

Inventor: Working with 3D Annotations and Model-Based Definitions

In modern 3D Modelling and Manufacturing Design Applications such as Autodesk Inventor, you are not required to wait until you are annotating your 2D drawings to define and display annotations next to your designs. Using Model Based Definitions (MBD) inside of Autodesk Inventor you can create 3D annotations to support the visual presentation of your model inside both Autodesk Inventor and 3D PDF files.

The Autodesk Inventor Working with 3D Annotations and Model-Based Definitions course guides users through the Model Based Definition Workflow, from creating 3D annotations to linking these directly to your 2D drawings or exporting out to 3D PDF and STEP formats.

Course Modules

- *Creating Dimensional Annotations*
- *Creating Hole/Thread Note Annotations*
- *Creating Surface Texture Annotations*
- *Creating Text-based Annotations to a Model to Communicate Additional Modelling Information*
- *Creating Tolerance Features to a Model*
- *Using the Tolerance Advisor to Review Informational Messages and Warnings on the Tolerance Features in a Model*
- *Creating a General Profile Note Annotation*

Prerequisites

This course assumes a good working knowledge of Autodesk Inventor, and is based on functionality only available in 2018 onwards.

Course Duration

1 day

Next Steps

A Bespoke Course

