Mechanical Design

CATIA - 3D Functional Tolerancing and Annotations

Define and manage tolerance specifications and annotations on 3D parts with dynamic assistance for standards compliance checking.

Product overview

CATIA - 3D Functional Tolerancing and Annotation 2 is a new-generation CATIA product addressing the easy definition and management of tolerance specifications and annotations of 3D parts and products.

The intuitive interface of CATIA - 3D Functional Tolerancing and Annotation 2 product provides an ideal solution for new CATIA customers in small and medium size industries, looking to reduce reliance on 2D drawings and increase the use of 3D as the master representation. 3D annotations can be extracted, using the annotation plane concept in CATIA GENERATIVE DRAFTING products. Besides syntactic and semantic verification command allows the user to check the correctness of all the annotations of the active product or part regarding to the standard used.

Finally, the 3D annotations can be reviewed using ENOVIA - DMU DIMENSIONING & TOLERANCING REVIEW 1 product (DT1) or DELMIA - DMU DIMENSIONING AND TOLERANCING REVIEW 2 product (MTR), which offers comprehensive tools for interpretation of annotations and tolerances on specific areas of the design or across the complete digital mockup.

Product Highlights

- Allows easy creation of associative 3D tolerance specifications and annotations
- Provides a comprehensive set of associative annotations, text, flag-notes and dress-up features in 3D format
- 3D Dimensioning in part, product and process documents
- Default tolerancing
- Display on/off of 3D annotations
- Display of the annotations in the 3D environment using annotation planes
- Can easily be modified and managed through Cut, Copy, Paste, Drag & Drop 3D annotations
- Totally associative to the 3D part or product
- Gives multiple display filtering options
- Mirrors the annotations to make them readable
- Intuitive customizable Windows native User Interface for short training cycles
- Fully compliant with latest revision of major international standards
- Allows visualization of the annotations in other workbenches (Part Design, Assembly Design,...)
- Provides a syntactic and semantic tolerancing checking of the annotations of active product or part regarding to the used
Product Key Customers Benefits

**Allows easy creation of associative 3D tolerance specifications and annotations**

By creating associative 3D tolerance specifications in part and product documents the user avoids the reliance on 2D drawings. He can apply tolerances to parts and products in the context of the assembly, using the types of tolerance that are provided.

- Text
- Datum
- Datum targets
- Geometrical tolerances
- Flag-note
- Roughness

**3D Dimensioning in part and product document:**

User can directly manipulate and update 3D Dimensioning. A permanent dress-up toolbar allows the user to directly access to most often used parameters.

**Comprehensive set of associative annotations for 3D views**

Users benefit from a full set of GD&Ts (Geometric Dimensioning & Tolerancing), including datum elements and datum targets and additional annotations to complete drawings (texts and flag-notes). He can use word processing capabilities (auto-wrapping, complex formatting inside text, etc.) to edit the annotations. The annotation positioning is associative to the geometry.

**Display on/off of 3D annotations**

The annotation display is, by default, activated when launching the CATIA - 3D FUNCTIONAL TOLERANCING & ANNOTATION 2 (FTA) product. The user can choose to un-activate or re-activate the displayed at any moment. This option is automatically un-activated, when quitting the functional dimensioning & tolerancing workbench. This option can be integrated in any other workbench (like Part Design, Assembly Design, etc.).

**Display of the annotations in the 3D environment using annotation planes**

Three annotation planes are created by default, when entering the functional tolerancing & annotation workbench, which correspond to the three planes of the coordinate system. Other annotation planes can be specified around the geometry, for automatic generation of the corresponding views, sections and cuts of the 2D drawing. These particular annotation planes are: Front View, Section View and Section Cut.

**Display modification of 3D annotations**

Zooming ability of the annotations as geometrical entities. Fast changes through Cut, Copy, Paste, Drag & Drop 3D annotations. 3D annotation transfer from one annotation plane vs. view to another one, when selecting (an) annotation(s) and activating the corresponding contextual menu (Transfer to View/Annotation Plane).

**Associativity to 3D part or product**

In-between the 3D part or product and the navigation tree. When picking an annotation around the 3D part or in the navigation tree, the related geometry features are highlighted in 3D. Associative leader and position are available. The trace of a surface is obtained through the intersection of the considered surface and the annotation plane containing...
the concerned 3D annotation. When manipulating the leader line linking an annotation to the surface/geometry, the trace of the (toleranced) surface is displayed and, by the way, indicates to the user all the possible attachment configurations, in the corresponding annotation plane. Associative highlight on/off of 3D annotations and geometry

When launching the CATIA - 3D FUNCTIONAL TOLERANCING & ANNOTATION 2 (FTA) product, the option is by default activated. This option is automatically un-activated, when quitting the functional tolerancing & annotation workbench. This option can be integrated in any other workbench (like part, assembly, etc.).

This associativity is an incomplete semantic associativity. This option allows queries on text, datum, datum target and GD&T. For a given annotation, only the related (toleranced) element(s) is highlighted, and reciprocally.

**Multiple display filtering options**

These filtering options allow the user to focus on his/her area of investigation. The user can filter the display of annotations in the 3D viewer using the following criteria : type (non semantic), by sub-type (text, data, datum targets, geometrical tolerances, flag-notes), feature or geometrical element, annotation plane, annotation set.

**Mirrors the annotations to make them readable**

Annotations may be unreadable (reversed or upside-down) for some views. Thanks to this command, clicking the "Mirror" icon will make all the annotations readable.

**Intuitive customizable Windows native User Interface for short training cycles**

Full access to 3D annotation edition function through "Edit Properties" panels. Icon based, flexible and scalable approach, from beginners & casual users using pull down menus and panels, to advanced users using contextual menus, keyboard shortcuts, and on Visual Basic Macro recording editing and replay. Permanent tool bar for direct access to most commonly used parameters when creating or modifying elements. Re-use of specifications of existing annotation elements as the standard for creation of other annotation elements (graphic attributes).

**Full compliance with latest revisions of major international standards**

ISO, ANSI, JIS supported. Full NLS compliance and DBCS support.

**Tolerancing annotations in other workbenches**

Tolerance visualization outside the FTA function, is by default activated, and allows to visualize the dimensioning and tolerancing annotations in other workbenches, through the export of the:

Display on/off of 3D annotations
Associative highlight on/off of 3D annotations and geometry

**User Assistance Capabilities**

The syntactic and semantic verification command allows the user to check the correctness of all the annotations on the active product or part regarding to the standard used. The Semantic and syntactic advisor command will Propose User applicable tolerance types regarding the selected surfaces, tolerance options when applicable, Tolerancing rules verification Help for an error-free tolerancing and Automatic support of annotation syntax (GD&T).

**Associative highlight capabilities**

Both inside and outside the FTA function, the associative highlight is by default activated, and allows to capture the links in between the entities and dimensioning and tolerancing annotations.

**Functional Tolerancing and Annotation generative dimensions from CATIA - Part**
Design
Support of sketch based part design features hole, pad, shaft and rib to automatically generate the relevant Functional Tolerancing and Annotations in FTA product. The tolerance values of the FTA/FT1 dimensions and the Part Design constraints are associative allowing the use of the mean dimensions command on FT1/FTA dimensions. The edit of the constraint will be also proposed. This command brings more productivity and reliability to the user due to the reutilisation of relevant previsouly defined information.

ENOVIA & DELMIA -DMU Tolerancing & Dimensioning review

This option helps users also to read D&T specifications created with CATIA - 3D FUNCTIONAL TOLERANCING & ANNOTATION 1 & 2 and link to ENOVIA - DMU DIMENSIONING & TOLERANCING REVIEW 1 (DT1) and DELMIA - DMU DIMENSIONING AND TOLERANCING REVIEW 2 product (MTR).
ABOUT CATIA V5R18

CATIA is Dassault Systemes’ PLM solution for digital product definition and simulation.

plm.3ds.com/CATIA