Drawings For Everyone (Still?)

Preview, presented to

Temecula Valley User Group

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Presenter: Charley Leonard, Leonard Design

Ohrqdug Ghvljq

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SolidWorks Certified Professional

Opening Comments

- Who I am
- “preview” disclaimer
- Audience query
  - Seen 2008 Rollout?
  - Using 2008?
  - 2007?
  - 2006?
  - Earlier?
- By no means all-inclusive
- Fast moving
- Request for feedback
- Ask questions any time
- Have fun

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Agenda

- Drawings Basics
- Drawing Templates and Sheet Formats
  - Not the same thing!
- Linking Notes To Document Properties
- What's new (… er old) in 2007 (Drawings)
- To draw or not to draw? What about Model-only documentation?
  - ASME Y14.41-2003
  - Annotation views
  - New in 2008: DimXpert
- *(As time permits):* Tips and Tricks….
  - Or "Additional Helpful Tools"
- Other ways to find information
- Solicitation for feedback

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Drawings Basics

- 3 main file types
  - .sldprt (part model)
  - .sldasm (assembly model)
  - **.slddrw (drawing)**

- A drawing references a part model, an assembly model or a combination of the two

Drawings Basics

- Starting a New Drawing
- Opening an Existing Drawing
- Adding Views
  - Standard
    - When no model is open
      - Menu
      - Toolbar
      - View palette (new in 2007)
      - Drag & drop (from explorer)
    - When model is open
      - All of the above
      - Drag & drop from open model
    - When several models are open
      - All of the above, all open models show in dialog box & palette
  - Section
  - Auxiliary
  - Detail

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Drawings Basics

Dimensioning & annotating

- Insert dimensions
  - Bring in from the model
    - Driving dimensions
      - » Bi-directionally associative when brought into sheet
    - Driven dimensions, or “reference dimensions”
  - Add at the drawing sheet (reference only)
- Insert annotations
  - At drawing
  - At model
- View focus, sheet focus
- Move dimensions
- Modify dimensions
- Hole callout
  - Modeling smart for ease of drawing annotation
    - Use up to next (sheet metal) or through on holes if you want “THRU” to show up on the drawing, instead of some depth
- Revision table

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Drawing Template

- Not the same thing as a sheet format – we will discuss next
- Drawing template is one of 3 types of document templates
  - Part Templates (*.prtdot)
  - Assembly Templates (*.asmdot)
  - **Drawing Templates** (***.drwdot**)
  - (There are also templates for various other items, such as tables. These are not considered *document* templates)
- Templates are files that include user-defined parameters and are the basis for new documents.
- Anything under tools/options/document properties can be saved in a document template.
- Each sheet in a drawing or drawing template can contain
  - Predefined views
  - Predefined text
    - Notes which might get edited
  - Blocks
  - Annotations
  - Sheet Format

Sheet Format

Should contain items to be edited rarely.

- page size and orientation
- standard text
- company logo & contact information
- borders
- title block
- General notes only if they are standard and not to be edited
  - Reloading the format resets everything at the format level to whatever is on the reloaded format.
- Think of a sheet format as a component of a sheet within a drawing or drawing template
- Think of a sheet as a component of a drawing or a drawing template
- A drawing or drawing template can have multiple sheets, each having a different format
- A drawing template can also have *no* sheet format. If this is the case, the user will be prompted for the sheet format size.
  - Tools, options, file locations, sheet formats determines the locations of default formats
  - You can browse if the format you need is in another location

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Sheet Format

- Create a new template and sheet format
- Setting up a new template folder
- Create a template from an existing drawing
- Create various templates for various types of documents
  - Various sheet sizes (although not necessary)
  - Various standards
  - Various sets of pre-inserted views
  - Different customers
    - I have a custom folder for each of my customers. It contains any templates, sheet formats, revision blocks, annotations, blocks, etc. specific to that customer

Linking Notes to Document Properties

All SolidWorks 2007 documents have these system-defined properties:

Property Name	Value
SW-Author	**Author** field in **Summary Information** dialog box
SW-Comments	**Comments** field in **Summary Information** dialog box
SW-Configuration	NameConfiguration name in ConfigurationManager of part or assembly
SW-Created Date	***Created** field in **Summary Information** dialog box
SW-File Name	document name, no extension
SW-Folder Name	document folder with backslash at the end
SW-Keywords	**Keywords** field in **Summary Information** dialog box
SW-Last Saved By	**Last Saved By** field in **Summary Information** dialog box
SW-Last Saved Date	***Last Saved** field in **Summary Information** dialog box
SW-Long Date	*current date in long format
SW-Short Date	*current date in short format
SW-Subject	**Subject** field in **Summary Information** dialog box
SW-Title	**Title** field in **Summary Information** dialog box

* Properties that include date formats are language and region dependent. For details, see the **Control Panel** settings on your computer.

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Linking Notes to Document Properties

Additionally, **drawings** have the following system-defined properties (2007):

Property Name	Value
SW-Current Sheet	sheet number of the active sheet
SW-Sheet Format Size	sheet size of the active sheet format
SW-Sheet Name	name of the active sheet
SW-Sheet Scale	scale of the active sheet
SW-Template Size	template size of the drawing template
SW-Total Sheets	total number of sheets in the active drawing document

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Custom Properties

- These are user-defined properties, which can also be referenced in notes.
- Out-of-the-box templates come with a predefined list, which you can edit.
- Create your own custom property if there isn’t one already in the list that’s appropriate for your need.
- When you create your own templates, you might want to delete the custom properties you won’t use, so the list is shorter and easier to navigate.

What's New (… er old) In SolidWorks 2007 (Drawings)

- Broken Views
  - Can create multiple horizontal and vertical breaks in the same drawing view.
  - When you insert the break lines, the breaks are immediately applied.
  - Specify horizontal or vertical breaks and type of break lines in the PropertyManager.
- Deleting Views
  - New **Confirm Delete** dialog box has option to **Delete detail sketch** or **Delete section sketch** so the sketch used to create the view is also deleted.

What's New (… er old) In SolidWorks 2007 (Drawings)

- Drawing sheets
  - Navigating
    - In drawings with multiple sheets that span beyond the length of the screen, use the navigation buttons at the bottom of the drawing window to jump to the first or last sheet
    - If you hold down the next or previous sheet buttons, the tabs scroll.
  - Renaming
    - You can rename a drawing sheet by right-clicking a tab at the bottom of the window and selecting Rename.
  - Reordering
    - When you reorder drawing sheets with the tabs at the bottom of the window, the pointer changes to indicate single sheet or multiple sheets.

What's New (… er old) In SolidWorks 2007 (Drawings)

- Relative Views
  - You can select reference planes to specify the view orientation of relative views. (In SolidWorks 2006, Workarounds were required for cylindrical structural members).
- Section Views
  - When you create a section view in a part or assembly, you can save the section view to include in the **View Palette**.

What's New (… er old) In SolidWorks 2007 (Drawings)

- View Palette
  - Contains images of
    - standard views
    - annotation views
    - section views
    - flat patterns (sheet metal parts)
  - Populates when you
    - Click **Make Drawing from Part/Assembly**.
    - Browse to a document from the **View Palette**.
    - Select from a list of open documents in the **View Palette**.

What's New In SolidWorks 2007 (Detailing)

- Read the "What's New" manual
- *More to come in future versions of this presentation*

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Tips and Tricks….
Or “Additional Helpful Tools”

- How to make a drawing from another drawing, referencing an created model, when the model was created from another model that had a drawing, but a drawing wasn’t created with the save as command
- Area boundry fill when hatching
- Select a group of ordinate dimensions, break alignment, then move the selection together
- Some annotation didn’t get attached to the correct view? Cut & paste it.
- Snap to min/max of arcs; shift (while dimensioning)

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Tips and Tricks….
Or “Additional Helpful Tools”

- Changing or adding text in a selected group of annotations, such as adding 2X, or changing number of decimal places
- Hiding/showing components in views
  - Useful when you want to dimension a hole but there’s a pem nut in it
  - Or when you want to highlight a specific component in one view only
  - Or when you what to show only certain components but don’t want to create a configuration
- copying things by dragging (such as symbols, notes)
- crop view

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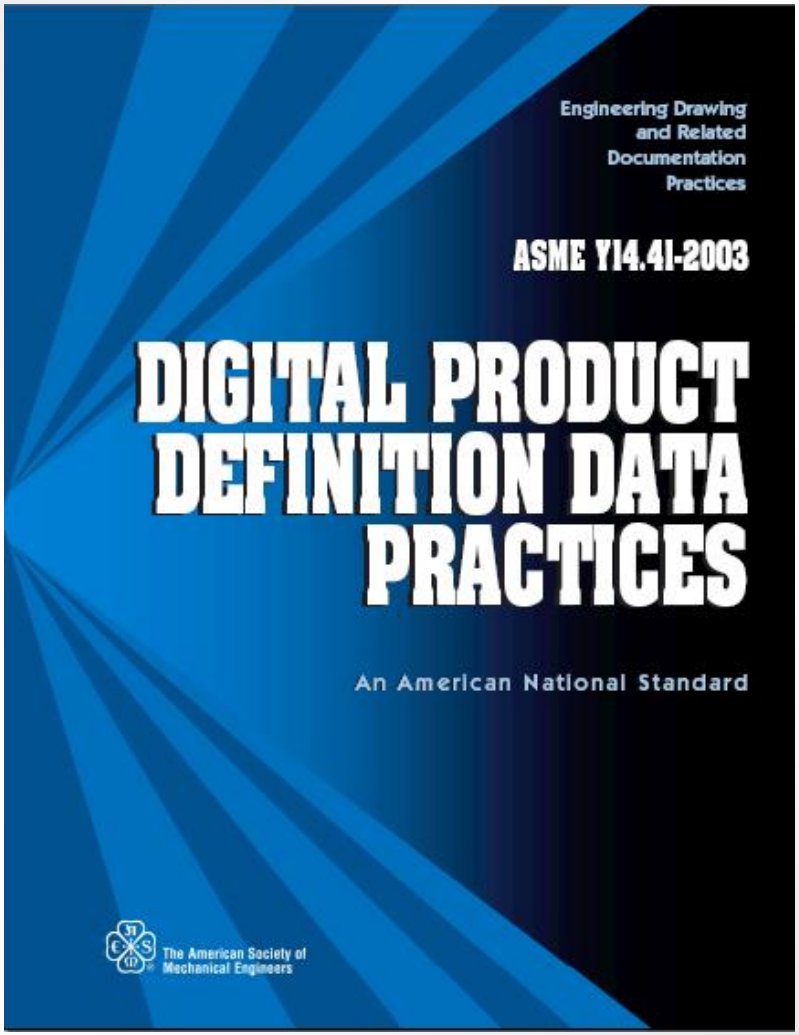
To Draw or Not to Draw?

- Audience Query:
  - How many agree with the statement, “Drawings are here to stay”?
  - Why or why not?

- Some reasons 2D drawings are still prevalently used, even in environments where 3D CAD is the primary design tool.
  - This is what users are used to seeing
  - Need to show tolerances, additional information such as material, special treatments, etc.
  - ?

Cover of ASME Y14.41-2003 Standard



Some excerpts from ASME Y14.41-2003, Digital Product Definition Data Practices

1 GENERAL

1.1 Scope

This Standard establishes requirements and references documents applicable to the preparation and revision of digital product definition data, hereafter referred to as data sets. This Standard defines exceptions and additional requirements to existing ASME standards for using product definition digital data sets or drawings in digital format. Where no exception or additional requirements are stated, existing ASME standards shall apply.

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Some excerpts from ASME Y14.41-2003, Digital Product Definition Data Practices

1.2 Structure of Standard

This Standard supports two methods of application: model only, and model and drawing in digital format. The structure starts with the requirements common to both methods, and then branches to the other sections that have differing requirements for each method. In addition, it provides a guide for the many CAD software packages to develop better modeling and annotation practices for computer aided design and engineering disciplines.

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Some excerpts from ASME Y14.41-2003, Digital Product Definition Data Practices

1.8 References

ASME Y14.1-1995 (R2002), Decimal Inch Drawing Sheet Size and Format

ASME Y14.1M-1995 (R2002), Metric Drawing Sheet Size and Format

ASME Y14.2M-1992 (R1998), Line Conventions and Lettering

ASME Y14.3M-1994 (R1999), Multiview and Sectional View Drawings

ASME Y14.4M-1989 (R1999), Pictorial Drawing

ASME Y14.5M-1994 (R1999), Dimensioning and Tolerancing

ASME Y14.8M-1996 (R2002), Castings and Forgings

ASME Y14.35M-1997, Revision of Engineering Drawings and Associated Documents

ASME Y14.38-1999, Abbreviations and Acronyms

ASME Y14.100-2000, Engineering Drawing Practices

Publisher: The American Society of Mechanical Engineers (ASME International), Three Park Avenue, New York, NY 10016-5990; Order Department: 22 Law Drive, Box 2300, Fairfield, NJ 07007-2300

IEEE/ASTM SI 10, Standard for Use of the International System of Units (SI): The Modern Metric System

Publisher: Institute of Electrical and Electronics Engineers (IEEE), 445 Hoes Lane, Piscataway, NJ 08854-1331

Publisher: The American Society for Testing and Materials (ASTM), 100 Barr Harbor Drive, West Conshohocken, PA 19428-2959

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Some excerpts from ASME Y14.41-2003, Digital Product Definition Data Practices

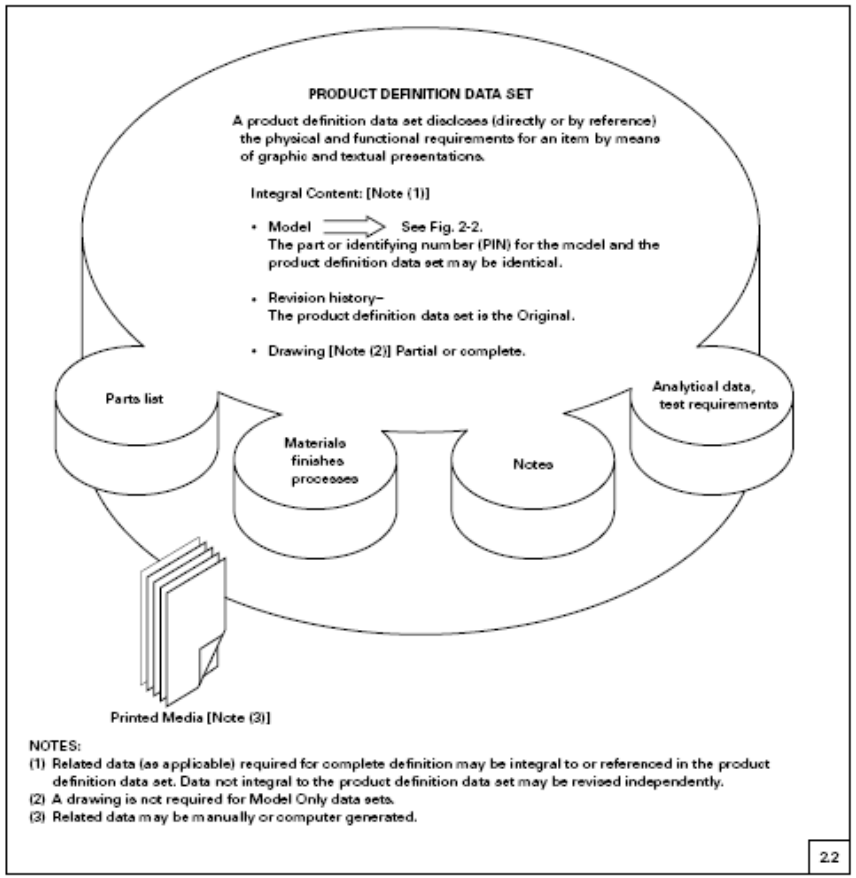

Fig. 2-1 Contents of a Product Definition Data Set

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Page from ASME Y14.41-2003 Standard

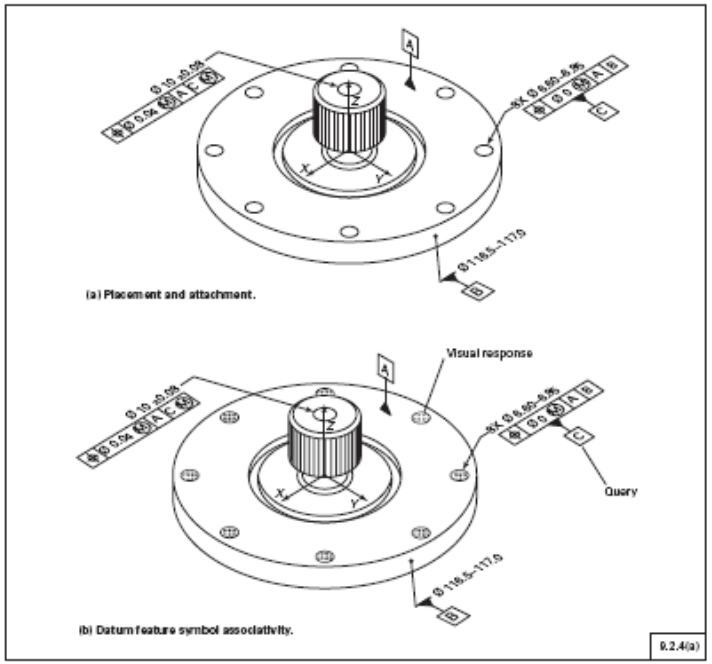

Fig. 9-7 Pattern of Features Establish a Datum Axis

show the amount of the tolerance zone outward from the surface. The remainder of the tolerance is inward. See Fig. 10-19 and Mandatory Appendix I.

(g) Profile of a Line Directed by Line Element. The feature control frame shall be placed on an annotation plane containing the represented line element, parallel and perpendicular to the absolute coordinate system or an established user-defined coordinate system. See Fig. 10-20.

(h) Profile of a Line Directed by Ordinate Axis. The feature control frame shall be placed on an annotation plane parallel and perpendicular to the absolute coordinate system or an established user-defined coordinate system. See Fig. 10-21.

10-2.4 Location Tolerances. The location feature control frame shall be placed on an annotation plane parallel or perpendicular to the referenced primary datum. Table 10-4 identifies location tolerance applications and the attachment method used.

(a) Positioning Feature Patterns Individually to Individual Datum Features. Each individual pattern of features of the model and the required individual datum feature of the model shall be collected as an associated group. A

Annotation Views

- SolidWorks part and assembly documents support 3D annotations according to the ASME Y14.41-2003 standard.
- 3D annotations are organized according to the model's orthographic views, such as front, bottom, isometric, etc. These orientations are called **annotation views**, and they replicate the standard drawing view orientations. Annotation views can be created automatically or manually.
- Double click any annotation view to see the annotations in the view. An arrow indicates when an annotation view is active.
- You can use annotation views in a drawing. They appear in the drawing as annotation views.

New in SolidWorks 2008:
SWIFT DimXpert

Quote from SolidWorks Press Release:

"SWIFT DimXpert automatically sets geometric dimensions and tolerances on parts, saving time and providing expert information every design team needs. It provides visual feedback on whether the model has been properly described and is ready for manufacturing. DimXpert adheres to the ASME Y14.41-2003 3D specification and automatically creates views, dimensions, and tolerances in 2D drawings for complete design documentation."

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Tips and Tricks….
Or “Additional Helpful Tools”

- alignment
  - Inferring when placing items such as circles, lines, text, etc.
- Adding text to detail & section labels
  - (right click & select “properties”, then what you add won’t disappear)
- Flag notes using symbols
- Putting photos in a drawing, annotating
  - How to move annotations to top (still an issue in 2006/20072008?)

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Where To Get Additional Information

- Help File
- Your VAR's tech support
- www.solidworks.com
- User groups, www.swugn.org
- Discussion forum, http://forum.solidworks.com/swforum

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Please critique me!!!

- Please email me at cleonard@2Dto3D.com or contact me through my web site with….
  - Anything I got wrong
  - Anything you thought should have been included or better covered
  - Anything you thought could have been presented more clearly

Thanks for coming!

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