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

COURSE DESCRIPTION

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. This course guides you through the Model Based Definitions 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
- Creating Tolerance Features to a Model
- Using the Tolerance Advisor to Review Messages and Warnings
- Creating a General Profile Note Annotation

OTHER INFORMATION

Prerequisites

This course assumes a good working knowledge of Inventor functionality from 2018 onwards.

Course Duration

1 Day

Next Steps

A bespoke course