CATIA Solutions Version 5 Release 19 Product Enhancement Overview

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- Product Knowledge Template

Infrastructure User's Guide

Enhanced Functionalities

Selecting Using the Selection Traps
>The **Polygon Selection Mode** now lets you select concave polygons.
>Ability to pan or zoom the model when working in **Polygon Selection** Mode.

Capturing Simple Images
>You can now choose the monochrome filter to be applied when capturing images in monochrome color mode.

Dragging and Dropping Objects onto Objects
>You can now drag an element from one location and drop it onto another location without changing its name.

Knowledgeware

Read-only design table edition
>You can now open a design table document (in read-only mode)
>
>whenever you want to refer to the file. By clicking **Edit** button available
>
>in design table editor dialog box, you can access the design table file even if you do not have
>
>write access to the file or if it is locked by another user.

Excel 2007 support
>Now the new Excel extensions (.xlsx and .xlsm) of Microsoft Excel
>
>2007 will be supported in the entire existing Knowledge GUI dealing
>
>with Excel worksheet documents.

Data Exchange Interface

Customizing Settings

STEP

    Cloud Of Points Properties (COPS) are available.

    You can decide to use Geometric or Assembly Validation Properties.

    Large Scale is taken into account by STEP settings.

    Export supports light assemblies.

IGES

    Large Scale is taken into account by IGES settings.

DXF

    Click here to read about the impact of changing scales.

    AutoCAD 2004 and AutoCAD 2007 are now supported.

    The semantic of 2D component (Detail/Ditto) can be exported.

IGES 2D

    Click here to read about the impact of changing scales.

ENOVIA-CATIA Interoperability

Enhanced Functionalities

Display Lock Owner in save messages
> Whenever a Save operation fails due to insufficient lock privileges, an error message is displayed indicating that the object has been locked. The name of the user who has locked the object is also displayed.

Improve Session Refresh command
> The enhanced capability of Refresh command allows you to retrieve the document from the database even if you have the modified the document in the session. Also you can visualize the iteration status of the document in the database before you retrieve the document.

Memorize error list window size
> Whenever you change size of the Reporting dialog box the changes will be stored in the CATSetting and consequent Reporting dialog box will be opened with settings preferred by you.

Improve Search and Filter in error list
> when an error message is displayed indicating insufficient lock privileges on an object, you can perform the Lock/Unlock operations from the Reporting dialog box using contextual menu.

Lock/Unlock object from Reporting dialog box
> Also you can search the impacted object in the database from the Reporting dialog box by using Search in DB button.

Move taken into account in refresh
> The enhanced capability of Refresh Documents command updates the modified position of an instance of a CATPart in an assembly while refreshing the document in the session.

Memory Management
> Memory Warning and Memory Stopper are used to warn you (the user) if maximal allowed use of memory is reached when saving a large data to ENOVIA V5 VPM, and opening of the data from the ENOVIA V5 VPM.

Saving Child Components Without Locking the Parent
> The CV5EV5_ADD_WITHOUT_PARENT_LOCK parameter is replaced by the new CV5EV5_PARENT_LOCK_REQUIRED parameter.

Photo Studio

Enhanced Functionalities

Rendering Images
> Allows you to store your favorite scenes, or customize a scene and save it.

Using Materials for Rendering
> Allows you to apply different types of materials to an object.

Photo Studio Optimizer

No enhancements in this release.

Product Data Filtering

No enhancements in this release.

Product Structure

Enhanced Functionality

Impacts Handling on Instance Name or Publication Name Change

You can handle impacts on publication or product instance renaming.

Real Time Rendering

New Functionalities

Material Variants
> Allows you to apply material variants to an object and also to browse variant for interactive reviews. It can also be store in a DMU Review allowing complex scenario where materials, colors and shapes can we switches all together.

Environment Selector
> A new dedicated command available in the workbench allows you to change de default reflection map by choosing a new one in a thumbnail list.

Pre-computed Lightmaps
> With this enhancement you can compute shadow maps (ambient occlusion and various shadows computed with ray tracing algorithms). Those shadow maps can be applied in real time onto the model or sub-part of the model thanks to a new type of Sticker called "shadow stickers".

Enhanced Functionality

Architecture & Design Materials
> New advanced materials are available in the material user interface. They are more focused on Architecture & Design usage. They are classified by type (plastic, glass, metal,…) and comes with dedicated parameters easiest t tune. They can be visualized both in real time and ray tracing.

Environment Editor
> Allows you to edit all the environment's parameters including textures from a single frame.

Light shadows on objects
> Allows you to activate real time shadows onto the models with multiple lights and over advanced materials such as shaders providing highly realistic real time rendering

Resources & Content

Advanced Material library
> A new library of materials is available. This library is filled with several CgFX shaders providing advanced and realistic material effects.

SMARTEAM - CATIA Supply Chain Engineering Exchange

Enhanced Functionalities

GUI enhancements which improve Reconciliator's usability:
- Reconciliator window size & position are remembered
- Global-Set may be run by double-clicking

Reconciliation of CATShape documents:

The Reconciliator helps the customer to simulate and to prepare the data to be saved into the database. The Reconciliator supports a set of document types. The current enhancement extends the document types recognized by the Reconciliator to include the CATShape. This development extends the Reconciliator scope to applications such as Piping and tubing.

Exposed Reconciliator CAA APIs

Enables the development of CAA applications which integrate Reconciliator functionality and enables the customization of existing Reconciliator functionality by adding new criteria or implementing existing user criteria.

Save & Restore Reconciliation session

Saves and restores settings that were previously defined for mapping & rules of the

interactive Reconciliator application.

SMARTEAM - CATIA Integration

Enhanced Functionalities

SmarTeam – CATIA integration enables suppliers to handle several CATIA environments. This allows suppliers to meet OEM requirements to work in a specific CATIA environment release, service pack and hotfix settings.

Customization of toolbars/menus allows the administrator to remove the SmarTeam menu and work with toolbars only

Added the ability to create a new folder from within the Project Manager window.

Open dialog enhancements:

- Added Items shortcut: enables users to browse items and open related AutoCAD documents
- Related documents pane: using the new Items shortcut, users can browse items, and the related documents appear in the Related Documents pane
- Open dialog Quick Search: added the ability to define the order of classes in the Quick Search and any class for Quick Search and Save last used class option

Added the ability to map text in title blocks

Support for VISTA 64-bit

VPM Supply Chain Engineering Exchange

New Functionality

Partial Replication Process Management

You can now create and simulate Replication Packages in VPM Navigator.

V4 Integration

No enhancements in this release.

2D Layout for 3D Design

New Functionality

Setting Large Scale in CATIA V5 Session
> The Large Scale feature allows you to create larger geometries, ranging in kilometers. In 2D Layout there are a few points, which you should consider.

Enhanced Functionalities

Making an Existing Annotation Associative by Creating Rigid Positional Links
> The Positional Link capability allows you to create rigid positional links between the annotation and another element. The annotation cannot be moved manually relative to the reference element.

Element Visualization in Specification Tree (Fix Together, 2D Component Instances, Use-edges)

Visualizing the Fix Together Constraints in the Specification Tree
> The elements constrained together using the Fix Together constraint can be visualized under the Fix Together Constraints node in the specification tree.

Visualizing the 2D Component Reference in the Specification Tree
> The 2D Component instances can be visualized under the Component Instances node in the specification tree.

Visualizing the Associative Use-Edges in the Specification Tree
> The use-edges created along with the creation types can be visualized under the Use Edges node in the specification tree.

Customizing Settings

General
> The Display features under views option allows you to display in the specification tree the representation for fixed elements, 2D component instances and use-edges.

Annotation and Dress-Up
> The Create rigid positional link option allows you to create rigid positional links whenever new positional links are created for text, table and geometrical tolerance annotations.

3D Functional Tolerancing & Annotation

New Functionality

Migration of V4 data to CATIA V5
>   CATIA V4 dimensioning, tolerancing and annotation data can be converted to CATIA V5 FTA data
>   part level. The converted data is associative to the converted CATIA V5 geometry.

Enhanced Functionalities

Save 3D Annotations Defined at Product Level as 3D XML
>   This capability takes into account the 3D annotations defined at product level while saving a product
>   as 3D XML.

Upgrade of Annotation Set
>   You can upgrade pre-R18 annotation sets using Upgrade contextual menu command, so that the
>   FTA features have the same behavior as those annotation sets created in R18 or later.

Managing 3D Annotations in Result Annotation Set
>   The FTA result features are light features that are dedicated for review and are not editable. From
>   R18 they are generated by Multi-cad applications or by exporting V6 products and 3D shape
>   representations to CATIA CATProduct and CATPart documents using the downward compatibility
>   tool. The Result Annotation Sets can be saved as .cgr and as .3dxml. They can also be kept in the
>   filtered document when using the product data filtering capabilities.

Assembly Design

New Functionalities

Associativity
>   This capability allows you to modify CATPart geometry in assembly context without modifying the reference CATPart. The Associativity command results in the creation of a new CATPart instantiated in the assembly, containing a copy obtained by Copy/Paste As Result With Link operation of a chosen geometry from all/ customized components of the active assembly.

Define Variant Generic Product
>   The Define Variant Generic Product command allows to create a new variant and select the components as per the required configuration.

Instantiate Variant
>   The Instantiate Variant command allows to instantiate variant generic product from file.

Replace Variant
>   The Replace Variant command replaces the existing solved by a new variant of another generic product.

Enhanced Functionalities

Add To Associated Part
>   When the assembly is saved in EV5 after the creation of assembly feature and associated part, the assembly feature is not saved in EV5. In this case, the Add to Associated Part command aids to add bodies from existing/new source parts in the product into the associated Part and maintains their Associatively.

Publication in Associated Part
>   The Allow publication in Associative Part allows for publication of the pasted bodies in the associated part in case they are published in the source part. When Only Published Features option is selected, only published geometries from the set of selected geometry options or custom list of geometries are imported in the associated part.

Associativity With PRC Context
>   The context of import of bodies and geometries is PRC. In case of structure exposed mode (mode of the active product), the feature is not created.

Customizing Settings

Redundancy Constraint Check
>   The option Disable redundancy check allows you to enable/disable the redundancy check performed while constraint creation.

Aerospace Sheetmetal Design

Enhanced Functionality

Managing the Joggle Parameters

This capability allows you to manage the joggle parameters valuated by a length (Depth, Runout, Start Radius, End Radius, Clearance) with pre-defined formulas stored in Design Tables.

Cast & Forged Part Optimizer

New Functionality

Upgrading Features

The Upgrade contextual command available on the part feature allows the activation of the last evolutions of the code available on the current level. Thus the feature is updated by manually upgrading it.

Enhanced Functionality

Wall Thickness Analysis

The Wall Thickness Analysis capability analyzes and provides a graphical feedback of the thickness of a part. You can measure the thickness of a part and easily adjust it to fit your needs as per the requirements of the manufacturing process.

Composites Design

New Functionalities

Plies:
Check Contours to check the validity of plies contours or flattening contours.
Data export:
XML export to export ply information (stacking and ply description) in a XML file.
Flattening:
GSD commands Unfold and Transfer are now directly accessible from the Composites Design workbench.
Unfold Composite Entity proposes a fast and associative alternative to Flattening.
Integration with 3D Functional Tolerancing & Annotation:
You can create annotations with associative attribute links to Composites Design features.

Enhanced Functionalities

Materials:
They can be defined as NCF, and used as such,
A value range can be added to material parameters,
Material parameters can be edited within that range.
On the Fly Information:
Selection filters have been added.
Limit Contour:
A Limit Contour Group is now created to enable global modification of limit contours.
Solid from Plies:

It now accepts multi-selection of edges of parts.

Skin swapping:
It now accepts edges of parts as input,
It provides two swapping modes,
It enables to reorder the stacking while swapping.
Numerical Analysis:
You can analyze zones in a CATPart context.
Creating a drawing with GVS Files:
More Composites attributes can be customized.
Ply Book has been rewritten.

Core & Cavity Design

Enhanced Functionality

Fillet Radius Reduction
          Creates now specific CCVFilletSurface features.

Enhanced Functionality

Functional Molded Part

No enhancements in this release.

Generative Sheetmetal Design

New Functionalities

Integration of Unfolded Curve in Drawing
During the creation of an unfolded curve, this capability allows you to specify if the element to be created is needed in the sheetmetal process or if it is only a construction element. The four options Construction element, Characteristic element, Marking or Engraving are available to put a semantic on the unfolded curve feature. It is represented in the drawing according to a standard which allows the parameter the visibility and the graphic properties according to the type of the curve or its position (top or bottom).

Enhanced Functionalities

Extrusion Explode Improvement
You can explode extrusion feature into sketch based wall on edge and sub extrusion (extrusion with a sub profile of the initial profile). This extrusion feature can be connected or not connected to the sheet metal part.

Stamp Recognize Improvement
It is not necessary to have a planar and single support to recognize a stamp geometry for creating a Generative Sheetmetal Design stamping feature. Stamps that are impacting bends except sharp stamps, opened stamps and stamps on the border can be recognized. You may need to force the faces to be recognized as stamps as all the possible stamps are not automatically recognized because of the confusion they can create with a combination of walls and bends.

Generative Drafting

New Functionality

Setting Large Scale in CATIA V5 Session
> The Large Scale feature allows you to create larger geometries, ranging in kilometers. In drafting there are a few points, which you should consider.

Enhanced Functionalities

Propagating View Modifications
> The Synchronize View Definition capability allows you to synchronize view from 3D (Functional Tolerancing and Annotation workbench and 2D Layout from 3D Design workbench) during the update process when the support plane in 3D is changed.

3D Clipping
> The Add 3D Clipping capability has been enhanced to allow you to select 2D or 3D geometry (point, line, edge, planer face, plane) as 3D clipping object to modify the view.

Customizing Settings

View
> This option allows you to synchronize the view generated from 3D, during the update process, when the supporting plane is changed in 3D.

Healing Assistant

No enhancements in this release.

Interactive Drafting

New Functionalities

Setting Large Scale in CATIA V5 Session
> The Large Scale feature allows you to create larger geometries, ranging in kilometers. In Drafting there are a few points, which you should consider.

Visualizing Broken Constraints between 2D and Generated Elements
> The broken constraints are visualized as a red warning symbol above the 2D geometry when a generative geometry is modified or deleted.

Enhanced Functionalities

Making an Existing Annotation Associative by Creating Rigid Positional Links
> The Positional Link capability allows you to create rigid positional links between the annotation and another element. The annotation cannot be moved manually relative to the reference element.

Element Visualization in Specification Tree (Fix Together, 2D Component Instances)

Visualizing the Fix Together Constraints in the Specification Tree
> The elements constrained together using the Fix Together constraint can be visualized under the Fix Together Constraints node in the specification tree.

Visualizing the 2D Component Reference in the Specification Tree
> The 2D Component instances can be visualized under the Component Instances node in the specification tree.

Customizing Settings

General
> The Display features under views option allows you to display in the specification tree the representation for fixed elements and 2D component instances.

Annotation and Dress-Up
> The Create rigid positional link option allows you to create rigid positional links whenever new positional links are created for text, table and geometrical tolerance annotations.

What's New?

New Functionalities

Icons
>   You can affect a customized icon to a Tooling component.

SmarTeam
>   You can manage the editability of impacts.

Technological results
>   You can display and export them.

Part Design

New Functionalities

Upgrading Features

The Upgrade contextual command available on the part feature allows the activation of the last evolutions of the code available on the current level.

Controlling Top Diameter of Tapered Hole

This capability allows you to control a top diameter of a tapered hole by selecting Top in the Type tab, and then entering the desired value in the Extension tab of a Hole Definition dialog box. Further geometry is computed by the end conditions.

Contextual Menu of Blend Corner(s)

For editing an edge fillet containing at least one blend corner, the Reframe On, Create by edges, Create by vertex, Edit and Remove options are available in the Blend corner(s) contextual menu. When no corner is created, only the Create by edges and Create by vertex commands are displayed in the contextual menu.

Intersection Fillets

The new Intersection selection mode allows the definition of edge fillets at the intersection of the selected features with current solid.

Enhanced Functionality

Displaying and Editing Parameters in Specification Tree

This capability allows you to edit the parameters of a feature from the specification tree by double-clicking it and changing its value in the Edit Parameter dialog box. Dress-Up features and Advanced Dress-Up features have their parameters displayed under their respective nodes in the specification tree.

Sketcher

New Functionalities

Upgrading Features

The Upgrade contextual command available on the sketch feature allows the activation of the last evolutions of the code available on the current level.

Exploding Sketch

The Explode… contextual command allows you to modify a sketch obtained by Copy/Paste As Result With Link. It converts all its as result with link geometry into regular sketched curves and points.

Enhanced Functionality

Positioning Sketch Created Using Copy/Paste As Result With Link Operation

You can now specify your own position of a sketch feature obtained by Copy/Paste As Result With Link OR retrieve its associativity in position with its sketch reference by using the new Positioned as reference option.

Wireframe and Surface

New Functionality

Operation Constructors
> Additional constructors are added to the formula editor.

Enhanced Functionalities

Creating Multiple Points and Planes
> Allows you to select any point as a reference point while creating multiple points and allows you to preview the multiple point instances.

Creating Planes Between Other Planes
> Allows you to preview the multiple planes.

Wireframe Constructors
> Additional constructors are added to the formula editor.

Surface Constructors
> Additional constructors are added to the formula editor.

Reordering elements using Drag and Drop
> Allows you to drag an element from one location and drop it at another location without changing its name.

Automotive Body In White Fastening

No enhancements in this release.

Digitized Shape Editor

New Functionalities

Mesh edition tool bar:
> Edit meshes (Add point, Move point, Remove element, Collapse element, Flip edge).

Scan Edition:
> Edit scans.

Discretize Curves:
> Discretizes curves.

Extract Data:
> Extracts visible points from a cloud of points to create a new one.

Disassemble Data:
> Disassembles multi-cells clouds into several mono-cell clouds.

View Management Toolbar:
> Now available.

Enhanced Functionalities

Import:
> Ascii User Format has been added.

3D Curve on Mesh:
> Curve optimization can be cleared,
> Constraints can be applied on end points.

Mesh Cleaner:
> A new tab has been added, Edition, to collapse thin triangles.

Graphic properties:
> They are now available from the Properties menu.

Customizing Settings

Two tabs are now available:
> General, that defines the Command Behavior and Warm Start,
> Display Modes that defines display modes for clouds of points.

FreeStyle Shaper, Optimizer & Profiler

New Functionalities

Filtering Trimmed and Untrimmed Curve and Surface
> A new toolbar Custom selection filters with two new icons Trimmed element Filter and No Trimmed element Filter has been added. These commands allow you to filter trimmed and untrimmed curve and surface and use this selection to perform any other command.

Displaying and Modifying Dress-Up Features
> The Dress-Up capability allows you to display and modify the dress-up options for curves, surfaces and faces.

Enhanced Functionalities

Cutting Plane Analysis
> New options have been added to Cutting Plane Analysis dialog box that informs you about the surfaces selected for analysis and further lets you add, remove or replace these surfaces. The new multi-selector field changes depending on the selection mode and allows you to select the reference elements.

Creating a Matching Constraint
> The Matching Constraint capability with the Mean Surface Solver option selected allows you to get better continuity in case of multiple constraints surface. The mean surface created is a the mean result of the two constraint results.

Styling a Fillet
> This Styling Fillet capability has been enhanced that displays the deviation values of continuity in the 3D area and Styling Fillet dialog box.

New 2D Diagram for 3D Curve Analysis

Performing a Curvature Analysis
> The 2D Diagram of Porcupine Curvature Analysis has been enhanced to allow 3D analysis (the magnitude curvature or radius) according to the position of a point on a 3D curve in a 2D graphical space.

Analyzing Distances Between Two Sets of Elements
> The 2D Diagram of Distance Analysis has been enhanced to allow analysis of the distance between any two geometric elements in a 2D graphical space.

Customizing Settings

Mean Surface Solver
> This option allows you to compute a result between multiple constraints surface as a mean surface between each constraint result.

Generative Shape Design, Optimizer, Developed Shapes & BiW Templates

New Functionality

Operation Constructors
> Additional constructors are added to the formula editor.

Enhanced Functionalities

Creating Blended Surfaces
> Allows you to avoid twists in geometry by automatically computing the coupling points.

Creating Swept Surfaces/Volumes
> Allows you to compute canonical portions for surfaces and volumes

Reshaping Corners
> Improves the blend corner(s) capability for the Edge Fillet and the Variable Fillet commands.

Autofilleting
> Allows you to fillet and round the edges of a part in one go.

Creating Multiple Points and Planes
> Allows you to select any point as a reference point while creating multiple points and allows you to preview the multiple point instances.

Creating Planes Between Other Planes
> Allows you to preview the multiple planes.

Wireframe Constructors
> Additional constructors are added to the formula editor.

Surface Constructors
> Additional constructors are added to the formula editor.

Reordering elements using Drag and Drop
> Allows you to drag an element from one location and drop it at another location without changing its name.

Imagine & Shape

New Functionalities

Moving Surfaces and Curves Using the 3D Compass
> You to manipulate (move and rotate) the subdivision surface and styling curve using the compass. The features can be manipulated using mouse and compass by dragging and dropping the compass onto the feature to be manipulated. The features can also be manipulated by using the Edit… contextual menu command of compass.

Selecting Elements
> A new edge selection mode is available where you select two edges and the path between (including) these two edges depending on set of criteria.

Selecting Specific Color for Selected Elements
> The Use specific color for all selected elements capability allows you to select color for the selected elements (edges and vertices) from a list.

Merging Subdivision Surfaces
> The Merge capability allows to merge two subdivision surfaces at their open edges.

Creating Subdivision from Curves

Subdivision Revolve
> The Revolve capability allows you to create subdivision surfaces by revolving a curve around an axis.

Subdivision Extrude
> The Extrude capability allows you to create subdivision surfaces by extruding a curve along a direction.

Enhanced Functionalities

General Ergonomic Improvements

Modifying the Weight of Vertices or Edges
> The weight of an elements is shown only when it is highlighted.

Validation while using the edition command (Extrude, Subdivide, Erase or Cut)
> Validation for extrusion, cutting, subdividing and erasing a subdivision can be done by reselecting the same element again. This information is also given in the help text when in detailed mode.

Scaling a Surface
> During scaling, the dimension corresponding to the selected manipulator is highlighted. This makes it easier to identify the dimensions being changed and also see its value.

Selecting Multiple Surfaces
> During selection of multiple subdivision surfaces, each subdivision mesh is displayed in different color for easy identification.

Selecting Visible Elements
> When using the View and Selection mode command, a new intermediate state is available which allows visualization of hidden edges, which are not selectable.

Associating Elements
> When two elements are associated, icon is displayed in the specification tree to represent the linked and linking elements along with tooltips indicating the one parent-child relationship.

Smart Selection of Multiple Surfaces
> The Modification capability has been enhanced to allow you to select multiple subdivision surfaces for modification in case a feature has more than one parent. Multiple selection can be made by using Ctrl key.

Projection and Dimension representation enhancements
> In case of Aligning Vertices the manipulators can be deactivated using the Ctrl key for easier element selection. The manipulator cannot be manipulated near the axis. The display of the

projection plane while Aligning Vertices, Dimensioning of Subdivisions and Performing Symmetry depends on the option selected for transparency in Tools > Options > Display > Performance tab, Transparency Quality area.

Displaying Coordinates

The Show Coord capability allows you to control the manipulation by displaying compass coordinates during translation, translating vertices along their normal, rotation and scaling.

Using Cyclic Toolbar and Menu Toolbar

There are two options Cyclic Menu and the Toolbar Menu that allow the visualization of the certain command levels cyclically in the same icon or in a separate expandable toolbar.

General Algorithms Improvements

Extruding Faces and Edges

The Extrusion capability has been enhanced to allow you to a compute a symmetric fill wherever the zone to be filled is symmetrical.

Erasing Faces and Edges

The Erasing capability allows you to erase multiple edges at the same time using Ctrl and Shift keys.

Performing A Symmetry

The Symmetry capability allows you to add weight to the intersection (scar) of symmetry between 0 and 100. You can select to have a sharp or smooth edge.

Dimensioning Subdivisions Along Half Axis

The Dimension capability allows you to modify the dimensions of subdivisions along half the axis.

Performing A Symmetry

The Convert Into Subdivision command allows you to convert a symmetry feature into a into a subdivision surface.

Customizing Settings

Selecting Multiple Surfaces in different colors

This option allows you to display each subdivision mesh in a different color in case of multiple selection of subdivision surfaces.

Displaying Selected Elements in Different Color

This options allows you to display elements with specific colors.

Show hidden lines in dashed lines

This option displays the hidden lines of subdivision surfaces as dashed lines.

Command Options Menu Mode

This option allows you visualize certain command levels in the same icon cyclically (Cyclic Menu) or in a separate expandable toolbar (Toolbar Menu).

Quick Surface Reconstruction

New Functionality

Curvature Mapping

Enhanced Functionalities

Segmentation by Curvature Criterion
> The parameter Influent Radius has been added to improve the result.
> A temporary curvature mapping is available.

3D Curve on Mesh
> Curve optimization can be cleared,
> constraints can be applied on end points.

Graphic properties
> They are now available from the Properties menu.

View Management Toolbar
> is now available.

Customizing Settings

Two tabs are now available:
> General, that defines the Command Behavior and Warm Start,
> Display Modes that defines display modes for clouds of points.

Realistic Shape Optimizer

New Functionality

Displacement Optimization
      Optimizes the computation of a vectors field from a deviation analysis.

Advanced Meshing Tools

Enhanced Functionalities

Creating 1D Mesh Parts
>You can use the minimal mesh method.

Solid Meshing

Creating Octree 3D Mesh Parts
>You can add a size distribution.
>A new option is available: global split.

Surface Meshing

Creating Octree 2D Mesh Parts
>You can add a size distribution.
>A new option is available: global split.

Customizing Settings

General
>Automatic naming of groups created under mesh parts.

Generative Structural Analysis

Enhanced Functionalities

Analysis Cases

Assembled Solutions
>You can assemble frequency solutions.

Model Manager

Creating 1D Mesh Parts
>You can use the minimal mesh method.

Creating Octree 3D Mesh Parts
>You can add a size distribution.

Creating Octree 2D Mesh Parts
>You can add a size distribution.

Creating 2D Properties
>You can define a thickness offset both in a 2D property and in a local 2D property.

Importing Composite Properties
>You can define a thickness offset and take into account a draping direction defined in the Composite
>Design workbench.

Mapping File Syntax
>A new keyword is available in the element identification of the XML mapping file.
>You can define an offset value for shell, inhomogeneous composite and and homogeneous
>composite properties in the XML mapping file.

Connection Properties

Contact Connection Properties
>You can take into account the friction in the computation of contact elements.

Result Visualization

Animating Images
>You can define and impose the duration of a full animation.

Editing Images
>The current occurrence and its associated value are displayed in the color map.
>You can define new conditions to filter values.

Generating Images
>The current occurrence and its associated value are displayed in the color map.

Available Images
>Instantaneous Von Mises stress and element centroid images are now available.

Information
>The content of the Information box has been enhanced.

Exporting Data
>You can export entities with no associated value in a .txt or .xls file.

Customizing Settings

General
>Automatic naming of groups created under mesh parts.

Compartment and Access

No enhancements in this release.

Electrical Cableway Routing

Enhanced Functionality

You need to set an environment variable to display the error message "180 degree turn is found in the cableways connected to the cableway element", while routing from the database.
The analyze cross document connections and manage document links functions now have an Open button that allows users to open linked documents that are in ENOVIA.

Electrical Connectivity Diagrams

Enhanced Functionality

The analyze cross document connections and manage document links functions now have an Open button that allows users to open linked documents that are in ENOVIA.

Electrical Harness Flattening

Enhanced Functionality

General Parameters

The **Regular** mode calculates angles in predefined target directions depending upon the number of bundle segments and branches.

Electrical Harness Installation

Enhanced Functionalities

Adding Light Protective Coverings
> After customization, the parameters and relations are displayed in a separate node in the specification tree.

Arranging Bundle Segments in Supports
> Allows you to show or hide the parameters of the of the bundle segment in the geometry area.

Migrating Step by Step...
> The migration mode is automatically switched to As Result when migration causes shape or length modifications in the As Spec mode in case of a flexible curve of the bundle segment.

Electrical Library

Enhanced Functionalities

Electrical Process Interfacing
> Allows you to access filebased XML files even in the ENOVIA environment.

Migrating Step by Step...
> Stores the part number of the CATPart in its file name after the migration process.

Migrating Step by Step...
> Deletes all the unnecessary created files after the migration from V4 to V5 process.

Creating Protective Coverings
> Allows you to customize the parameters and attributes for protection parts.

Migrating Step by Step...
> Allows you to migrate V4 protections as V5 internal protections.

Editing Electrical Properties
> The Reference Designator attribute can be synchronized using a knowledgeware rule when it is changed in either electrical or mechanical application systems.

Electrical Wire Routing

Enhanced Functionalities

Electrical Process Interfacing
> Allows you to access filebased XML files even in the ENOVIA environment

Automatically Routing Wires and Wire Groups
> Allows you to route wires though devices and assemblies.

Electrical Package in Knowledge Expert
> Returns the name/part number of the end device connected from or to the wire. Also, checks for the validity of the bundle segment diameter.

Equipment Arrangement

Enhanced Functionality

The Update feature allows you to update insulation after the base part has been modified.
A setting allows you to find parts during parts placement by using the compatibility table, or by using values in your design.
The analyze cross document connections and manage document links functions now have an Open button that allows users to open linked documents that are in ENOVIA.

Hanger Design

Enhanced Functionality

A setting allows you to find parts during parts placement by using the compatibility table, or by using values in your design.
The analyze cross document connections and manage document links functions now have an Open button that allows users to open linked documents that are in ENOVIA.

HVAC Design

Enhanced Functionality

The Update feature allows you to update insulation after the base part has been modified.
A setting allows you to find parts during parts placement by using the compatibility table, or by using values in your design.
The option "Update adjacent segments direction" in the Offset Segment dialog box allows you to change the direction of adjacent segments when you are making a segment parallel to a reference plane.
The analyze cross document connections and manage document links functions now have an Open button that allows users to open linked documents that are in ENOVIA.

HVAC Diagrams

Enhanced Functionality

The analyze cross document connections and manage document links functions now have an Open button that allows users to open linked documents that are in ENOVIA.

Piping and Instrumentation Diagrams

Enhanced Functionality

The analyze cross document connections and manage document links functions now have an Open button that allows users to open linked documents that are in ENOVIA.

Piping Design

Enhanced Functionalities

The Update feature allows you to update insulation after the base part has been modified.
A setting allows you to find parts during parts placement by using the compatibility table, or by using values in your design.
When you are generating an isometric drawing, the generated object obtains nominal size from the pipe dimension table. The column IsoGenSize needs to have values added to it.
The option "Update adjacent segments direction" in the Offset Segment dialog box allows you to change the direction of adjacent segments when you are making a segment parallel to a reference plane.
The analyze cross document connections and manage document links functions now have an Open button that allows users to open linked documents that are in ENOVIA.

Plant Layout

No enhancements in this release.

Raceway and Conduit Design

Enhanced Functionalities

The Update feature allows you to update insulation after the base part has been modified.
A setting allows you to find parts during parts placement by using the compatibility table, or by using values in your design.
The option "Update adjacent segments direction" in the Offset Segment dialog box allows you to change the direction of adjacent segments when you are making a segment parallel to a reference plane.
The analyze cross document connections and manage document links functions now have an Open button that allows users to open linked documents that are in ENOVIA.

Ship Structure Detail Design

New Functionalities

During ENOVIA Integration, the **Applicative Domain** and **Applicative Type** attributes are assigned specific values. These values can be used for searching, querying, and filtering data.
You can use the End Cut command to Place End Cuts on profiles, such as stiffeners, stiffeners on a free edge, and beams. You can also Edit and Delete End Cuts and Place End Cuts on Multiple Profiles by using this command.
You can Define Built-in End Cuts by assigning appropriate values to the built-in end cut parameters.

Enhanced Functionalities

You can now add built-in end cuts to the catalog.
While creating a stiffener, you can select the **Fast Preview** check box in the Stiffener Dialog Box, which lets you preview stiffener creation.
The Effect of Openings on Stiffeners explains how openings are organized and how these impact the stiffeners.
When generating Graphic Replacement for Beams, Plates, and Stiffeners you can compute **AcrossSide** parameters in Logical and FTA section modes. The end cut symbol applied on the extremity of a beam depends on the 3D parameter value.
When Placing Small Assemblies, such as chocks, you can use the **Display all inputs** check box to manually select the faces of the shape required as input.
You can define and modify the orientation of profiles, such as stiffeners and beams by Using Section Manipulators in your design.
You can also Multi-edit Material Properties of profiles, such as stiffeners and beams. You can also Multi-edit Material Properties of Plates.
When defining standard or contextual End Cuts based on a User Defined Feature, you need to follow the naming conventions.

Structure Functional Design

New Functionalities

Mesh Parts, Properties, and Materials task explains how you can obtain section parameters of mesh parts for analysis by using the **Manage Section CATPart** command.

The computed attributes, IsASuperObject, SupportOffset, and SupportedPlate are added to the List of Attributes. These attributes help in filtering your search results.

During ENOVIA Integration, the **Applicative Domain** and **Applicative Type** attributes are assigned specific values. These values can be used for searching, querying, and filtering data.

You can Generate Shell Expansion Drawings by using the Shell Expansion drawing command. This drawing is generated by expanding the shell plate surfaces in the transverse direction.

You can use the End Cut command to Place End Cuts on profiles, such as stiffeners, stiffeners on a free edge, and beams. You can also Edit and Delete End Cuts and Place End Cuts on Multiple Profiles by using this command.

You can Define Built-in End Cuts by assigning appropriate values to the built-in end cut parameters.

Enhanced Functionalities

You can now add built-in end cuts to the catalog.

While creating a stiffener, you can select the **Fast Preview** check box in the Stiffener Dialog Box, which lets you preview stiffener creation.

The Effect of Openings on Stiffeners task explains how openings are organized and how these impact the stiffeners. In SFD, a stiffener is not split into more child stiffeners as a result of the opening.

When generating Graphic Replacement for Beams, Plates, and Stiffeners you can compute **AcrossSide** parameters in Logical and FTA section modes. The end cut symbol applied on the extremity of the beam depends on the 3D parameter value.

You can define and modify the orientation of profiles, such as stiffeners and beams by Using Section Manipulators in your design.

You can also Multi-edit Material Properties of profiles, such as stiffeners and beams. You can also Multi-edit Material Properties of Plates.

When defining standard or contextual End Cuts based on a User Defined Feature, you need to follow the naming conventions.

Systems Routing

No enhancements in this release.

System Space Reservation

No enhancements in this release.

Tubing Design

Enhanced Functionalities

The Update feature allows you to update insulation after the base part has been modified.
A setting allows you to find parts during parts placement by using the compatibility table, or by using values in your design.
The option "Update adjacent segments direction" in the Offset Segment dialog box allows you to change the direction of adjacent segments when you are making a segment parallel to a reference plane.
The analyze cross document connections and manage document links functions now have an Open button that allows users to open linked documents that are in ENOVIA.

Tubing Diagrams

Enhanced Functionality

The analyze cross document connections and manage document links functions now have an Open button that allows users to open linked documents that are in ENOVIA.

Waveguide Design

Enhanced Functionalities

The Update feature allows you to update insulation after the base part has been modified.
A setting allows you to find parts during parts placement by using the compatibility table, or by using values in your design.
The option "Update adjacent segments direction" in the Offset Segment dialog box allows you to change the direction of adjacent segments when you are making a segment parallel to a reference plane.
The analyze cross document connections and manage document links functions now have an Open button that allows users to open linked documents that are in ENOVIA.

Waveguide Diagrams

Enhanced Functionality

The analyze cross document connections and manage document links functions now have an Open button that allows users to open linked documents that are in ENOVIA.

Circuit Board Design

No enhancements in this release.

3 Axis Surface Machining

New Functionalities

Part Operation:
> You can now use the Part defined at the Part Operation level.
> Collision Checking is available at the Part Operation level.

Four probing operations:
> Holes or Pins Probing,
> Slots or Ribs Probing,
> Corner Probing,
> Multi-Points Probing.

Enhanced Functionalities

3/5-Axis Converter:
> it is now available for all End Mill tools, TSlotters and Lollipops,
> a new strategy guide is available: Normal to drive surface,
> an Angle parameter has been added to the Thru a guide strategy mode,
> ZLevel tool path can be optimized.

Tool Path Editor lets you now:
> edit feedrates of tool paths, either locally or globally,
> edit PP Words in tool paths,
> edit V4 NCMill operations.

Roughing
> management of start point is enhanced,
> you can choose to compute a tool path free of collision with the tool holder,
> for pockets, you can create engagements from external zones, and add a circular approach for automatic macros,
> you can skip small holes in the rough stock, instead of avoiding them.

Plunge Milling
> a new grid type has been added: By Offset,
> you can visualize plunging points before computing the tool path,
> a new parameter is available for retract motions: Axial corner radius,
> you can modify the feedrate of the clearance macro.

Contour-Driven machining operations:
> improved guide definition for parallel contour mode
> (guide defined by faces boundaries, zig-zag motif defined by either two points or one point and an axis),
> new Strategy side parameter for parallel contour mode,
> Option Reference now available in mode Parallel contour with Constant 3D strategy,
> improved selection of faces in a polygon trap: new option to select visible faces only,
> in Between contours mode, with a constant 2D stepover, the offset on guides can be applied to stops.

ZLevel machining operations:
> an offset on tool path in the machining plane is now available,
> you can compute a circular interpolation of the tool path, or approximate one,
> the **Reverse Machining Conditions** command inverts the Climb and Conventional cutting modes.

Spiral Milling
> a new strategy mode has been added: Back and Forth,
> a new option Always stay on bottom has been added to the Helical tool path style,
> Face Mill Tool is now available,
> clearance feedrate management via a contextual menu,
> new Ramping up to a plane macro is available.

Advanced Machining

New Functionalities

Intermediate Stock Management
> It is now possible to compute and visualize the input/output intermediate stock for milling and turning operations.
> The computed stock is taken into account to optimize and secure the tool path.

Part Operation
> You can now use the Part defined at the Part Operation level.
> Collision Checking is available at the Part Operation level.

Four probing operations:
> Holes or Pins Probing,
> Slots or Ribs Probing,
> Corner Probing,
> Multi-Points Probing.

Enhanced Functionalities

Enhancements brought to multi-axis machining operations

Tool axis guidance for multi-axis machining operations
> A Lead angle parameter has been added to the Thru a guide strategy for multi-axis sweeping operations, multi-axis isoparametric operations, multi-axis contour-driven operations and multi-axis curve operations.
> A new strategy Normal to drive surface has been added to multi-axis sweeping operations, multi-axis isoparametric operations and multi-axis contour-driven operations.

Multi-axis Contour-Driven
> In Between contours mode, the offset on guides can be applied to stops.

Macros for 5-axis machining operations
> Additional macros are now available for 5-axis machining operations:
> Add normal motion helps avoiding collisions in the linking macros of tube machining.
> Add Circular motion is very useful in return in a level macros of tube machining, when the **Guiding strategy** is set to **Along guide.**
> Enable 5-axis simultaneous motion.
> An additional clearance option is available: Smooth tool axis moves.

Enhancements brought to 3-axis machining operations

3/5-Axis Converter:
> it is now available for all End Mill tools, TSlotters and Lollipops,
> a new strategy guide is available: Normal to drive surface,
> an Angle parameter has been added to the Thru a guide strategy mode,
> ZLevel tool path can be optimized.

Tool Path Editor lets you now:
> edit feedrates of tool paths, either locally or globally,
> edit PP Words in tool paths,
> edit V4 NCMill operations.

Roughing
> management of start point is enhanced,
> you can choose to compute a tool path free of collision with the tool holder,
> for pockets, you can create engagements from external zones, and add a circular approach for automatic macros,
> you can skip small holes in the rough stock, instead of avoiding them.

Plunge Milling

a new grid type has been added: By Offset,

you can visualize plunging points before computing the tool path,

a new parameter is available for retract motions: Axial corner radius,

you can modify the feedrate of the clearance macro.

Contour-Driven machining operations:

improved guide definition for parallel contour mode

(guide defined by faces boundaries, zig-zag motif defined by either two points or one point and an axis),

new Strategy side parameter for parallel contour mode,

Option Reference now available in mode Parallel contour with Constant 3D strategy,

improved selection of faces in a polygon trap: new option to select visible faces only,

in Between contours mode, with a constant 2D stepover, the offset on guides can be applied to stops.

ZLevel machining operations:

an offset on tool path in the machining plane is now available,

you can compute a circular interpolation of the tool path, or approximate one,

the **Reverse Machining Conditions** command inverts the Climb and Conventional cutting modes.

Spiral Milling

a new strategy mode has been added: Back and Forth,

a new option Always stay on bottom has been added to the Helical tool path style,

Face Mill Tool is now available,

clearance feedrate management via a contextual menu,

new Ramping up to a plane macro is available.

Enhancements brought to 2.5-axis machining operations

Helical Interpolation Output for Machining Operations

A helical interpolation instruction can now be generated in the output file (APT source and Clfile) for machining operations which support helical tool motion. These operations are Thread Milling, Circular Milling in Helical mode, Sequential Groove with Helical motions, and Profile Contouring defined with a Helix tool path style.

Multiple Radial Passes for Thread and Circular Milling Operations

This enhancement allows the user to create multiple passes (rough and finish) for Thread Milling and Circular Milling in Helical mode, and provides capability to create a spring pass. The user no longer needs to create separate operations for rough, finish and spring passes.

Support Multiple Tool Axis in Point to Point Operation

This enhancement enables the user to select different tool axes for multiple points in a single operation, and simplifies the selection procedure.

Extended Tool Type Support for Machining Operations

This enhancement gives user a wider selection of cutting tools on the following machining operations: Circular Milling, Countersinking, Counter Boring, Sequential Axial, Sequential Groove, and Pocketing.

Lathe Machining

Enhanced Functionalities

B-axis management for turning operations
>   New capability to define a tool axis on each Turning operation. This allows the B-axis orientation to be managed and eliminates the need for a tool change activity.

Turret C-axis management for turning operations
>   A new C-axis inversion option on the Strategy tab page of the Machining Operation editor enables a C-axis rotation (0/180 degrees) without requiring a tool change activity. This allows machining of different areas (front and back areas of a part, or part on main spindle and part on a counter-spindle) using the same tool assembly. Only the tool orientation (inversion) differs when machining the different areas or parts.

Automatic Generation of ROTABL and TLAXIS for Changeover from Lathe to Mill
>   This enhancement allows automatic generation of ROTABL and TLAXIS statements whenever there is change-over between Turning and Milling Operations.

Multi-Axis Surface Machining

Enhanced Functionalities

Part Operation:
> You can now use the Part defined at the Part Operation level,
> Collision Checking is available at the Part Operation level.dd>

Tool axis guidance:
> A Lead angle parameter has been added to the Thru a guide strategy for multi-axis sweeping operations, multi-axis isoparametric operations, multi-axis contour-driven operations and multi-axis curve operations.
> A new strategy Normal to drive surface has been added to multi-axis sweeping operations, multi-axis isoparametric operations and multi-axis contour-driven operations.

Contour-Driven
> In Between contours mode, the offset on guides can be applied to stops.

Macros
> Additional macros are now available for 5-axis machining operations:
> Add normal motion helps avoiding collisions in the linking macros of tube machining.
> Add Circular motion is very useful in return in a level macros of tube machining, when the Guiding strategy is set to Along guide.
> Enable 5-axis simultaneous motion.
> An additional clearance option is available: Smooth tool axis moves.

Multi-Pockets Machining

New Functionality

Multi-Pocket Flank Contouring

Enhanced Functionalities

Power Machining:
      You can choose to compute a tool path free of collision with the tool holder,
      For pockets, you can create engagements from external zones, and add a circular approach for
      automatic macros,
      You can skip small holes in the rough stock, instead of avoiding them.

Part Operation
      You can now use the Part defined at the Part Operation level,
      Collision Checking is available at the Part Operation level.

Multi-Slide Lathe Machining

Enhanced Functionality

Improvement of output based on Counter Spindle

 This enhancement is provided by the **Use Spindle Axis system according to the Spindle involved on the Machining Operation** checkbox in the Option tab of the Part Operation editor for a Multi-slide lathe machine or a Mill-Turn machine.

 If this check box is selected, tool tip points are computed based on the spindle that is set on the machining operation.

 If this check box is not selected, the main spindle axis is used. This is determined by the default reference machining axis system set on the Part Operation.

NC Machine Tool Builder

New Functionalities

Defining Machines with Interchangeable Heads
> Previously, the creation of Milling Machines and Milling Heads, with the capability to do a head change during simulation, was not supported. This functionality enables to create such machines to be used in simulation. It allows to create a new Milling Machine Head, which can then be inserted into the milling machine CATProduct.

Defining Tool and Work-piece Mount Points
> This section describes how to create, modify and delete multiple tool and work-piece mount points on milling machines, using the Create Mount Point command.

NC Machine Tool Simulation

New Functionality

Simulation with Interchangeable heads
>   This functionality allows you to simulate Milling machines with interchangeable heads, thus enabling to verify NC tool paths for travel limits, collisions and modify if necessary.

Enhanced Functionalities

User-selectable Machining Axis System for Workpiece automount
>   This provides the ability to mount the workpiece on a Milling machine bed or Mill-Turn machine with respect to a user-selected machining axis system.

Machine Instruction Support for Turrets
>   For Mill-Turn Simulation, this enhancement allows you to define Machine Instruction activities on Turrets.

Machine configuration management for Machining Operation
>   This functionality allows you to associate and persist a machine configuration with a particular tool path point, from within the Modify Tool Path dialog box. This machine configuration will be applied during machine simulation, when that particular tool path point is to be reached.

Simulating Machine Motion for Milling and Mill-Turn Machines
>   This provides support to simulate machine motion with Integrated Material Removal for Milling and Mill Turn machines.

NC Manufacturing Infrastructure

Enhanced Functionalities

ToolPath Editor: Feedrate Modification
>   This enhancement to feedrates is integrated to the Area Modification command of the tool path editor and works on one or several tool paths. Tool path editor is available on APT imports and milling operations. Tool paths can be modified partially or globally.

ToolPath Editor: V4 NC Mill Support
>   This enhancement enables Tool Path edition on manufacturing data from V4 NC MILL models.

ToolPath Editor: PP Word Management
>   This enhancement enables merging several PP instructions and editing PP words inside a tool path.

Video Mode: Usability Improvements
>   This enhancement provides a number of usability improvements such as improved error messages; open fixtures support; extended support of turning tool holders; save CATProduct at the same location as the Process; interruptible progress bar; capability to disconnect non cutting diameter; and support of probing operations.

Feed Support for Transition Paths
>   This enhancement provides a number of improvements for transition feedrates. In particular, you can now locally set the feedrate for a transition path to a machining operation B from a machining operation A or from a tool change activity. This is done by selecting the **Transition** check box in the Machining Operation dialog box for operation B.

Head Change Activity
>   This enhancement provides a new **Head Change** command in the Auxiliary Operations tool bar. Clicking on this command inserting a new Head Change activity in the program, if an NC machine with at least one head is assigned to the Part Operation.

Automatic Generation of ROTABL and TLAXIS for Changeover from Lathe to Mill
>   This enhancement allows automatic generation of ROTABL and TLAXIS statements whenever there is change-over between Turning and Milling Operations.

Ordered List of PP Tables in Machine Editor
>   Prior to this release, Post Processor (PP) word table names were not listed in an order, thus making it difficult for the user to locate a PP word table. This enhancement addresses this usability issue and lists the PP word tables in an alphanumerically sorted order. Controller Emulator and Post Processor selections are also listed in sorted order.

Improved Error Messages for Output Generation
>   This highlight improves the message issued after NC data output generation by quantifying the number of errors/warnings along with success/failure information.

Replace the Tool Assembly of Multiple MOs
>   Prior to this release, the **Replace Tool** contextual command could be used to replace the tool of selected Machining Operations. This enhancement allows the tool assembly of selected operations to be replaced.

Tool Number in Process Table
>   This enhancement allows the following additional parameters to be accessed in the Process Table: Tool Number, Corrector Id, Corrector Number, and Radius Number. The tool number can be edited.

Prismatic Machining

New Functionality

Intermediate Stock Management
>It is now possible to compute and visualize the input/output intermediate stock for milling and turning operations.
>The computed stock is taken into account to optimize and secure the tool path.

Enhanced Functionalities

Helical Interpolation Output for Machining Operations
>A helical interpolation instruction can now be generated in the output file (APT source and Clfile) for machining operations which support helical tool motion. These operations are Thread Milling, Circular Milling in Helical mode, Sequential Groove with Helical motions, and Profile Contouring defined with a Helix tool path style.

Multiple Radial Passes for Thread and Circular Milling Operations
>This enhancement allows the user to create multiple passes (rough and finish) for Thread Milling and Circular Milling in Helical mode, and provides capability to create a spring pass. The user no longer needs to create separate operations for rough, finish and spring passes.

Support Multiple Tool Axis in Point to Point Operation
>This enhancement enables the user to select different tool axes for multiple points in a single operation, and simplifies the selection procedure.

Extended Tool Type Support for Machining Operations
>This enhancement gives user a wider selection of cutting tools on the following machining operations: Circular Milling, Countersinking, Counter Boring, Sequential Axial, Sequential Groove, and Pocketing.

Prismatic Machining Preparation Assistant

Enhanced Functionality

New option to recognize features based on Technological Results

This enhancement provides the capability to create axial machinable features corresponding to existing Technological Results of holes of the selected body.

STL Rapid Prototyping

New Functionality

View Management Toolbar
is now available.

Enhanced Functionality

Graphic properties
They are now available from the Properties menu.

Customizing Settings

Two tabs are now available:
General, that defines the Command Behavior and Warm Start,
Display Modes that defines display modes for clouds of points.

Business Process Knowledge Template

No enhancements in this release.

DMU Composites Review

No enhancements in this release.

Kinematics Simulator

Enhanced Functionalities

Deleting Joints

      The Delete All Children option is now selected by default in the Delete dialog box.

Copying and Pasting a Mechanism
      You can now copy and paste a dress-up.

DMU Navigator

No enhanced functionalities in this release.

DMU Optimizer

No enhancements in this release.

DMU Space Analysis

Enhanced Functionalities

Sectioning

Exporting Section Results
> You can now export sectioning results to VPM Navigator using the **Open and Export** command available in the **Result** tab in the **Sectioning Definition** dialog box.

Exporting Section Results
> Polyline results have been improved and thus increased the quality of a smoothed curve.

DMU Space Analysis Interoperability

Performing DMU Space Analysis Tasks with VPM Navigator
> Interference object is now integrated within VPM Navigator (you can thus perform open, send to and delete operations).

Human Activity Analysis

New Functionalities

3D Display of Biomechanics Results
>A better way to display the results in 3D, each segment results are highlighted in the list when the related segment in the 3D model is selected.

Defining Loads on Other Segments
>This provides a more realistic behavior of the Biomechanics Analysis by allowing the use of loads positioned on segments other then just the hands. Therefore, Biomechanics Analysis can be applied on manikin having loads on shoulders, forearms and back.

Enhanced Functionality

Increased the number of analyzed joints - wrists
>For a more complete analysis of the human biomechanics Results a new analysis will cover both wrists in flexion-extension and Radial-Ulnar deviation

Human Builder

New Functionalities

Displaying Balance
> This displays the support polygon of manikin's balance in the viewer. The color of the support
> polygon will change from green to red if the manikin loses his balance.

Defining the color of the Manikin Referential
> Choosing the color of the manikin referential for each manikin enables the client to differentiate
> them and to make sure that it doesn't blend with the environment.

Auto Grasp Offset
> The need for this highlight comes from wanting more control over an existing function that
> automatically generates a grasp posture of the hand by checking the collision between the hand and
> one or several objects to be grasped. A new option would allow you to edit the offset that control
> the reaching position of the hand which is currently automatically set to a default value.

Knowledge Advisor

Enhanced Functionality

R19-Knowledge Language improvements

> The improvement adds new functions/methods in the Engineering Knowledge Language. With the new Knowledge functions you will be able to add two lists, create a list by extracting information from existing list, sort, delete and remove the duplicate element from the list.

Product Engineering Optimizer

No enhancements in this release.

Product Knowledge Template

No enhancements in this release.