Ensure GD&T Practices in Models Directly with SOLIDWORKS MBD

Oboe Wu
Senior Product Portfolio Manager
DS SOLIDWORKS
Oboe.Wu@3DS.com
Why Does GD&T Matter?

- Significant economic consequences
- Small details entailing big changes
- Different from drawings to MBD
Does It Look Familiar?

A telescope lens barrel in the outer space

Story courtesy of Tec-Ease
An Actual Machined Part
How to Inspect?

 Supplier

 $8,000,000 LAWSUIT

 Client

 PASS?

 FAIL?
Datum, Datum Feature, and Datum Feature Simulator
How Does SOLIDWORKS Define It?
Three MBD Advantages over Drawings on GD&T

- Define features unambiguously
- Ensure compliances with built-in GD&T intelligence
- Facilitate communications with visual aids
Define Features Unambiguously
Define Features Unambiguously

Confusing datum feature symbol

Define datum features clearly
Are They The Same?
Define A Width Feature as A Datum Feature

The dimension lines are automatically aligned
Define A Pattern as A Datum Feature
Define Features Unambiguously
Ensure Compliances with Built-in GD&T Intelligence
Ensure Compliances with Built-in GD&T Intelligence
Fig. 7-28  Bidirectional Positional Tolerancing, Rectangular Coordinate Method

This on the drawing:

Means this:

Axes of MMC holes must lie within the 0.4 X 0.2 rectangular tolerance zone basically constrained in rotation and translation in relation to the specified datum reference frame.
Flag Conflicting Tolerance Zones
Flag Invalid Maximum Material Conditions

No size tolerances defined for feature Simple HOLE

3X ☄ .20

3X ☄ .04 M A B C

C
Flag Invalid Maximum Material Conditions

Notes:
1. Query the CAD model for basic dimensions.
2. Unless otherwise specified, the default profile tolerance is \[ \pm 0.02 \] A B C D.
Facilitate Communications with Visual Aids
Automatically Create and Display Coordinate Systems
Locate Datum Features from References Automatically
Cross Highlight
Summary

Define features unambiguously

Ensure compliances with built-in GD&T intelligence

Facilitate communications with visual aids
Expect a Learning Curve

2D to 3D
- Geometry vs. Feature
- Orientation and Views

Tolerancing Practices
- Datum features
- Dimensions
- GD&T Intelligence

New tools
- DimXpert
- 3D PDF
- Downstream applications
- PDM/PLM Integrations
An Exercise Idea: MBD Test Models by NIST
Backup Slides
Avoid “view dependent tolerances”
NOTES (UNLESS OTHERWISE SPECIFIED)
1. OBTAIN DIMENSIONS FOR ALL UNDIMENSIONED FEATURES FROM THE MODEL. ALL DIMENSIONS OBTAINED FROM THE MODEL ARE BASIC UNLESS OTHERWISE SPECIFIED.
2. ASME Y14.41-2003 APPLIES TO DATASET
3. ASME Y14.5M-2009 APPLIES TO DIMENSIONING AND TOLERANCING.
Automatically Flag GD&T Issues (Datum Features)
Datum, Datum Feature, and Datum Feature Simulator