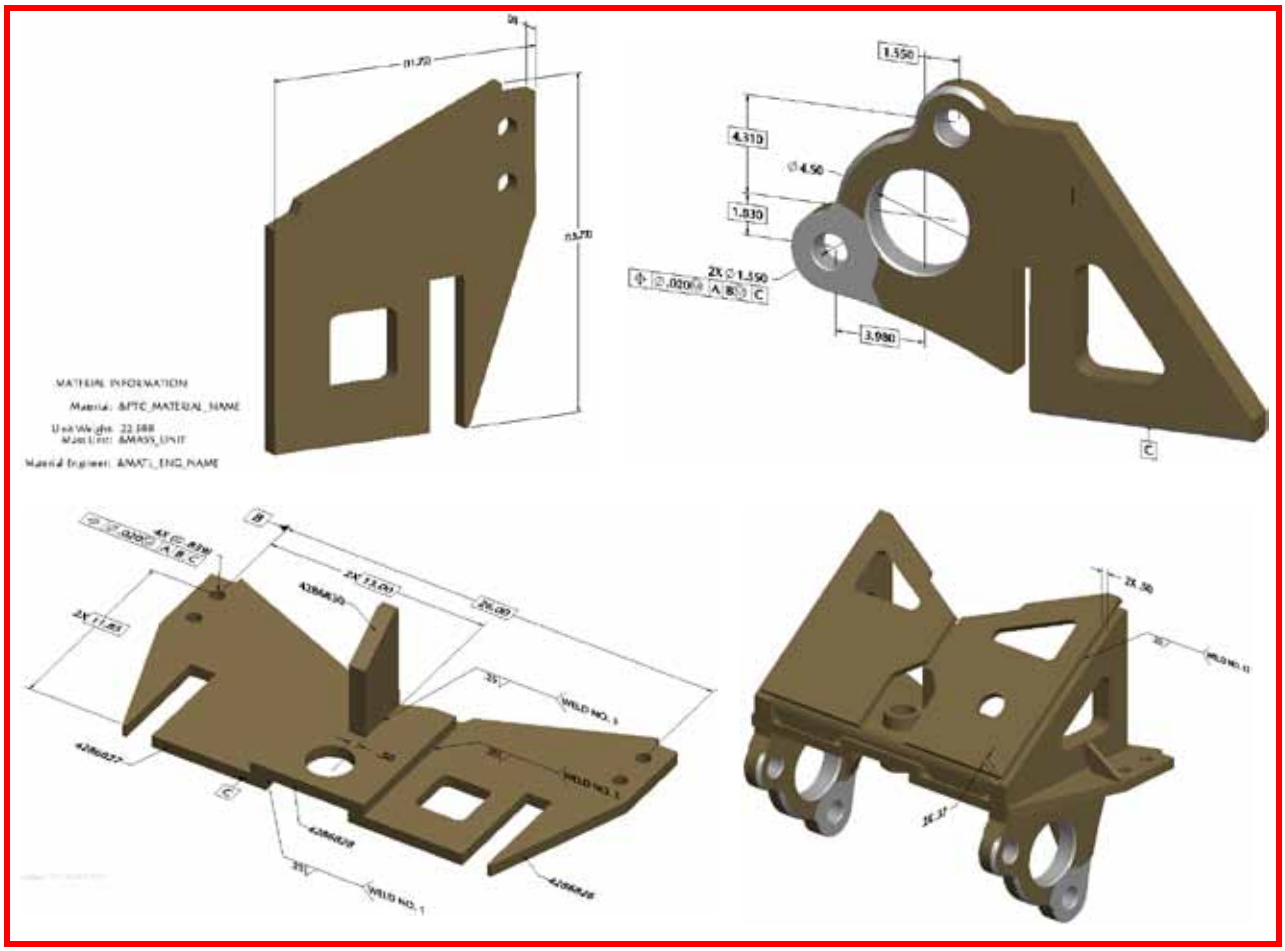

Figure 7: MBD Methods

7.1 Naming Conventions

Every item supporting MBD should be named in a consistent manor in order to facilitate both understanding and automating data extraction. Features should be named consistently to describe there intended purpose. All other annotations and supporting information should begin with “D#_Description” to identify it as MBD data. Future definition prefixes will be named appropriately (See Figure 8 for some examples)

· **Design**	**D#_**
· **Manufacturing**	**M#_**
· **Inspection**	**I#_**
· **Technical Publications**	**P#_**
· **Logistics**	**L#_**
· **Testing**	**T#_**
· **Others?**	**?#_**

Figure 8: Definition Prefixes

7.2 Layers

Layers are used to manage the display of annotation data and support definition for combination views. Layers support model display requirements for combination Views to comply with ASME Y 14.41-2003 section 5.1.1.

One can organize and control product data items by assigning them to layers. One can hide and show layers as necessary, which allows One control the display of multiple data items simultaneously. For example, One can assign notes to a specific layer and then show/hide them as desired with other data. Using layers, One can organize items and streamline selection because all the layer items are treated as a group. Layers can include other items in a Pro/ENGINEER database such as features, dimensions, notes, geometric tolerances, and other layers. Figure 9 shows an example of a layer tree for an annotated assembly model.

The naming convention shall be D#_Description (should coincide with Combination Views). One exception is Assembly only layers which should be named DA#_Description to aid in display control on higher level assemblies.

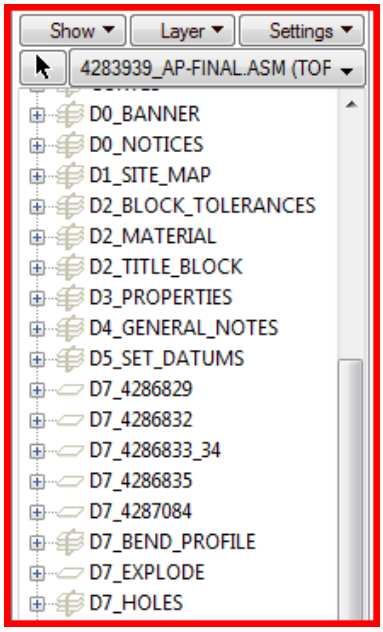

Figure 9: Layer Tree

7.3 Views States

View States are used to facilitate the presentation of the model and its annotation to comply with ASME Y14.41-2003 section 3.6.3.

One can show different view states using the **View Manager** dialog box. A view state may be saved in a part or assembly model. The saved view state may be utilized or referenced in a combination view or “all-state” view defined by the user. Figure 10 shows a typical list of view orientations.

View States support model display requirements for combination Views. The following views states shall be used to manage data as required:

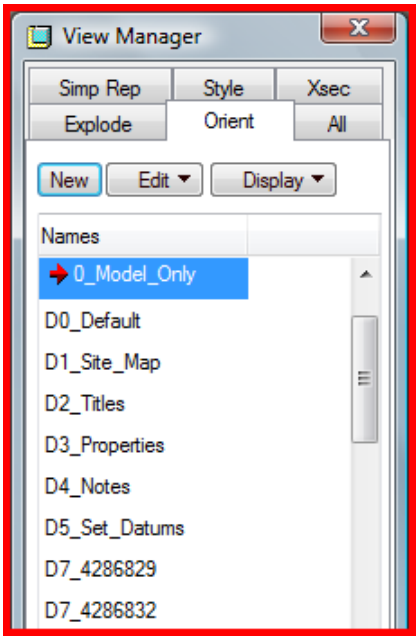

Figure 10: View State Menu

- **Orientation** – Views should be shown in an orientation that best displays the detail being defined. Orthographic view arrangements may be used but are not required with MBD. Views may be set in a 3D orientation. In additional to the principle views, zooming in on a specific area to clearly represent specific details may be added.
- **Cross Sections** –Section views should be used when interior detail cannot be shown clearly on the typical 3D views. One can streamline the display of cross-sections by using 3D cross-sections created in the model. By default, 3D cross-sections, are available for display in the model. One can display and control the cross-hatching of these 3D cross-sections.
- **Display Styles** – The display style for viewing components may be controlled using the display style menu. One can store a component with an assigned display style and retrieve that style by name so that One can return the model to the regular display style without losing defined settings.

- **Simplified Reps** - Simplified representations are used to control which members of an assembly are brought into session and displayed. Each simplified representation can correspond to an area or level of detail of the assembly in which individual areas or groups are being defined.
- **Explode States** - An exploded view of an assembly shows each component of the model separated from other components. An exploded view affects only the assembly appearance. Design intent and the true distance between assembled components does not change.

The naming convention shall be D#_Description (should coincide with it's corresponding Combination View).

7.4 Combination Views

Combination Views are used for defining an associative relationship between digital elements to comply with ASME Y14.41-2003 section 5.2.1.

A combination view allows one to combine and apply multiple **View Manager** display states and layers. Multiple combination views may be saved to display all the required product definition data. Figure 11 shows a typical list of combination views.

Combination views allow one to quickly navigate customized model display states to display required information by combing multiple View Manager display states. Combination views shall be used to manage the following data:

- View States (orientation, x-sections, styles, simplified representations and explode states)
- Layer Status
- Annotations (dimensions, GD&T, notes, text or symbols)

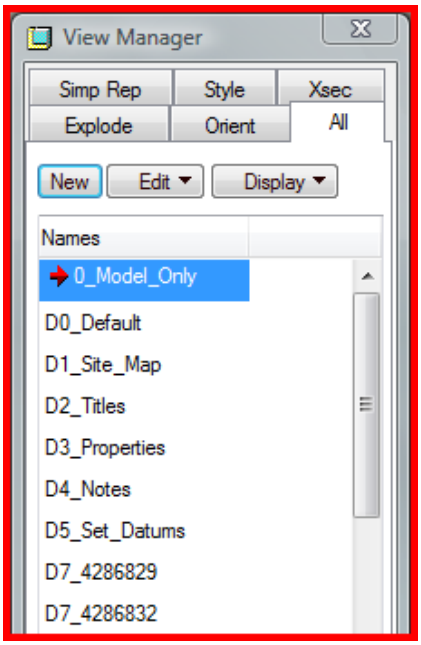

Figure 11: Combination Views

The naming convention shall be D#_Description

The Combination View Menu shall contain the following menu selections (located in the **All** tab in the **View Manager** dialog box in Pro/E):

Menu Name	Description of Contents Displayed
· 0_Model_Only	Although it is not required for MBD this view is recommended to quickly hide all annotation displaying only the model
· D0_Default	Legal notice, Proprietary notice, ITAR notice and/or other company specific notices. **NOTE:** File to be saved in this state
· D1_Site_Map	Notes identifying all available combination views
· D2_Titles	Title Block information (Company name, design description, model number, cage code, design signatures, block tolerances, material and finish requirements)
· D3_Properties	Overall boundary dimensions, mass properties, material, finish requirements and title block information

Combination View Menu continued:

- D3_Characteristics Optional – Key characteristics
- D4_General_Notes General note information
- D5_Set_Datums Set datums
- D6_User_Derfined Exploded Views, Section Views and Assembly Unique views
- D7_User_Defined Define the details of the Machining operations using annotations
- D8_User_Defined Define the details of the Welding operations using annotations

7.5 Annotations

An annotation is a note-type entity created to store model information without geometry. Annotation elements and features allow the user to define dimensions and supporting data in 3D space. They are parametric and will rotate with the model (Figure 12). Annotations are used to provide complete product definition to comply with ASME Y14.41-2003 section 3.

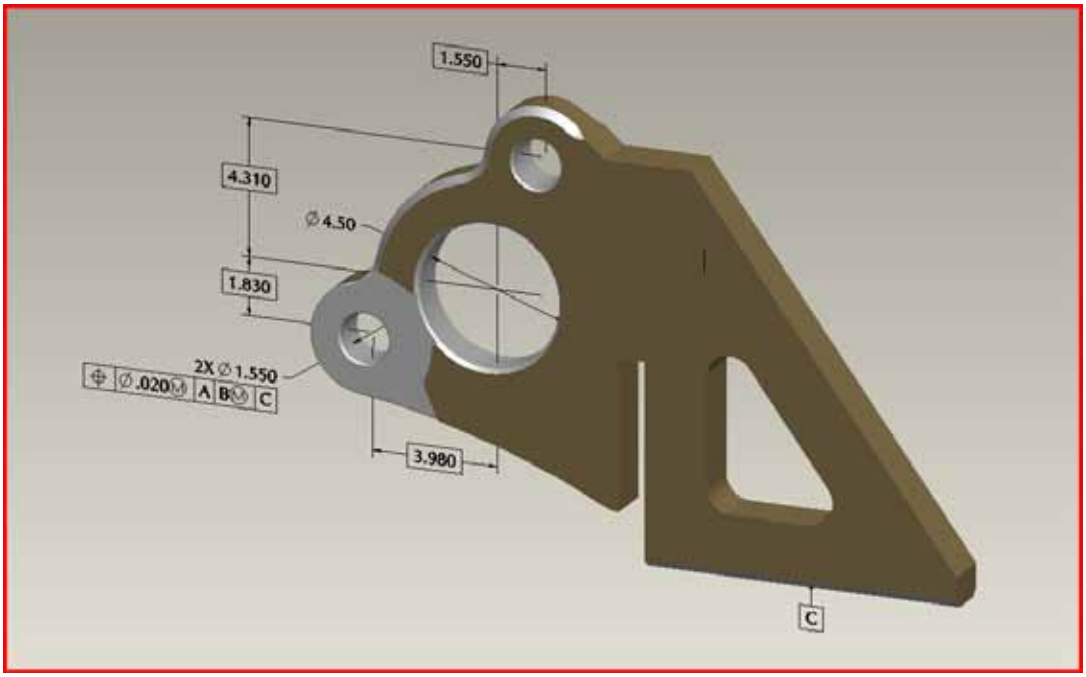

Figure 12: Annotations

Annotations can exist independently within the model as an annotation element, or be included in Annotation features. If an annotation is part of an Annotation feature, it becomes the graphic representation of an Annotation Element.

Each annotation data type shall be placed on an appropriate layer and orientation plane corresponding to the view it represents:

Driving Dimensions - are model driving dimensions created and owned by features in Annotation Elements by selecting the dimensions and placing them on annotation planes. Such Annotation Elements are called Driving Dimension Annotation Elements and are created in the feature that owns the dimension.

Driven Dimensions - are used by Pro/ENGINEER to measure the size and shape of features within a model. The value of a driven dimension changes when the size and shape of the features are modified. Driven dimensions can have tolerances, to which manufactured components can be accepted or rejected.

Notes - are text strings that can either placed as a fixed note or attached to an object. One can place any number of notes in a model and orient them in multiple orientations.

Symbols - are collections of draft geometry and text that either serve as a simple labeling object, or that represent more complicated objects such as assemblies or electrical components. The symbol may show parameters as text notes. When the instance is placed, the parameters read the values associated with the drawing.

Geometric Tolerances - (GTOLs) provide a comprehensive method of specifying where on a part the critical surfaces are, how they relate to one another, and how the part must be inspected to determine if it is acceptable. They provide a method for controlling the location, form, profile, orientation, and run out of features.

Surface Finishes - are associated with surfaces in the part. Each surface symbol applies to the entire surface. One can add surface finish symbols to a model using standard surface finish symbols available in Pro/E, or One can create and save Oner own surface finish symbols.

7.6 Annotation Orientation

An annotation orientation refers to the plane or the parallel plane in which the annotation lies, the viewing direction and the right direction or text rotation. One should maintain text orientation in a view that provides consistency for viewing.

One can define the orientation for annotations by selecting a datum plane or flat surface which is parallel to the annotation. The annotation orientation defined by a reference plane can be driven by plane or frozen. Figure 13 show an example of an annotation orientation plane.

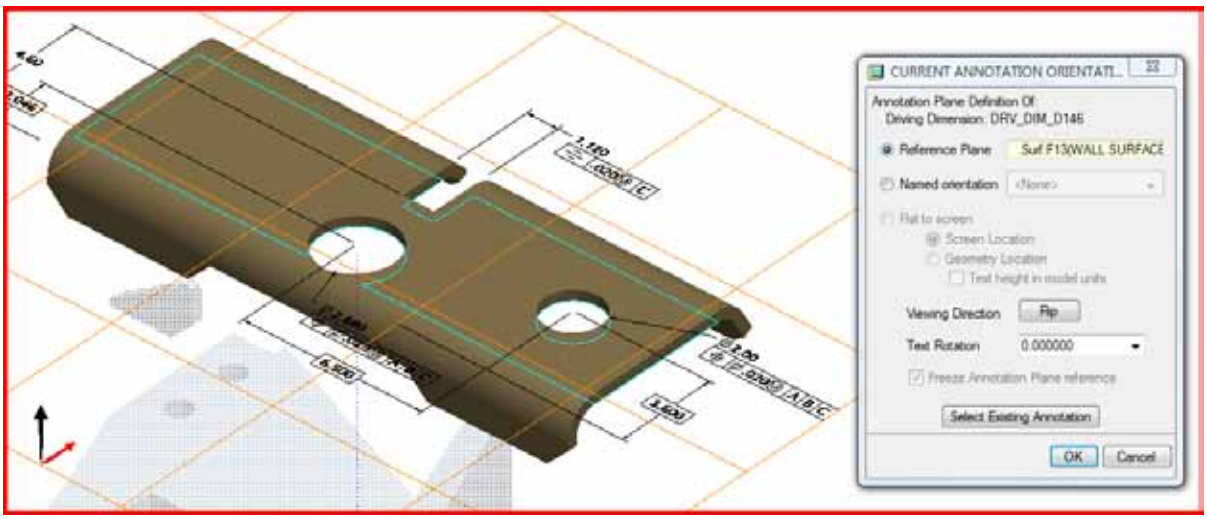

Figure 13: Annotation Orientation Plane

The orientation of the annotation plane shall be maintained relative to the model geometry as the model is manipulated in 3D to comply with ASME Y14.41-2003 section 5.2.3.

The following annotation data types shall be placed on an appropriate orientation plane:

- D0_Notices - Flat to screen by screen location
- D1_Site_Map – Appropriate orientation plane
- D2_Titles – Flat to screen by screen location
- D3_Properties - Appropriate orientation plane
- D4_General_Notes - Flat to screen by screen location
- D5_Set_Datums - Appropriate orientation plane
- D6_Assembly - Appropriate orientation plane
- D7_Machining - Appropriate orientation plane

- D8_Weld - Appropriate orientation plane

7.7 Model Notes

Model notes are text strings that can be attached to objects or displayed in a fixed location flat to screen. One can attach any number of notes to any object in the model to comply with ASME Y14.41-2003, section 3.4.2.

One can include model notes in Annotation features. This way, one can define a library of company-specific notes, and then place them as annotations in various models, as appropriate.

When inserting notes, one has a series of options to set up the note properties, such as attachment type and location, leader style, and text position.

All models shall have the following notes to accommodate MBD:

Note Name	Description of Note
· Banner	A fixed note such as your company name or other indicator such as Top Secret
· Site_Map	Note/s identifying all available combination views
· Notice	ITAR notice or other classified specific restricted data information
· Legal	Competition sensitive, government rights and/or distribution statement
· Proprietary	Company proprietary information
· Title_Block	Company name, part number, revision, date, part description, cage code and/or contract number. Define as parameters
· Design_Signatures	Designer name, design approval record, engineer name, engineer release record, checker name, check date. Define as parameters
· Block_Tolerances	Format type, format units, 1 place tolerance, 2 place tolerance, Angle tolerance. Define as a parameter for extraction and reuse.
· Material	Material specification, coating, heat treating, unit weight, mass units, material engineer record. Define as a parameter for extraction and reuse.
· Finish	Finish specification, preparation call out, paint note, etc. Define as a parameter for extraction and reuse.
· General_Notes	Company specific notes including: ASME Y14.41-2003, model query note - When model query is required, a notation stating the requirement for query of the model or associated data shall be added to the general notes (per ASME Y14.41-2003, sec 3.1.1, 5). Values queried from the model for any feature(s) without any tolerance or datum target specifications assigned shall be reference dimensions (per ASME Y14.41-2003, sec 3.1.1, 8).

7.8 Meta Data

Meta Data is data that supports the definition, administrative or supplemental data package. Metadata includes all relations, parameters and system information used in a model. This data resides at the model and feature level.

Certain information such as title block data, material, block tolerances, etc. should be defined as parameters to allow the option for future extraction of the data. In addition, this allows the information to be displayed in any number of combination view and to easily update all instances via an associative link to the parameter name.

All models shall contain the following parameters to accommodate MBD:

Critical Data Elements List		
ID	**Element**	**Description**
P1	ALT_MATERIAL	Use as needed to define alternative materials
P2	BOM_NOMENCLATURE	Nomenclature for use in the drawing parts list
P3	CAGE_CODE	Company Cage Code
P4	CHK_DATE	Checked date
P5	CHK_NAME	Checked by
P6	CONTRACT_NO	Parameter for contract number
P7	CONTROL_ACTIVITY	The organization code of the group with the control authority
P8	CRIGHT_DATE	Copyright year
P9	DATA_RIGHTS	The code, which identifies the rights status of the information on the Model or contained in the file identified by this record.
P10	DESIGN_ACTIVITY	The name of the design activity whose CAGE is assigned to the Model.
P11	DIST_CODE	The distribution statement code letter (A, B, C, D, E, F, or X) of the model identified in this record.
P12	DOC_TYPE	A code entered that identifies the class or type of engineering model (e.g., product drawing, parts list, wire list, safety data sheet, etc.).
P13	DRAWING_NUMBER	Drawing Number
P14	DWN_DATE	date the drawing was created
P15	DWN_NAME	Name of the drawing author
P16	ENG_DATE	Date of the approval engineer's signature
P17	ENG_NAME	Name of the approval engineer
P18	EQV_PART_NUM	USE if needed to define an alternant part for this item
P19	FILE_FORMAT	The specific product data file format with version, e.g., HP 7586, TIFF Group 4, RS-274X, AutoCAD 2000, DXF r14, EDIF 400 Schematic, Intel Hex-32, etc. No version needs to appear for DXF or JT files since many conversion programs do not specify the versioning.

Critical Data Elements List		
ID	**Element**	**Description**
P20	ITAR_NOTE	Legal note parameter for ITAR statements
P21	KG_WT	The weight of the product in kilograms
P22	LB_WT	The weight of the product in pounds
P23	LEGAL_NOTES	General note parameter for contractual and legal statements
P24	LT_0	Liner tolerance
P25	LT_X	Liner tolerance
P26	LT_XX	Liner tolerance
P27	LT_XXX	Liner tolerance
P28	MATERIAL	General note parameter for material
P29	MATL_ENG	Name of the material engineer
P30	MATL_ENG_DATE	Date of the material engineer's approval
P31	MATL_SPEC	Name of material specifications
P32	MATL_TYPE	Material Type
P33	MFG_DATE	Date of the manufacturing engineer's approval
P34	MFG_NAME	Name of the manufacturing engineer
P35	MODEL_NUMBER	The model number of the product
P36	NEXT_ASSY1	The number of the next higher assembly using this product
P37	NEXT_ASSY1_QTY	Quantity of Usage per Next Higher Assembly
P38	NEXT_ASSY2	The number of additional next higher assemblies using this product
P39	NEXT_ASSY2_QTY	Quantity of Usage per Next Higher Assembly
P40	NEXT_ASSY3	The number of additional next higher assemblies using this product
P41	NEXT_ASSY3_QTY	Quantity of Usage per Next Higher Assembly
P42	NEXT_ASSY4	The number of additional next higher assemblies using this product
P43	NEXT_ASSY4_QTY	Quantity of Usage per Next Higher Assembly
P44	NEXT_ASSY5	The number of additional next higher assemblies using this product
P45	NEXT_ASSY5_QTY	Quantity of Usage per Next Higher Assembly
P46	NOMENCLATURE	The nomenclature description of the product
P47	OA_HEIGHT	Over All Height of the product
P48	OA_LENGTH	Over all length of the product
P49	OA_WIDTH	Over all width of the product

Critical Data Elements List		
ID	**Element**	**Description**
P50	PIN	The PIN is an identifier assigned by the responsible design activity or by the controlling nationally recognized standard, which uniquely identifies (relative to that design activity) a specific item.
P51	PIN_REV	The revision level assigned to a PIN.
P52	PROD_DATA_TYPE	The appropriate product data type, e.g., native drawing, neutral drawing, view only image drawing, raster drawing, native model, neutral model, loft data, ASCII text, programming data file, tubing data, Gerber artwork data, etc.
P53	PROPRIETARY	Note parameter for proprietary statements
P54	QA_DATE	Date of quality approval
P55	QA_NAME	Name of Quality Engineer
P56	RAW_MATL_NO	The part number of the raw material
P57	REV	Revision Level of the drawing
P58	REV_APPROVAL	Name of the revision approver
P59	REV_DATE	Date of the revision approval
P60	REV_DESC	Description of the Revision
P61	SECURITY_LVL	The code that identifies the classification of the model.
P62	SW_NAME	The name of the native software CAD program used to generate the vector file.
P63	SW_OS	The computer operating system for the CAD software.
P64	SW_OS_VER	The version of the computer operating system for the CAD software.
P65	SW_VENDOR	The vendor of the native CAD software program.
P66	SW_VERSION	The version number and/or letters of the native software CAD program used to generate the file.
P67	TOL_HEADING_1	Tolerance information
P68	TOL_HEADING_2	Tolerance information
P69	TOL_TABLE	Metric tolerance table used
P70	UNIT_WT	Unit weight of the part for BOMs
P71	UOM	Unit of measure for the model
P72	UOW	Unit of weight for the model
P73	VOLUME	Volume of the product
P74	WELD_ENG	Name of the weld engineer
P75	WELD_ENG_DATE	Date of the weld engineer's approval
P76	WEP_SYS_MODEL	If this file is unique to a weapon system model.
P77	WEP_SYS_NAME	The weapon system name that the drawing is applicable to.

Appendix A: Glossary

- Annotation - A note, dimension, symbol, or other representation that defines some characteristic of the design intent
- Annotated Model - A Computer Aided Design 3D Solid model that documents the product and manufacturing definition in a human readable manner similar to, but without reliance on, a conventional drawing.
- CAD – Computer Aided Design
- Fully Annotated Model - An annotated model that documents the complete product definition.
- Minimally Annotated Model - An annotated model that documents only basic product and manufacturing definition and requires that the model be interrogated to get more information. (i.e. they do not contain tolerances or symbology other than envelope dimensions, material and finish)
- Model Based Definition - A methodology of creating a single source master CAD model which describes the product definition that does not rely on a conventional drawing representation and/or paper based documentation.
- Model Based Enterprise - A fully integrated and collaborative environment founded on model based definition that is shared and reused across the enterprise; to enable rapid, seamless, and affordable deployment of products from concept to disposal.
- Partially Annotated Model - An annotated model that documents all key, critical and nonstandard product and manufacturing definition data while covering all other aspects with general note statements.

Appendix B: Reference Documents

- ASME Y14.41-2003 Standard on Digital Product Definition Data Practices