

CATIA Solutions Version 5 Release 18

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Infrastructure User Guide

New Functionality

Navigating

The Align Viewpoint command lets you modify the viewpoint so that its orientation is automatically set to the nearest horizontal or vertical orientation.

Enhanced Functionalities

Capturing Simple Images

In non-standard scale mode, you can only capture pixel images.

Creating, Opening, and Saving Documents

For dialog boxes allowing you to select a file or a directory, the last accessed path is now stored in preference files.

Customizing Fonts for Displaying Geometry Area Texts

TrueType font versioning: the display of a text for a particular font in a particular version remains unchanged in future Version 5 releases or service packs.

Customizing Print Settings Before Printing Documents

In non-standard scale mode, only the Rasterization rendering mode is supported.

Managing Document Save

The Apply Save to editor scope only option now applies to the Save Management command.

When you resize the Save Management dialog box, the new size is kept.

Running Batches

The Generic Batch lets you run executable files in batch mode.

When running the Print Batch in non-standard scale mode, only the Shading rendering mode is supported.

Selecting Objects Using a Filter

The Tangent Intersection Edges activation mode is now available.

Selecting Objects Using the Search... Command

The Pre-highlight option lets you activate or deactivate pre-highlighting of elements found by the search.

Navigation options are now provided to display the search results.

Using the Paste Special... Command

Ability to edit the link created when pasting a body As Result With Link in the same document.

Customizing

Document

Apply Save All to editor scope only has been renamed to Apply Save to editor scope only because it now impacts the Save Management command as well.

Graphics Formats

In non-standard scale mode, options available in the Export area are deactivated.

Managing User-Defined Toolbars

Toolbar positioning has been modified when using the Restore position capability.

Scale

This new tab lets you create smaller geometries with a high accuracy.

Search

Two new options let you specify the maximum number of elements to be pre-highlighted and the maximum number of results displayed in the Search dialog box.

Data Exchange Interfaces

Enhanced Functionalities

DXF-IGES-STEP Batch

IGES and STEP files can be imported into 3D XML files (non compressed tessellation).

STEP Import

Faceted_brep are now imported as V5 solids.

STEP Report File

Geometric Validation Properties: The estimation of computation error on GVP is available in the report file.

Assemblies: information on assemblies is available in the conversion summary at import and at export.

STEP Import and Export

STEP support small scales files.

IGES 3D Export

V4 models referenced in a CATProduct are exported to .igs files.

DXF/DWG Import

Proxy graphics are supported.

Autocad2007 is supported.

CATIA - ENOVIA Interoperability

New Functionality

Sorting and Filtering Error Messages

This improved dialog box informs the user about the possible causes for Interoperability failure by classifying them as Error, Warning or Information.

Enhanced Functionalities

New Tracing Capability in Customer Environment

This capability is available for problem tracking purposes in a customer environment.

P&O Privilege Errors

In case a Save operation fails due to security privilege, an improved error message including the missing P&O privilege name.

Settings for Instances Synchronization at Save

The setting allows to select the synchronization policy for instances at Save regarding positions and/or references.

A Warning Message When Server is Lost

A warning appears when the transaction is aborted due to server crash or failure.

Refreshing Documents in Session

You can use the new Analysis window to sort and refresh documents in session.

MULTICAx AD Plug-in

Enhanced Functionality

MULTICAx Acis/DXF3D (A) Plug-in supports Windows 64 bits.

MULTICAx ID Plug-in

Enhanced Functionality

IDI MS12 is now supported.

MULTICAx Pro/Engineer Plug-in

Enhanced Functionalities

ProE Wildfire3 is supported in Direct Mode on all OS (Windows & Unix) 32 and 64 bits.

Pro/ENGINEER Wildfire 3.0 converter support is also available in Windows 64-bit mode.

MULTICAx SolidEdge Plug-in

Enhanced Functionalities

MULTICAx SolidEdge (SE) Plug-in supports Windows 64 bits.

MULTICAx SolidWorks Plug-in

Enhanced Functionalities

MULTICAx SolidWorks (S) Plug-in supports Windows 64 bits.

MULTICAx Unigraphics Plug-in

Enhanced Functionality

MULTICAx Unigraphics UD or U Plug-in support Windows 64 bits.

Photo Studio

New Functionality

Rendering Images

A new toolbar allows you to render images from any part or product workbench.

Photo Studio Optimizer

Enhanced Functionality

Installing and Uninstalling Satellites on UNIX/Windows

Allows you to use up to 24 CPUs simultaneously in a network.

Product Data Filtering

Enhanced Functionality

Better processing of 3D Functional Tolerancing & Annotation Features in Product to Product and Part to Part, in interactive and in batch mode.

Product Structure

New Functionality

Using the Instance Copy in Visualization Mode when the Cache option is activated in CATIA

This command allows you to create an instance copy in Visualization mode without loading the product reference.

Enhanced Functionalities

Inserting an Existing Document and Part Version from ENOVIA V5

The Insert Component command allows you to select a document or a part version stored in the database when VPM Navigator is connected to ENOVIA.

Deactivating / Activating a Terminal Node: Interrupt on Activate Terminal Node

Users can manage the parts they want to visualize thanks to the functions Activate / Deactivate a Node function. The Activate / Deactivate a Node function allows to visualize all parts in a specification tree. A progress bar appears to show how many objects are still to visualize. A new button, Interrupt, is available to stop the visualization.

Real Time Rendering

New Functionalities

Aligning Icons

Allows you to align icons to the nearest grid location.

Snapping Geometry

Allows you to snap the bottom wall of the environment to the geometry.

Enhanced Functionality

Modifying Lighting Properties

Allows you to choose, or even store particular color combinations.

Customizing Settings

General

Allows you to view devices, particularly cars, as touching the ground.

Shading Display

Allows you to control the shading parameters without having to apply extra material.

SMARTTEAM - CATIA Supply Chain Engineering Exchange

New Functionalities

Customizing Queries

Queries are used to map objects loaded in CATIA to objects existing in the VPDM. The SMARTTEAM reconciliator provides one query, Latest available revision, which can be launched without an interface or interactively. Custom queries can be defined based on the Latest available revision query. Custom queries can define which parameters make up the query and/or supply default values for the query parameters.

NewFrom Reconciliation Rule

The New From rule can replace the "fake product" methodology used to reconcile products and assemblies saved in structure exposed mode in ENOVIA V5 VPM. NewFrom creates a new document in SMARTTEAM during the SMARTTEAM save operation (the same as using the New rule) and it creates linkage (history) between the new document and the SMARTTEAM document it is mapped to. The New From rule is also available when using batch to reconcile or when automating the valuation of reconciliation rules.

SMARTTEAM - CATIA Integration

Enhancements

Global Refresh Interface

The following modifications have been made to the interface for Global Refresh to improve usability:

- Warning and error messages are displayed on the same line as the object they are referring to.
- The More/Less button has been removed since it is no longer necessary. Relevant information is always displayed in the panel.
- Icons are used for quick identification of the object status.

Save

SMARTTEAM save is made up of four basic tasks:

- Saving the objects in the SMARTTEAM database.
- Save the files.
- Generating the preview.
- Updating the document links.

These tasks are now reflected in the SMARTTEAM save status panel with informative messages indicating which task is currently being processed.

VPM Supply Chain Engineering Exchange

New Functionalities

Reconciling a Filtered View of a Product Structure

You can reconcile data extracted from a filtered product structure in ENOVIA V5 VPM using an attribute, volume or configuration filter. The process for the reconciliation remains the same except that after mapping your structure to ENOVIA V5 VPM you must apply the correct filter to the VPDM view.

Executing Global Sets on a Set of Objects

You can now execute a global set on a pre-selected list of objects (instead of on all objects in the CATIA view) by using a filter to select the proper objects in the CATIA view. You can locate objects based on attributes, document id, document information and mapped attributes. You can also choose to select all documents in the CATIA view or all objects in a sub tree. This feature is available both interactively and when running the reconciliator in batch (for ENOVIAvpm or ENOVIA V5 VPM).

Reconciliation of Product View Result (PVR) Documents

When working with structure exposed data in ENOVIA V5 VPM, Product View Result (PVR) documents can be created to preserve applicative (kinematic and drafting, for example) data. PVR documents can be modified by suppliers and integrated back into ENOVIA V5 VPM using the reconciliator (both interactively and in batch).

Enhanced Functionalities

Free Naming Support

If your site uses the free naming feature to map the identifiers of parts and documents in CATIA to the V_ID, V_name or V_description in ENOVIA V5 VPM, the Easy Query will take this mapping into account. The identifier will be automatically set in the correct field to take into account the mapping.

Customizing Global Sets and Filters

You can customize global sets and filters by inheriting existing (base product) global sets and filters and fixing their input values.

V4 Integration

New Functionality

Migrating V4 data into V5 in a User Defined mode

This new command, `SendToV4TV5Migration`, allows you to use the Structuration capability of the V4 To V5 Migration Batch, on a part of a V4 model.

Enhanced Functionalities

Migrating from V4 To V5 in Batch Mode

With this new PLM batch architecture, the interface has been slightly modified.

Extract Version 4 Models from Sequential File in Batch Mode

With this new PLM batch architecture, the interface has been slightly modified.

2D Layout for 3D Design

New Functionalities

Reordering sheets and views

Reorder sheets or views in the required sequence.

Clipping view

This capability allows you to clip the 2D and 3D backgrounds of layout views using a circular or polygonal profile.

2D mode for view background

The 3D elements and 2D elements that are not in the same plane as the view are filtered from view background, by activating the 2D mode property for the required view.

Analyzing View Geometries

This capability allows you to analyze view geometries whether it is under-constrained, over-constrained or iso-constrained. Complete information about every element of the geometry or constraint of a view is given. Information on each projection or intersection (use-edges) is also given.

Associative use edges

This capability allows you to create use-edges which are associated to geometry and customize the color at creation.

Setting Small Scale in CATIA V5 Session

The Small Scale feature allows you to create smaller geometries with a high accuracy of less than 1 micron. In 2D Layout for 3D Design there are a few points which you should consider.

Enhanced Functionalities

Pasting driving dimensions

This capability allows you to paste driving dimensions along with geometry in Sketcher workbench.

True Type Fonts support on UNIX

The True Type fonts are now supported on UNIX also.

2D Layout persistent rendering mode

This capability allows the saving of the rendering mode in the document and is used to display the layout when opening it.

Coordinate dimension value display format

This capability allows you to define the display format of the coordinate dimensions. The new Numerical Properties toolbar manages the display format for all the dimensions including the coordinate dimensions.

Generating the FTA elements

This capability allows you to export the FTA annotations and constructed geometry from 2D layout views to drafting.

Annotations blanking enhancements

This capability allows you to blank geometries in addition to area fill. Blanking can also be defined for both dimensions and leader symbols.

Background display and Cutting Plane

The Cutting Plane and Display Backgrounds as Specified for Each View switches are automatically activated when loading a layout.

Projection views enhancement

Customize the definition of the view box

This capability allows you to customize the definition of the view box with manipulators, while creating primary views using 3D reference. When no selection is made in 3D, you can change the primary view type, with respect to the view box, using the manipulators.

Clean up filters

Clean up of filter deletes unused filter or filters.

Customizing Settings

Background and Cutting Plane Display when loading layout

This options allows you to automatically activate the Cutting Plane and Display Backgrounds as Specified for Each View switches when loading a layout.

Clipping view outline customization

This option allows you to predefine the display of the clipping outline to be used for creating new clipped views.

2D visualization mode option

This option allows you to predefine the value to be used for 2D Mode when creating new views. The 2D Mode property allows you to display only the 2D planar elements that are in the same plane as this view in 3D.

Geometry

This new tab allows you to create associative use edges and customize its color.

3D Functional Tolerancing & Annotation

New Functionalities

Naming a tolerancing capture

This capability allows you to name and rename tolerancing captures while maintaining the uniqueness of capture names.

Support of FTA features when replacing a geometrical element

FTA features can be migrated when replacing one geometrical element by another.

Support of Thread Technological Result features

The thread technological result feature is supported in the Thread Representation Creation and the Constructed Geometry Creation commands.

Removal of the link of NOA instance to NOA reference

The NOA detail instance points to the reference CATDrawing or catalog and not to the local reference.

Geometry selection propagation capability for feature creation

This capability allows you to automatically propagate geometry selection corresponding to the same surface.

New icon for Product Functional Tolerancing & Annotation workbench

A new icon has been defined for the Product Functional Tolerancing & Annotation workbench.

Dimension line-up capability

Lining Up Dimensions (Free)

This capability allows you to line-up the dimensions relative to a point in the free space.

Lining Up Dimensions (Reference)

This capability allows you to line-up the dimensions relative to a given reference.

3D preview of view and capture feature

View selection

Clipping plane and section profile are displayed when a view is selected.

Capture selection

Clipping plane and section profile of the active view in a capture are displayed when a capture is selected.

Enhanced Functionalities

Coordinate dimension value display format

This capability allows you to define the display format of the coordinate dimensions.

The Numerical Properties toolbar manages the display format for all the dimensions including the coordinate dimensions.

Automatic creation of framed dimensions

This capability allows you to automatically create framed dimensions.

Consistent units for angular dimension

The units for angular dimension has been made consistent between the Numerical Display Description of the Numerical Properties toolbar and the tolerance values in the Limit of Size Definition dialog box.

Support of ISO 2768 and ANSI B4.3 - 1978 general tolerances for angular dimensions

The tolerance standards ISO 2768 and ANSI B4.3 - 1978 are supported for angular dimensions.

Save 3D Annotations as 3D XML

This capability allows you to save 3D Annotations as 3DXML for the Dynamic Tessellation and Static Tessellation geometry representation formats.

Customizing Settings

Control the display of view axis system

This new option allows you to control the display of the view axis system.

Assembly Design

Enhanced Functionalities

FTA Toolbar Completeness

Text, Flag Note, View/Annotation Planes have got more menu options.

Modifying a Symmetry

The geometry options selected with the previous symmetry reappears with the new symmetry. A new option 'Set Configuration as Default' is provided in the Assembly Symmetry Wizard in order to facilitate that.

Fix Part or Product by Coordinate or Angle

Fix a component specifying co-ordinates.

Supporting Assembly documents with different scales

Assembly of documents with different scales are not supported.

Aerospace Sheetmetal Design

Enhanced Functionalities

Non deformed cylindrical hole

You can create a cylindrical hole in folded and unfolded view by checking No Deformation option in the Deformation tab of the Hole Definition dialog box.

Double curvature Web

You can select a non ruled surface as Support of the Web.

Joggle: definition plane as start

Previously the joggle plane was always the end plane. Now, it is possible to define the plane as start or end of the Joggle.

Characteristic curves enhancements

The characteristic curves-related behaviours of the surfacic flange can also be applicable for swept flange. The CLB (Center Line of Bend) characteristic curve can be computed on unfolded view of both Surfacic Flanges and Swept Flanges.

Cast & Forged Part Optimizer

New Functionality

Autodrafting (for casting)

The Automatic draft capability speeds up the draft angle process to manage the whole part in one shot and to capture higher level semantic.

Enhanced Functionalities

Autofilleting with Parting Element

The Automatic Filletting capability can involve parting lines and parting surfaces. A parting line is a chain of sharp edges included on a parting surface. These edges are not rounded. The sharp edges that are incident to the parting line are rounded. This operation modifies the parting line in such a way that it remains on the parting surface.

Draft Both Sides Neutral/Neutral Fitted

The Fitted computation method is available if you are using the Neutral/Neutral mode. It performs a draft operation on two opposite sides of the part while adjusting the resulting faces on the parting element you chose.

Draft Both Side: Additional Tolerance for Forging Process

The resulting drafted faces of a draft both sides fitted are adjusted on the parting surface. This constraint may lead to transversal sharp edges. In some cases (Forging for instance), this constraint may not be useful and these transversal sharp edges may be unwanted. In such cases, you may need to set the Parting Element Adjustment to remove some transversal sharp edges.

Draft Both Side: Automatic Relimitation

This capability is applicable when the ribbon propagation encounters the first twist in the draft skin while creating a draft for a single body. It will reduce some steps and design time. With this capability, drafts will not fail in case of twist, but instead drafts the maximum ribbon and leaves undrafted the twisted zones.

Fillet face-face hold-curve with keep edge

You can specify the edges of support body to be kept and creates fillet surfaces with Hold Curve and Keep Edge as support wherever necessary.

Composites Design

New Functionalities

On the Fly Information

Displays information on composites elements while moving the cursor over them.

Darts

Darts can be created with a specific command, or within the producibility command.

Ply Section

Creates section views of the stacking or of the plies without exploding them beforehand.

Enhanced Functionalities

Material library management

Instead of selecting a whole catalog, you will select only the materials you intend to use, from one or several catalogs. They will be stored as Composites Parameters so that the Composites data can be understood and updated even if the catalogs are not available.

This impacts the way you modify materials in plies.

Pre-R18 models must be upgraded.

Plies Group

The Plies Group dialog box lets you lock the draping direction of all the plies of the plies group.

Producibility

Provides:

the capability to determine whether the ply/cut-piece shape would fit into the given material roll width of the ply.

the capability to create a dart within the command.

Flattening

Provides the capability to determine whether the ply shape would fit into the given material roll width of the ply.

Core and Cavity Design

New Functionality

Chordal Fillet

This Generative Shape Design functionality allows you to create chordal fillets..

Enhanced Functionalities

Fillet Radius Reduction

The dialog box has been simplified, allowing you to process fillets belonging to surfaces or faces, or located in one Geometrical Set.

The naming constraint on this Geometrical Set has been removed.

You can choose the name and color of the Geometric Set that will contain the reduced fillet.

Functional Molded Part

New Functionalities

Scan Integration

You can scan the part. Integrating the Functional Features in the Scan process gives a quick and efficient way to understand the design of a Functional Molded Part. It is useful for the analysis of the better understanding of the complex design.

Creating Technological Results

This capability supplies technological information about the features included in a body. This information can then be reused at any stage of your design by downstream applications.

Creating Chordal Fillets

The Chordal Fillet command is used to control the width of the fillet (distance between the 2 rolling edges) which is also called as Chordal length.

Enhanced Functionalities

Creating Edge Fillets

The Conical parameter option allows you to vary the section of the fillet.

Creating Face-Face Fillets

The Conical parameter option allows you to vary the section of the fillet.

Creating Variable Radius Fillets

The Conical parameter option allows you to vary the section of the fillet.

Fillet face-face hold-curve with keep edge

You can specify the edge(s) of support body to be kept and creates Fillet Surfaces with Hold Curve and Keep Edge as support wherever necessary.

Ability to select the draft properties of other bodies

You can access the draft property which is defined in another solid functional set. All draft properties are displayed and can be selected for feature definition even though draft properties are defined in different solid functional sets.

Display Body Under Functional Features

It will simplify the specification tree and improve management of the insertion point by providing a mechanism for users to gather bi-dimensional and tri-dimensional elements within the same work environment. The bodies referenced by FMP features will be displayed in the specification tree only under the pointing feature.

Importing a Single Specification

A Functional Import consists of a copy of geometry from one part to another part.

To Shell Option

To Shell option for length in functional features will be available only when shellable feature is present in solid functional set. If no shellable feature is present in PartBody, the To Shell option will not be displayed in the length for concerned functional features.

Workbench simplification

This improves the usability by simplifying the Functional Molded Part workbench with less icons. Only important features are presented as icons and all other features are accessed through the menus. This improves the ergonomics of the Functional Molded Part workbench. You need not switch over another workbench for accessing certain features.

Support of all 3D sketched geometries

You can select 2Doutput, 2DoutputProfile and 2DlayoutProfile for the input profile definition wherever applicable and also for limiting element.

Generative Sheetmetal Design

New Functionality

Isometries

You can make a basic transformation operations like translation, rotation, symmetry or axis to axis transformation.

Enhanced Functionalities

Extruding

You can select a profile containing any kind of geometry, invert the extrusion direction, specify the fixed area, automatic bend, automatic connection to the sheet metal part, tears selection, KFactor management, exploded mode with the options added in the Extrusion Definition dialog box.

Creating wall with middle position option

You can define the wall sketch at the middle of the thickness.

Creating multi tangent wall

You can connect a wall to several faces. In order to define the multi connected wall, you can select in Tangent to frame: top faces, bottom faces, and sheetmetal features.

Creating a Hopper

You can create a hopper from a double curvature surface.

Generative Drafting

New Functionalities

View callout re-creation

This capability allows you to recreate the view callouts.

PLM compliant batch drawing update

This capability allows you to update a list of CATDrawing documents which are file or ENOVIA LCA based in batch environment.

Propagation of view definition modifications

This capability allows you to propagate the view modifications to other views.

Views of scenes saved in Technological Package

This capability allows the creation of views of scenes saved in Technological Packages (ENOVIA LCA, ENOVIA VPM).

Setting Small Scale in CATIA V5 Session

The Small Scale feature allows you to create smaller geometries with a high accuracy of less than 1 micron. In drafting there are a few points which you should consider.

Enhanced Functionalities

Persistency of the last used Generative View Style

Last used generative view style is given as default option for next view creation.

Thread dimension enhancement

The 3D standard management for thread dimension generation has been enhanced.

Detail view from Broken view

This capability allows the creation of detail view from a broken view.

Detail and quick detail view

Clipping view / Clipping view profile

This capability allows the creation of detail and quick detail views on a view where clipping is applied.

Quick clipping view / Quick clipping view profile

This capability allows the creation of detail and quick detail views on a view where clipping is applied.

Profile associativity

Offset Section view / cut

The section profile can be created using the new Tools Palette which keeps profile associative to the geometry.

Aligned Section view / cut

The section profile can be created using the new Tools Palette which keeps profile associative to the geometry.

Section profile start point repositioning

Offset Section view / cut

The start point of the section profile can be repositioned.

Aligned Section view / cut

The start point of the section profile can be repositioned.

Customizing Settings

2D generative view from FTA property to control validity checking of generated annotations

This option allows you to visualize the crossed annotations in 2D generative view while creating View from 3D.

Healing Assistant

Enhanced Functionality

Compare Parts

You have now the choice between a quick comparison that analyzes surfaces and a detailed comparison that analyzes areas.

Interactive Drafting

New Functionalities

Reordering sheets or views

Reorder sheets or views in the required sequence.

Analyzing View Geometries

This capability allows you to analyze view geometries whether it is under-constrained, over-constrained or iso-constrained. Complete information about every element of the geometry or constraint of a view is given.

Setting Small Scale in CATIA V5 Session

The Small Scale feature allows you to create smaller geometries with a high accuracy of less than 1 micron. In drafting there are a few points which you should consider.

Enhanced Functionalities

Coordinate dimension value display format

This capability allows you to define the display format of the coordinate dimensions. The new Numerical Properties toolbar manages the display format for all the dimensions including the coordinate dimensions.

True Type Fonts support on UNIX

The True Type fonts are now supported on UNIX also.

Thread dimension enhancement

The thread dimensions on generated views are created as feature dimensions with the color of display as the one selected for the Technological feature dimensions type.

Annotation blanking

This capability allows you to blank geometries in addition to area fill. Blanking can also be defined for both dimensions and leader symbols.

Mold Tooling Design

New Functionalities

Generate Final Holes & Pads

In order to improve time performances, lets you use a simplified representation of holes and pads throughout the construction of the mold and then replace the simplified representations by the actual holes and pads.

Replace Plate

This command enables you to replace a mold plate by another one while keeping the consistency of the mold.

Enhanced Functionalities

Generic dialog box

The behavior of the Manage All option has been improved and now applies to rotation, axis modification and direction inversion of the components.

You can adapt the geometrical parameters of a component to your mold context.

Explode Holes

This tool is no longer available as technological results make this tool obsolete.

Part Design

New Functionalities

Creating Chordal Fillets

The Chordal Fillet command is used to control the width of the fillet (distance between the 2 rolling edges) which is also called as Chordal length.

Transforming Elements From an Axis to Another

You can transform geometry positioned according to a given axis system into a new axis system. The geometry is duplicated and positioned according to the new axis system.

Transforming Geometry by Affinity

You can transform geometry by means of an affinity operation.

Enhanced Functionalities

Creating Edge Fillets

The Conical parameter option allows you to vary the section of the fillet.

Creating Face-Face Fillets

The Conical parameter option allows you to vary the section of the fillet.

Creating Variable Radius Fillets

The Conical parameter option allows you to vary the section of the fillet.

Thread Up To Plane & Support Depth

You can use a planar geometrical element as limiting thread/tap depth element or set the thread depth with the length of the support. This capability is available for Creating Threads and Taps and Creating Threaded Holes.

Editing Copy/Paste Body Link

This capability gives more flexibility to the generic solid feature generated with a Copy/Paste Special as Result with Link operation on a body, by being able to edit it.

Reordering Features

This capability aims at replugging features that use sub-elements (such as edges, faces and vertices) on the features on which they resolve themselves.

Customizing Settings

Inherit Colors from the Reference Feature

This option assigns the color of the reference feature to its copy obtained using Copy Paste As Result With Link.

Setting Small Scale

The Small Scale option allows you to create smaller geometries with a high accuracy of less than 1 micron.

Sketcher

Enhanced Functionality

Displaying Tolerances

- Tolerance values are now displayed on dimensional constraints in the graphical window.
- You can now easily edit tolerance values applied on dimensional constraints using the Edit Multi-Constraint dialog box.

Removed Functionality

Fix Together

When editing a Fix Together constraint or creating a new Fix Together constraint, the Fix Together Definition dialog box appears without the Add/Remove Dependencies check box.

Wireframe and Surfaces

New Functionalities

Manually Updating a Part

Allows you to manually update a part that has just been modified.

Setting Small Scale Support

Allows you to define a unique geometrical range from 0.01m to 1m.

Enhanced Functionalities

Creating Points

Allows you to create points using the Compass Location button.

Splitting Geometry

Ignore no intersecting elements option allows you to split many surfaces when the cutting element does not intersect all the surfaces, while Keep elements in half space option allows you to keep the elements in one half space defined by the cutting plane.

Creating Filling Surfaces

Displays gaps present on surfaces, and allows you to fill the gaps.

Connect Checker

Allows you to check connections between surfaces, between curves, and between curves and surfaces. Also allows you to set tolerance values.

Automotive Body In White Fastening

New Functionalities

Measuring Distance Between Spot Fasteners

This capability allows you to measure the linear and curvilinear distance between spot fasteners.

Creating Fastener Annotations Automatically

This capability allows you to create BiW fastener annotation automatically at the time of fastener creation or edition.

Editing Diameter using Formula

This capability allows you to specify the diameter using formula.

Enhanced Functionalities

Repeating Spot Points

This capability allows you to maintain the distance between last created spot point and the reference end point.

Customizing the BiW Fastening Application Standard File

An additional customized sample GBF_STD file is supported.

Customizing Settings

General

This option allows you to lock the fastener identifier to avoid renaming it.

Digitized Shape Editor

New Functionalities

Distance Analysis

This command is replaced by Deviation Analysis.

Curve On Mesh

This command lets you create curves on mesh suitable for Reverse Engineering.

Enhanced Functionalities

Activate, Remove, Split

A new selection mode is available: Flood.

Import

Ascii RGB file format is now supported and lets you import color information.

Cloud Display Options

For cloud of points containing color information (e.g. imported from a Ascii RGB file), a new check box Colored lets you take this color information into account.

Align using the Compass

Selective Display has been replaced by a Hide/Show icon.

The option Keep Initial has been added.

Align by Best Fit

Selective Display has been replaced by a Hide/Show icon.

Activate icons have been added to activate areas on the clouds to process.

The option Keep Initial has been added.

Align with Constraints

Selective Display has been replaced by a Hide/Show icon.

The option Keep Initial has been added.

Align with RPS

Selective Display has been replaced by a Hide/Show icon.

The option Keep Initial has been added.

Align using Spheres

Selective Display has been replaced by a Hide/Show icon.

A Sphere icon has been added to start the definition of the spheres.

The option Keep Initial has been added.

Align with Previous Transformation

A dialog box has been created with a Hide/Show icon and the option Keep Initial. The message to accept alignment has been removed.

FreeStyle Shaper Optimizer & Profiler

New Functionalities

Connect Checker

Connect Checker

This section describes about the various CCK types between two curves and/or surfaces.

Check connections between surfaces

Select two surfaces, specify the type of analysis (distance, tangency, curvature) and set the analysis parameters.

Check connections between curves

Select two curves, specify the type of analysis (distance, tangency, curvature) and set the analysis parameters.

Checking connections between curves and surfaces

Select one curve and one surface, specify the type of analysis (distance, tangency, curvature) and set the analysis parameters.

Tolerances

This sections describes about Tolerance values.

Enhanced Functionalities

Control Points Commands

The new control point command should provide more standard behaviors such as selection, undo/redo, integration in CATIA.

Using Control Points

Selection and advanced customization of control points.

Creating Delta Analysis/Delta Deflection

Usage of options like Delta Analysis and Translator Delta Display.

Extend with negative value

Negative extension of curve.

Isophotes Analysis

The new options that have been added are: Multi band type, Show/hide the manipulator, Light Sources On/Off, and Highlight On/Off effect.

Untrim enhancements

Selection, Create Curves and Create Extract options have been added to the Untrim dialog box.

Break Enhancements

Breaking curves and surfaces.

Continuity Constraint Evolution

Creating FreeStyle continuity constraint.

Styling a Fillet

Create an ACA fillet surface between two given surfaces using Styling Fillet.

Draft Analysis Improvement

Performing draft analysis shader.

Generative Shape Design

New Functionalities

Manually Updating a Part

Allows you to manually update a part that has just been modified.

Creating Chordal Fillets

Allows you to create chordal fillets.

Styling a Fillet

Allows you create an ACA fillet surface between two given surfaces.

Setting Small Scale Support

Allows you to define a unique geometrical range from 0.01m to 1m.

Enhanced Functionalities

Creating Points

Allows you to create points using the Compass Location button.

Creating Edge Fillets

Allows you to vary the section of the fillet using the Conical parameter option. You can also create a tangency continuous intersection edge ensuring that the intersection edge remains tangency continuous even after update.

Creating Adaptive Swept Surfaces

Allows you to deactivate or activate adaptive sweep sections.

Creating Swept Surfaces

Allows you to fill twisted areas

Creating Fill Surfaces

Displays gaps on surfaces, and allows you to fill the gaps.

Extracting Geometry

Allows you to create a tangency continuous intersection edge ensuring that the intersection edge remains tangency continuous even after update.

Extracting Multiple Elements

Allows you to create a tangency continuous intersection edge ensuring that the intersection edge remains tangency continuous even after update.

Splitting Geometry

Ignore no intersecting elements option allows you to split many surfaces when the cutting element does not intersect all the surfaces, while Keep elements in half space option allows you to keep the elements in one half space defined by the cutting plane.

Creating a Diabolo

Allows you to create a diabolo by specifying different angles on various edges. You can also invert the orientation of the diabolo.

Creating a Bead

Allows you to select a Reference Element to stabilize the orientation of the bead.

Creating a Mating Flange

Allows you to create a mating flange on a surface with a hole.

Unfolding a Surface

Allows you to unfold a surface even if the reference origin lies on a edge to tear.

Reordering Features

Allows you to replugin features that use sub-elements (such as edges, faces, and vertices) on the features on which they resolve themselves.

Creating Blended Surfaces

Allows you to create a ruled developable surface.

Smoothing Deviation

Allows you to smooth out surfaces in order to perform 1 or 2 offsets while creating a thick surface, thickness, or shell.

Creating a Face-Face Fillet

Allows you to specify the edge(s) of support body to be kept, and creates Fillet Surfaces with Hold Curve and Keep Edge as support wherever necessary

Synchronization

Allows the Scan or Define In Work Object command to work independently on each side of the Comparison window.

Transferring Elements

Allows you to map any wireframe feature from 3D to 2D and vice versa.

Creating Variable Offset Surfaces

Creating Variable Offset Surfaces

Allows you to offset a surface by specifying a variable or constant offset to sub-parts of a surface. You can also reverse the offset direction.

Filleting Surfaces

Allows you to vary the section of the fillet using the Conical parameter option while creating face-face fillets, variable radius fillets and bitangent shape fillets.

Updating Parts

Allows you to view the parents of the element in error and thus understand the origin of the error.

Connect Checker

Allows you to check connections between surfaces, between curves, and between curves and surfaces. Also allows you to set tolerance values.

Imagine & Shape

New Functionalities

Dimensioning Subdivisions

This capability allows to control the dimension on a subdivision surface and resizing it.

Selecting Multiple Surfaces

This capability allows you to select multiple surfaces and perform modification.

Migrating Pre-R18 Features

This capability allows you to migrate pre-R18 elements to current level giving better integration and stability to IMA features.

Selecting Visible Elements

This capability allows you to hide the hidden elements thus making it easier to select only the visible elements in a subdivision surface.

Enhanced Functionalities

Extruding Faces and Edges

This capability allows you to automatically fill a hole with a undefined number of faces.

Aligning Vertices

This capability allows you to re-organize the vertices of a mesh i.e. project the vertices.

Apply Command

This capability allows you to validate the previewed selection and apply the command incase of extruding faces and edges, face cutting, face subdivision and erasing faces and edges.

Cursor improvements

The cursor movements have been enhanced to give better usability in face cutting, face subdivision and weight addition commands.

Quick Surface Reconstruction

New Functionalities

Deviation Check

Once you have performed a deviation analysis, you may want to add annotations on peculiar locations of this deviation analysis, or issue a report. Three new commands will help you do that: Annotation Set, Annotation and Deviation Report.

Curve On Mesh

This command lets you create curves on mesh suitable for Reverse Engineering.

Enhanced Functionalities

Activate, Remove, Split

A new selection mode is available: Flood.

Basic Surface Recognition

You can now select areas to process directly from the Basic Surface Recognition dialog box.

Sketch from Scan

You can now select meshes to process their free edges.

Non planar scans are now accepted as input.

Fixed split points are no longer proposed for closed scan, thus reducing the number of primitives and improving the result.

Deviation Analysis

More data can be selected as Reference data or data To Measure.

Once the analysis is started, deviation values are displayed automatically under the cursor.

An option Direction has been added to define the projection mode.

Modal values have been implemented in the color map.

Realistic Shape Optimizer

Enhanced Functionality

Digitized Morphing

You can now use a limit curve to restrict the area to deform.

Advanced Meshing Tools

New Functionalities

Mesh Visualization Tools

Visualizing Meshes and Groups

You can visualize meshes and group contents and modify the graphic properties of groups to increase the group visualization.

Hiding/Showing Meshes

You can show or hide the mesh result of a mesh part.

Enhanced Functionalities

Solid Meshing

Sweep 3D Mesher

Lets you capture neighboring lateral and top meshes to provide a solid mesh without interference.
You can initialize and preview the capture tolerance.

Mesh Transformations

Capture

You can initialize and check the capture tolerance in all the transformation and extrusion commands: Translation Mesher, Rotation Mesher, Symmetry Mesher, Extrude Mesher by Translation, Extrude Mesher by Rotation, Extrude Mesher by Symmetry, Extrude Mesher along a Spine.

Customizing Settings

General

Automatic naming of mesh parts, properties and user materials.

Generative Structural Analysis

New Functionality

Analysis Connections

Inserting Sets of Analysis Connections

You can insert several sets of analysis connections.

Enhanced Functionalities

Before You Begin

Overcome the 2Gb storage size limitation for .CATAnalysis documents.

Analysis Cases

Inserting Combined Cases

You can now select multi-load static solutions, assembled solutions and imported solutions.

Model Manager

Importing Composite Property

You can now take into account producibility defined in the *Composite Design* workbench.

XML Mapping Property

Mapping File Syntax

The syntax of the XML mapping file has been enhanced (new attributes; possibility to reference publications, geometrical sets and multi-output features...).

Exporting XML Mapping File

The properties already applied are now taken into account in the exported XML mapping file and you can select groups under mesh parts as support of the mapping property.

Connection Properties

About Connection Properties

You can release degrees of freedom on beam inside a user-defined connection properties except for the Beam-Spring-Beam combination.

Results Visualization

ELFINI Solver Log

A warning message is displayed when you launch a computation if equilibrium in the system is not reached.

Cut Plane Analysis

The last cut plane properties are stored in the settings and you can restore the default position and orientation of the cut plane.

Available Images

You can generate several images of failure under analysis solutions with different criteria: Hoffman, Maximum failure, Tresca, Tsai-Hill, Tsai-Wu and Von Mises criterion.

A new physical type is available: Failure criterion.

New images are available for the Stress physical type.

Failure Criteria

Formulas of the failure criterion images are described.

Exporting Data

You can export value locations of a user-defined axis system.

Customizing Settings

General

Automatic naming of mesh parts, connection properties, connection mesh parts and user materials.

Compartment and Access

New Functionality

You can divide a ship into compartments using the Compartment command. You can also modify compartments by replacing selected limits, split a compartment by a functional plate, and create a complex compartment from two or more existing compartments. You can also create boundaries for compartments automatically using the Boundary command.

Enhanced Functionality

Attribute Driven Graphic Replacement explains how to add attribute driven graphic replacement to 2-D drawings that you generate. The graphic replacement settings in your GVS file enable you to view boundary attributes. These boundary attributes display in your 2-D drawings as offset curves and patterns.

Electrical Cableway Routing

New Functionality

A design validation tool has been added. This allows you to check for specific errors in your design.

The Copy/Paste function, together with the Translate function, lets you copy and paste elements in the same design document.

You can also copy from one work package and paste into another.

You can use the Raceway and Conduit Design application to create cableway networks. Customizing Settings explains how to set Options on the Electrical Cableway Routing Tab to route cables through cableway networks created in Raceway and Conduit Design.

You can change the status of a partially routed cable to Complete Route using the Complete Partial Route Cable command.

Enhanced Functionality

You can load work packages from the light network to modify data using Load Enovia Document for Network Object command.

Electrical Connectivity Diagrams

New Functionality

You now have the capability of inserting an image in a title block, and editing the background view.

Moving the Middle Points of a Route explains how to move the middle points of a route using node manipulators.

Defining a Flexible Route explains how to define and view a flexible route in your document.

You can use the Import Component Image command to place images of a schematic object in different schematic documents.

Additionally, this task explains how you can copy multiple components in a schematic diagram, and paste images of these components in another diagram.

Electrical Harness Flattening

New Functionalities

Length Management

Allows you to apply or remove length tolerance to take into account bundle segment length adjustments made during the manufacturing process.

Network Assistant for validation before extracting

Before extraction, allows you to automatically or manually check the network for connection errors.

Enhanced Functionalities

Workbench Description

Toolbar commands have been re-organized and are now also available in menus.

Synchronizing the Environment

The orientation angle between the single insert connector and the backshell is synchronized with the 3D harness model.

Integrating Protective Coverings

Electrical Harness Flattening workbench commands (Extract, Flatten, Straight, Roll, etc.) are integrated with Protective Coverings. Also, Electrical Dress-Up commands are integrated with Protective Coverings.

Selecting Bundle Segments

You can now select a bundle segment even if it is covered by a protection.

Integrating the Straight Bend Mode

Allows you to extract, flatten, and manipulate branches built using the Straight Bend mode.

Flattening Orientation

Allows you to define the reference plane used for device orientation during flattening.

Customizing Settings

Electrical Harness Flattening

The default mode for the Extract operation is now Synchronize (without mechanical constraints).

Electrical Harness Installation

New Functionality

Adding Light Protective Coverings

Allows you to instantiate a new type of protective covering for bundle segments. For main differences with regular protective coverings, see Instantiating Protective Coverings.

Enhanced Functionalities

Editing bundle segments

Allows you to select a bundle segment even if it is covered with a protection.

Arranging Bundle Segments in Supports

Allows you to maintain the orientation of the bundle segments with respect to the support even if the support is moved. Rotating the support and orienting it in any direction keeps the arrangement relative to the new support position.

Status of the electrical connections when linking and unlinking

Allows you to view the status of the electrical connection when linking and unlinking bundle segments.

Creating a New Multi-Branchable Document and First Branch

New construction mode allows you to compute the bundle segment length from the centreline curve with a fixed minimum bend radius.

Electrical Package in Knowledge Expert

New rules and attributes are added to the electrical knowledge dictionary.

Customizing Settings

Electrical Harness Installation

New setting improves user experience by highlighting potential issues with segment geometry.

Electrical Library

New Functionalities

Defining Protection Products

Allows you to define a more flexible type of internal protective covering for bundle segments. For main differences with protection parts, see [Creating Protective Coverings](#).

Integration with Collaborative Enterprise Sourcing (CES) for placement of devices

Allows you to access the Collaborative Enterprise Sourcing (CES) environment for placement of devices using the [Manage Links](#) command or the [Smart Place](#) command.

Electrical Wire Routing

New Functionality

Network Assistant for validation before wire routing

Highlights the network and checks for connection errors before wire routing.

Enhanced Functionality

Automatically Routing Wires and Wire Groups

Allows you to automatically check the network for wires that are not routed and to repair them.

Customizing Settings

Electrical Wire Routing

New diameter computation mode allows you to obtain a better accuracy while computing the bundle segment diameter after wire routing.

Equipment Arrangement

New Functionality

A design validation tool has been added. This allows you to check for specific errors in your design.

The Copy/Paste function, together with the Translate function, lets you copy and paste elements in the same design document.

You can also copy from one work package and paste into another.

You can now generate an integration report for a schematic driven design. An integration report shows you the results of validation checks on your schematic and the 3D document generated from it.

The Measure command lets you measure distance between elements or along runs in your design.

Hanger Design

New Functionality

A design validation tool has been added. This allows you to check for specific errors in your design. The Measure command lets you measure distance between elements or along runs in your design.

Enhanced Functionality

The Penetration Management function has been moved to the DMU Space Analysis workbench. Although the function is available for you to use in this application, you must have a DMU Space Analysis license to be able to do so. Also, for documentation you need to refer to the DMU Space Analysis User's Guide.

HVAC Design

New Functionalities

A design validation tool has been added. This allows you to check for specific errors in your design.

The Copy/Paste function, together with the Translate function, lets you copy and paste elements in the same design document.

You can also copy from one work package and paste into another.

You can now generate an integration report for a schematic driven design. An integration report shows you the results of validation checks on your schematic and the 3D document generated from it.

The Measure command lets you measure distance between elements or along runs in your design.

You can adjust your layout by shifting parts in it, or by changing the angle of certain parts.

You can route parallel to an existing flexible, thereby creating a 'bundle'. You can create a bundle with a new route, or with existing routes. You can also add a flexible to an existing bundle, and modify a bundle.

Enhanced Functionality

The Penetration Management function has been moved to the DMU Space Analysis workbench. Although the function is available for you to use in this application, you must have a DMU Space Analysis license to be able to do so. Also, for documentation you need to refer to the DMU Space Analysis User's Guide.

HVAC Diagrams

New Functionalities

You now have the capability of inserting an image in a title block, editing the background view and updating a schematic view that has been created from a 3D document.

Moving the Middle Points of a Route explains how to move the middle points of a route using node manipulators.

Defining a Flexible Route explains how to define and view a flexible route in your document.

You can use the Import Component Image command to place images of a schematic object in different schematic documents.

Additionally, this task explains how you can copy multiple components in a schematic diagram, and paste images of these components in another diagram.

Piping and Instrumentation Diagrams

New Functionalities

You now have the capability of inserting an image in a title block, editing the background view and updating a schematic view that has been created from a 3D document.

Moving the Middle Points of a Route explains how to move the middle points of a route using node manipulators.

Defining a Flexible Route explains how to define and view a flexible route in your document.

You can use the Import Component Image command to place images of a schematic object in different schematic documents.

Additionally, this task explains how you can copy multiple components in a schematic diagram, and paste images of these components in another diagram.

Piping Design

New Functionalities

A design validation tool has been added. This allows you to check for specific errors in your design.

The Copy/Paste function, together with the Translate function, lets you copy and paste elements in the same design document.

You can also copy from one work package and paste into another.

You can now generate an integration report for a schematic driven design. An integration report shows you the results of validation checks on your schematic and the 3D document generated from it.

The Measure command lets you measure distance between elements or along runs in your design.

You can adjust your layout by shifting parts in it, or by changing the angle of certain parts.

Enhanced Functionality

The Penetration Management function has been moved to the DMU Space Analysis workbench. Although the function is available for you to use in this application, you must have a DMU Space Analysis license to be able to do so. Also, for documentation you need to refer to the DMU Space Analysis User's Guide.

Plant Layout

New Functionality

The Measure command lets you measure distance between elements or along routables in your design.

Raceway and Conduit Design

New Functionalities

A design validation tool has been added. This allows you to check for specific errors in your design.

The Copy/Paste function, together with the Translate function, lets you copy and paste elements in the same design document.

You can also copy from one work package and paste into another.

The Measure command lets you measure distance between elements or along runs in your design.

You can adjust your layout by shifting parts in it, or by changing the angle of certain parts.

You can route parallel to an existing flexible, thereby creating a 'bundle'. You can create a bundle with a new route, or with existing routes. You can also add a flexible to an existing bundle, and modify a bundle.

You can create cableway networks and use them with the Electrical Cableway Routing workbench. You can display and modify cableway network attributes defined in a raceway run, and create Raceway and Conduit networks that support the Electrical Cable Database.

Enhanced Functionality

The Penetration Management function has been moved to the DMU Space Analysis workbench. Although the function is available for you to use in this application, you must have a DMU Space Analysis license to be able to do so. Also, for documentation you need to refer to the DMU Space Analysis User's Guide.

Ship Structure Detail Design

New Functionality

You can create an opening with a standard opening. You can multi-select references and create multiple openings.

Enhanced Functionalities

You can define the offset value of each plate limit. A positive value adds material. A negative value removes material. Additionally, limit tags display in your 3-D session to assist with plate limit management.

You can multi-select objects and apply an opening to more than one object at a time when creating an opening using a sketch and creating an opening with a 3D object.

You can define the offset value of each stiffener on free edge limit.

You can define the offset value of each stiffener limit.

Using the Structure Detailing Features Catalog task explains how you modify Chapter and Family information (and add Custom and Built-in families) in the sample catalog, StructureDetailingFeatures.catalog (a catalog of slots, endcuts and small assemblies).

Adding Slots, Endcuts and Small Assemblies Templates to a Catalog shows you how to store new user defined features (for slots, endcuts and small assemblies) in the sample catalog called, StructureDetailingFeatures.catalog.

You can select multiple penetrated objects when placing slots in your 3-D session.

You can create beams and pillars using point and limit, point and length, and two beams and a plane. Additionally, Beam dialog box (Beam command) allows you to select from a list of GSD stacking commands.

When generating graphic replacement for reference plane systems and structural objects, the XML file now offers versioning for future changes, consistent terminology, and better organization to better customize the file.

Slots and endcuts are preserved, where possible, when using copy and paste with a plate, opening, or stiffener.

Structure Functional Design

New Functionalities

You can automatically generate a CATIA mesh model, containing appropriate mesh constraints and properties, from a Structure design model. This makes for a more efficient Finite Element Model (FEM) creation process, compared to a manual approach. You can create an opening with a standard opening. You can multi-select references and create multiple openings.

Enhanced Functionalities

You can define the offset value of each plate limit. A positive value adds material. A negative value removes material. Additionally, limit tags display in your 3-D session to assist with plate limit management.

You can multi-select objects and apply an opening to more than one object at a time when creating an opening using a sketch and creating an opening with a 3D object.

You can define the offset value of each stiffener on free edge limit.

You can define the offset value of each stiffener limit.

Using the Structure Detailing Features Catalog task explains how you modify Chapter and Family information (and add Custom and Built-in families) in the sample catalog, StructureDetailingFeatures.catalog (a catalog of slots, endcuts and small assemblies).

Adding Slots, Endcuts and Small Assemblies Templates to a Catalog shows you how to store new user defined features (for slots, endcuts and small assemblies) in the sample catalog called, StructureDetailingFeatures.catalog.

You can select multiple penetrated objects when placing slots in your 3-D session.

You can create beams and pillars using point and limit, point and length, and two beams and a plane. Additionally, Beam dialog box (Beam command) allows you to select from a list of GSD stacking commands.

When generating graphic replacement for reference plane systems and structural objects, the XML file now offers versioning for future changes, consistent terminology, and better organization to better customize the file.

Slots and endcuts are preserved, where possible, when using copy and paste with a plate, opening, or stiffener.

Systems Routing

New Functionality

The Measure command lets you measure distance between elements or along runs in your design.

System Space Reservation

New Functionality

The Measure command lets you measure distance between elements or along routables in your design.

Tubing Design

New Functionalities

You can route parallel to an existing flexible, thereby creating a "bundle". Capabilities have been added to allow you to create a bundle with new or existing routes, add a flexible to an existing bundle, and modify a bundle.

A design validation tool has been added. This allows you to check for specific errors in your design.

The Copy/Paste function, together with the Translate function, lets you copy and paste elements in the same design document.

You can also copy from one work package and paste into another.

You can now generate an integration report for a schematic driven design. An integration report shows you the results of validation checks on your schematic and the 3D document generated from it.

The Measure command lets you measure distance between elements or along runs in your design.

You can adjust your layout by shifting parts in it, or by changing the angle of certain parts.

Enhanced Functionality

The Penetration Management function has been moved to the DMU Space Analysis workbench. Although the function is available for you to use in this application, you must have a DMU Space Analysis license to be able to do so. Also, for documentation you need to refer to the DMU Space Analysis User's Guide.

Tubing Diagrams

New Functionalities

You now have the capability of inserting an image in a title block, editing the background view and updating a schematic view that has been created from a 3D document.

Moving the Middle Points of a Route explains how to move the middle points of a route using node manipulators.

Defining a Flexible Route explains how to define and view a flexible route in your document.

You can use the Import Component Image command to place images of a schematic object in different schematic documents.

Additionally, this task explains how you can copy multiple components in a schematic diagram, and paste images of these components in another diagram.

Waveguide Design

New Functionalities

You can route parallel to an existing flexible, thereby creating a "bundle". Capabilities have been added to allow you to create a bundle with new or existing routes, add a flexible to an existing bundle, and modify a bundle.

A design validation tool has been added. This allows you to check for specific errors in your design.

The Copy/Paste function, together with the Translate function, lets you copy and paste elements in the same design document.

You can also copy from one work package and paste into another.

You can now generate an integration report for a schematic driven design. An integration report shows you the results of validation checks on your schematic and the 3D document generated from it.

You can adjust your layout by shifting parts in it, or by changing the angle of certain parts.

Enhanced Functionality

The Penetration Management function has been moved to the DMU Space Analysis workbench. Although the function is available for you to use in this application, you must have a DMU Space Analysis license to be able to do so. Also, for documentation you need to refer to the DMU Space Analysis User's Guide.

Waveguide Diagrams

New Functionalities

You now have the capability of inserting an image in a title block, editing the background view and updating a schematic view that has been created from a 3D document.

Moving the Middle Points of a Route explains how to move the middle points of a route using node manipulators.

Defining a Flexible Route explains how to define and view a flexible route in your document.

You can use the Import Component Image command to place images of a schematic object in different schematic documents.

Additionally, this task explains how you can copy multiple components in a schematic diagram, and paste images of these components in another diagram.

Circuit Board Design

New Functionality

Defining an Axis System

Allows you to define the origin of the board before exporting as IDF files.

Enhanced Functionalities

Exporting Data (as IDF)

Allows you to perform the IDF Export independent of the language of the session.

Exporting Data (as IDF)

Allows you to export CBD objects built on pads based on 3D curves.

Integration of Constraint Area patterns

Allows you to export/import patterned pads to/from IDF files.

3 Axis Surface Machining

New Functionalities

3/5-Axis Converter

Available in the dialog boxes of the Roughing, Sweeping, Pencil, ZLevel, Contour-driven, Spiral Milling, Isoparametric Machining and Profile Contouring operations, this is an extra process that can be included in the tool path computation of those operations. It allows the collision checking of the tool holder with different strategies to avoid these collisions (5-axis motion or tool path split and reconnection) and a global conversion of a 3-axis tool path in a 5-axis tool path.

Enable multi-editing of machining operations of the same type

It is now possible to perform multi-edition of certain parameters on two or more machining operations of the same type. This capability is available on Multi-Axis Flank Contouring, Multi-Axis Curve Machining, Isoparametric Machining, and Profile Contouring.

Enhanced Functionalities

Definition of the tool axis, machining direction or line or axis direction, ...

An icon has been added in the dialog box to set those directions directly to the normal to the screen, in Manual mode.

Split on Collision Points

You can now select several tools to perform the Split on Collisions Points.

Options are proposed to allow splitting the tool path between an approach and a retract motion and to close automatically the resulting tool paths.

Defining macros

PP words can be added.

ZLevel

TSlotter tool is now available and may be used as a torical tool,

Enhancement of the management of undercut areas of a part,

The TSlotter tool checks collisions with the shank of the tool (allowance value).

Machining with tool tip in constant plane perpendicular to the tool axis is now possible.

Variable distance between planes has been added.

It is now possible to add Axial + Radial motions in approaches and retract movements to avoid collisions.

Between path HSM macro is now composed by 3 elementary motions.

Advanced Machining

New Functionalities

Multi-Axis Spiral Milling

New operation is provided to machine pockets (with computation of bitangency between the pocket bottom and the pocket side) or to engrave complex surfaces.

Sequential Axial and SEquential Groove Operations

Two new operations are provided in order to define all elementary motions and PP word syntaxes to be applied at each point of a machining pattern of holes or grooves.

4-axis Pocket Machining Operation

New 4-axis Pocketing operation is provided in order to compute a tool path to mill a pocket on a cylindrical or conical surface.

3/5-Axis Converter

Available in the dialog boxes of the Roughing, Sweeping, Pencil, ZLevel, Contour-driven, Spiral Milling, Isoparametric Machining and Profile Contouring operations, this is an extra process that can be included in the tool path computation of those operations. It allows the collision checking of the tool holder with different strategies to avoid these collisions (5-axis motion or tool path split and reconnection) and a global conversion of a 3-axis tool path in a 5-axis tool path.

Enable multi-editing of machining operations of the same type

It is now possible to perform multi-edition of certain parameters on two or more machining operations of the same type. This capability is available on Multi-Axis Flank Contouring, Multi-Axis Curve Machining, Isoparametric Machining, and Profile Contouring.

Enhanced Functionalities

Improved drive element selection in Multi-Axis Flank Contouring operation

A new command in the Face Selection toolbar allows selecting the same face more than once during drive element selection in Multi-axis Flank Contouring operations.

Multi-Axis Helix Machining

The definition of interpolation axes has been made easier and proposes an interference check.

Definition of the tool axis, machining direction or line or axis direction

An icon has been added in the dialog box to set those directions directly to the normal to the screen, in Manual mode.

Split on Collision Points

You can now select several tools to perform the Split on Collisions Points.

Options are proposed to allow splitting the tool path between an approach and a retract motion and to close automatically the resulting tool paths.

ZLevel

TSlotter tool is now available and may be used as a torical tool,

Enhancement of the management of undercut areas of a part,

The TSlotter tool checks collisions with the shank of the tool (allowance value).

Machining with tool tip in constant plane perpendicular to the tool axis is now possible.

Variable distance between planes has been added.

It is now possible to add Axial + Radial motions in approaches and retract movements to avoid collisions.

Between path HSM macro is now composed by 3 elementary motions.

Multi-Axis Curve Machining

The definition of interpolation axes has been made easier and proposes a check of interferences.

A new tool axis guidance strategy is available: Thru a guide.

In Contact and in Between curve and part modes, you can now select a plane as the part to machine.

Multi-Axis Sweeping

A new tool is available: barrel tool.

A new tool axis guidance strategy is available: Thru a guide.

You can now select a limiting contour with a possible offset.

Multi-Axis Contour Driven

A new tool axis guidance strategy is available: Thru a guide.

You can now select a limiting contour with a possible offset.

Multi-Axis Tube Machining

Guiding strategy: Along guide has been added.

Driving tool points: two computation methods are now available: Tool tip or Contact on part.

Zone: three machining zones are now available: Tube, Cavity, Tube and cavity.

Isoparametric Machining

Isoparametric Machining

The definition of interpolation axes has been made easier and proposes a check of interferences.

A new tool axis guidance strategy is available: Thru a guide.

Tool compensation as Guiding Point

New option Compensation application mode: Guiding point/Output point is available on all axial machining operations.

Allows managing toolpath computation according to the tool compensation point.

Support of Helical strategy in Profile Contouring

Helix tool path style is now possible in Between two Planes mode.

Helix approach macro

When a cutter approaches raw material, a new Helix approach macro can be used rather than a Ramping approach macro. This capability is available in Pocketing, Profile Contouring, Multi-axis Curve Machining and Multi-Axis Flank Contouring operations.

Defining macros

PP words can be added.

Capability to manage machinable features in Manufacturing View

Capability allows user to sort and filter Machinable Axial features and Machinable Milling features. Also user can now delete unused Machinable Area features.

Lathe Machining

Enhanced Functionalities

Reference Point in Sequential turning

The tool positioning for the first motion of a Sequential Turning Operation with respect to TO/ON/PAST can now be specified according to a Reference Point.

Single center plunge for narrow groove in Groove Turning

New Single plunge option for center plunge (suitable for machining a narrow groove).

Choice between linear and angular feedrate units

Feedrates can now be set in linear or angular units.

Macro Local Feedrate

Local feedrates for macros can now be set in linear or angular units.

Enhanced behavior of Linking Macro in Groove Turning

Optimization of interrupted linking macro by returning to next cutting position instead of previous tool path).

New interrupt option in Groove Turning

Linking macro can now be interrupted after a number of specified levels.

Optimized feedrates in Grooving by level mode in Groove Turning

Optimization of feedrates during return motion at the end of a level thanks to a Max approach distance parameter.

Groove bottom conditions for Groove Finish Turning

The sum of the Clearance and Overlap should be less than or equal to the groove bottom width. Otherwise a warning message is issued.

Groove Insert Positioning on Start and End Limits

More accurate Groove insert positioning, with respect to the IN/ON/OUT setting on the Start/End Limits, while taking into account the tool compensation number and the insert geometry.

Stock extension for Rough Turning operations

New option in Geometry tab page for Rough Turning and Ramp Rough Turning to extend the stock profile in machining direction and to have the extension till desired length.

Improved Profile Contouring with asymmetric tool

This enhancement ensures complete profile finishing in a recess when using an asymmetric groove tool whose nose radius (left or right) greater than half its width.

Capability to compute maximum turning envelope

New command Maximum turning envelope provides a quick and efficient way to compute the Turning profile of a part.

Support approach and retract motion for stock computation

Capability incorporates Approach/Retract motions in the Stock Computations, leading to improved Stock Update.

Reference compensation point for the gauges on Lathe tool assembly

User can now specify the reference tool compensation point on which the gauge values are defined.

It provides new output parameters to define in lathe tool change and lathe reference point modification syntaxes (linked to PP word table). The syntaxes are generated with parameter values that are automatically computed according to the active compensation point.

Automatic stock computation for the last operation of the program

User can now compute the final Stock after the last operation in the Program has been performed by means of the Compute Final Stock command in the contextual menu of the Program.

Multi-Axis Surface Machining

New Functionality

Multi-Axis Spiral Milling

Enables you to machine pockets (with computation of bitangency between the pocket bottom and the pocket side) or to engrave complex surfaces.

Enhanced Functionalities

Definition of the tool axis, machining direction or line or axis direction, ...

An icon has been added in the dialog box to set those directions directly to the normal to the screen, in Manual mode.

Enable multi-editing of machining operations of the same type

It is now possible to perform multi-edition of certain parameters on two or more machining operations of the same type. This capability is available on Multi-Axis Flank Contouring, Multi-Axis Curve Machining, Isoparametric Machining, and Profile Contouring.

Multi-Axis Curve Machining

The definition of interpolation axes has been made easier and proposes a check of interferences.

A new tool axis guidance strategy is available: Thru a guide.

In Contact and in Between curve and part modes, you can now select a plane as the part to machine.

Multi-Axis Sweeping

A new tool is available: barrel tool.

A new tool axis guidance strategy is available: Thru a guide.

You can now select a limiting contour with a possible offset.

Multi-Axis Contour Driven

A new tool axis guidance strategy is available: Thru a guide.

You can now select a limiting contour with a possible offset.

Multi-Axis Tube Machining

Guiding strategy: Along guide has been added.

Driving tool points: two computation methods are now available: Tool tip or Contact on part.

Zone: three machining zones are now available: Tube, Cavity, Tube and cavity.

Isoparametric Machining

The definition of interpolation axes has been made easier and proposes a check of interferences.

A new tool axis guidance strategy is available: Thru a guide.

Multi-Pocket Machining

Enhanced Functionality

Definition of the tool axis, machining direction or line or axis direction, ...

An icon has been added in the dialog box to set those directions directly to the normal to the screen, in **Manual** mode.

Multi-Slide Lathe Machining

Enhanced Functionality

Time-based video simulation support for Multi-Slide Lathe Machines

Capability to perform time-based video simulations of multi-turret and multi-spindle scenarios with synchronized operations as defined in the Machining Gantt viewer.

The user can set the time step for video simulation for customized visualization.

Other video functionalities such as Save Video Result as CATProduct, collision lists, and video analysis commands are also available.

NC Machine Tool Builder

New Functionalities

New MillTurn Machine

This new functionality is available from the New Creation toolbar.

New Spindle

This new functionality is available from the New Creation toolbar.

New Turret

This new functionality is available from the New Creation toolbar.

Insert Spindle

This new functionality is available from the Component Management toolbar.

Insert Turret

This new functionality is available from the Component Management toolbar.

Remove Spindle

This new functionality is available from the Component Management toolbar.

Remove Turret

This new functionality is available from the Component Management toolbar.

Create Mount Point

This new functionality is available from the Device Attributes toolbar.

These commands provide the ability to create a Mill Turn Machine, create turrets and spindles and define the placement with the Mount Tool Point and add specifications as described in Creating a MillTurn Machine.

NC Machine Tool Simulation

New Functionalities

Multi-resource Machine Simulation command

This command enables users to simulate operations on lathe and mill-turning machines.

Generate Simulations in Batch Mode

This command enables users to select aspects of a simulation to run in batch mode so that the user can save the time required to visualize each simulation.

Enhanced Functionality

The Fault List command works for mill-turn machines

The Fault List command was called the Collision List command in previous releases. It has been renamed because it now lists faults that are not collisions, such as violation of travel limitations. The Fault List command has been enhanced so that collisions can be detected after Multi-resource Machine Simulations have been run.

Subprograms can be generated in NC Code

This is handled by the postprocessor and can be taken into account during simulation and collision checking.

NC Manufacturing Infrastructure

Enhanced Functionalities

Support chamfers on T-Slotter tool

It is now possible to define four additional parameters on a T-slotter tool:

- Top and Bottom Chamfer angles
- Diameters at the top and bottom.

Support kinematic Multi-slide lathe machine on the Part Operation

It is now possible to assign a kinematic Multi-slide lathe machine to a Part operation. This machine must be a CATProduct representation that was created using the NC Machine Tool Builder product.

Support tool assignments to Turret-Stations for a Manufacturing Program

Capability to add tools/tool assemblies present in the Resource list to a turret and also view the tools and their stations for a specific turret using a Turret Station editor.

Ability to select/modify a station for a tool on Tool-change activity

Capability to select or modify a station for a tool/tool assembly on a Tool Change activity.

This change will be reflected in the Turret Station editor of the corresponding Manufacturing Program.

Capability to associate Video Result CATProduct to a machining operation

Possibility to associate Video Result CATProduct to an operation:

- by using the Associate Video Result to Machining Operation command after a material removal simulation using the Full Video command or Video from last saved result command
- by using Associate a Video Result to the last MO of the program option in the Generate NC Output Interactively or Generate NC Output in Batch Mode dialog box.

Capability to load Video Result CATProduct associated to a machining operation for analysis

Load Simulation Result command is now available in the contextual menu of a machining operation which has an associated Video Result CATProduct. This command opens the Video Result CATProduct in a Video window for analysis of the machined stock, collision results, and so on.

Sub-program support for NC code based simulation

NC code based simulation of ISO file containing subprograms or ISO file split into several smaller files.

Support of barrel mill tool

New barrel mill tool is available and can be used in Multi-axis Sweeping operations.

Enable multi-editing of machining operations of the same type

It is now possible to perform multi-edition of certain parameters on two or more machining operations of the same type. This capability is available on Multi-Axis Flank Contouring, Multi-Axis Curve Machining, Isoparametric Machining, and Profile Contouring.

Identical Parts Machining

New functionalities allow methodology for defining the process for machining an assembly of identical or similar parts.

Copy Program: extends the Copy Transformation function to Manufacturing Program.

Merge Program: allows merging machining programs.

Improved management of NURBS output for machining operations

Capability to choose which operations will be output in APT with NURBS format and which ones will not contain NURBS.

Change feedrate and tool gage without recomputing the toolpath

Capability to modify feedrates, spindle speeds and tool gage G1 without breaking a toolpath.

The toolpath is updated with the new values.

Improved Performance of Toolpath Analysis

Subsequent polygonal rendering for the users are now significantly faster or instantaneous when compared to the first time polygonal rendering.

Reference compensation point for the gauges on Lathe tool assembly

Capability to define the reference tool compensation point on which the gauges values are defined.

It provides new output parameters to define in lathe tool change and lathe reference point modification syntaxes (linked to the PP word table). The syntaxes are generated with parameter values that are automatically computed according to the active compensation point.

Capability to fixed Machining axis on part for Table rotations

A new option Machining Axis Origin with Fixed on the part after table rotation checkbox is available in Tools > Options > Machining > Output to specify that machining axis origin will be linked to the machined part when a table rotation is done. It is no longer necessary to create a machining axis and insert a Machining Axis Change after an activity on which a table rotation is done.

Create and import tool assemblies

It is now possible to create a tool assembly from scratch and also to import an existing Tool Assembly from a catalog or database without creating any machining activity.

Prismatic Machining

New Functionalities

Sequential Axial and Sequential Groove Operations

Two new operations are provided in order to define all elementary motions and PP word syntaxes to be applied at each point of a machining pattern of holes or grooves.

4-axis Pocket Machining Operation

This new 4-axis Pocketing operation is provided in order to compute a tool path to mill a pocket on a cylindrical or conical surface.

Enhanced Functionalities

Tool compensation as Guiding Point

New option Compensation application mode: Guiding point/Output point is available on all axial machining operations. Allows managing toolpath computation according to the tool compensation point.

Support of Helical strategy in Profile Contouring

Helix tool path style is now possible in Between two Planes mode.

Helix approach macro

When a cutter approaches raw material, a new Helix approach macro can be used rather than a Ramping approach macro. This capability is available in Pocketing and Profile Contouring operations.

Capability to manage machinable features in Manufacturing View

Capability allows user to sort and filter Machinable Axial features and Machinable Milling features. Also user can now delete unused Machinable Area features.

Prismatic Machining Preparation Assistant

Enhanced Functionality

Improved Milling Feature Recognition

When running Feature Recognition, the milling areas (with not planar top element) are now taken into account for feature creation (Prismatic Machining Area). Previously, conditional checking was done on top and bottom face. These conditions were replaced by stringent condition checking on bottom face only, whereas conditions are not applied on top face. The top face can be any surface, it does not need to be planar which is parallel to bottom face. Feature recognition will now provide a Prismatic Machining Area corresponding to a Complex Pocket. The boundary will be defined by a hard boundary only. An imaginary contour at the intersection of Hole and Slot, is automatically defined to close the hard contour.

STL Rapid Prototyping

Enhanced Functionality

Activate, Remove, Split

A new selection mode is available: Flood.

Business Process Knowledge Template

New Functionalities

Object Type Functions

AddExtension

Enables you to add a BKT type on all mechanical features and products.

RemoveExtension

Enables you to remove a BKT type on all mechanical features and products.

Enhanced Functionality

Supported Objects by the Typing Commands

The Typing commands now support an extended range of objects.

DMU Composites Review

Welcome to the DMU Composites Review User's Guide!

This guide is intended for users who need to become quickly familiar with the product.

This overview provides the following information:

- DMU Composites Review in a Nutshell
- Before Reading this Guide
- Getting the Most Out of this Guide
- Accessing Sample Documents
- Conventions Used in this Guide

DMU Composites Review in a Nutshell

DMU Composites Review aims at reviewing a CATProduct with Composites geometry in at least one of its CATParts.

Before Reading this Guide

Before reading this guide, you should be familiar with basic Version 5 concepts such as document windows, standard and view toolbars. Therefore, we recommend that you read the Infrastructure User's Guide that describes generic capabilities common to all Version 5 products. It also describes the general layout of V5 and the interoperability between workbenches.

You should also be familiar with Composites Design (CPD), Composites Engineering Design (CPE) or Composites Design for Manufacturing (CPM).

Getting the Most Out of this Guide

To get the most out of this guide, we suggest that you start reading the User Tasks section, which deals with handling all the product functions.

The Workbench Description section, which describes the DMU Composites Review workbench, will also certainly prove useful to find your way around the DMU Composites Review workbench.

Navigating in the Split View mode is recommended. This mode offers a framed layout allowing direct access from the table of contents to the information.

Accessing Sample Documents

To perform the scenarios, sample documents are provided all along this documentation. For more information on accessing sample documents, refer to Accessing Sample Documents in the Infrastructure User's Guide.

Conventions Used in this Guide

To learn more about the conventions used in this guide, please refer to Conventions section.

DMU Kinematics Simulator

New Functionality

Designing a V5 mechanism

Copying and pasting a mechanism

You can now easily copy and paste a mechanism within an open product in session.

DMU Navigator

Enhanced Functionalities

Inserting Components

You can now interrupt the activate terminal node process.

Running the CATDMUUtility Batch Process

3D XML format now supported as an output option for CATParts.

Running the CATDMUUtility2D Batch Process

3D XML format now supported as an output option for CATDrawings.

Saving as 3D XML

You can save documents as 3D XML files, explanations are included for exchanging 3D XML data between DMU Review and 3DLive.

3D XML

Settings for the usage of the 3D XML format have evolved.

DMU Optimizer

Enhanced Functionalities

Cutting through a Model with 3D Cut

3D Cut functionality now takes advantage of a new Boolean computation to cut the triangles on the box limits.

More about the Spatial Split option

Spatial split now uses a new algorithm which improves the quality of the envelope.

DMU Space Analysis

New Functionalities

Penetration Management Tools

Are now available within DMU Space Analysis (in User Tasks section).

Enhanced Functionalities

3D XML Alignment

Documents containing sections and/or measurements made with **Measure Between** or **Measure Item** can be exported in the new 3D XML format version. This enables upward compatibility with CATIA products.

Measure Tools

Measure Between

It is now possible to display selection content and components under the form of annotations in the geometry area 3D using the **Customize** button.

Knowledgware Capabilities

Knowledgware Rule-based Clash

You can now optimize your clash rules, refer to Optimizing Rule-based clash.

Customizing Settings

XML Settings

To improve the visualization of clash results in XML files: XML settings have been enriched with two new options (Zoom on interferences (Second XML type) only and Transparent background available in **Tools > Options > Digital Mockup > DMU Space Analysis > DMU - Clash Process**)

To classify better XML result files, the Aggregate files in folders option has been added in **Tools > Options > Digital Mockup > DMU Space Analysis > DMU - Clash Process**.

First XML Type

Two new options are available: Transparent background and Aggregate files in folders.

Second XML Type

Visualization section has been created listing the various setting combinations to improve results visualization. XML Grammar section has been created.

DMU Space Analysis Interoperability

Exporting Clash Results Using First XML- Products from ENOVIA V5 database

A new scenario has been added to show how to export clash results in first XML format.

Exporting Clash Results Using Second XML - Products from ENOVIA V5 database

A new scenario has been added to show how to export clash results in second XML format.

Human Activity Analysis

Enhanced Functionality

Display additional NIOSH Information

Until today, the intermediate factors, the Recommended Weight Limit (WRL), and the Lifting Index (LI) in the Score section of the Lift/Lower Analysis panel were calculated internally by the software and never displayed. There is a need to display these values. This way, you can be fully informed of a particular factor change, say the frequency multiplier, and can appreciate precisely how any of the intermediate factors influences the final result.

Human Builder

New Functionality

Restoring an Attached Position of an Object

With restoring an attached position, the behavior of the attach functionality will not change. But when an object is first attached to a segment, the initial relative position between the segment and the object will be calculated and kept. You will be able to re-apply this relative position at any given point in time. If the object's position relative to its master segment has not changed, the object will not move; but if it has changed, the object will go back to its initial relative position.

Vehicle Occupant Accommodation (license required)

Vehicle Occupant Accommodation commands

The Vehicle Occupant Accommodation toolbar is only available with the Vehicle Occupant Accommodation product license.

Human Data Catalogs (licenses required)

Human Anthropometry Catalog

Human Posture Catalog

Human Preferred Angles Catalog

Manikin simulation and/or posture-creation can be time-consuming. The human catalogs improve the user experience and reduce simulation creation time.

Enhanced Functionality

The Vision Window Behavior

Previously, this vision behavior consists to keep the camera constantly at the horizontal according the world coordinates system. This means that, even if the manikin leans its head (Y in the local coordinates system), the view in the vision window will remain the same. You will always see the scene at the horizontal even if the manikin is bending its head. Now, the peripheral contour angle of the vision window will be readjust automatically to the head orientation.

Width and Type of Line Management for Constraints

You will be able to set up independently the type and thickness of the constraints view in the 3d screen, for the status for the width and line for constraints.

Enhanced Management of Look At behavior

You can explicitly choose which hand is to be the target of the Look At, the left, right, or either hand.

That way, you will be able to activate the right option from the start, and will not have to switch back and forth between the IK Mode (or Reach Mode) and the general options page.

Keeping the Settings of a Hand grasp

This stores the latest grasping posture angle specified in the Standard Pose dialog box for each posture type: spherical, cylindrical and pinch. These values will be then use as the default posture of the Hand Grasp command. In order to facilitate still more the hand positioning on an object, we also apply an appropriate offset for each posture type of the Hand Grasp. This will ease the hand positioning on an object while using the Reach command.

Manikin Constraints dialog box - Priority

Specific explanation of the priority settings in the dialog box.

Knowledge Advisor

New Functionality

Object Type Functions

ListAttributeNames

It returns a list of strings corresponding to the name of the available attributes for a given object.

Product Engineering Optimizer

New Functionality

Update Mode

The Update Mode allows you to choose the scope of the update used by the optimization algorithms ranging from global part update to local update (output only).

Product Knowledge Template

Enhanced Functionalities

Creating a Power Copy

- Add all authorized inputs command.

- While creating a Power Copy, you can automatically add all authorized inputs to the selected components.

- Reorder Power Copy inputs at creation time.

- While creating a Power Copy, you can reorder inputs at creation time and keep this order at instantiation time.

Creating a User Feature

- Add all authorized inputs command.

- While creating a User Feature, you can automatically add all authorized inputs to the selected components.

- Reorder User Feature inputs at creation time

- While creating a User Feature, you can reorder inputs at creation time and keep this order at instantiation time.

New Functionalities

Generative Functions

CreatePathString

- Enables you to generate a path string in order to access either a feature or a publication for a given assembly.

CreatePublication

- Enables you to publish either a feature or a publication for a given product.

CreateConstraint

- Enables you to create a constraint for a given product.

ManageInstance

- It returns the created or replaced product instance.

RemoveInstance

- Allows you to remove a product instance.