# Enhancements in CATIA V5-6 R2024

[R34]
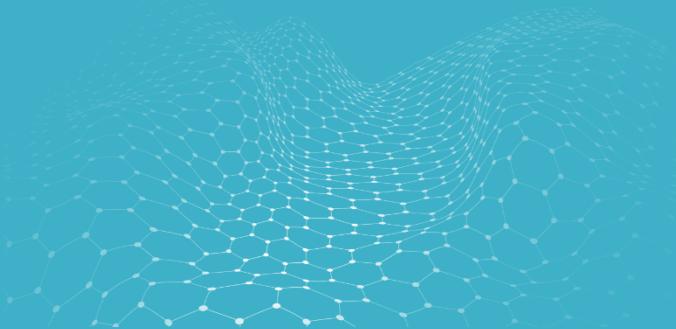
What's New - Overview

New Functionality	Update Warnings for Invalid FTA Features
Enhanced Functionality	Supporting Additional Graphical Symbols of ISO 21920-1:2021
	Improvement in Callout Creation and Multiplier of View Layout.
	Upgrading an Annotation
	Displaying FTA Features in the Parent/Children Window
	Creating FTA Views for Holes
	Selecting Multiple Edge with Propagation Selection
	Checking the Tabulated Values for Annotation Validity Status
	Selecting a Sketch as a Reference for Views
	Automatic Rerouting of Welding Links
	Integrating Static Geometrical Link in Analysis Display Mode
	Propagating The Selection

EXPERIENCE MATTERS
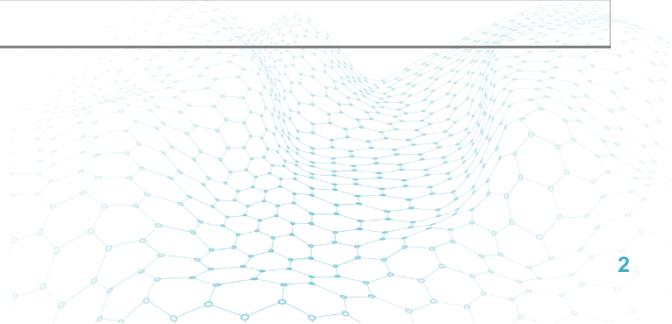
Update Warnings for Invalid FTA Features
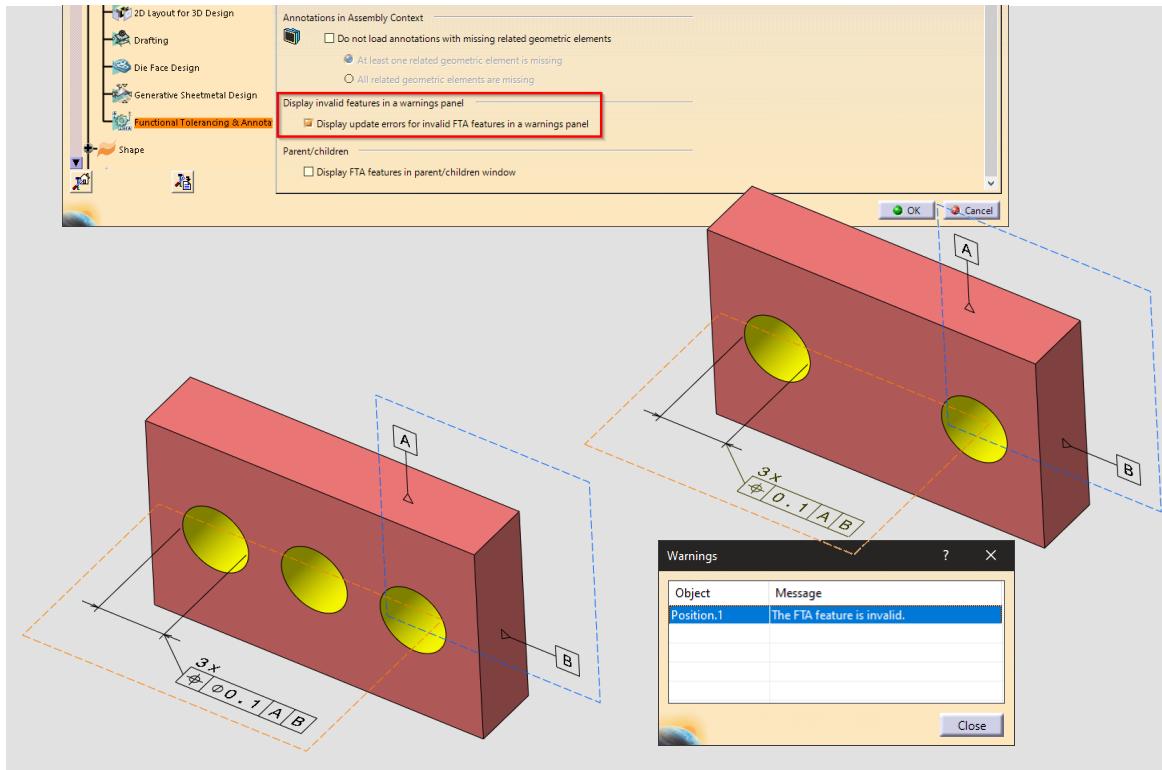
You now get update warnings in a panel for FTA features which are invalid after update.

You will get update warnings in a panel for FTA features which become or remain invalid after update. The warning panel will list all the FTA features such as Annotations, Dimensions, Restricted areas or Constructed Geometries that are invalid.

Display update errors for invalid FTA features in a warnings panel check box at Tools -> Options -> Mechanical Design -> Functional Tolerancing & Annotation -> Display -> Display invalid features in a warnings panel

# Supporting Additional Graphical Symbols of ISO 21920-1:2021
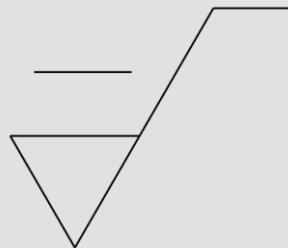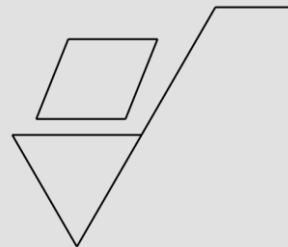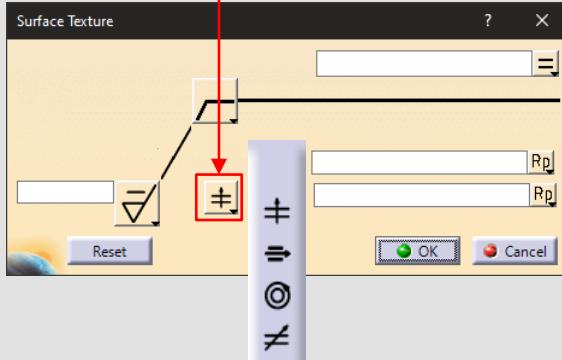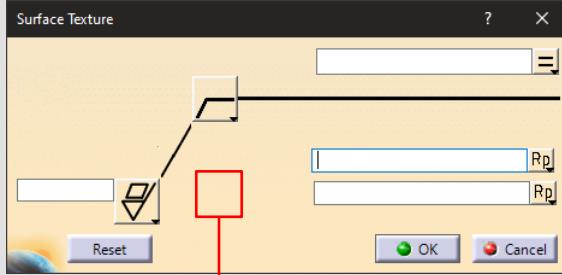
The app has added six new entries in a graphical symbol for the ISO standards. For the profile direction, only four symbols are available now. Old symbols of the profile directions are now enlisted under the working processes. Your drawing gets to comply with the global ISO 21920-1:2021 standard.
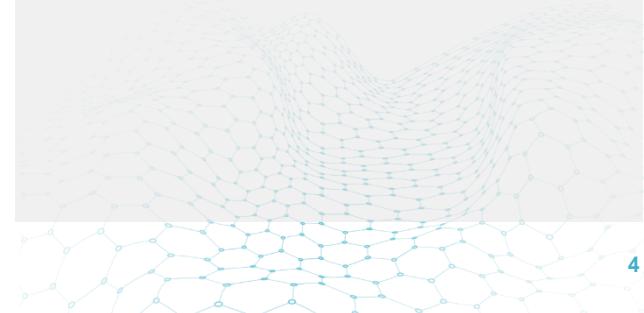
# Improvement in Callout Creation and Multiplier of View Layout
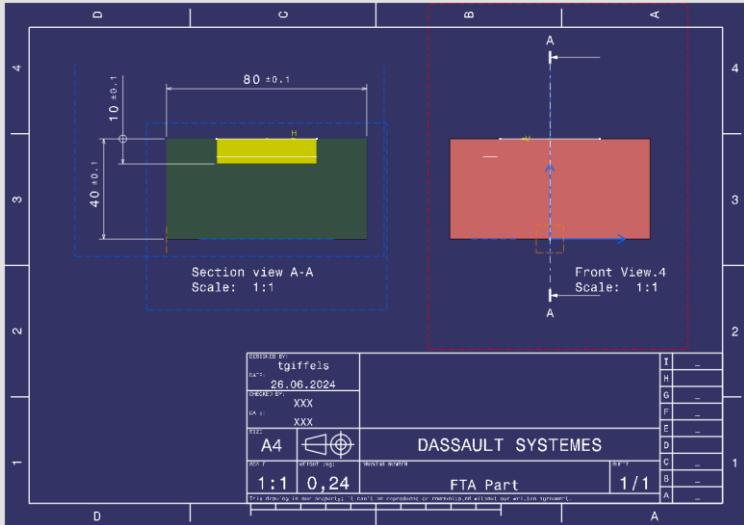
You can now use the new option Create callout even if callout intersects annotations for the creation of callout intersection annotations, when Do not apply default multiplier to shrink view in sheet size option is cleared, the extremities do not intersect any annotation or dimensions bounding box or other callout extremities. Place layouted views minimizing sheet empty space regardless of tolerancing views order option creates and arranges the view in a space saving manner. You can fit a view within the sheet frame. You have a choice to create the callout even if it intersects with the annotations. Space management is now possible.

Upgrading an Annotation
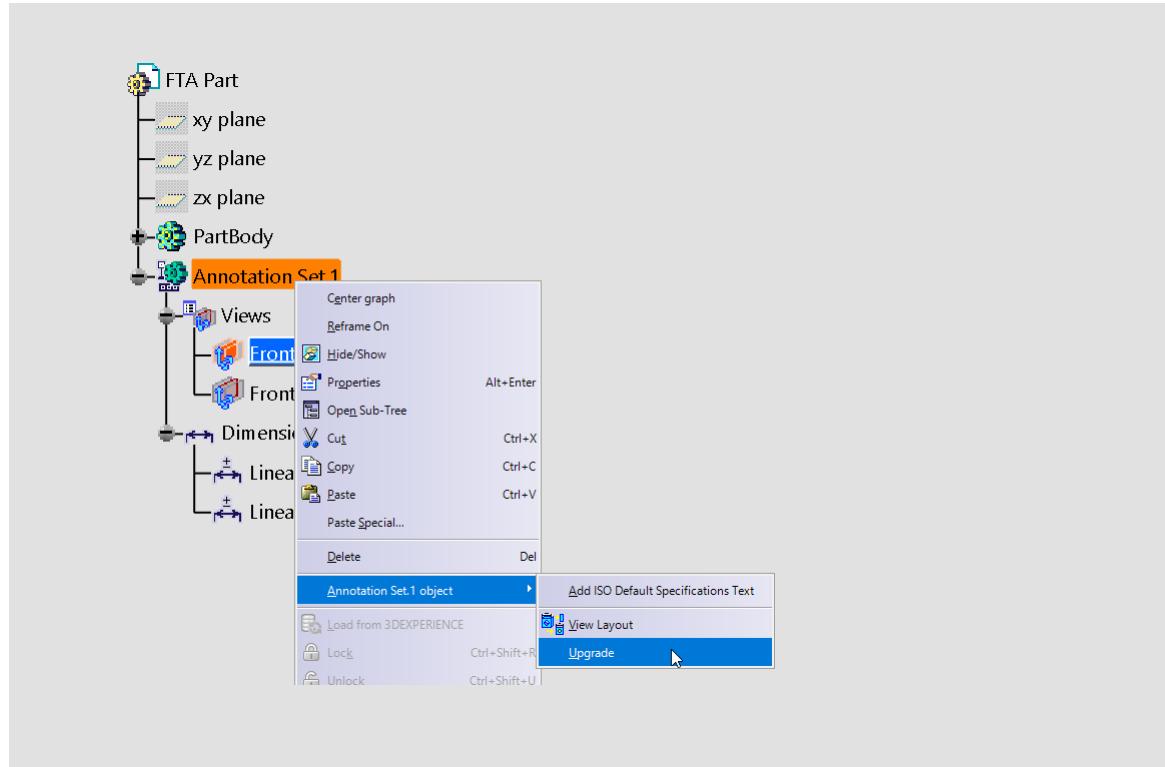
You can now upgrade all the contents of an annotation. Working with annotations is easier as you can now access the whole content.

Displaying FTA Features in the Parent/Children Window
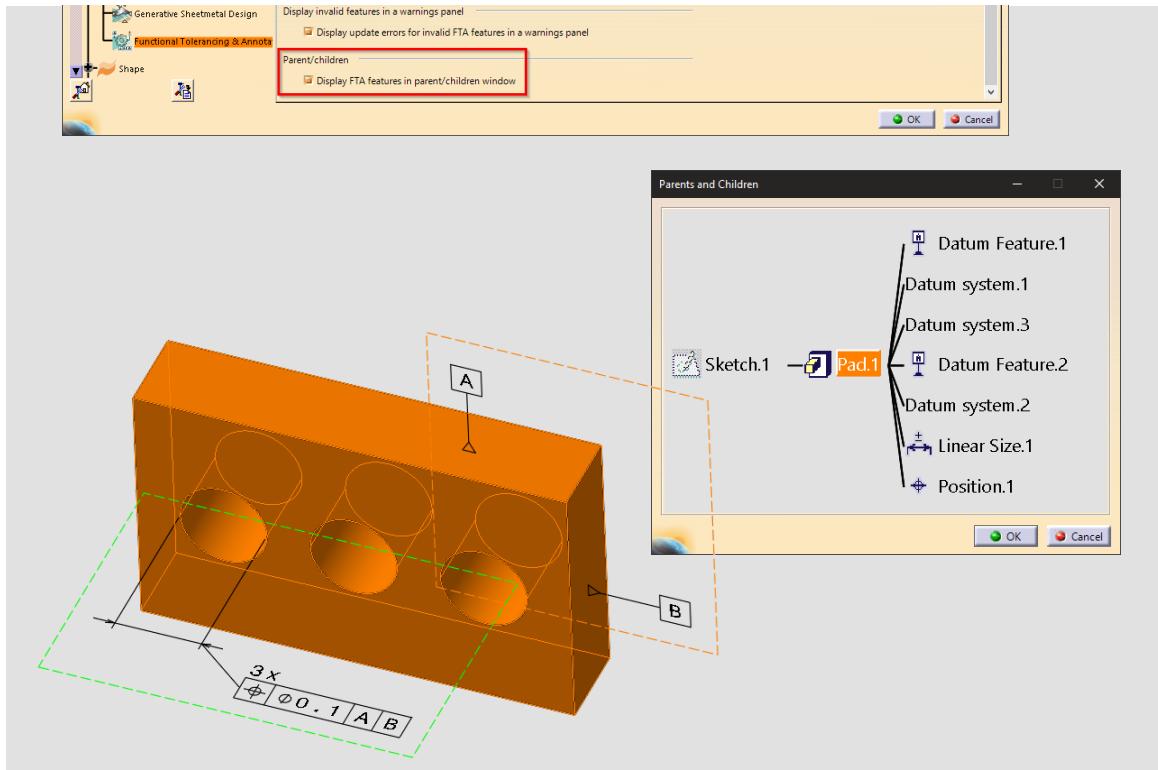
An FTA feature (annotation, constructed geometry, restricted area, view, and capture) with the geometrical link is visible in the parent/children panel of the geometric feature. You can now have a better understanding of the relation of the FTA features and the geometric features.

Displaying must be activated at Tools -> Options -> Mechanical Design -> Functional Tolerancing & Annotation -> Display -> Parent/children

Creating FTA Views for Holes
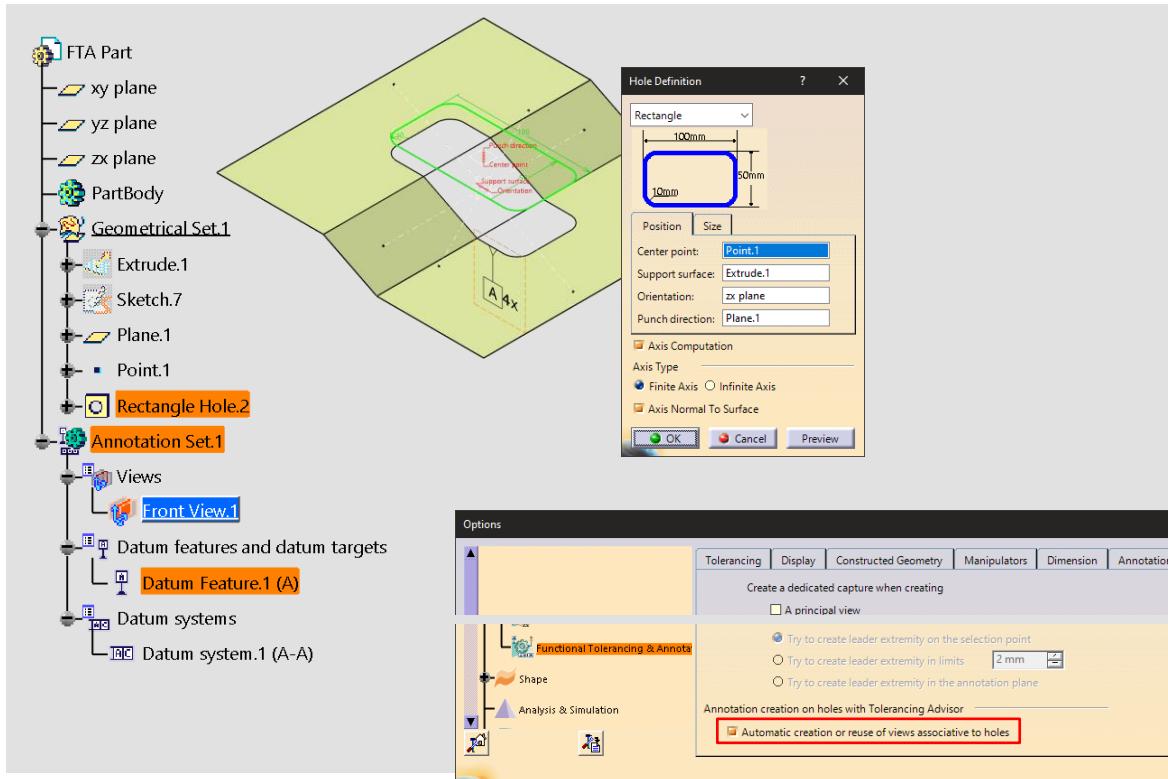
The app creates an FTA view for each hole while creating the semantic dimensions for the four types of hole. A new option Automatic creation or reuse of views associative to holes is available in the View/Annotation Plane tab.

# Selecting Multiple Edge with Propagation Selection
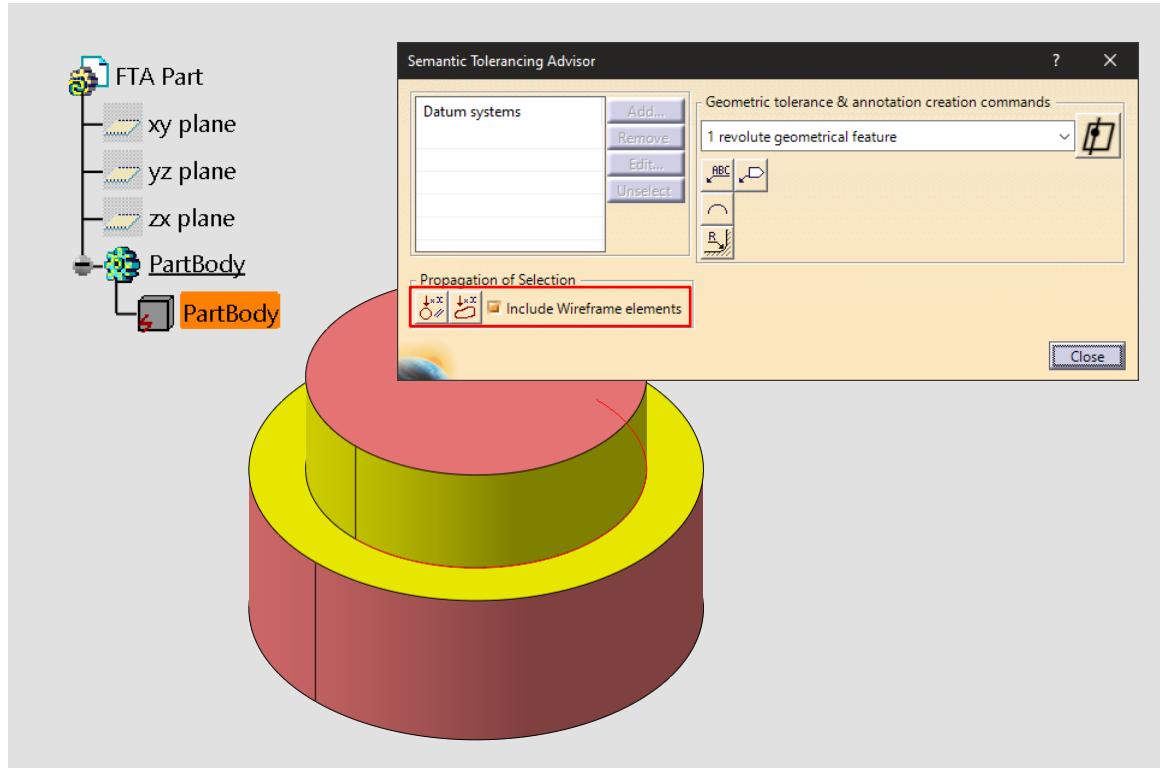
You can select multiple edges in a single click, if the edge is considered as a cylinder in the Semantic Tolerancing Advisor.
Also, a new option Exclude wire frame geometries is available in the Semantic Tolerancing Advisor panel.

- The selected circular border edge of non-parametric surface is treated as a cylinder when the bounded surface is planar. This cylinder axis is perpendicular to the edge and passes through the center of the circular edge. For such features, you can create the diameter dimension.

- The selected linear border edge of non-parametric surface is treated as a plane when the surface bounded by it is planar. This plane is oriented in a way that it passes through the selected edge and perpendicular to the bounded planar surface. When you select the two such parallel border edges, it is considered as two parallel planes are selected. For such features, you can create the distance linear dimension.

Checking the Tabulated Values for Annotation Validity Status

2024

2023
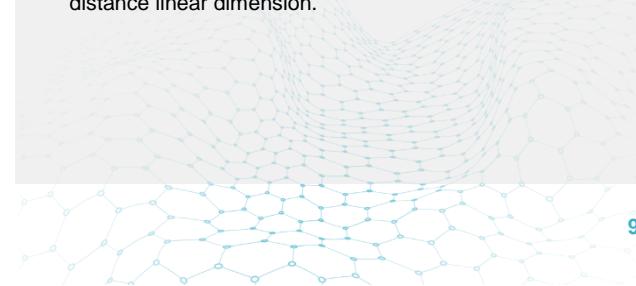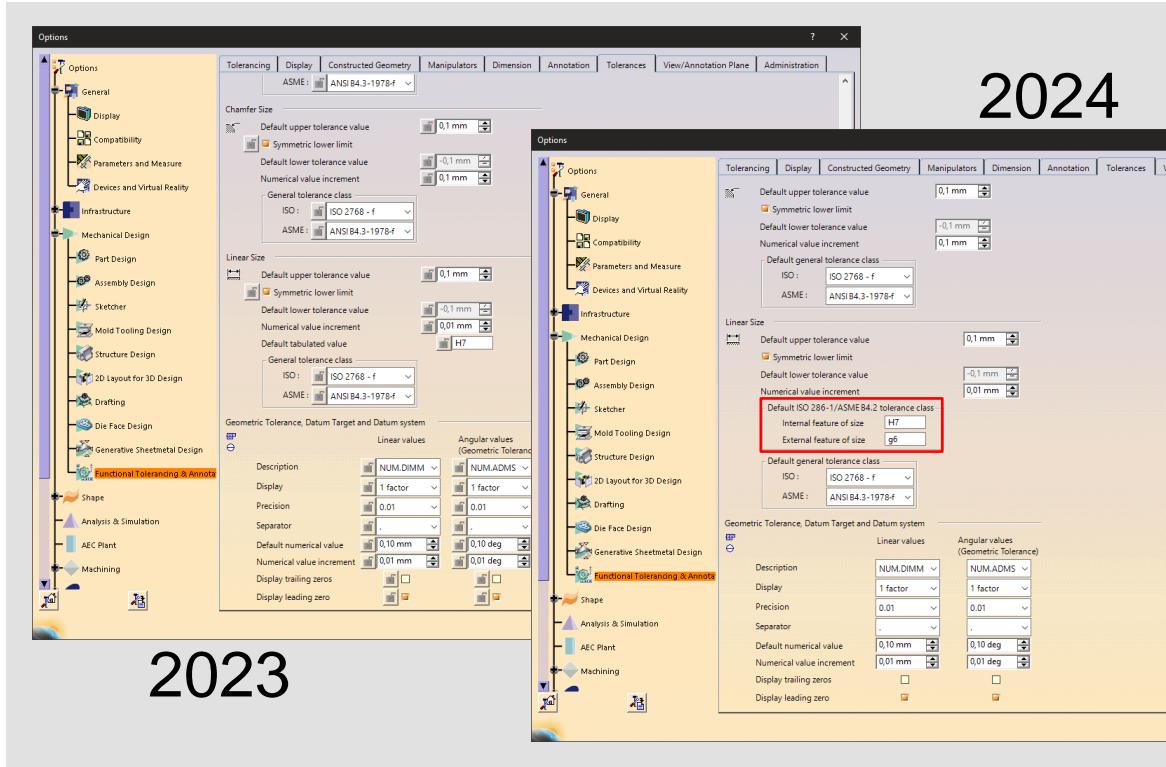
You can decide the default tabulated value according to the gender of the feature. The usability is improved as only the syntactically correct values are accepted. Two new options Internal feature of size and External feature of size are available in Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > Tolerances > Liner Size Section > Default ISO 286-1/ASME B4.2 tolerance class.

# Selecting a Sketch as a Reference for Views
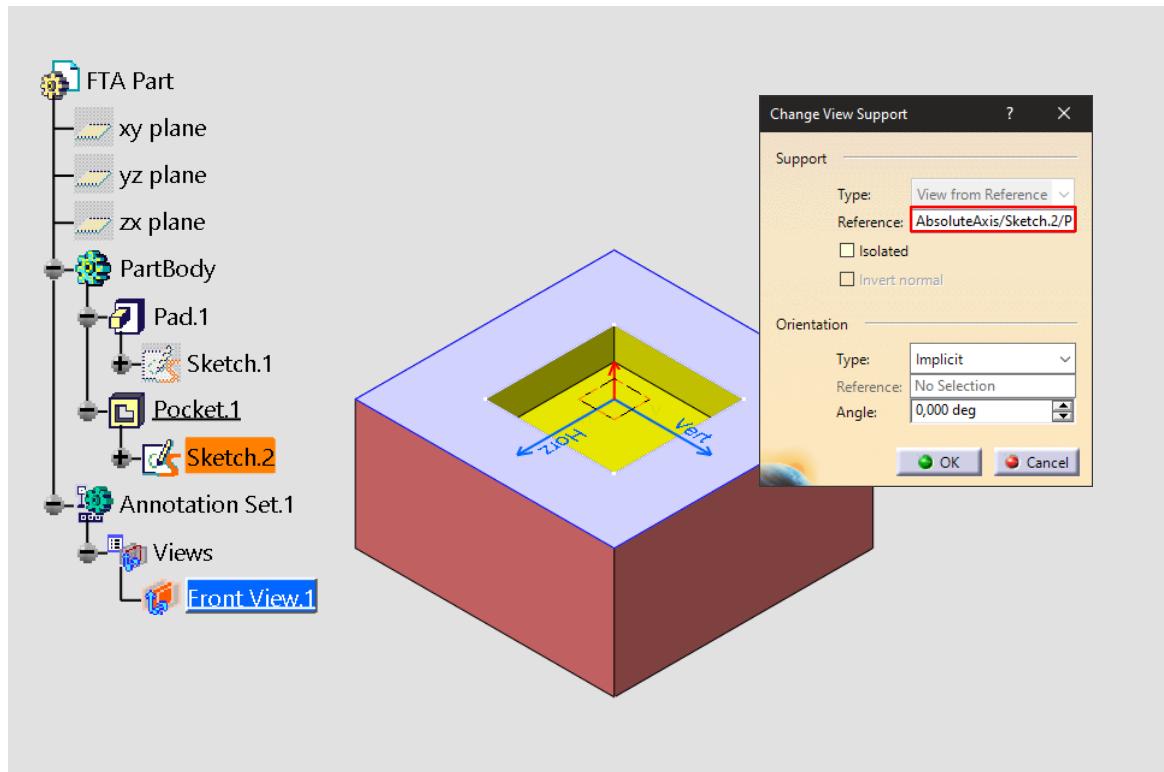
You can now select a sketch as a reference for the view creation. The app allows you to select a sketch to modify a reference in the Change View Support panel. Using the sketch as a reference for view creation saves time and minimizes the efforts of the creation of a new reference.
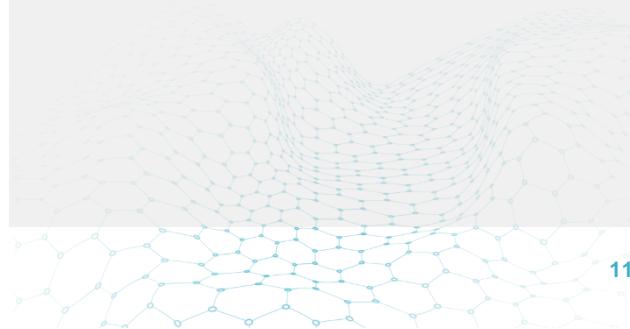
Automatic Rerouting of Welding Links

You can now automatically re-establish the welding link between the annotation and the weld feature in PowerCopy instantiation, Geometry Connection Management, Weld Feature Creation is now automatic. It is not required to reroute welding links manually anymore.

Integrating Static Geometrical Link in Analysis Display Mode

2024

2023

CAUTION: no further description available
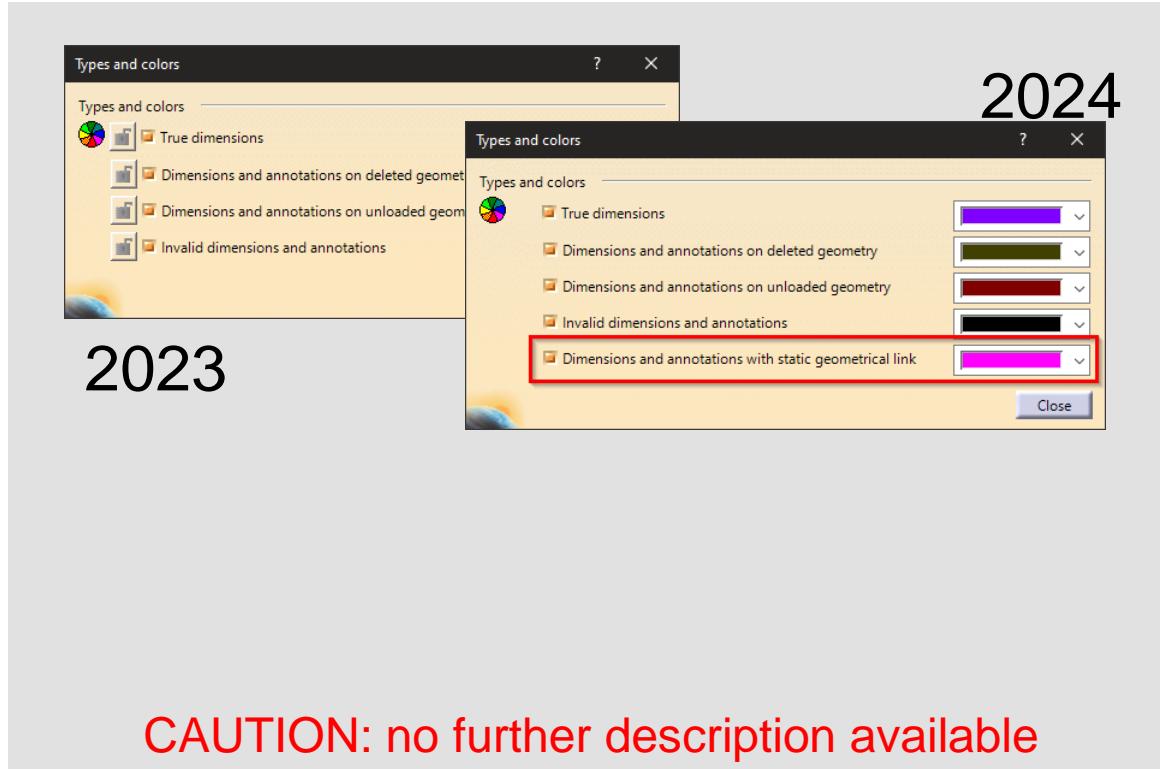
You can now have the new type and color for the FTA features like annotations, dimensions, constructed geometries with a static geometrical link in FTA Element's Analysis. Also, when you are creating an FTA feature, an information message is visible about static geometric link creation. You can have a better visibility of the elements in the view as they are now displayed with app specified colors.
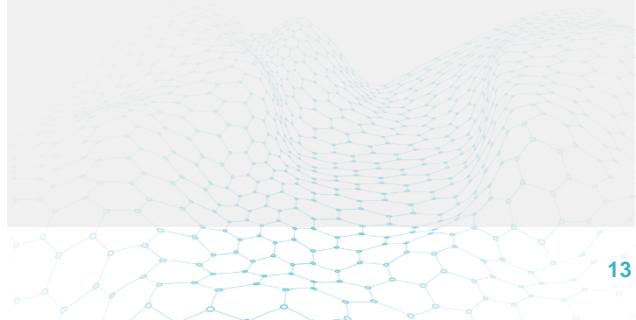
Propagating The Selection
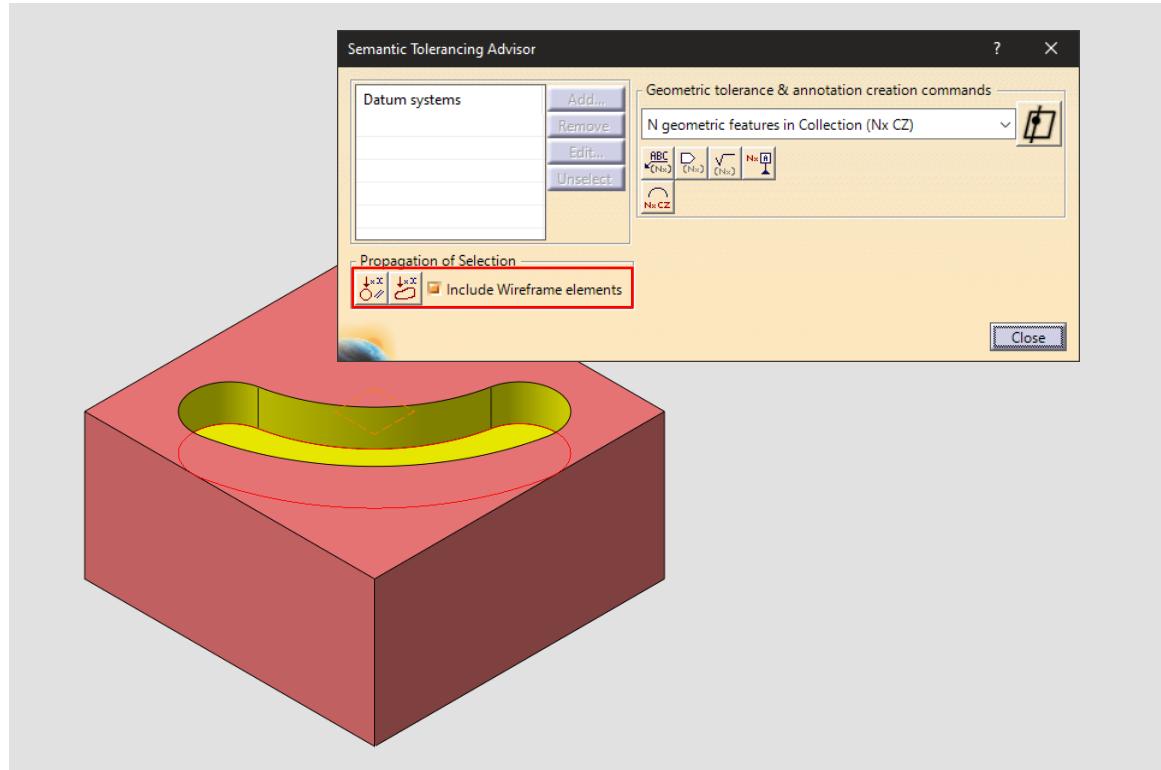
You can now propagate the selection to all the surface and solid edges of the closed contour.
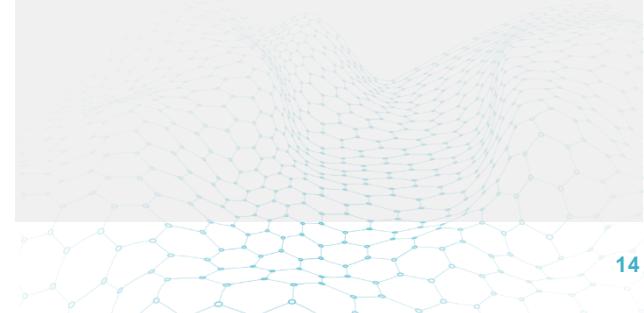
Enhancements in 3DEXPERIENCE

3DEXPERIENCE®

# 3DEXPERIENCE R2024 (Stand FD02)

## Adding Transition Specification to Symbols
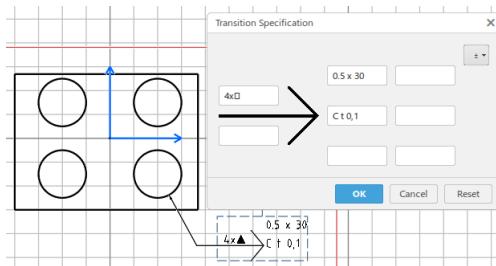
## Selecting Multiple Edges with Propagation
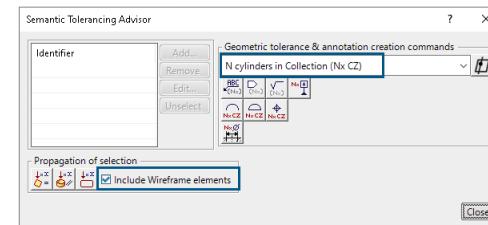
## Automatic Rerouting of Welding Links
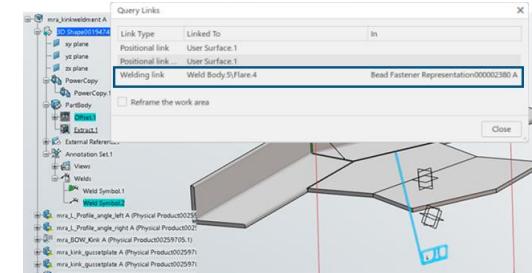
Creating FTA Views for Holes
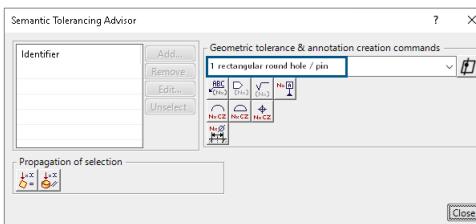
## Checking the Tabulated Values for Annotation Validity Status
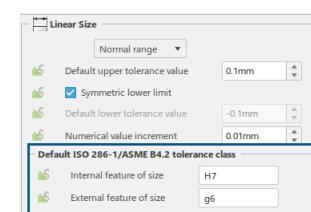
## Integrating Static Geometrical Link in Analysis Display Mode

EXPERIENCE MATTERS
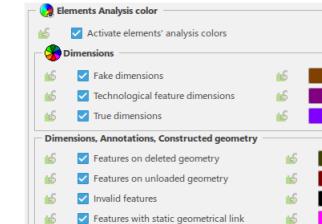
3DEXPERIENCE R2024 (Stand FD02)
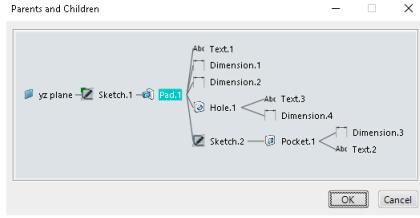
Displaying FTA Features in the Parent/Children Window	Selecting a Sketch as a Reference for Views	Generative Functional Tolerancing Credits
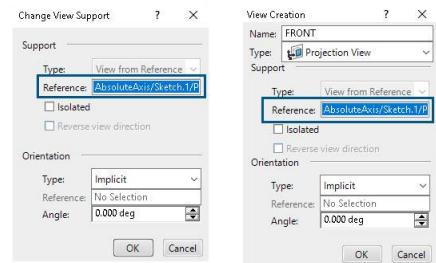
Displaying Update Errors	Supporting Additional Graphical Symbols of ISO 21920-1:2021	

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