

Inventor 2018 – Model Based Definition

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Positioning

Building the Digital Enterprise

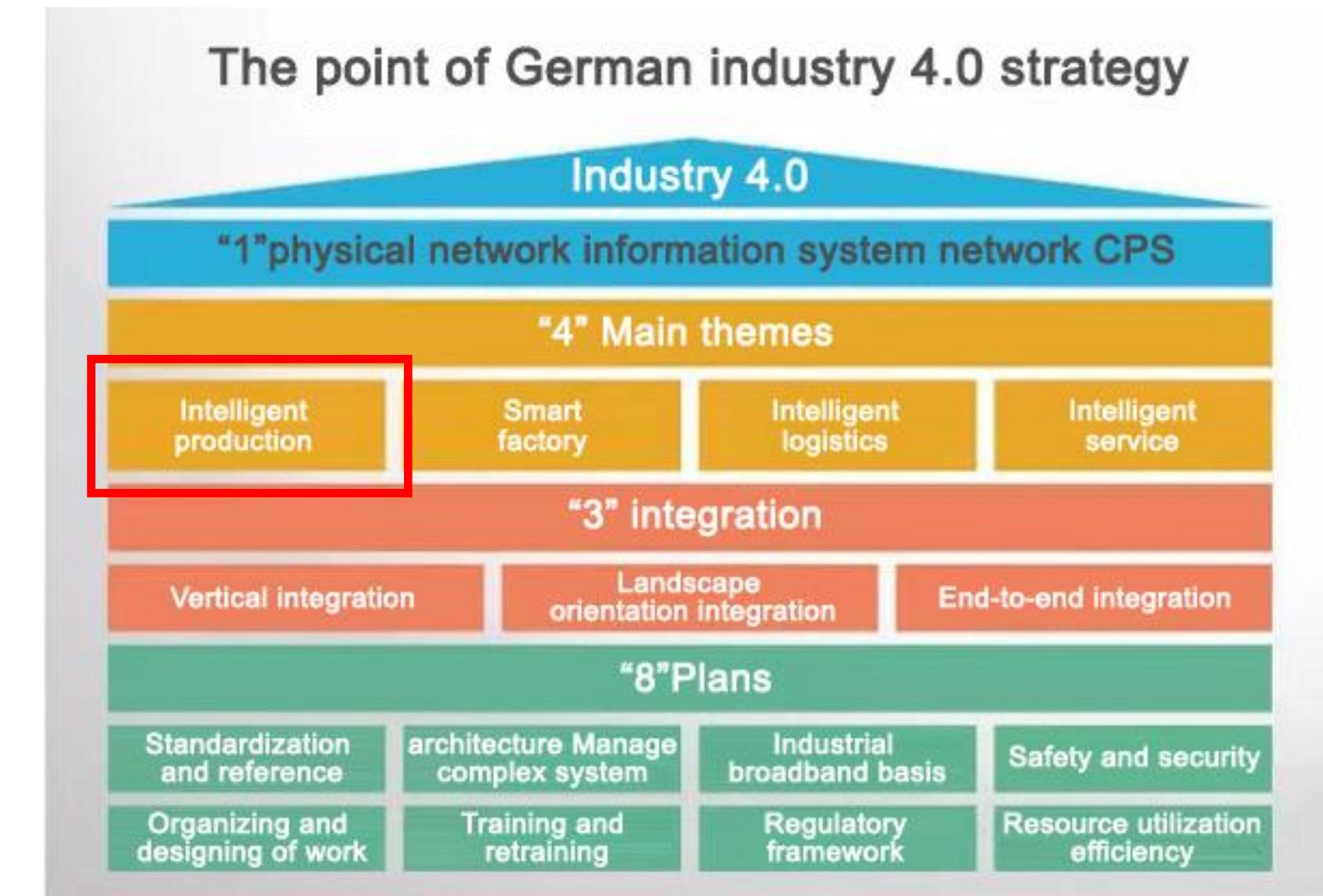


SMARTFACTORY

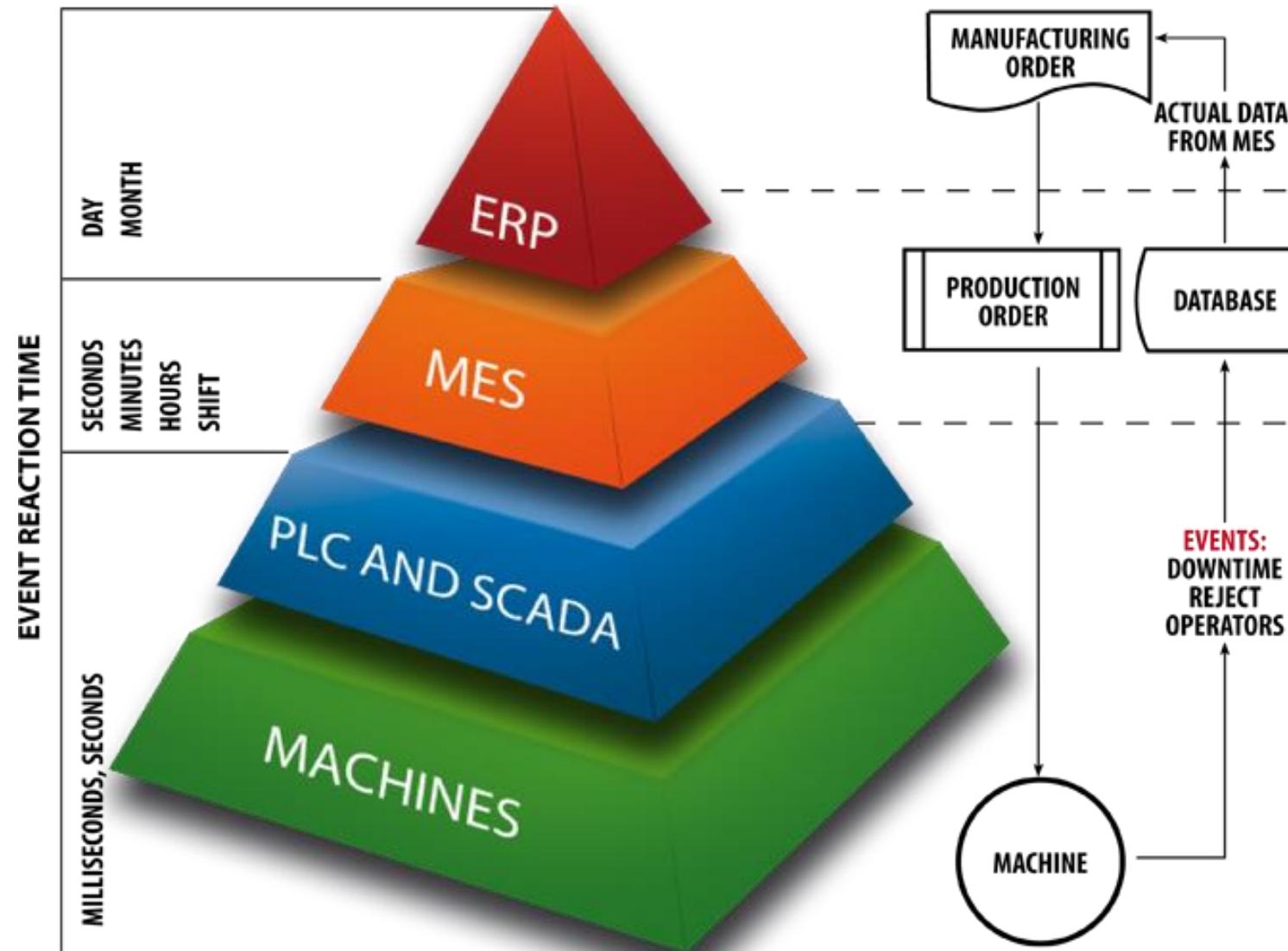
EFFICIENCY THROUGH INNOVATION

Building the Digital Enterprise

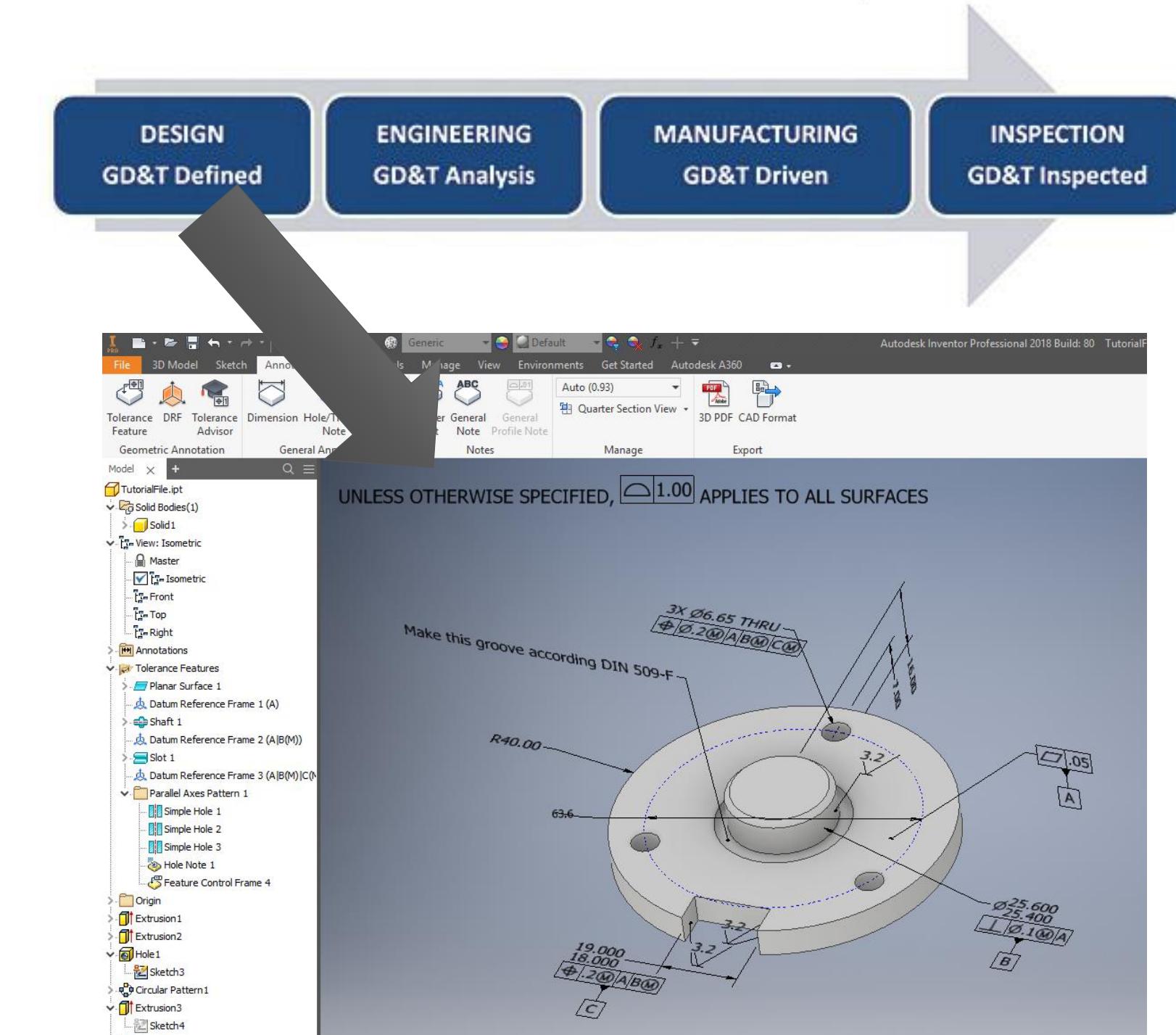
- Industry 4.0 is the current trend of automation and data exchange in manufacturing technologies
- Industry 4.0 creates what has been called a "smart factory"
- Industry 4.0 one of the most important pillars is Intelligent Production



Manufacturing Execution System (MES)



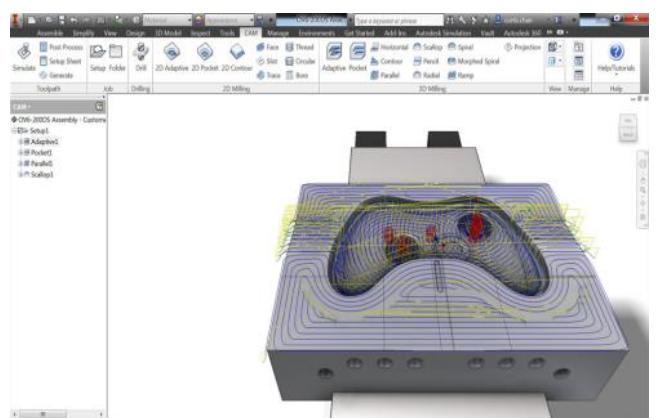
Model Based GD&T Enterprise



Model Based Enterprise



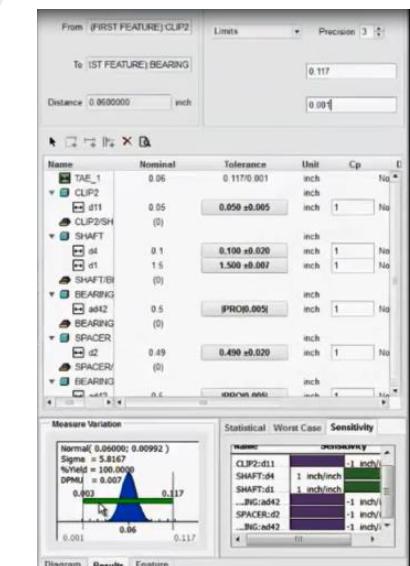
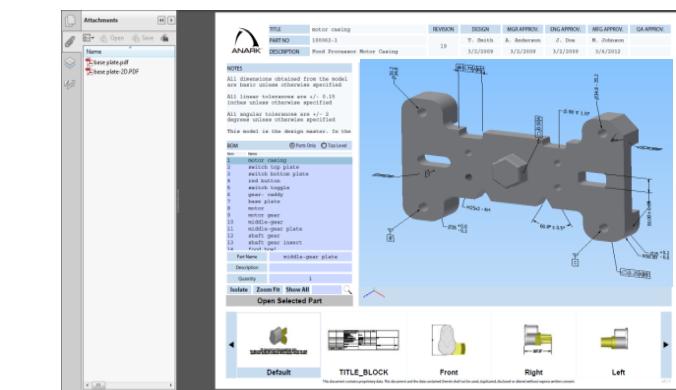
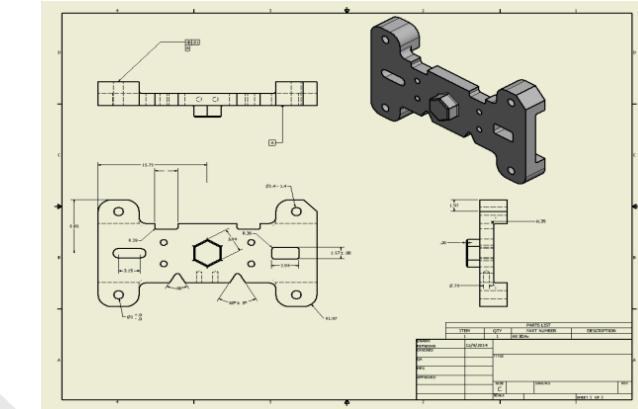
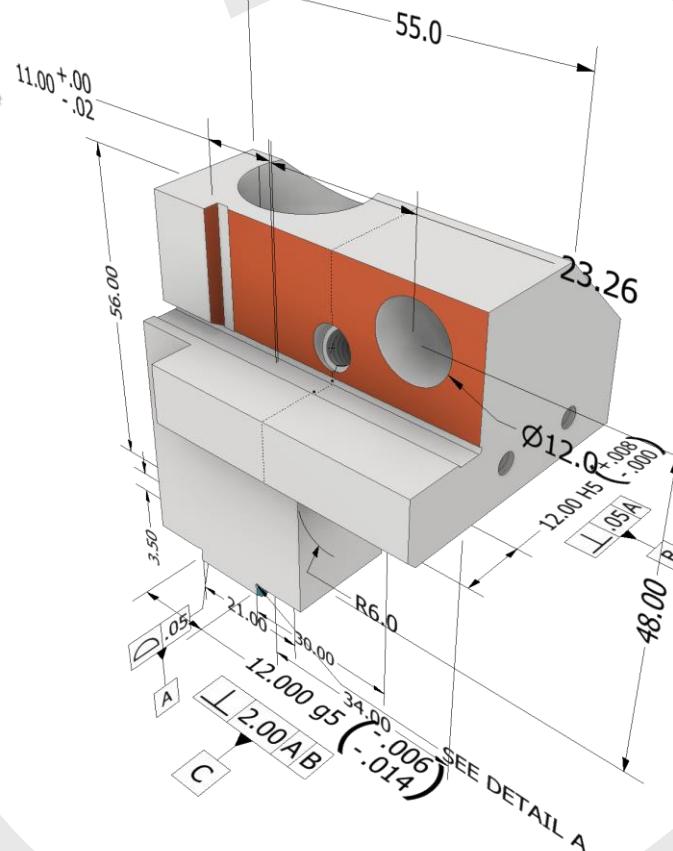
CMM Paths



NC Toolpaths



STEP
AP242





What is MBD?

What is Model Based Definition?

- Model-Based Definition (MBD) refers to a fully defined 3D model of individual components and product assemblies. The types of information can include:
 - Meta Data (iProperties)
 - Tolerance Dimensions
 - Geometric dimensioning and tolerancing (GD&T)
 - Surface Finishing
 - Hole and thread notes
 - General Notes

Back to School

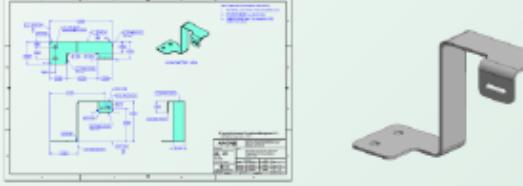
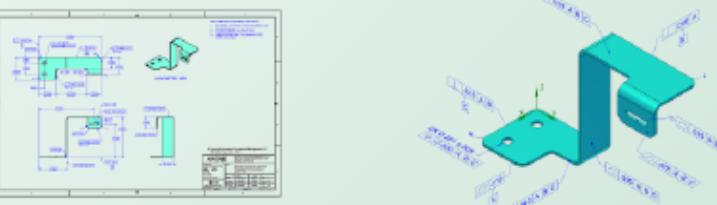
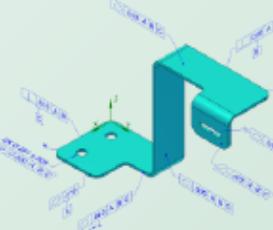
- New standards, New rules
- Don't try to apply the 2D annotation rules in a 3D environment
- Learn the new rules



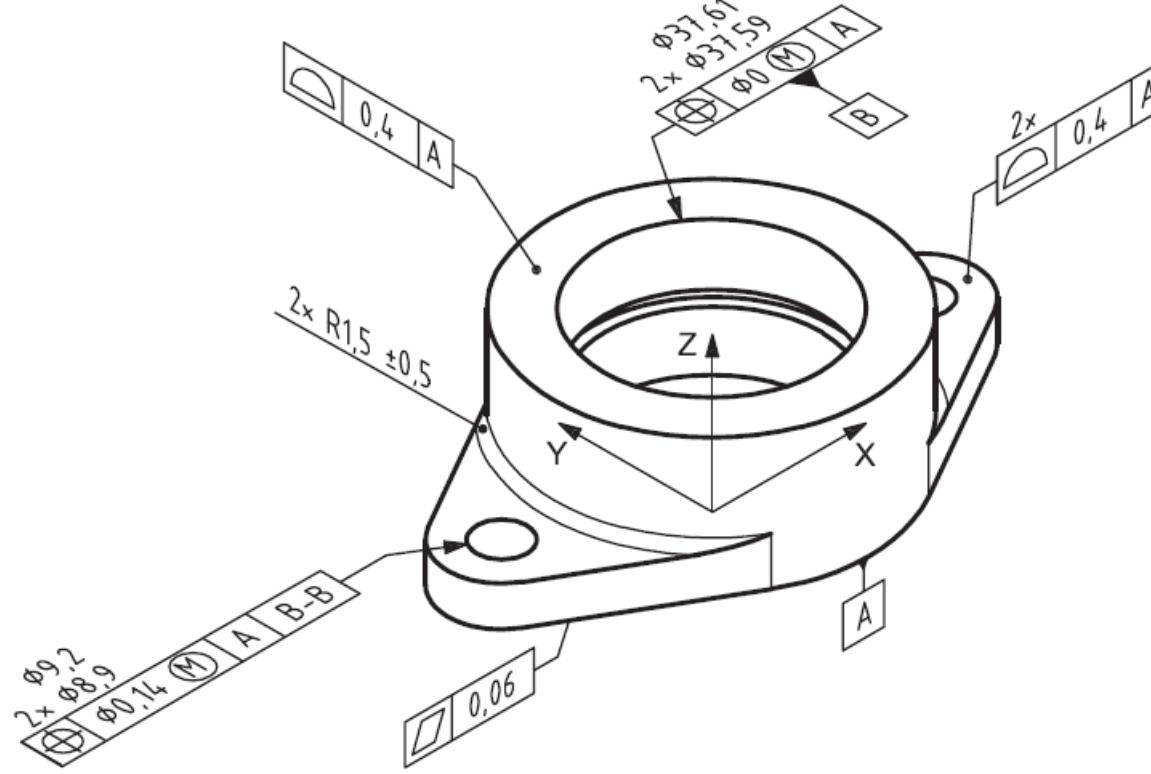
Working according the new standards

- Two types of delivery
 - Model only
 - *Product definition data, including but not limited to notes, parts lists, marking requirements, dimensions, and tolerances, shall be contained or referenced in the data set*
 - Model and drawing
 - *A complete definition of a product shall contain a model and a drawing that may contain orthographic views, axonometric views, or a combination thereof. Annotation may be applied to the model or on the drawing or a combination thereof*

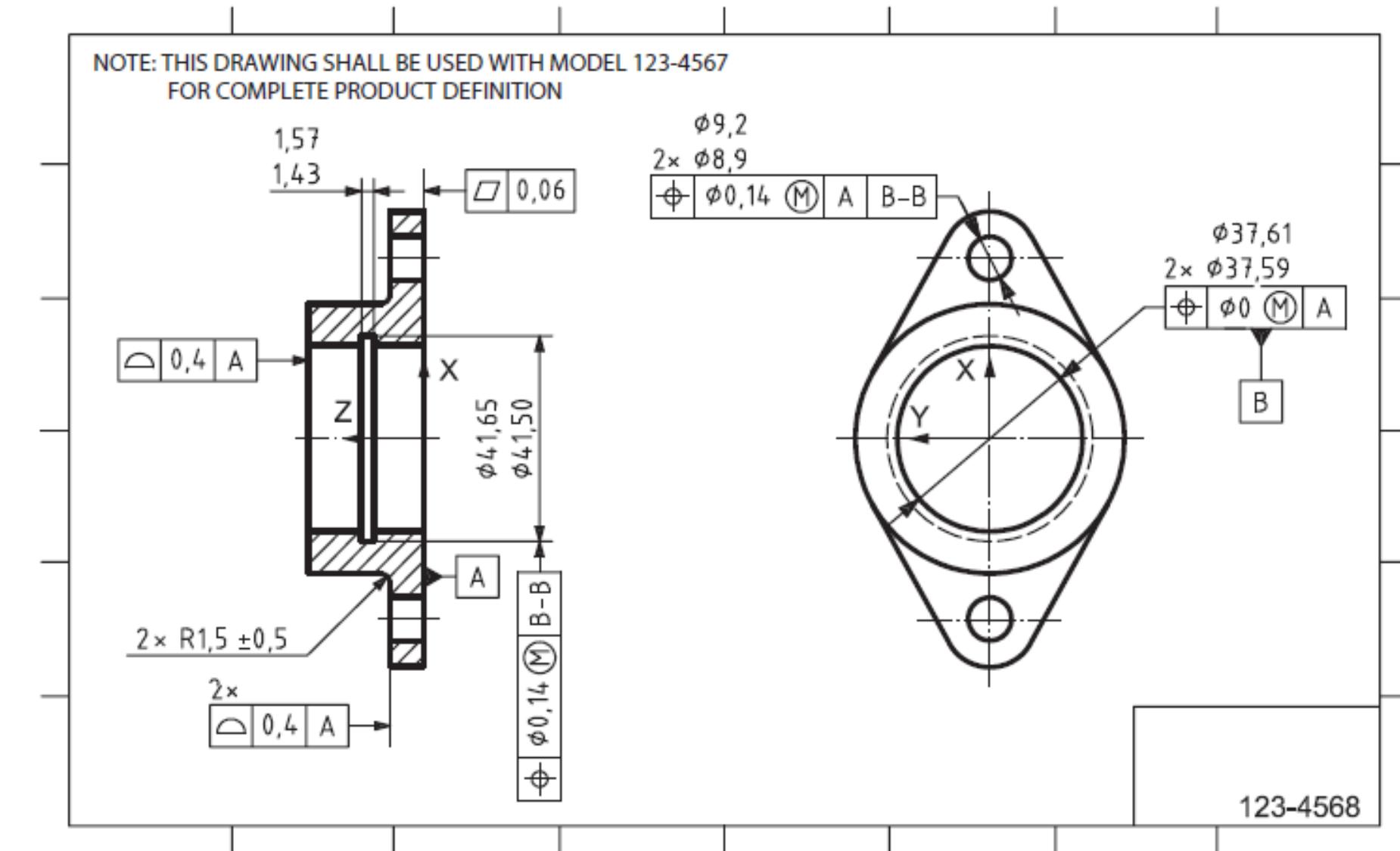
Dataset Classification Code (DCC) Stages

Dataset Class	Master
DCC 1 2D Drawing Complete definition	Drawing 
DCC 2 2D Drawing with 3D Model as reference	Drawing 
DCC 3 2D Drawing + 3D Model Partial definition in both	Both 
DCC 4 2D Drawing + 3D Model Complete definition in both	Both 
DCC 5 3D Model Complete definition	Model 

Working according the new standards



Model 123-4567

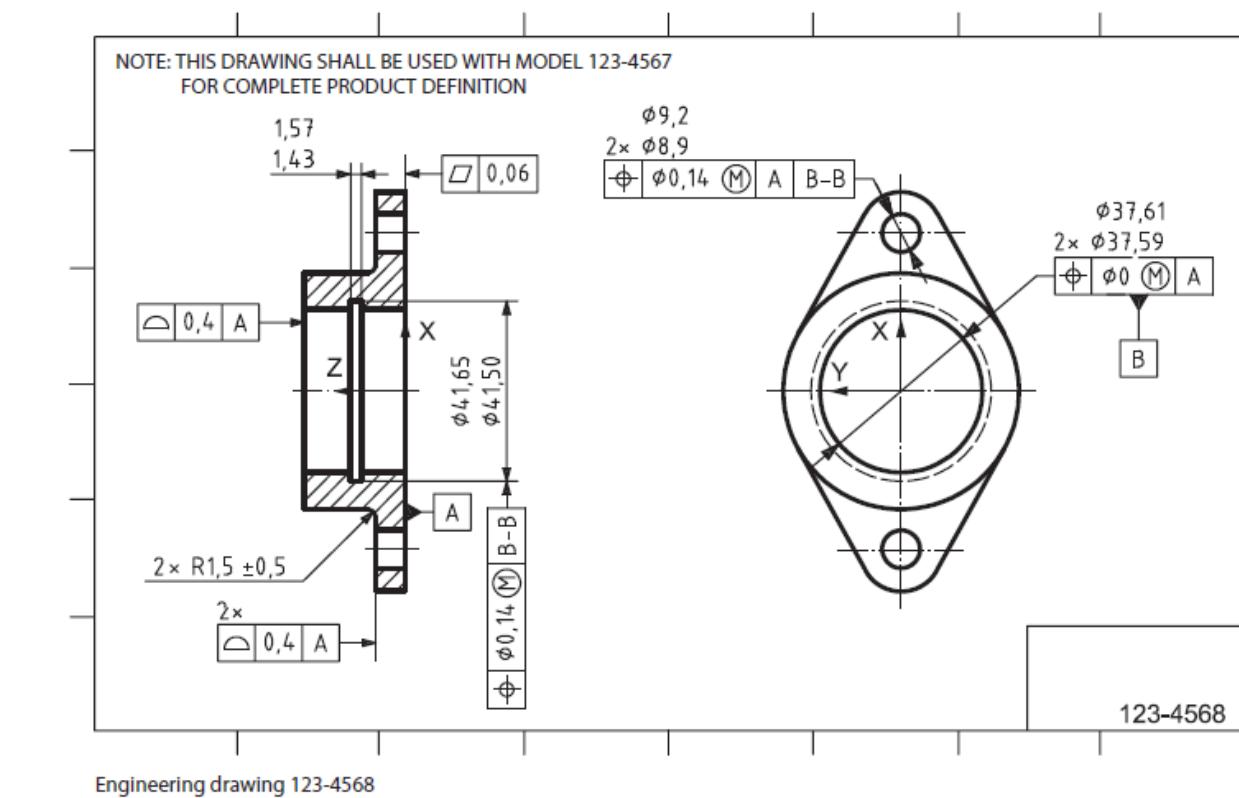


Engineering drawing 123-4568

Source: NEN-ISO 16792_2015 en

New drawing requirements

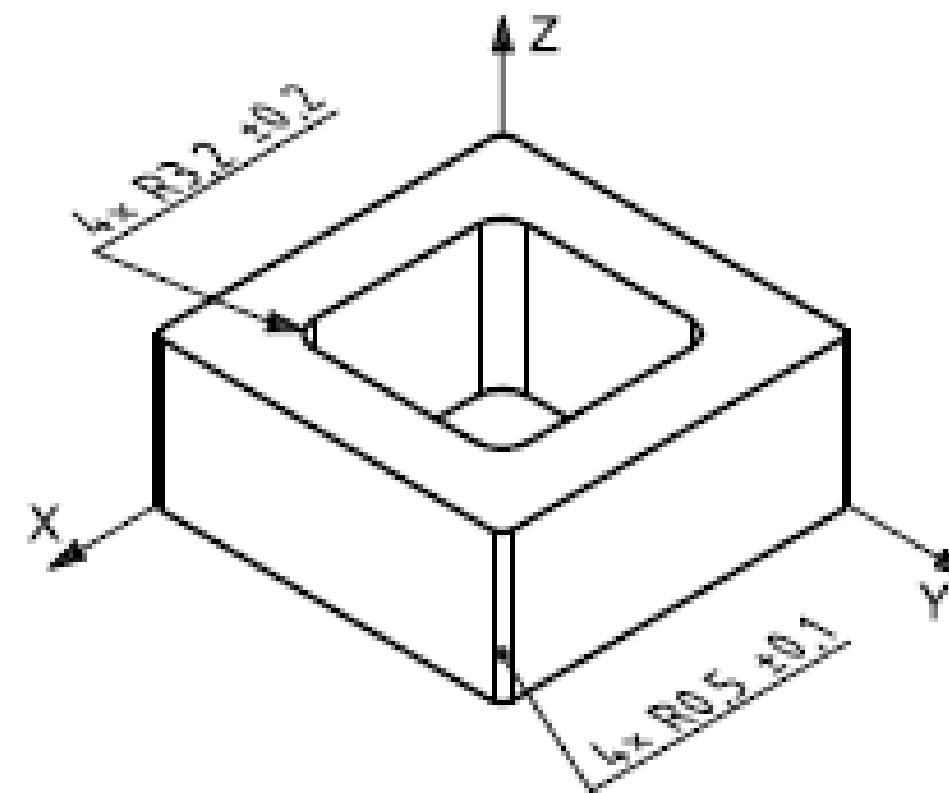
- Features can be shown in their actual position, without foreshortening or alignment, when the section is made from an axonometric view



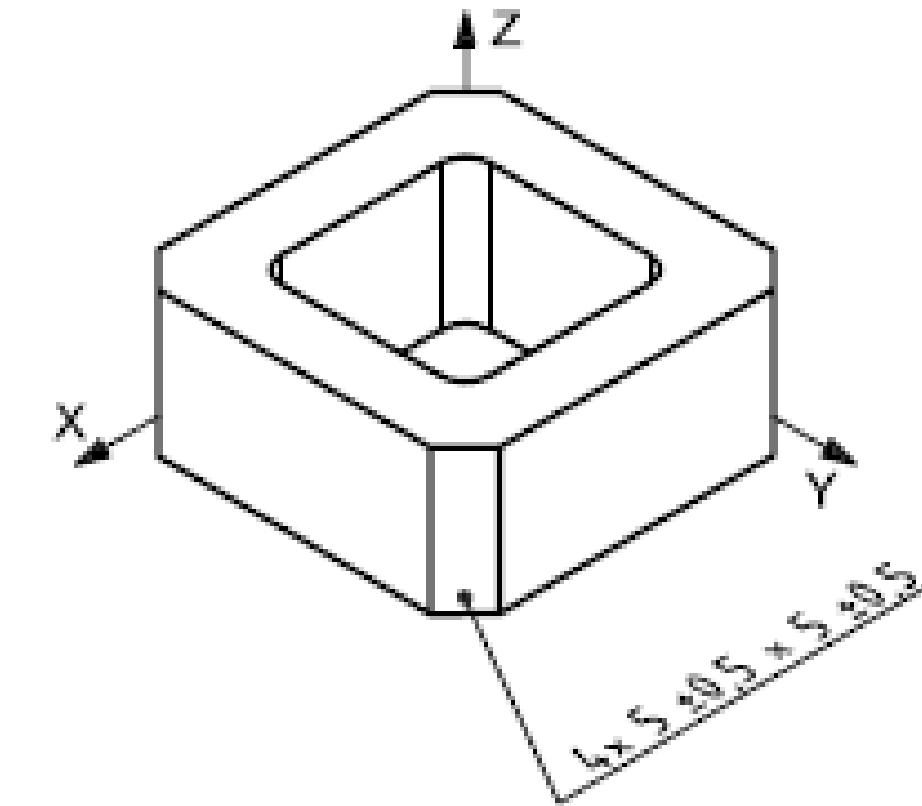
Source: NEN-ISO 16792_2015 en

Example's of how to apply the new rules

- Fillet and chamfer notes



a) Fillets and rounds

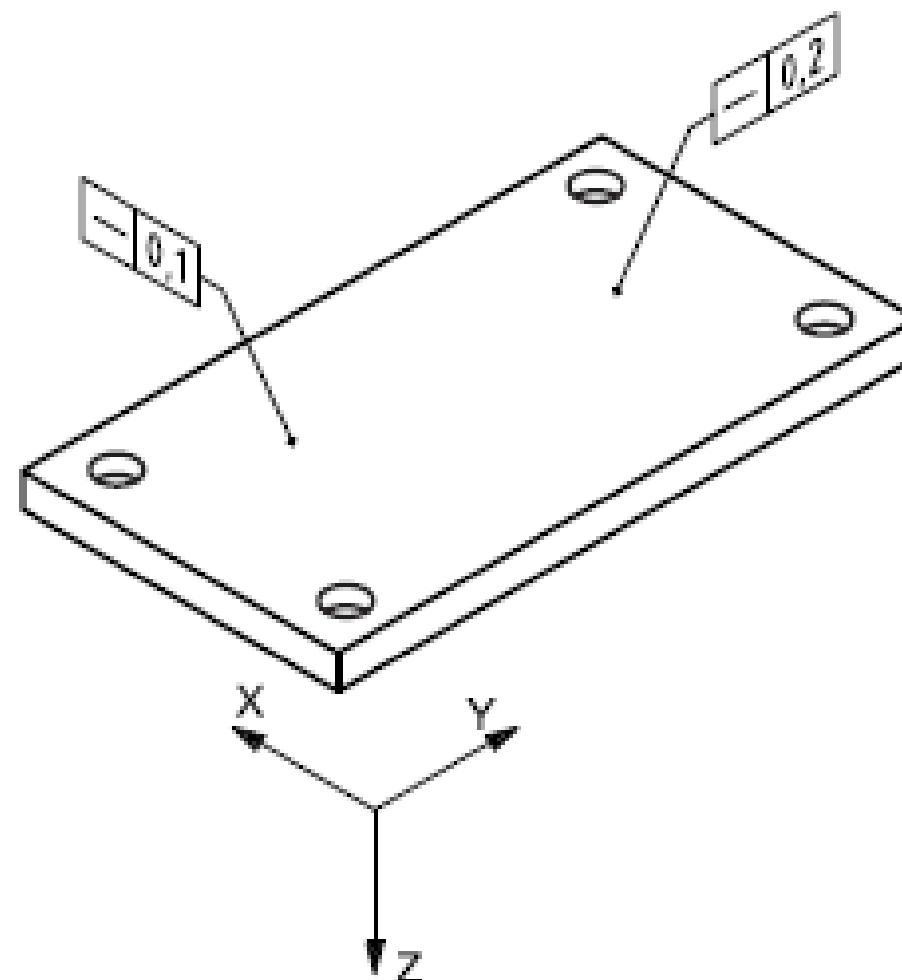


b) Chamfer — Equal offsets

Source: NEN-ISO 16792_2015 en

Example's of how to apply the new rules

- Straightness defined by annotation plane direction



a) Placement and attachment

Source: NEN-ISO 16792_2015 en



Functionality

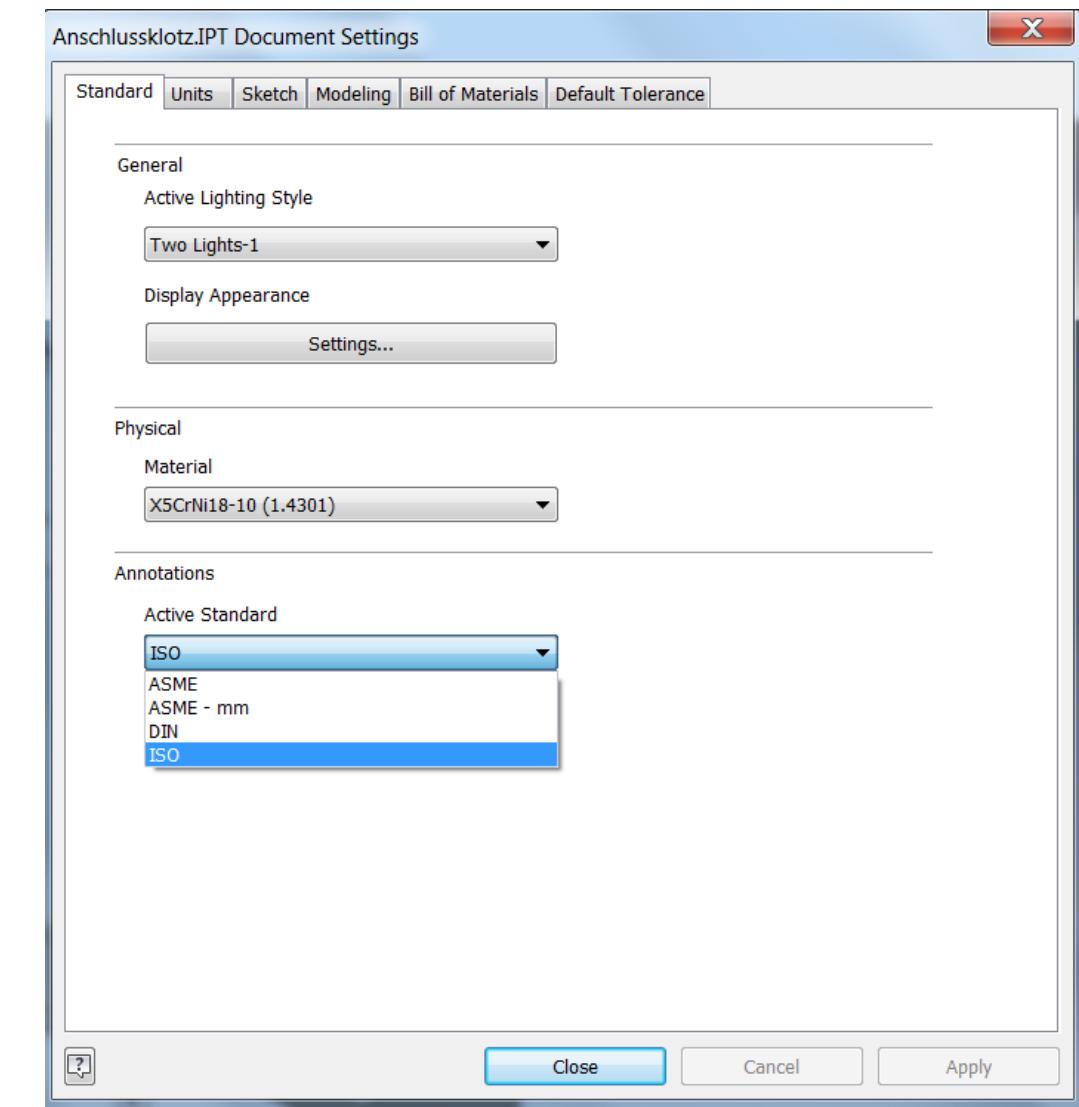
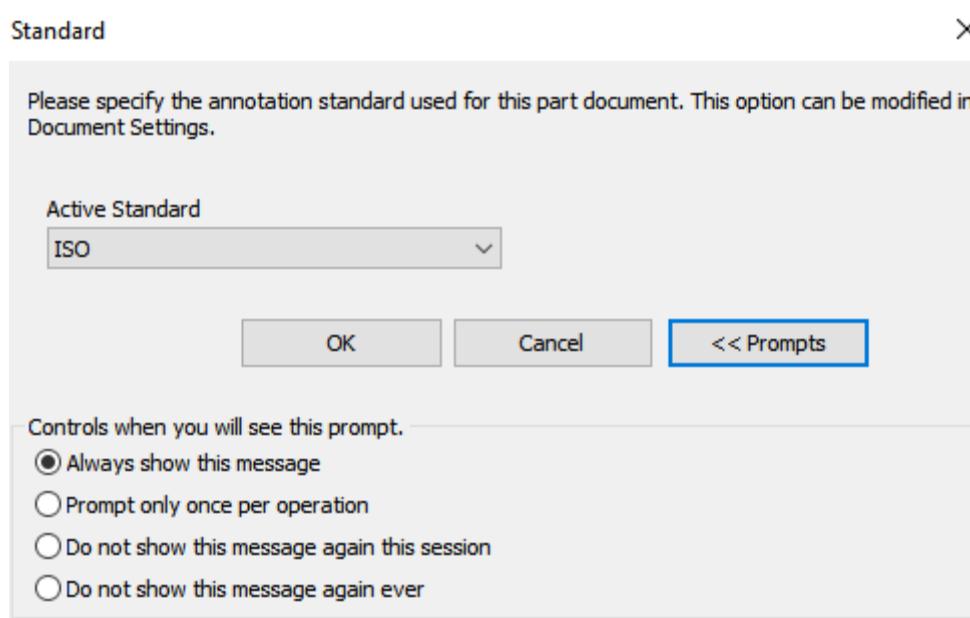
Supported...

- Environment
 - Part
 - Sheet Metal
 - Folded
 - Unfolded
 - In-context part edit
- Standards
 - ASME Y14.41:2003
 - DIN ISO 16792:2015 en
 - ISO 1101:2017

Style Definition

- Annotation style definition in Document Settings

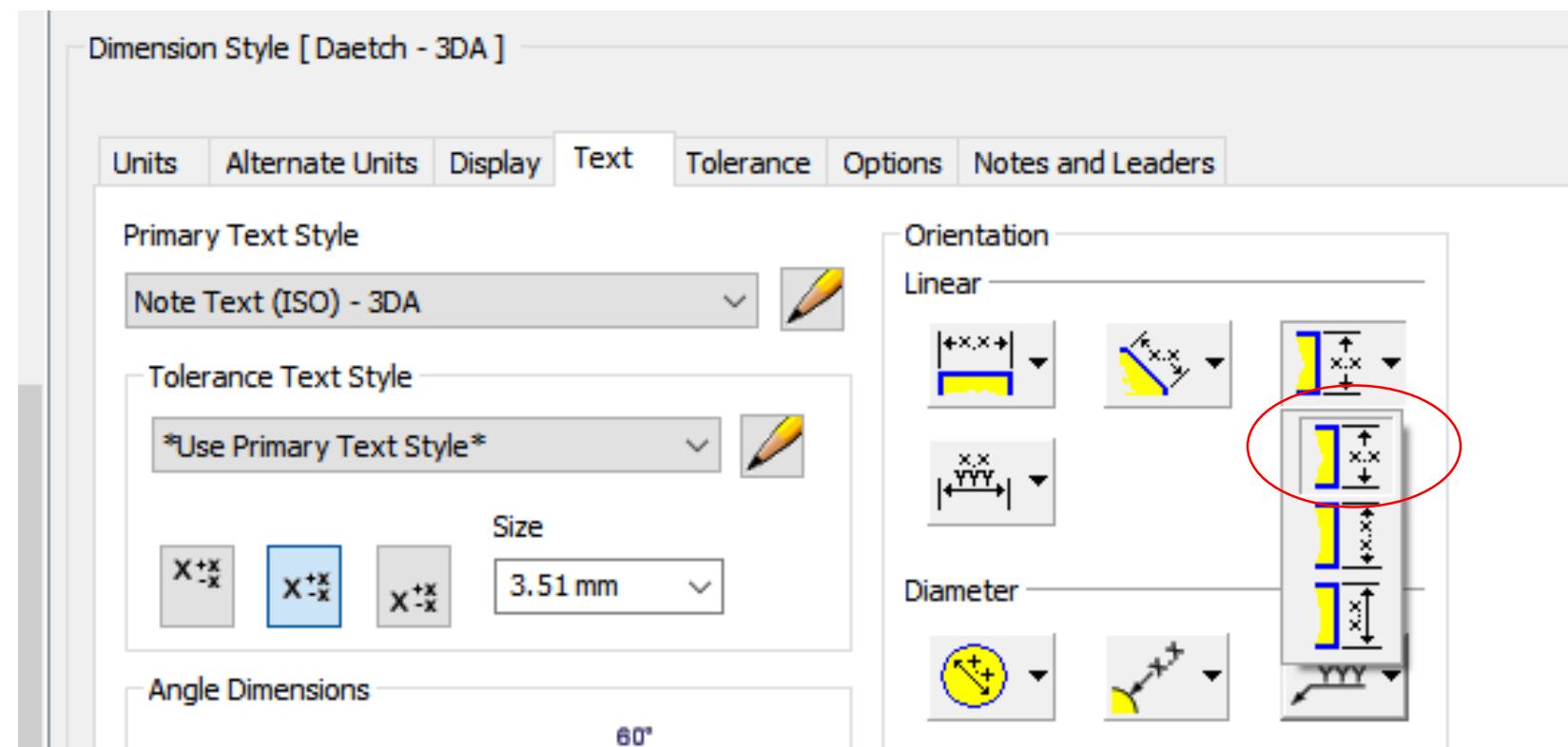
- Prompted to set for previous version files on start with annotation



- Change styles in drawing environment

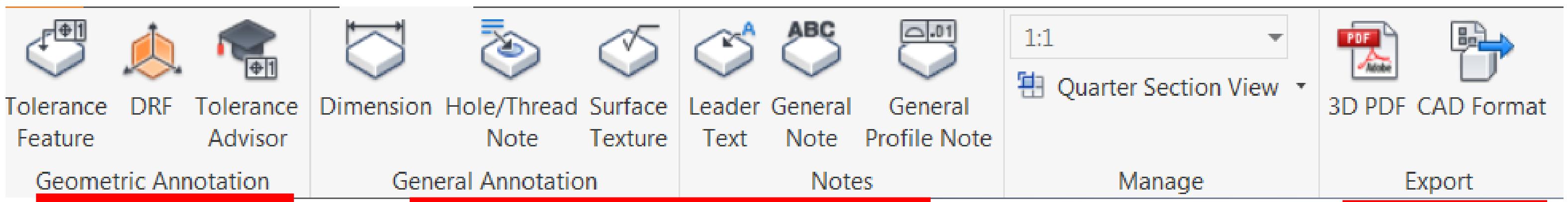
Style Management

- Edit the original “Standards” style like ISO, ASME or DIN with the 3DA extension
- Sub Styles can be created from scratch.



MBD Functionality

- Check the health of your tolerance scheme



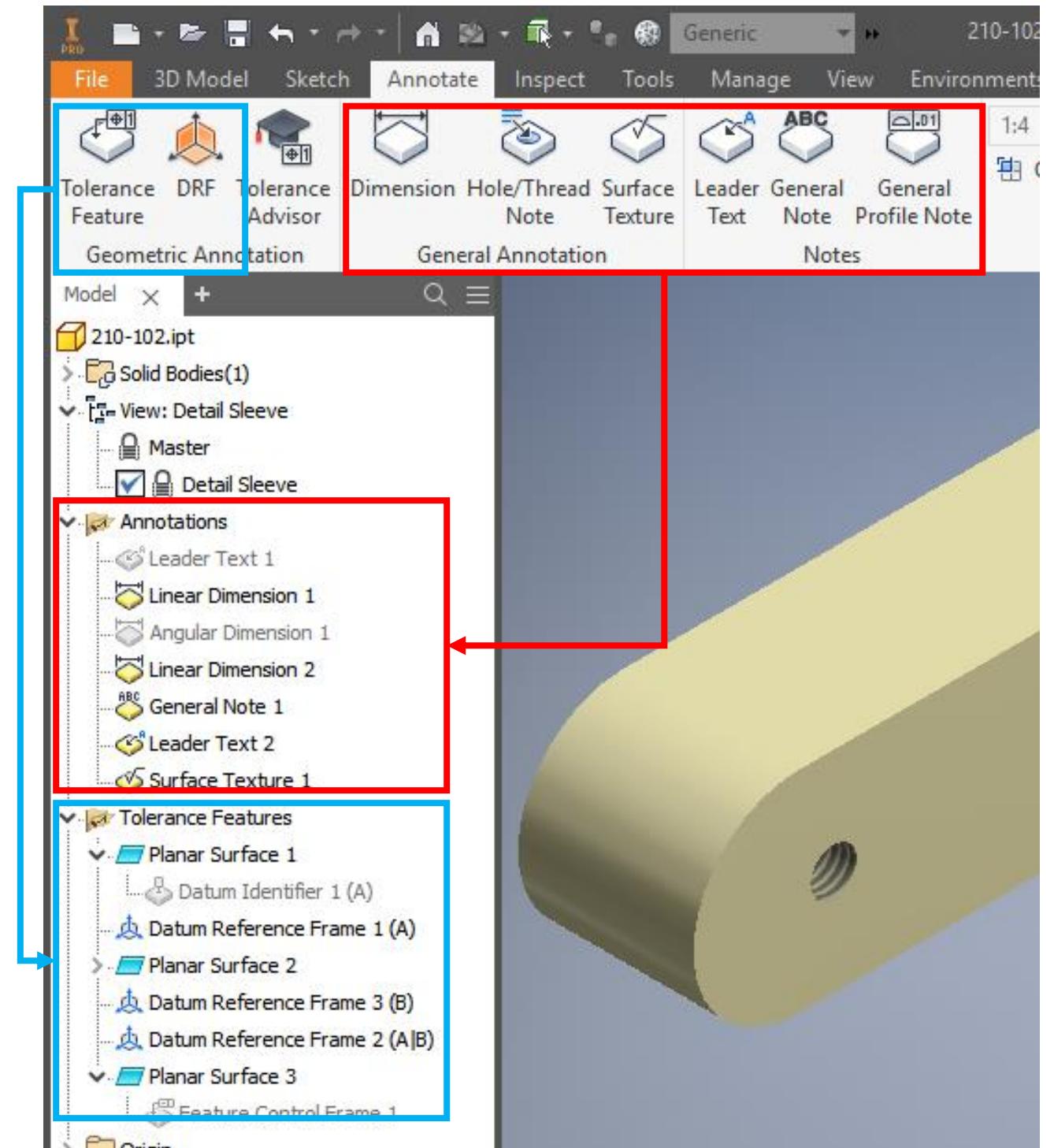
- Includes GD&T/GPS
- Standards-based validation & advice
- Automatic recognition of tolerance features
- Typically for downstream consumption
- Powered by 

- No GD&T/GPS
- No validation & advice
- Manual creation
- Typically for documentation & mark-up

- Downstream consumption

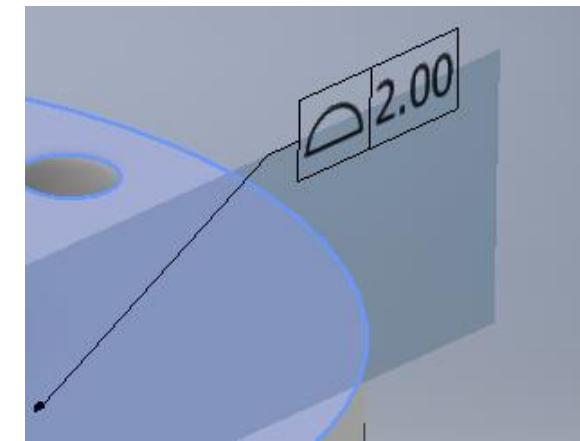
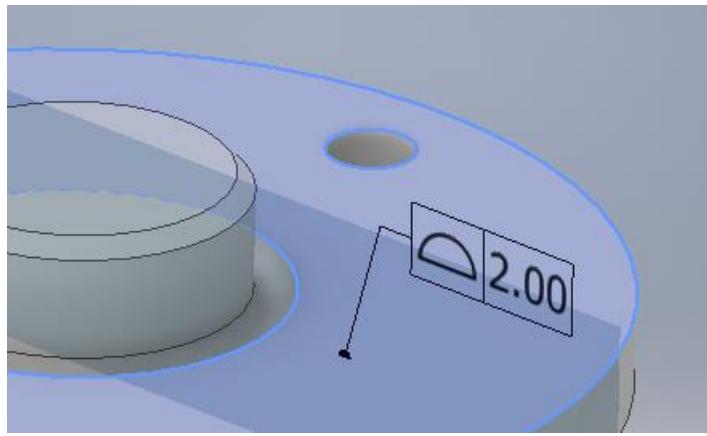
Feature Tree – Tolerance Features Folder

- Annotations folder
 - General Annotations
 - Notes
- Tolerance Features
 - Geometric Annotations
- Control visibility

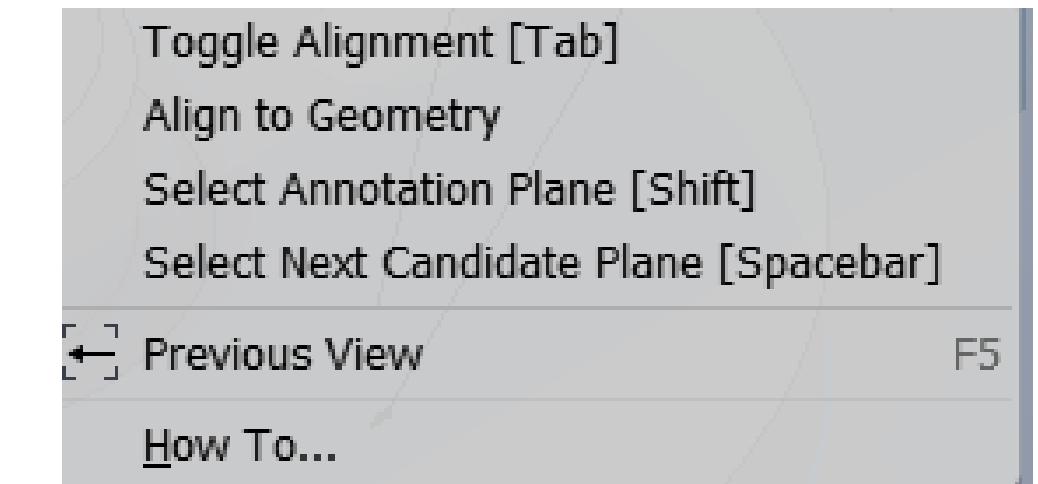
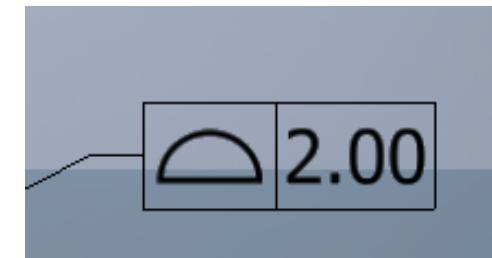
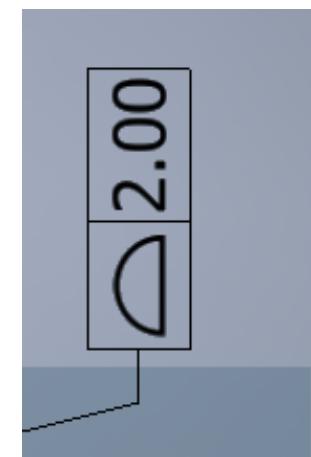


Tolerance Feature Placement – Orientation

- Spacebar to toggle between the next candidate plane



- Tab to toggle between alignment*

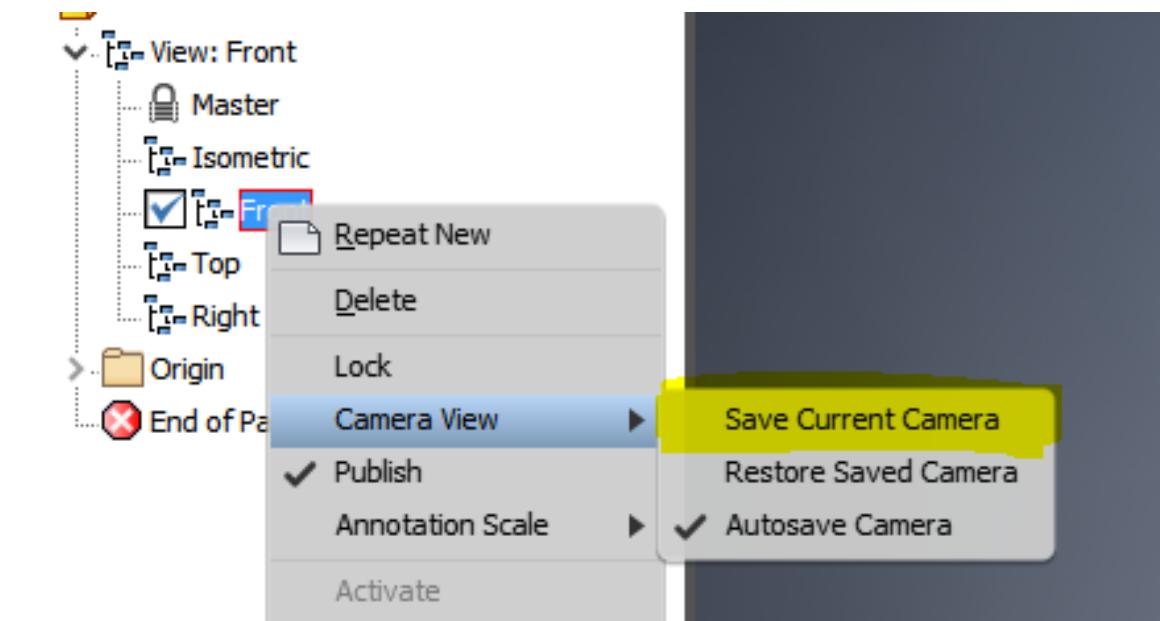


- Shift for selecting an annotation plan

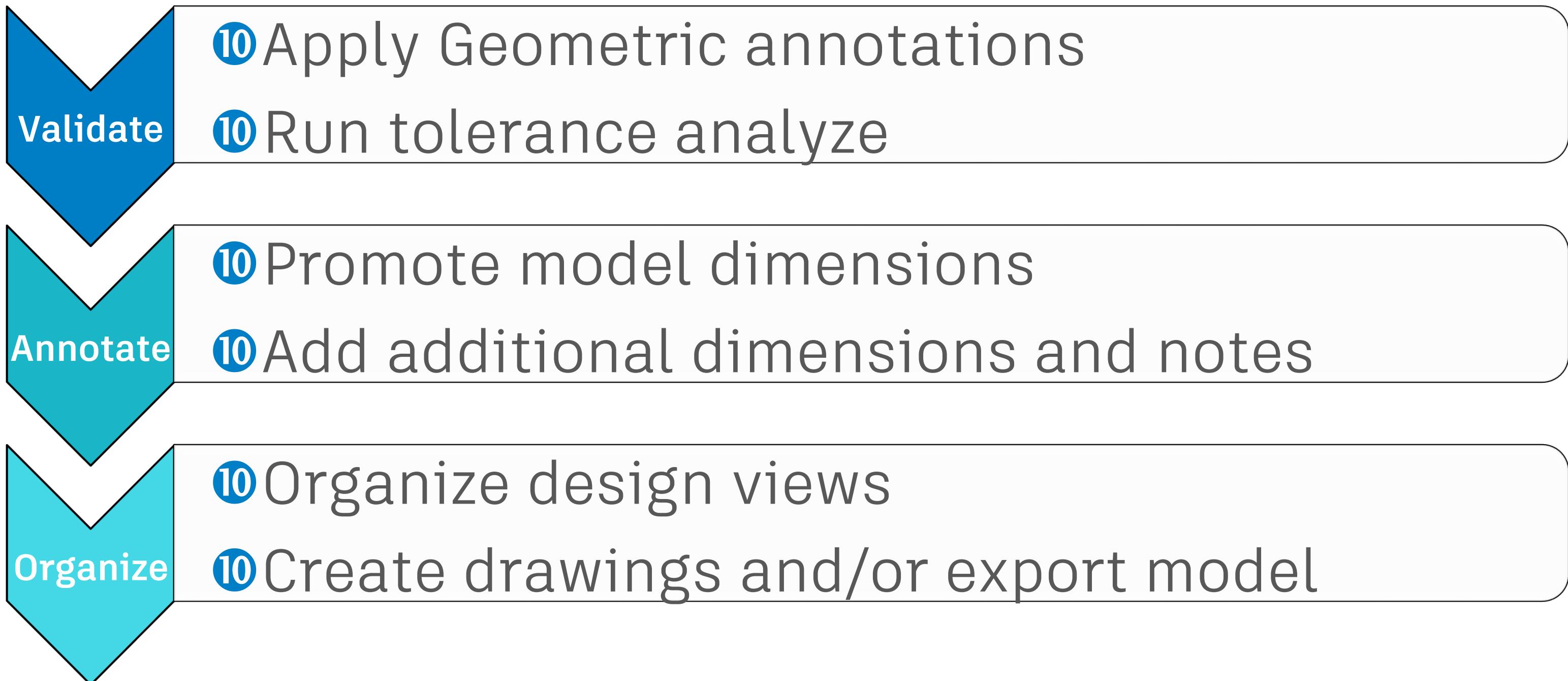
* Availability depend on the style settings. Only available with “inline – horizontal” orientation setting. This is set by default for the ASME standard.

Design Views

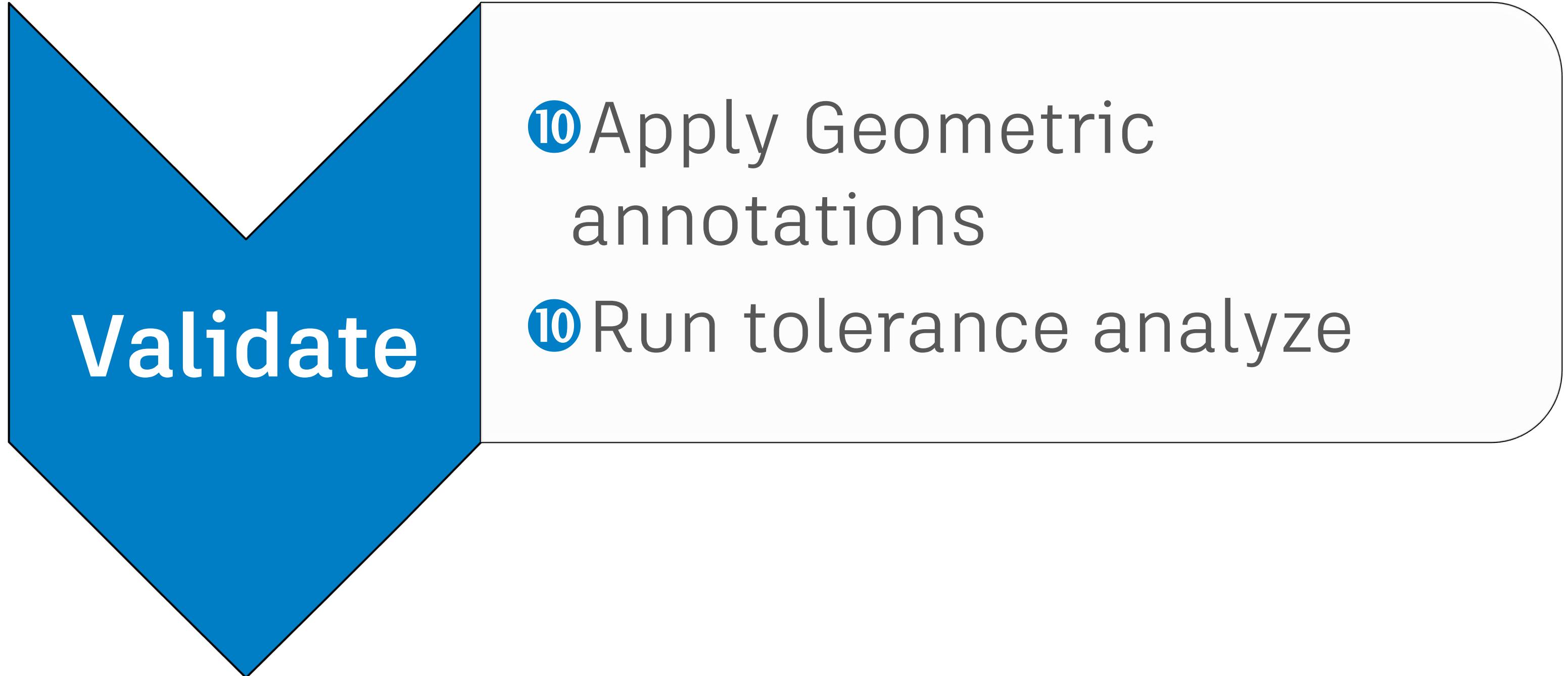
- Lock your view
 - New annotations are automatically hidden
- Change your camera view to “save current camera”
 - Change in your template
- Publish
 - the view will be automatically marked for publication to 3d PDF, before creating you can adjust this manually
- Annotation scale
 - Set your annotation (text) display height. “Auto” is the default.



Workflow

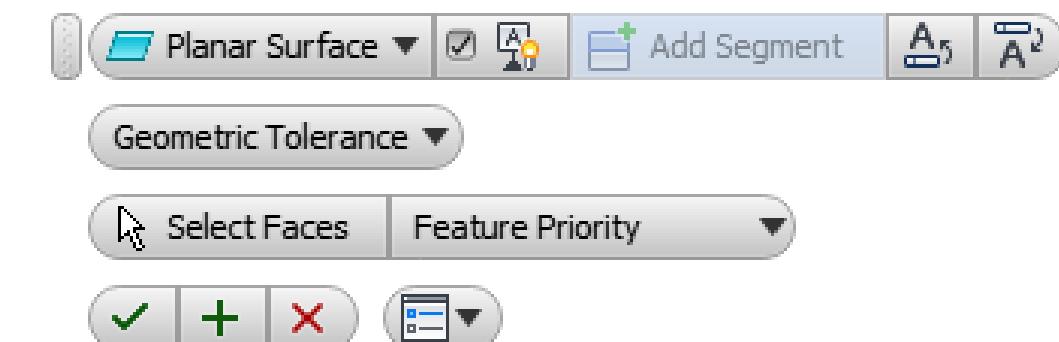
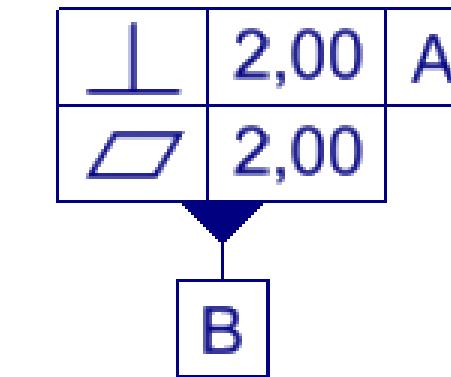
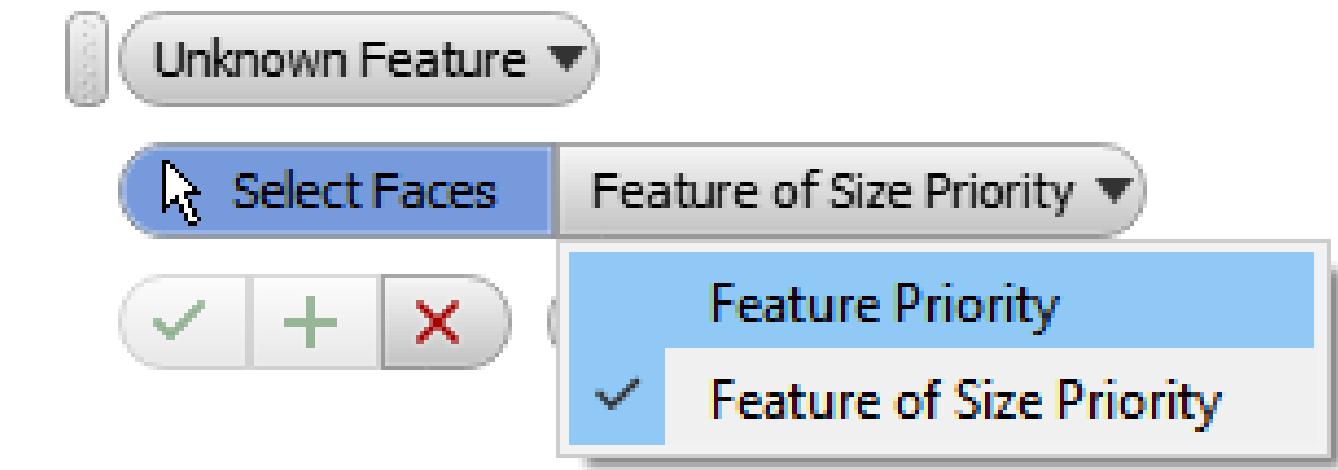


Workflow



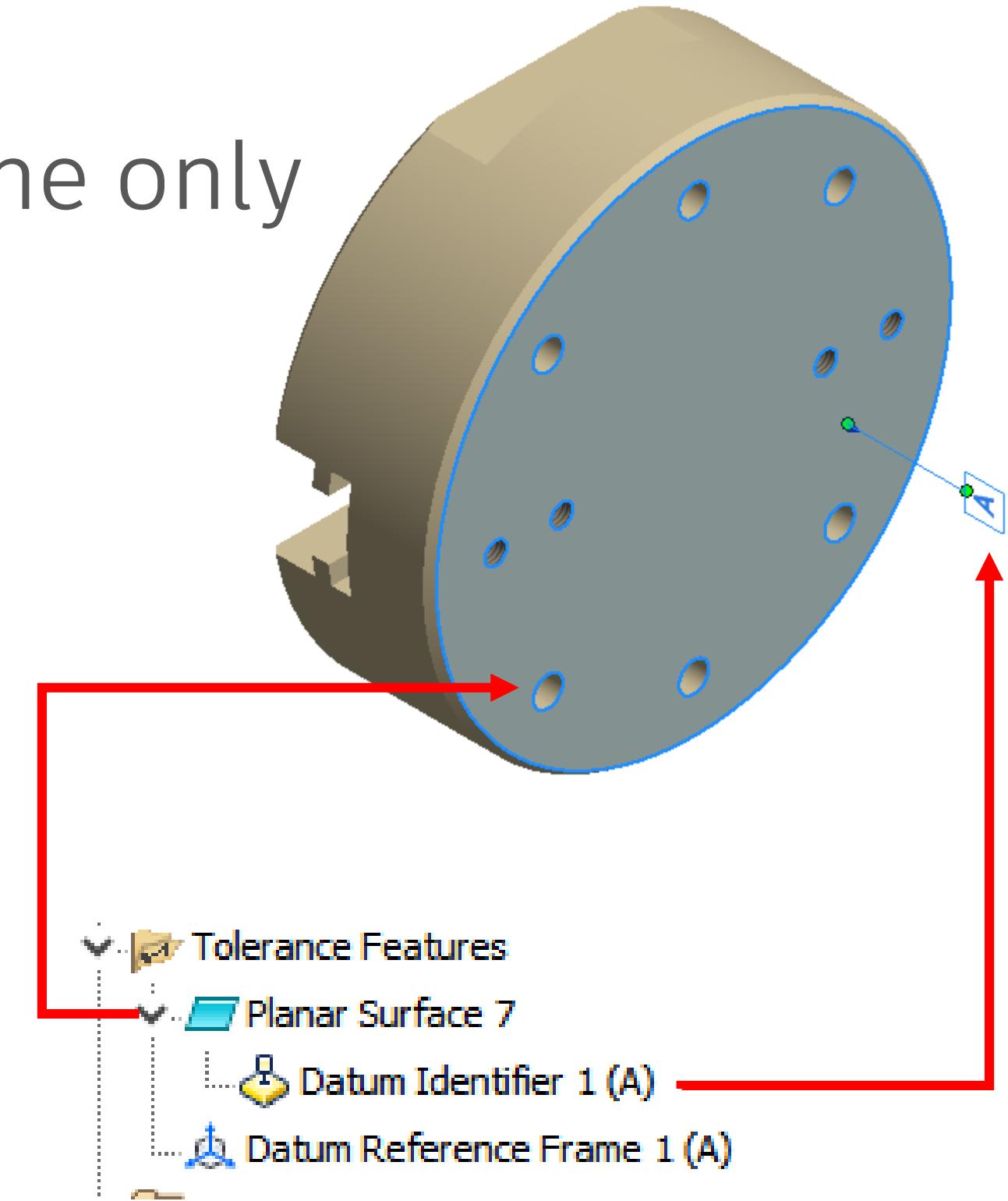
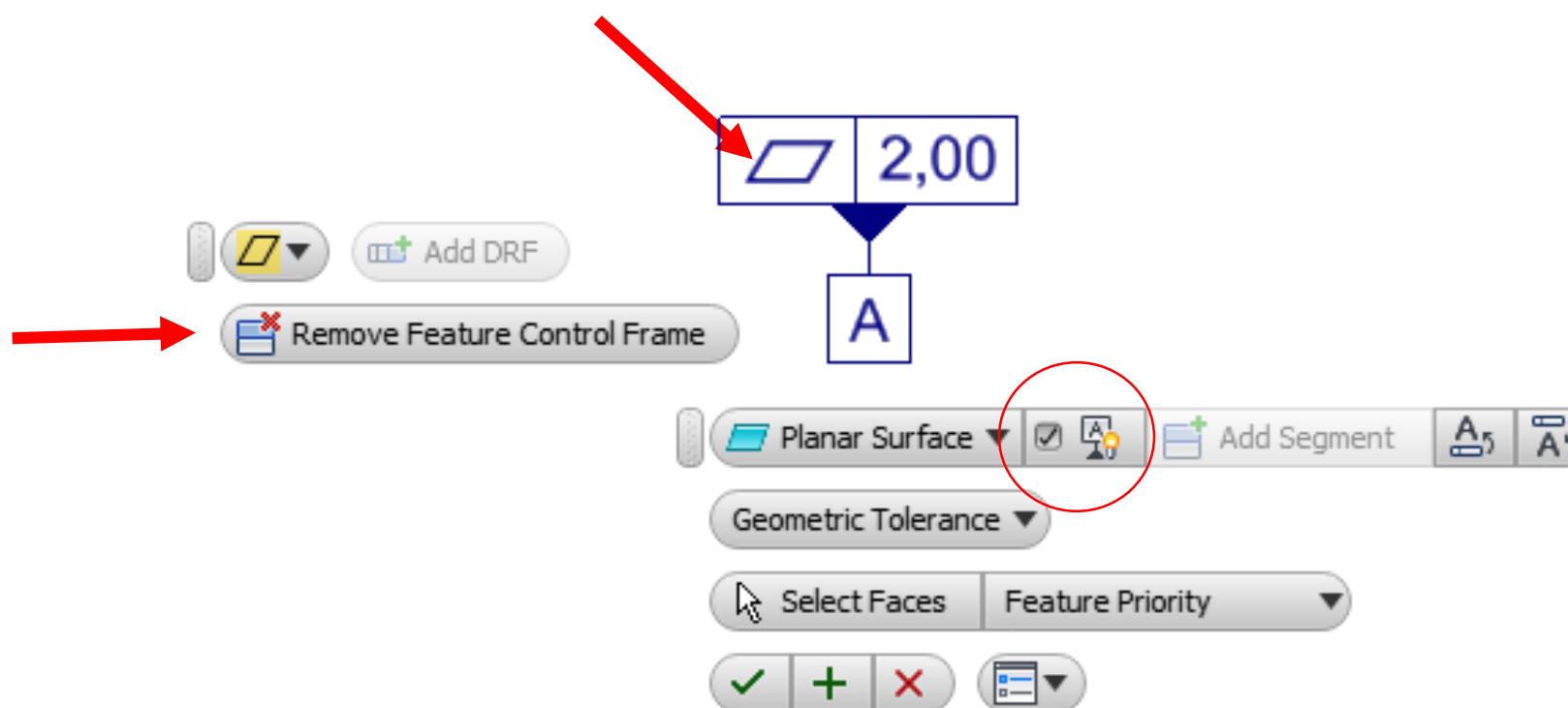
Tolerance Feature

- Define Tolerance feature
 - Geometric
 - Dimensional
 - Linear
 - Hole notes
- Datum reference plane automatically created
- One Tolerance annotation per face
 - Multiple segments



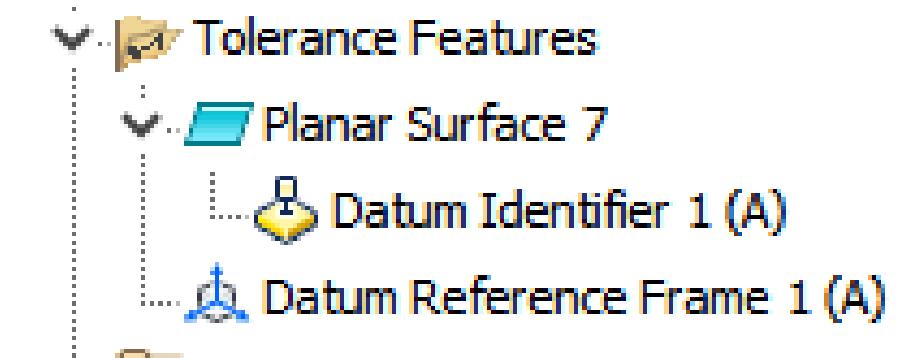
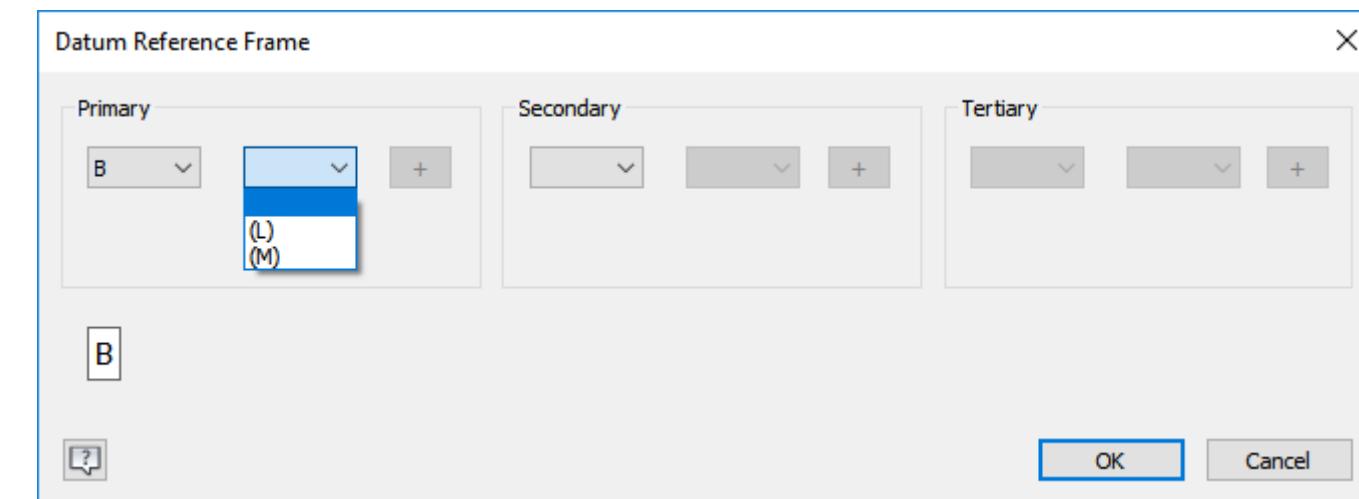
Tolerance Feature

- Define a datum reference plane only
 - Add tolerance feature
 - Select Feature control frame
 - Remove Feature Control Frame



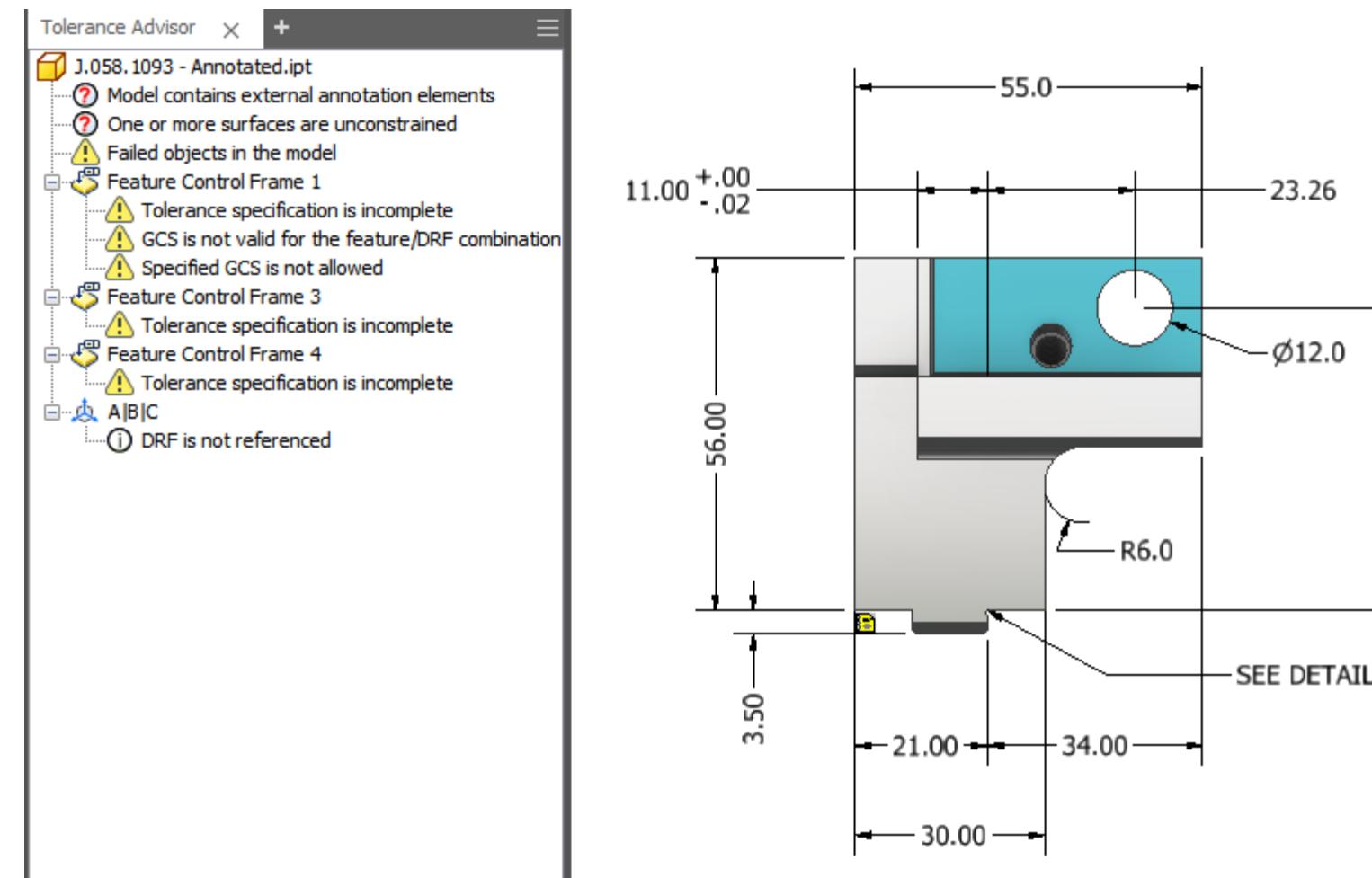
DRF (Datum Reference Frame)

- Define new DRF based on existing Datum Identifiers
- Add material condition
 - Minimum
 - Maximum



Tolerance Advisor

- Checks the health of your tolerance scheme
- Analyses Geometric Dimensions and tolerances



Example
“Validate”

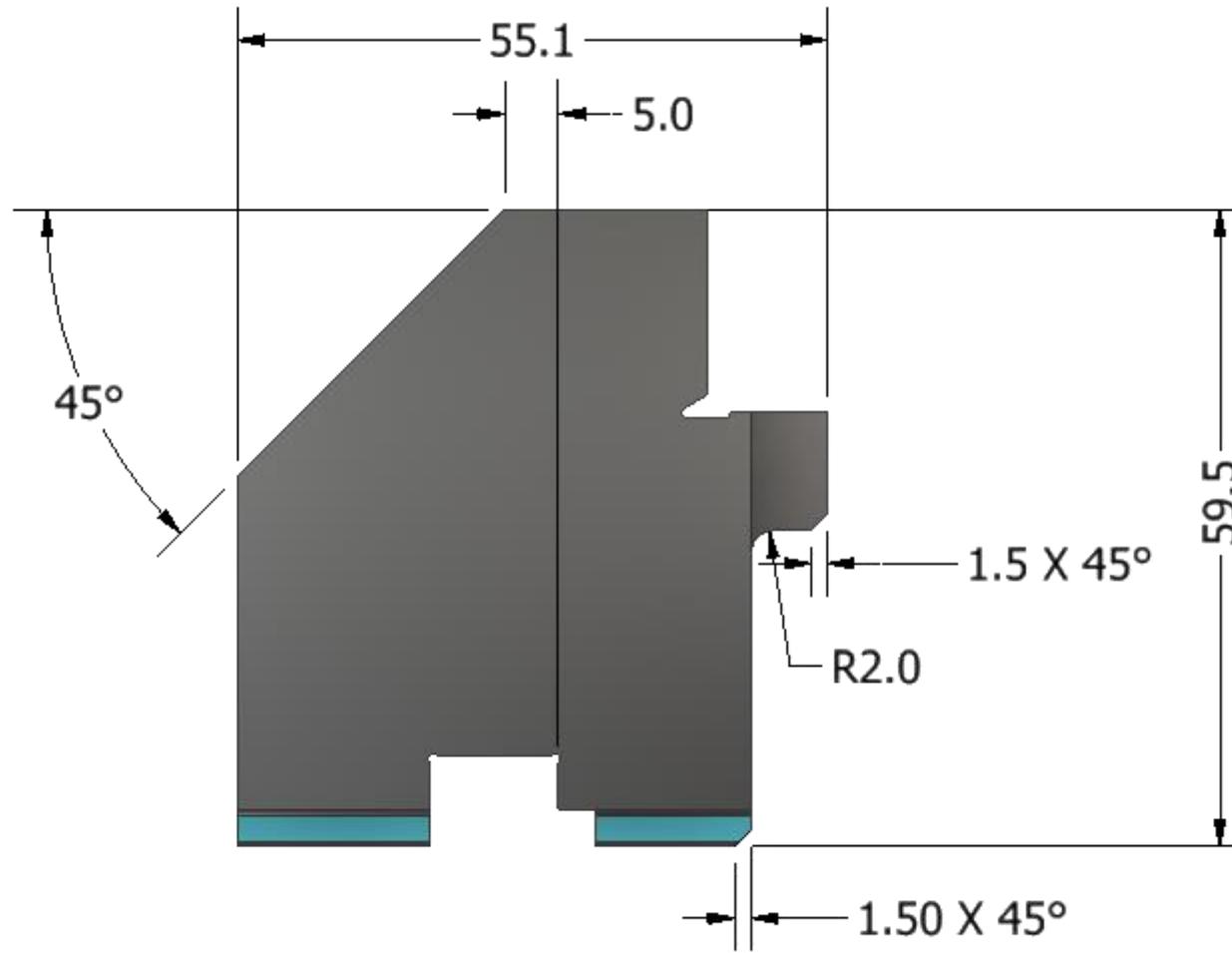
Workflow

Annotate

- ⑩ Promote model dimensions
- ⑩ Add additional dimensions and notes

Dimensions

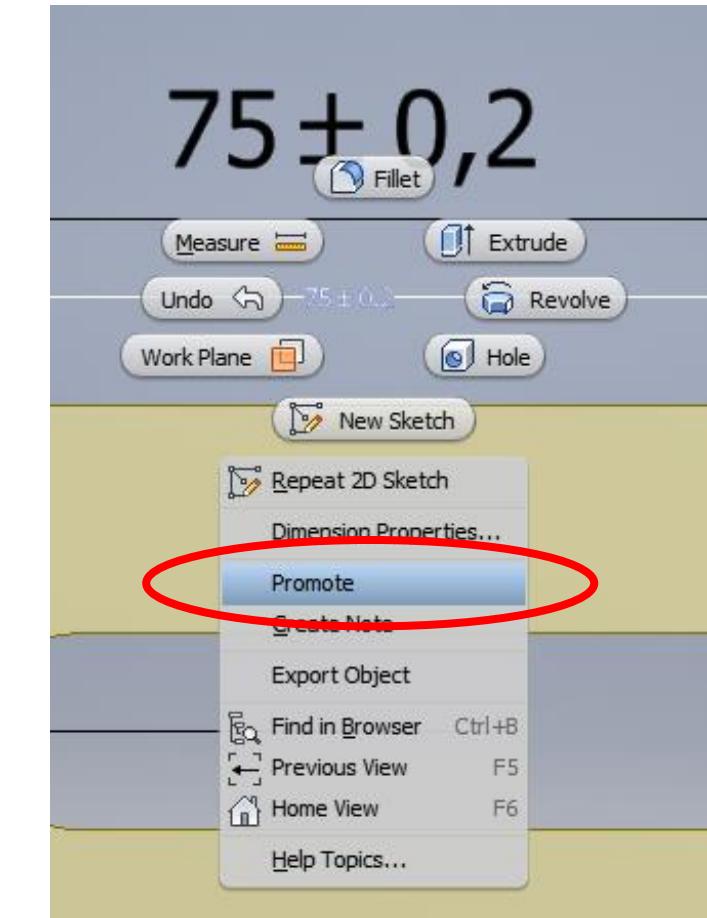
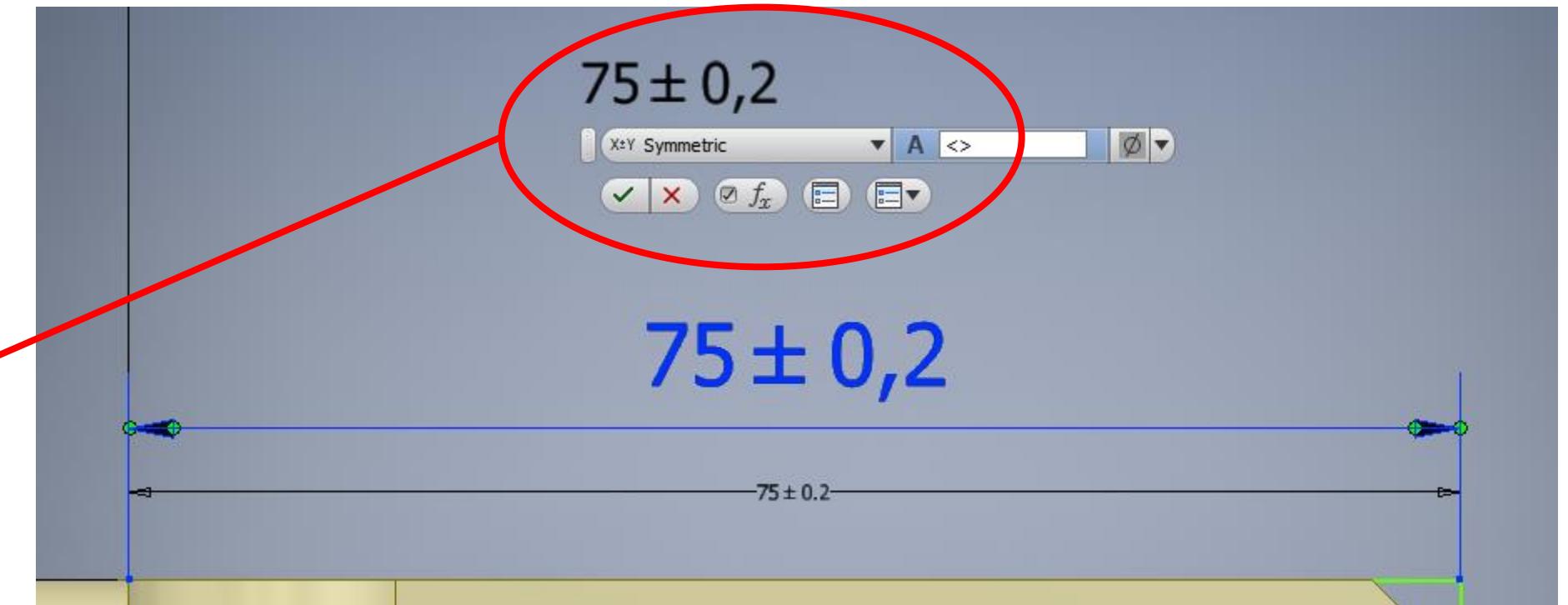
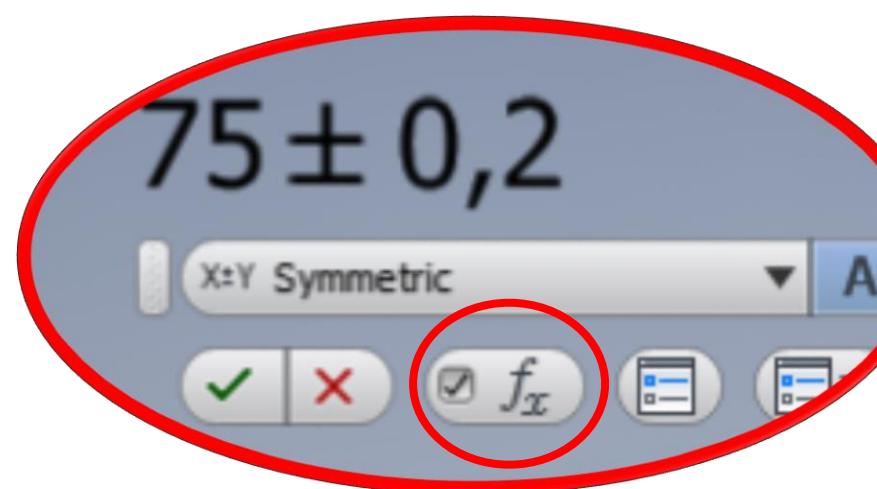
- Linear
- Radial
- Diametric
- Angular



- NOTE: no midpoint and quadrant snapping, create work point before dimensioning
- TIP: dimension in a isometric view, not an orthographic view

Dimensions

- Promote Dimensions incl Tolerance
 - Select feature
 - Show dimensions
 - Select dimension
 - Promote
- Sync tolerance

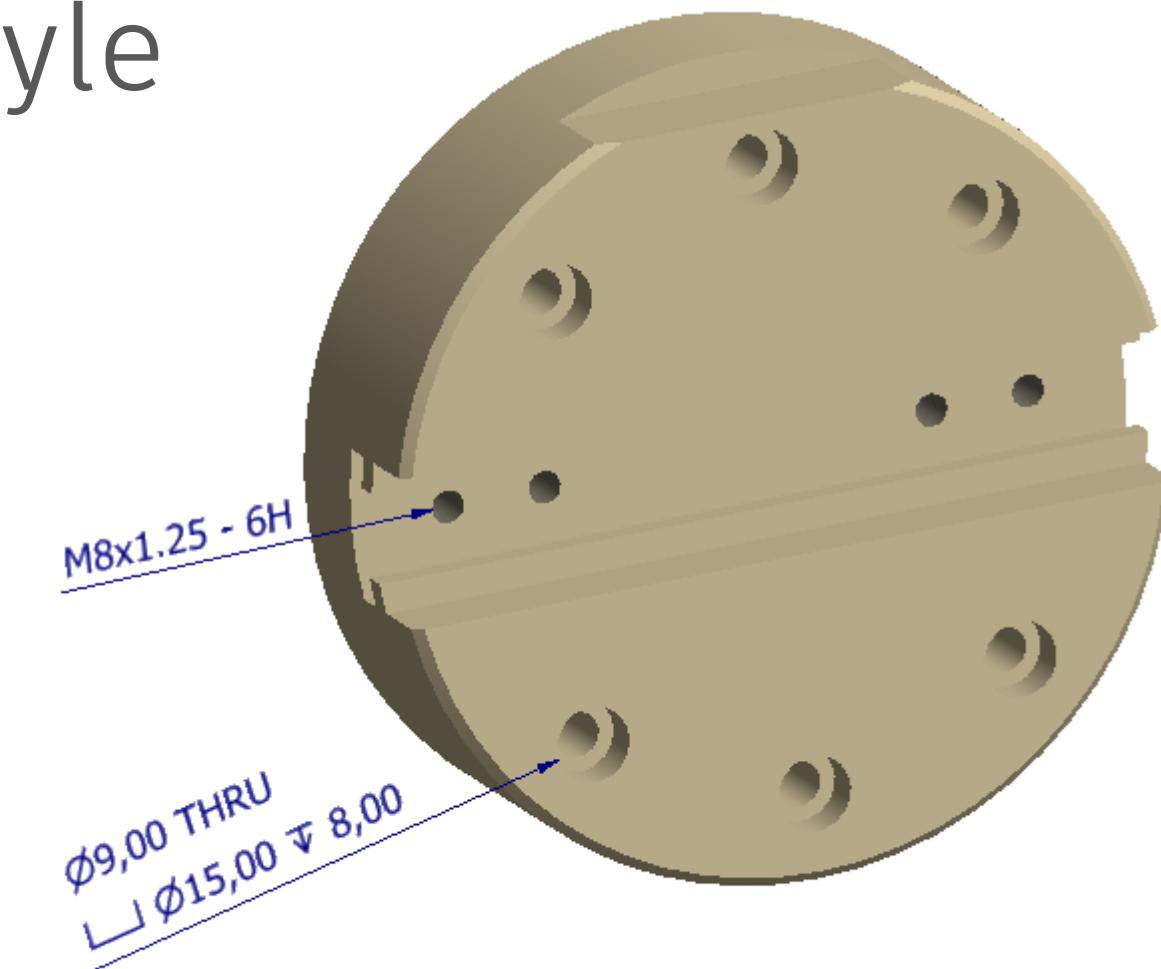


Dimensions – Style limitation

	General Annotation Dimension	Promoted Dimension
Units: Linear Unit setting	✓	✓
Units: Precision	✓	✗
Units: Trailing zeros settings	✓	✓
Alternative Units: Linear unit setting	✓	✓
Alternative Units: Precision	✓	✓
Alternative Units: Trailing zeros settings	✓	✓

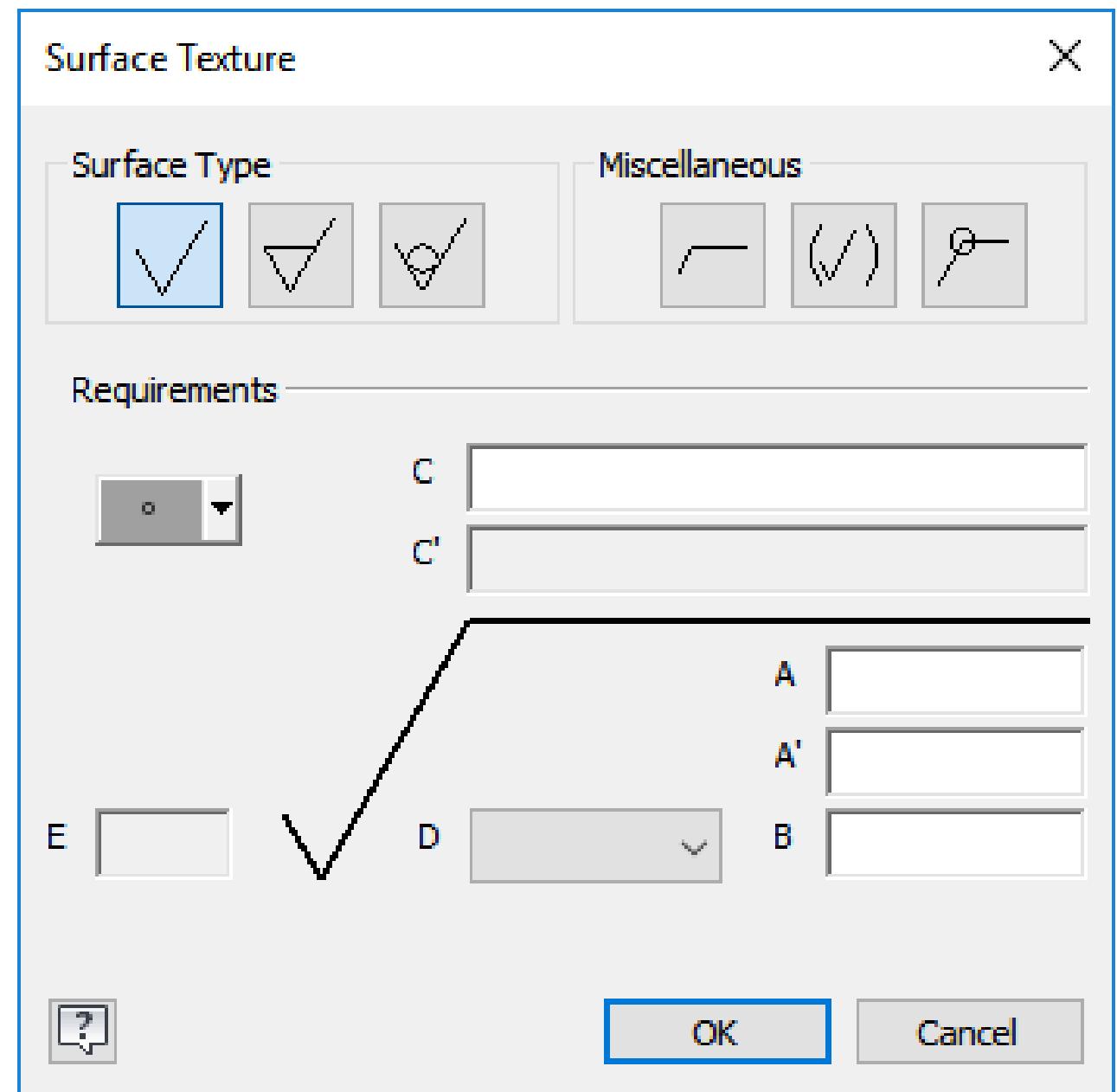
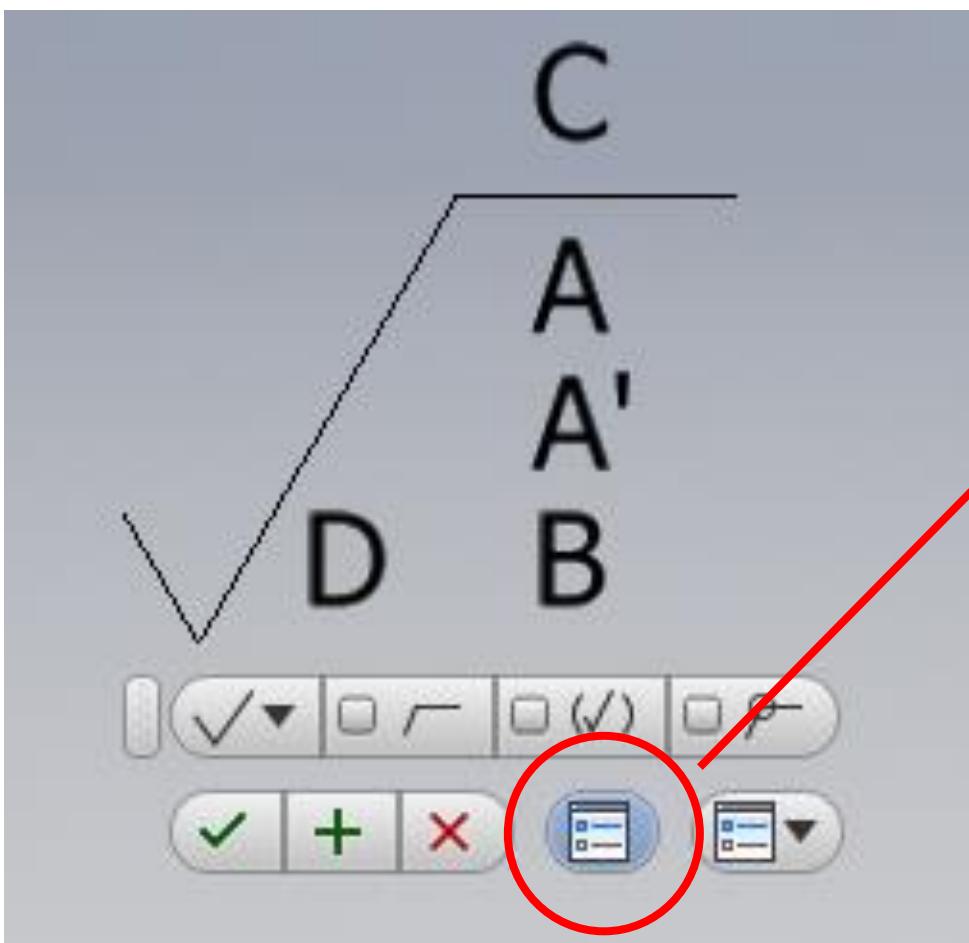
Hole/Thread notes

- Equal to Hole note in drawings, but no QTY support
- Use Tolerance feature to show QTY
- Definition in Dimension Style



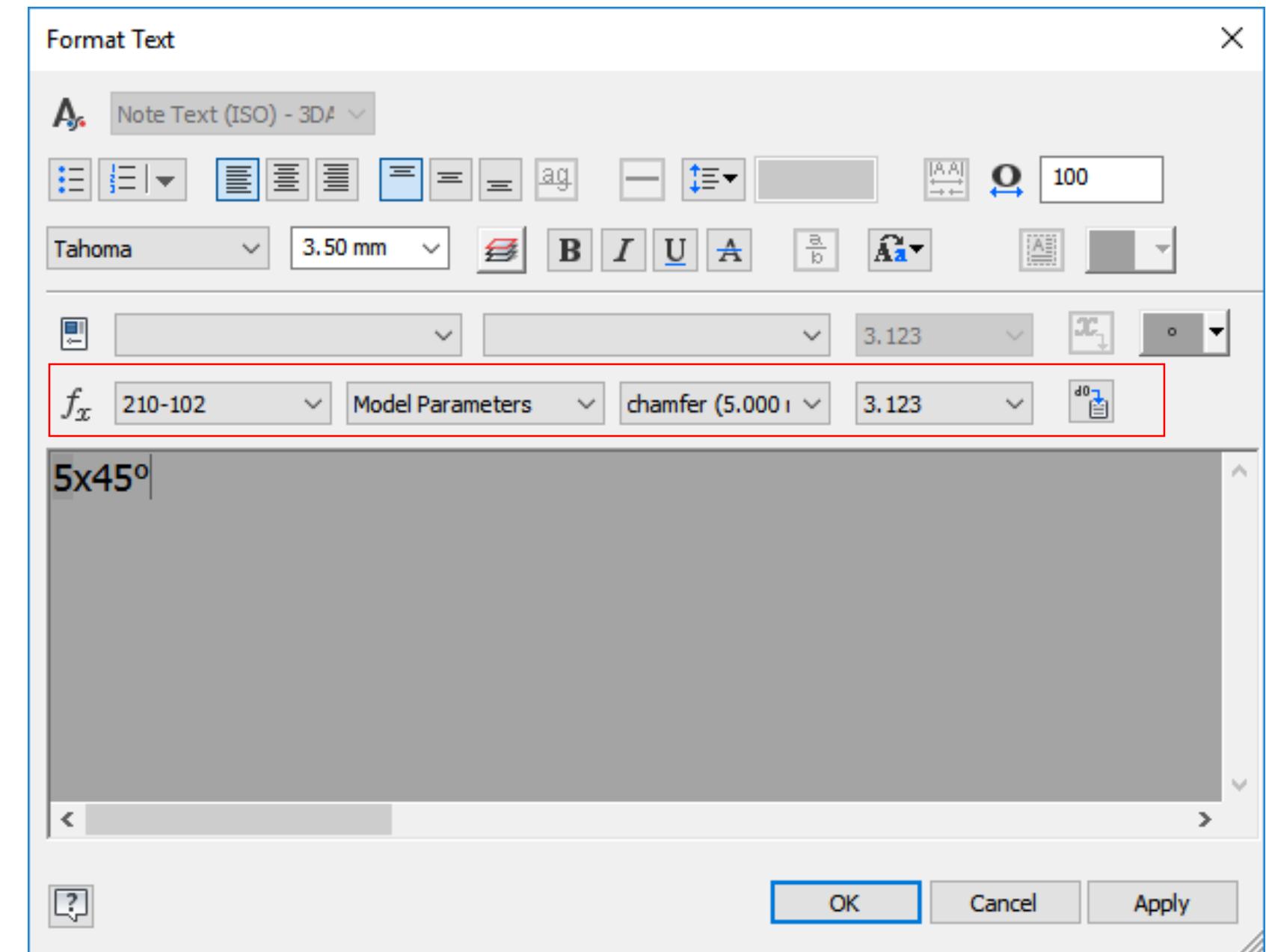
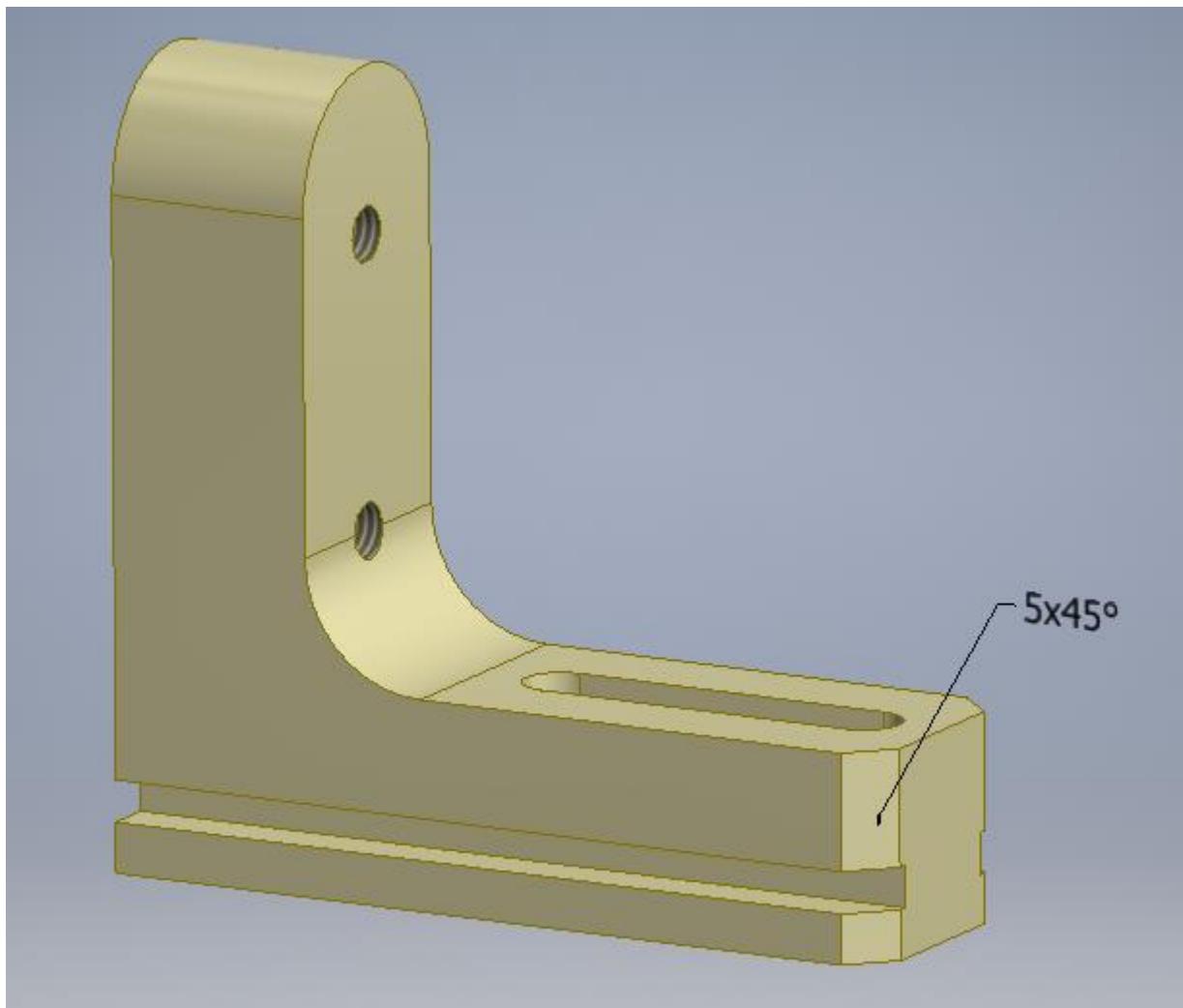
Surface Texture

- Align to geometry
- Single or multiline leader



Leader Text

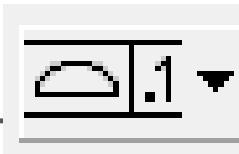
- Parameter support

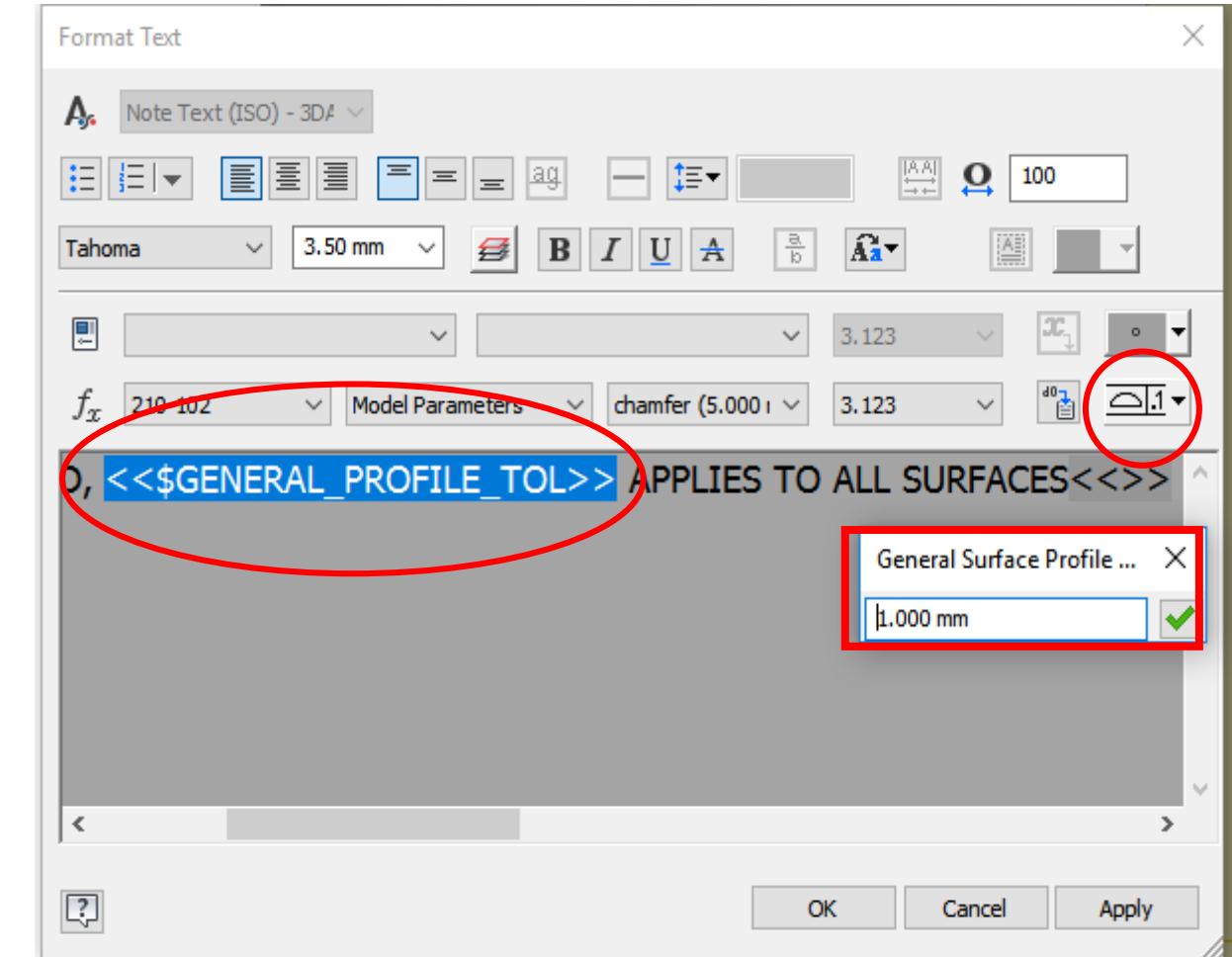


General Note

- Four quadrants available for notes
- To add or edits in quadrant, activate general note command and select quadrant
- Visibility controlled by design views

General Profile Note

- One Profile Note
- Add note:
 - Select quadrant
 - Enter note
- Edit or delete note:
 - Activate General note command or choose edit in browser
 - Select quadrant 
 - Edit note: Select 
 - Delete “<<\$GENERAL_PROFILE_TOL>>”



Handson
“Annotate”

Workflow

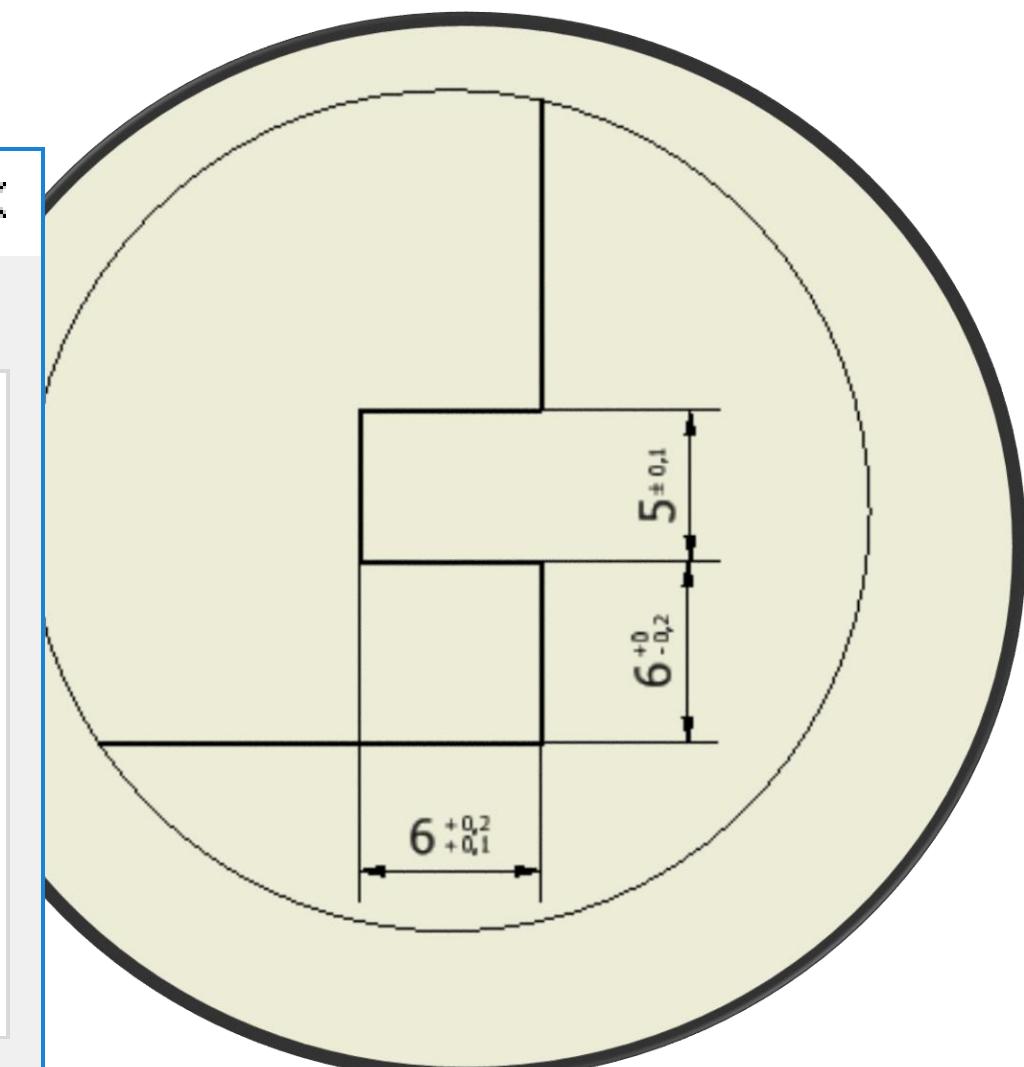
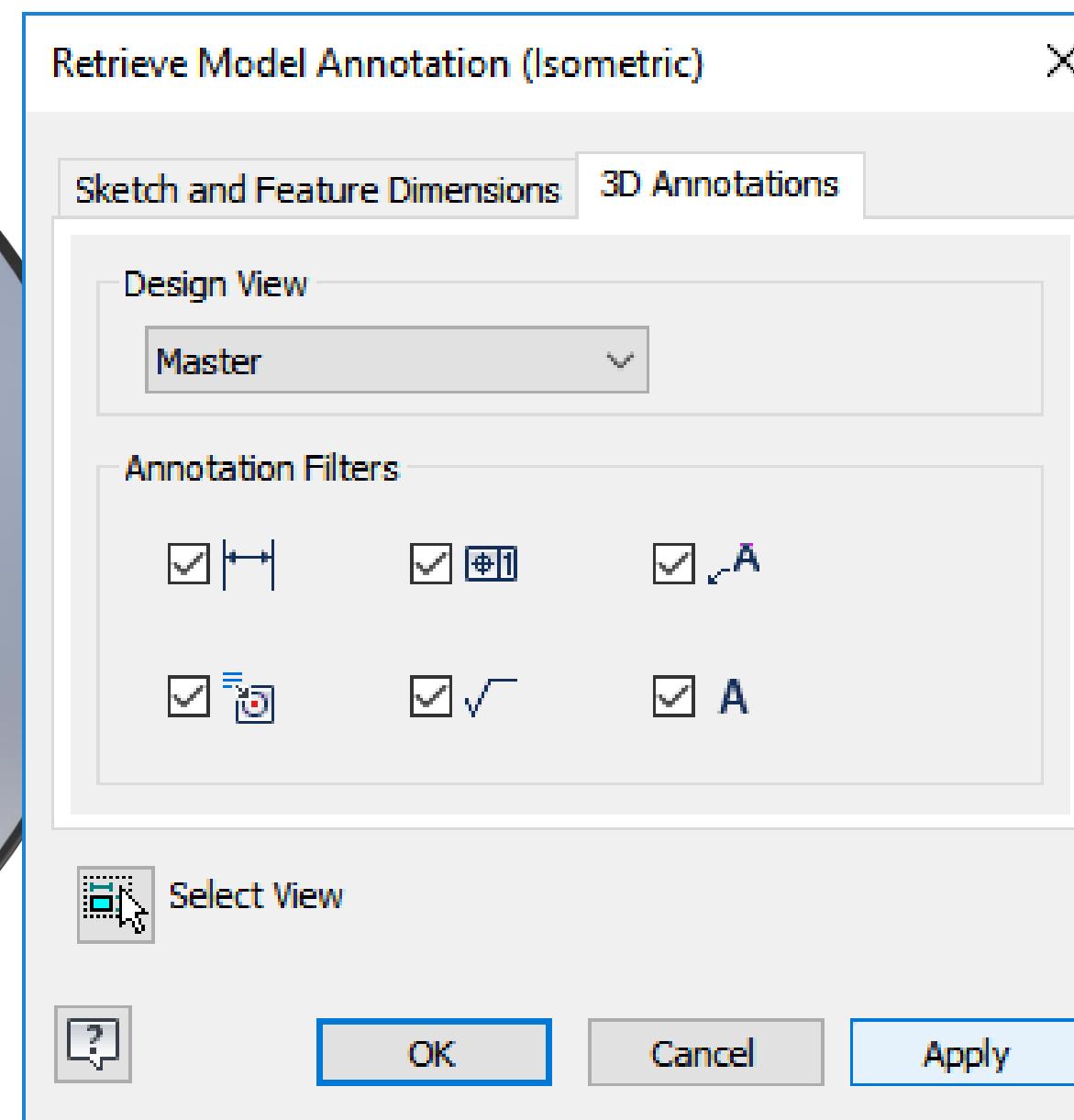
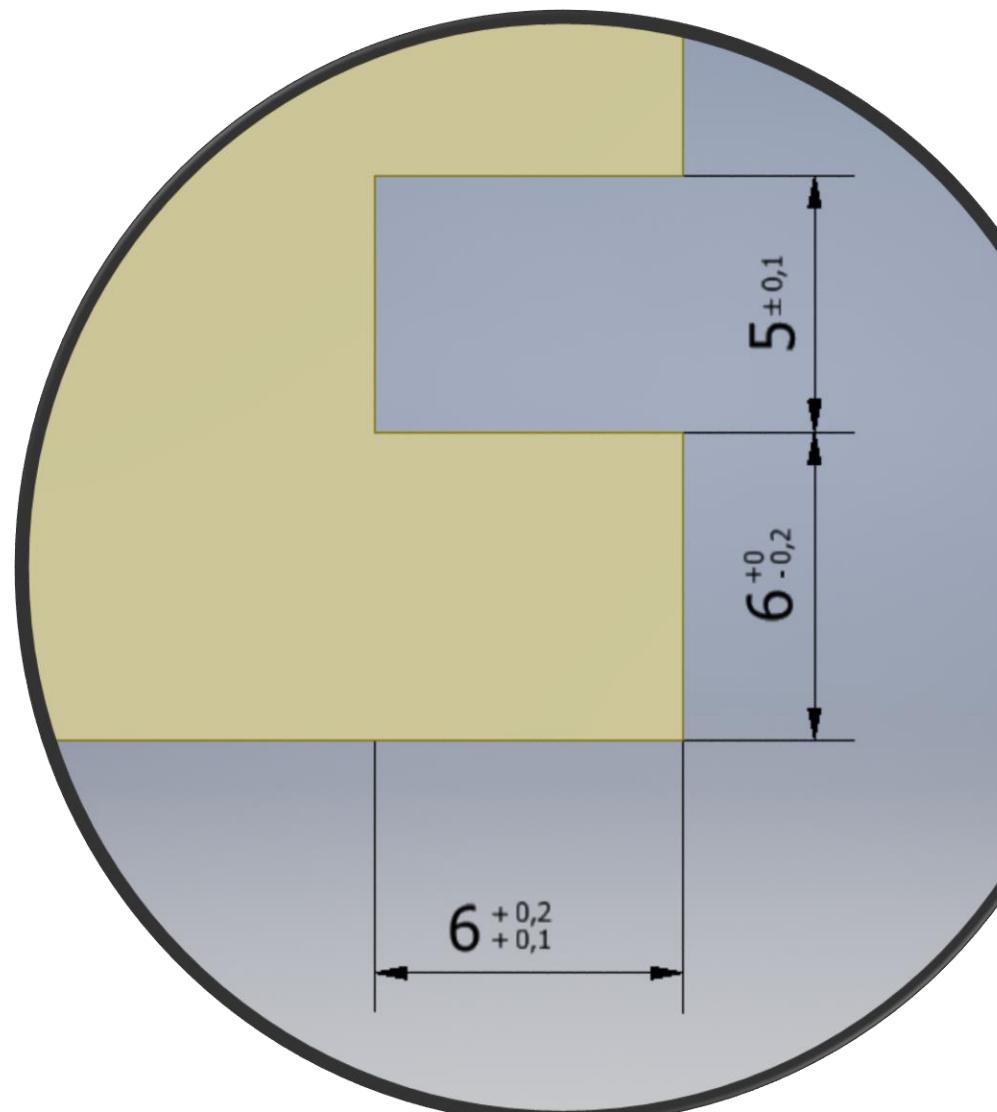


Organize

- ⑩ Organize design views
- ⑩ Create drawings and/or export model

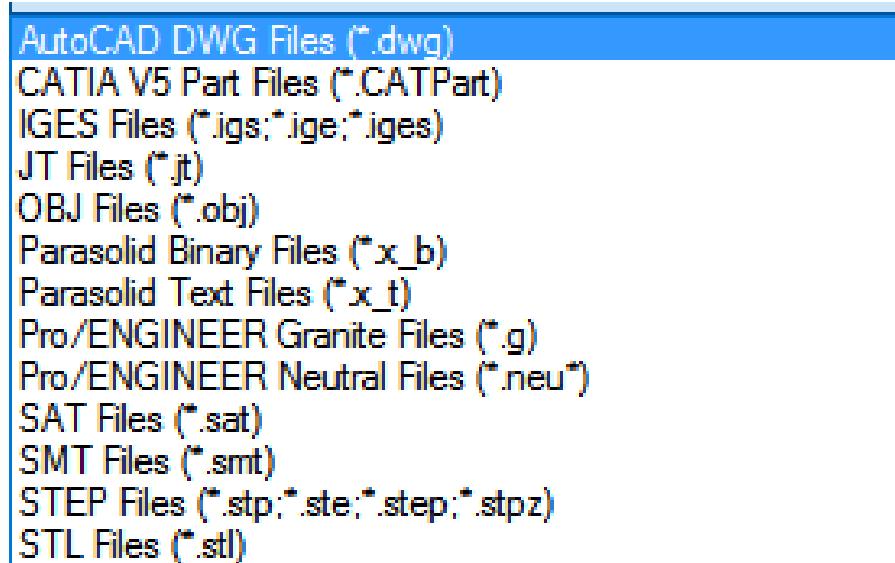
2D Drawing

■ Retrieve Model Annotations

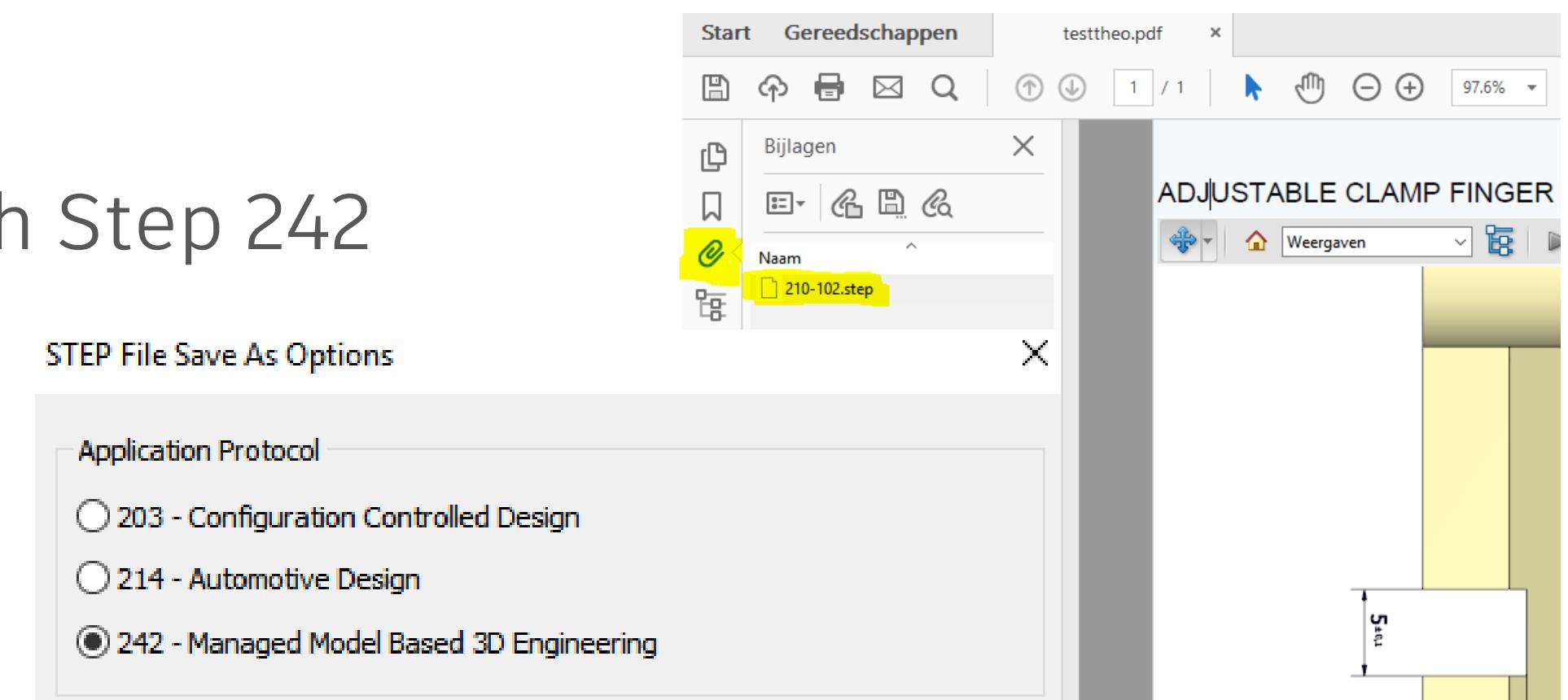


Export

- CAD

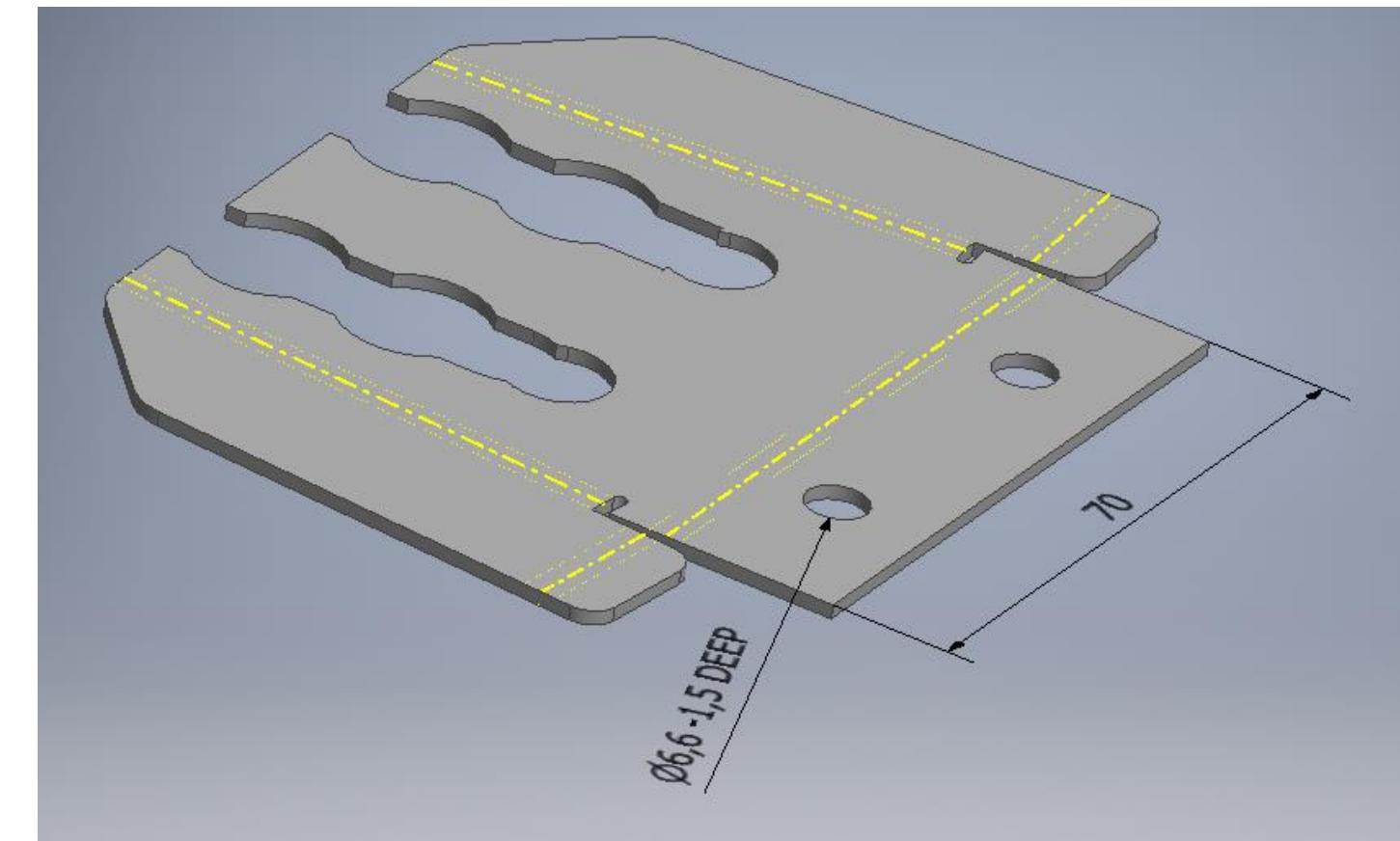


- 3D pdf
 - Generate and attach Step 242



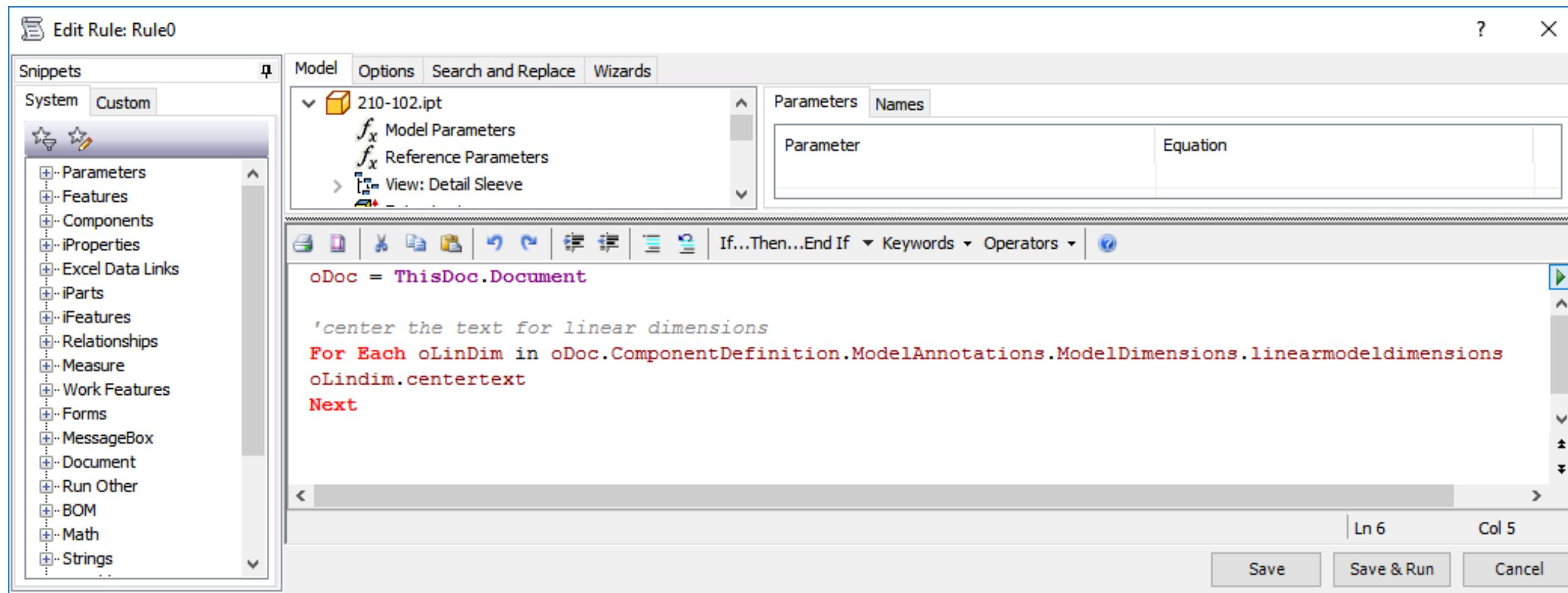
Export

- Sheet Metal parts
 - Only 3d model is exported
 - Annotation in Flat pattern **are not** exported



API support

- PartComponentDefinition.AnnotationPlanes
- PartComponentDefinition.ModelAnnotations
- PartComponentDefinition.ModelToleranceFeatures



Handson
“Organize”



Future Path



*Global Experts in GD&T
and Mechanical Variation*

EZtol for Inventor
A new 1D tolerance analysis tool
coming in Q3 2017

A Better 1D Tolerance Analysis Tool



Easier than Excel to create, manage, and report on multiple 1D tolerance analyses in an assembly

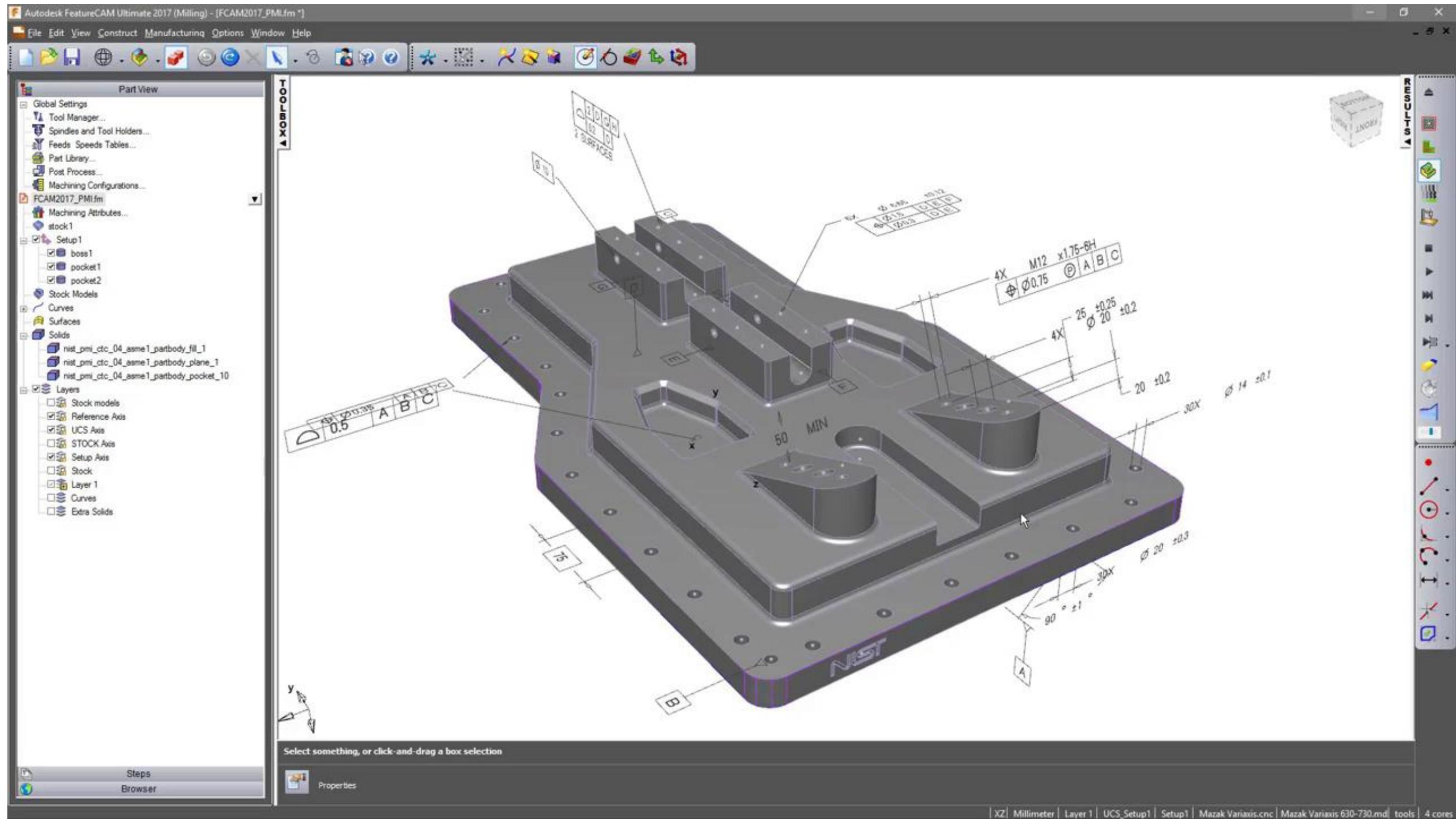
- Works with 3D models with or without tolerance information
 - Utilize PMI tolerance information if available
 - Correctly handle same part appearing multiple times in loop(s)
- Supports definition of multiple analyses
 - Dimensions common to more than one analysis handled correctly
 - Summarize results in dashboard table
- Analysis Results report includes tolerance analysis table with associated images
- Warns user when problem appears to be more than 1-Dimensional

Sigmatrix EZtol – Model Based Tolerance Analysis

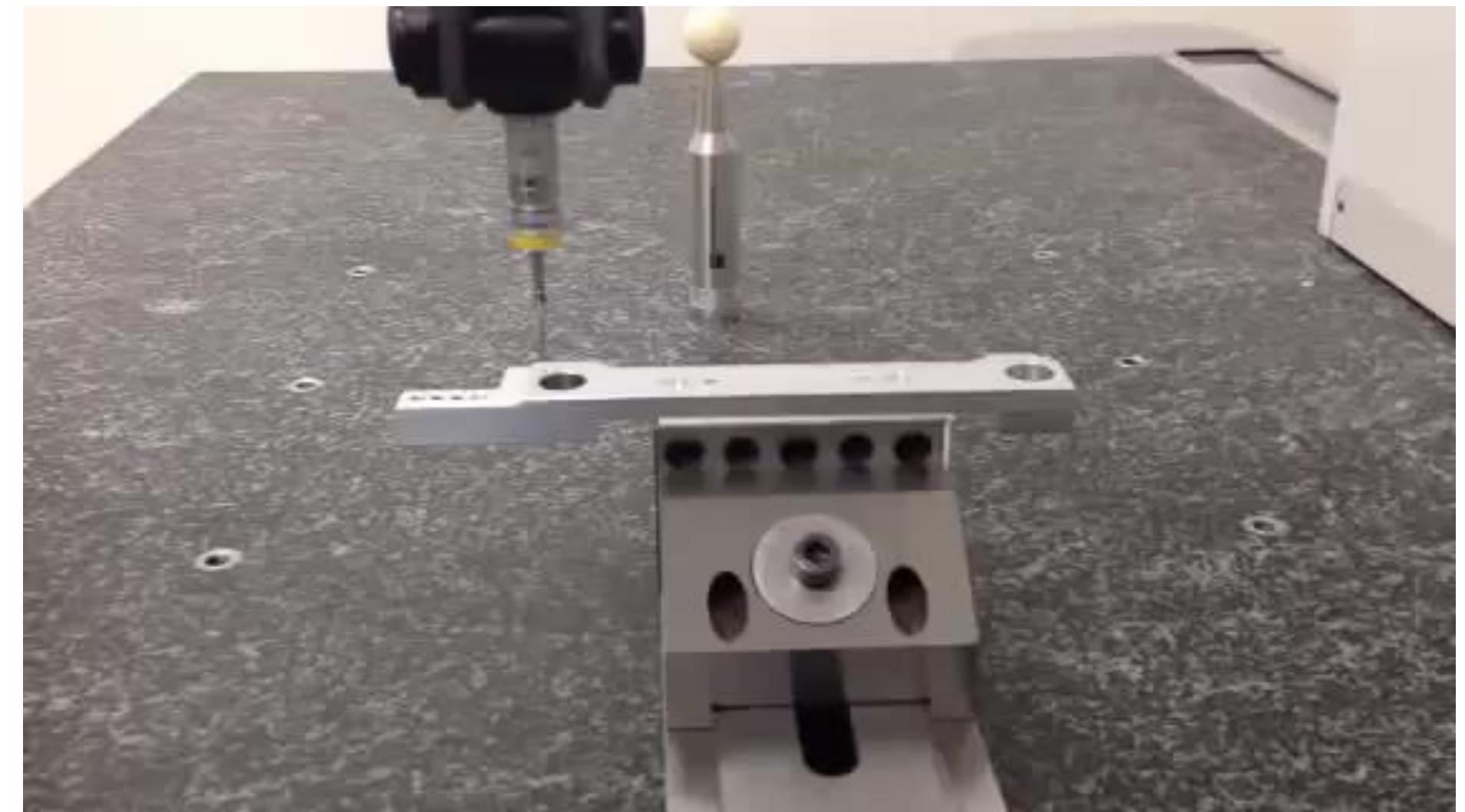


 *Sigmatrix, global experts in GD&T and mechanical variation, is proud to introduce the latest solution in our product portfolio, and the first available to the Inventor community –*

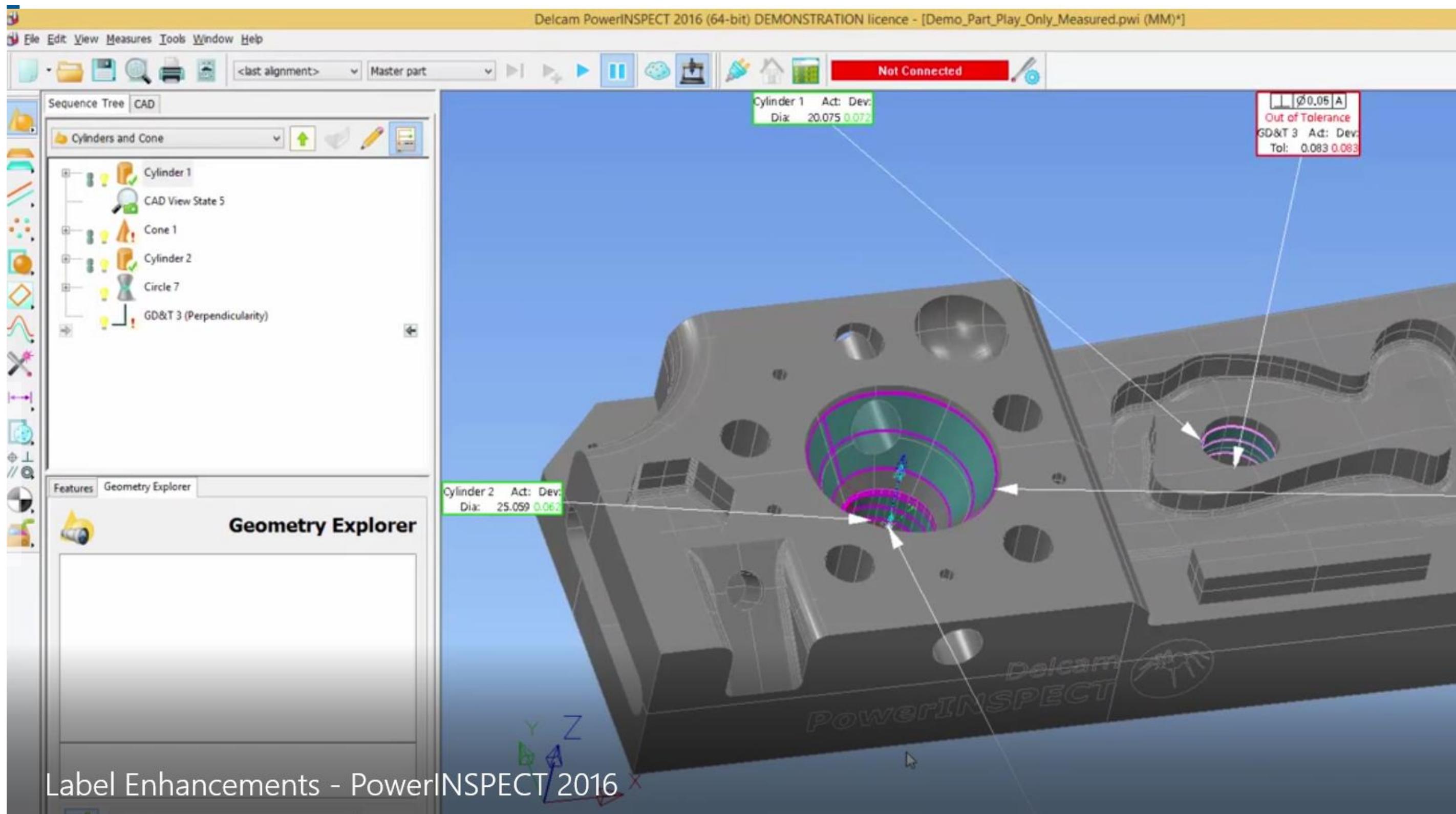
Autodesk FeatureCam – Model Based Production



CMM – Model Based Inspection



CMM – Model Based Inspection





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