

CATIA Version 5 Release 21

Product Enhancement Overview

This document lists both new and enhanced product functionality for the current release.

Infrastructure

- ENOVIA-CATIA Interoperability
- Installation and Deployment
- Infrastructure
- Product Structure
- V4 Integration
- Data Exchange Interfaces
- Product Data Filtering
- VPM Supply Chain Engineering Exchange
- SMARTEAM CATIA Integration
- SMARTEAM CATIA Supply Chain Engineering Exchange
- SMARTEAM CATIA Web Integration Product

Mechanical Design

- Interactive Drafting
- Generative Drafting
- 2D Layout for 3D Design
- Structure Design
- Wireframe and Surface
- Generative Sheetmetal Design
- 3D Functional Tolerancing & Annotation
- Mold Tooling Design
- Core & Cavity Design
- Healing Assistant
- Composites Design

Shape Design & Styling

- FreeStyle Shaper Optimizer & Profiler
- Generative Shape Design & Optimizer
- Shape Sculptor
- Imagine & Shape
- Digitized Shape Editor
- Quick Surface Reconstruction
- Realistic Shape Optimizer
- ICEM Shape Design Center
- ICEM Shape Design AeroExpert

Analysis

- Generative Structural Analysis
- Nonlinear Structural Analysis and Thermal Analysis
- Advanced Meshing Tools

Equipment & Systems Engineering

- Electrical Harness Installation
- Electrical Harness Flattening
- System Routing
- Systems Space Reservation
- Raceway & Conduit Design
- Waveguide Design
- Piping Design
- Tubing Design
- HVAC Design
- Ship Structure Detail Design

Machining

- NC Manufacturing Infrastructure
- Prismatic Machining Preparation Assistant
- Prismatic Machining
- 3 Axis Surface Machining
- Multi-Axis Surface Machining
- Multi-Pocket Machining
- Advanced Machining
- Lathe Machining
- Multi-Slide Lathe Machining
- STL Rapid Prototyping
- DPM Process and Resource Definition

Product Synthesis

- Knowledge Advisor
- Knowledge Expert
- DMU Engineering Analysis Review
- DMU Composites Review
- Human Builder

ENOVIA-CATIA Interoperability

New Functionality

Saving Creation, Modification, or Deletion of Constraints to ENOVIA V5 VPM

Enables you to save the modifications on constraints although the constraints may be owned by some other user or organization.

Discard Save

Enables you to select or reject specific data to ENOVIA V5 VPM from the group of data you have modified in your CATIA session.

Alternative Save

Enables you to save a modified document in ENOVIA V5 VPM as a new document while leaving the source object unmodified.

Enhanced Functionality

Sorting data in the Set PDM Properties dialog box

You can sort the data alphabetically in all the columns of the Set PDM Properties dialog box.

Viewing details of an object in the Save in ENOVIA V5 VPM dialog box

In the Save in ENOVIA V5 VPM dialog box, you can view the other details related to the objects while saving. This information helps you understand details related to which objects are really modified and will be part of the actual save process.

Installation and Deployment

New Functionality

New Licensing Middleware

Dassault Systèmes License Server (DSLS) is introduced as the new default Licensing middleware in addition to License Use Management (IBM LUM).

Supporting Platforms on Vault Server

Vault (64-bit) is supported on Windows (64-bit) and AIX (64-bit) platforms.

Supporting Windows Server 2008

ENOVIA V5 Server (32-bit) installation is also supported on Windows Server 2008 SP2 / R2 (64-bit).

Supporting DB 2 9.7

DB 2 9.7 is also supported by Vault and Windows Server.

Enhanced Functionality

There are no enhancements in R21.

Infrastructure

Customizing

Document

A new option lets you load a file even if you are not authorized to access some of its sub-documents.

Product Structure

Enhanced Functionality

Defining Contextual Links

You can **reroute** broken unpublished external references using the `Reroute` button.

V4 Integration

New Functionalities

Migrating Draw Data from .Model Files to a Layout

Explains the process of migrating drawing data from .model files to a layout.

Data Exchange Interfaces

New Functionalities

IGES 2D Import

IGES 2D files can be imported directly into 2D Layout for 3D Design via `Tools > Import from File`.

Enhanced Functionalities

STEP

Infinite Planes are now **imported** and **exported**.

DXF Import

DXf tolerances are now imported as semantic geometrical tolerances.

Customizing Settings

STEP

AP214 edition 3 is now supported.

User Defined Attributes are now imported and exported.

DXF

The new option `Map DXF Layers with 2D Layout for 3D Design Sheets` maps geometries and annotations with 2D Layout for 3D Design sheets, from the number of the layer the geometries and annotations belong to, if any.

Product Data Filtering

Enhanced Functionalities

Product to Product, Product to Part, Part to Part, Batch Operation

You can choose to transfer specifications of sketches.

Product to Product, Part to Part

An empty default PartBody is no longer created in the resulting CATPart.
3D working supports are transferred.

Batch Operation

You can choose to transfer constraints on sketches.

CATIA - ENOVIAvpm Supply Chain Engineering Exchange

New Functionality

Selecting Magic Reconciliation

The Magic Reconciliation provides the Global set in interactive mode and in batch mode (both in ENOVIA V5 and VPMV4) to manage persistency, mapping and automatic choices of reconciliation rules. The Magic Reconciliation button in Reconciliator Toolbar is integrated with Global Set to customize the reconciliation UI.

Supporting CGR in Reconciliation

The Reconciliator enables you to re-integrate the received CATIA data as CGR documents (where CGR documents can be either standalone or alternate representation in Structure or Publication Exposed assembly). This sort of CGR documents are supported for reconciliation in ENOVIA V5 VPM and ENOVIA VPM V4.

Enhanced Functionality

Improving Reconciliator Usability

The **Calendar** enables you to select the creation or modification date from which the product needs to be searched in **Easy Query**. The **Calendar** is so designed to avert the iteration for identifying the right format while selecting a date.

The **3D Viewer Toolbar** enables you to define the different viewpoints and visualization modes. These visualization modes allows you to perform an efficient visual comparison of the mapped object.

The **Ambiguities** tab enables you to identify the types of ambiguities occurred while reconciling an object and review and confirm the automatic choice of rules.

Selecting Keep as External Rule

The **Keep as External** rule is applicable for both missing and loaded documents in ENOVIA VPM V4, when the storage mode is **Publication Exposed**.

SMARTEAM CATIA Integration

New Functionality

Creation Work In Progress (WIP) Assembly

You can create the Work In Progress assembly which is a CATIA Product containing collection of sub assemblies from the main assembly.

WIP Document Management

You can manage WIP documents by choosing its Open modes, refreshing the documents and also insert documents from the original assembly.

Synchronization of Large Assembly

You can synchronize the WIP document with the latest revision of Original Assembly.

Representation Management for Large Assembly

You can use `Manage Representation` functions to visualize a sub-assembly in Context of its Root assembly.

Migration from LUM to DSLS Licensing

By default, SMARTEAM CATIA Integration now uses DSLS Licensing Mode. You can activate back LUM Licensing independently from SmarTeam Licensing mode. For more information, refer to CATIA Licensing documentation.

Using High Secure Connection Mode

SMARTEAM CATIA Integration supports High Secure Connection Mode. High Secure Connection Mode can be set in SmarTeam Configuration Editor. For more information, see *SmarTeam Editor User Guide*.

Enhanced Functionality

Insert Existing Component and Replace Component

These commands are available through Product Structure's commands in Product Structure's Menu, toolbar, and contextual menu.

Check-In and Releasing Component

When you Check-In an individual part which is under an assembly and is not pointing to any other component, only that part gets saved and Checked-In in SmarTeam.

SMARTEAM CATIA Supply Chain Engineering Exchange

New Functionality

Supporting CGR in Reconciliation

SMARTEAM - CATIA Supply Chain Engineering Exchange enables you to reconcile CGR documents.

Using One-Click Reconciliation

You can use One Click Reconciliation command to set mapping and reconciliation rules automatically to reconcile the object with just one click.

Migration from LUM to DSLS Licensing

By default, SMARTEAM CATIA Supply Chain Engineering Exchange now uses DSLS Licensing Mode. You can activate back LUM Licensing independently from SmarTeam Licensing mode. For more information, refer to CATIA Licensing documentation.

SMARTEAM CATIA Web Integration Product

New Functionality

Migration from LUM to DSLS Licensing

By default, SMARTEAM CATIA Web Integration now uses DSLS Licensing Mode. You can activate back LUM Licensing independently from SmarTeam Licensing mode. For more information, refer to CATIA Licensing documentation.

Interactive Drafting

Enhanced Functionality

Creating a Welding Symbol

You can now specify a **scarf**, a **stud**, an **edge** and a **melt thru**weld symbol in AWS A2.4 standard and an **edge**, a **surface joint** and an **inclined joint** weld symbol in ISO 2553 standard or JIS Z3021.

Annotation Manipulators

You can now customize the display of manipulators for annotations, whether it is for annotation selection or text edition.

Engineering Symbols

As per new revision of the ISO standard related to Geometrical Product Specification - Dimensional Tolerancing (14405-1), you can now have some new engineering symbols, to provide a standardized display and print in any application.

You can also have some new **application zone symbols All Around, All About** and **All Over** to support ISO 10135.

Zoomable High Resolution Preview in Catalog Browser

You can now identify smaller details within a large and complex 2D component by zooming on its high resolution preview.

Customizing Settings

Tools > Options > Mechanical Design > Drafting > Manipulators tab > Annotation Manipulators

The new `Annotation Manipulators` options allow you to customize the display of manipulators, whether it is for annotation selection or text edition.

Generative Drafting

New Functionality

Reduction of Required View Update Cycles after 3D Graphic Attribute Changes

You can now avoid updating a view which is not impacted by a show/no-show modification done in the 3D. This improves productivity.

Enhanced Functionality

Fillet Symbolic Representation Enhancements

You can now avoid extraneous fillet representation which overlaps with the actual geometry edge, thus enabling you to create associative dimensions on views.

2D Layout for 3D Design

New Functionality

2D Background Generation

You can now generate a drawing view which displays the 2D layout geometry and annotations seen in the background of a 2D Layout for 3D Design view.

Offsetting 3D Elements Associatively

You can now create an offset use-edge that will be associative to a 3D element (surface or edge).

About Multi-Domain Use-Edges

You can now create use-edges (silhouette, intersections, and projections) with multiple domains in geometrical result.

Enhanced Functionality

About View Callouts

You can now:

- **Create callouts automatically** in the active view, whenever possible.
- Create a callout with a size computed from the selected 3D reference, if any.
- **Position callout's text manually.**
- **Move callout's extremities** interactively so as to resize it.

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Customizing Settings

Tools > Options > Mechanical Design > 2D Layout for 3D Design > Manipulators tab > Annotation Manipulators

The new `Annotation Manipulators` options allow you to customize the display of manipulators, whether it is for annotation selection or text edition.

Tools > Options > Mechanical Design > 2D Layout for 3D Design > View Creation tab > Callout area

The `Create callouts in active view at the end of "New View From"` command option allows you to create a callout in the active view at the end of `New View From` command.

Structure Design

Enhanced Functionality

Generating Piece Parts

You can now view the summary of information related to processed objects after completion of piece part generation. If it fails to convert all the SFD / SDD components into the SR1 components, you can review all the parts and their status by clicking the `More` button, available in the `Piece Part Progress Bar` dialog box.

Additional Computed Attributes

You can now get the following attributes in the `Properties` dialog box after piece part generation for the various STR objects listed below:

- For Plates: Painted Area, Welding Length and Plate Perimeter
- For Stiffeners / Stiffeners on Free Edges: Painted Area and Welding Length
- For Beams: Painted Area

Wireframe and Surface

Enhanced Functionality

Defining an Axis System

It is now possible to have an associativity between the default computed inputs and user-defined inputs.

Selecting Using Multi-Output

You can now delete one or more elements in one instance while selecting using multi-output.

Replacing Elements

A new button enables you to cancel the inversion of the locally inverted feature.

Creating Fill Surfaces

The `Canonical portion detection` check box allows you to compute canonical portions.

Creating the Nearest or the Farthest Entity of a Multiple Element

You can now create the entity that is far from the reference surface.

Generative Sheetmetal Design

New Functionality

Creating a Free Form Surface

You can now create complex volumes from complex surfaces created with an other workbench.

3D Functional Tolerancing & Annotation

New Functionality

Dimensioning and Tolerancing Pattern of Non Cylindrical Holes

You can now create semantic annotations on pattern of non cylindrical holes.

Interrupting Extension Lines

You can now interrupt one or two extension lines of several dimensions, and also remove these interruptions.

Enhanced Functionality

Importing an Annotation Set

You can now import and **merge** annotation features from another CATPart.

About Welding Symbols

You can now specify a **scarf**, a **stud**, an **edge** and a **melt thru** weld symbol in AWS 2.4 standard and an **edge**, a **surface joint** and an **inclined joint** weld symbol in ISO 2553 or JIS Z3021 standard.

Annotation Manipulators

You can now customize the display of manipulators for annotations, whether it is for annotation selection or text edition.

Engineering Symbols

As per new revision of the ISO standard related to Geometrical Product Specification - Dimensional Tolerancing (14405-1), you can now have some new engineering symbols, to provide a standardized display and print.

You can also have some new **application zone symbols** **All Around**, **All Over** and **All About** to support ISO 10135.

Creating Threads using Thread Representation or Constructed Geometry Creation

You can now have CG thread representation as per Drafting standard, i.e. using collection of circles or collection of circular arcs depending on the value of the dedicated parameter defined in the selected drafting standard.

Reconnecting the Broken Links

You can now automatically start and validate the broken links reconnection using the `Geometry Connection Management` command, thus minimizing user interactions.

Customizing Settings

Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > Manipulators tab > Annotation Manipulators

The new `Annotation Manipulators` options allow you to customize the display of manipulators, whether it is for annotation selection or text edition.

Tools > Options > Mechanical Design > Functional Tolerancing & Annotation > Annotation tab > Geometry Connection Management

The new `Automatically propose reconnection of broken links` option allows you to define whether the reconnection of broken links is to be proposed automatically or not.

Mold Tooling Design

New Functionalities

Split Untrimmed Ejectors

Lets you split all untrimmed ejectors in one shot, while modifying their Offset_Parting.

Tool Viewer

Lets you filter tooling elements by type and change their display properties.

Tooling Bill Of Material

This Mold Tooling Design specific command provides you with a more accurate count of the components, a real management of the numbering and the capability to add user-defined columns.

Core & Cavity Design

New Functionalities

Model Properties

Displays the properties of a model and generates a HTML report, as well as the hidden parameters.

Generative Form Block

Creates a pad, andn if necessary, a DrillHole body (in the Mold Tooling Design sense) from a surface and a profile. The part can then be turned into an insert in Mold Tooling Design.

Sew Surface

Splits the faces of a solid to create a sew surface, that can be colored.

Plug Faces

Plugs holes in faces.

Fill Surfaces

Fill holes in surfaces.

Parting Line by Color

Creates an outer parting line by joining the outer edges of each colored mold area of a shape.

Enhanced Functionalities

Bounding Box

Creates now an associative bounding box which supports the replacement of the input shape or axis system.

Main Pulling Direction and Slider Lifter Direction

Both commands offer the following enhancements:

- Easier specification of the pulling direction,
- Editable colors of the areas,
- Areas can be extracted or colored.

Search Fillets

Offers the following enhancements:

- More user-friendly dialog box,
- Fillets can be extracted or colored.

Healing Assistant

Enhanced functionalities

Compare & Review

For a better understanding of the color map, a dialog box displays the color scale with the distance values.
The report opens automatically after generation.

Surface Connection Checker

You can now select solids as input, thus selecting a large number of faces in one shot.

Composites Design

New Functionalities

Junction Lines Wizard

This wizard:
helps you create junction lines,
checks and validates existing or new junction lines,
shows locations where junction lines are missing or incorrect,
provides tools for an easy modification of iso-thickness areas for simple cases.

Grid Angle Cut Wizard

This wizard is a quicker and foolproof alternative to the manual command.

Material Addition Wizard

This wizard is a quicker and foolproof alternative to the manual command.

Multiple Core Sampling

Creates or reviews several core samplings.

Gap Offset

Creates a gap offset on either a no splice zone or a butt splice zone to take the manufacturing tolerance into account.

Enhanced Functionalities

Workbench

For a better efficiency, commands are grouped differently and made available only when needed.

Panel Definition

A reference element can now be split into sub-elements, with specific staggering information.

Close reference elements are now managed.

Staggering information is displayed in the main Panel Definition dialog box.

Flattening

An option lets you now create and store the flatten curves as a sketch.

Interactive Ply Table

Tools have been added for an easier selection of Composites data.

The dialog box has been enhanced with more attributes.

Material Excess

Butt splice zones with **Gap Offset** are supported as input.

Import/Export of Composites Design data

.xlsx files are now supported in **Import Laminate**, **Grid Definition**, **Stack-up file from zones**, **Stack-up File from Core Samples**, **Plies From Zones**, **Interactive Ply Table**, **Ply Table**, **Ply Table Import**, **Limit Contours from Input File**, **Numerical Analysis**, Export after a **Core Sampling**, **Define Stagger Origin Points**.

FreeStyle Shaper Optimizer & Profiler

New Functionality

Performing Quick Mirror Analysis

Using `Soft Mirror` command, you can now perform quick mirror analysis of geometrical elements without adding the feature to specification tree. This helps you to take a quick check on mirrored elements.

Enhanced Functionality

Multiple Analysis Root Nodes

You can now add and reorder multiple nodes inside root analysis node.

Displaying Geometric Information On Elements

Now you can identify canonical elements and their types using the `Geometric Information` command.

Displaying and Modifying Dress-Up Features

All the options of the `Curves and surfaces dress-up` dialog box are now available on the same interface for easy access.

The `Variants` are created to quickly apply a pre-defined set of attributes using the `Dress-Up` command.

Displaying Furtive Elements While Creating Geometry

You can temporarily display furtive elements such as control points, segmentation, iso-curves and the free edges of the geometry being created while in a `FreeStyle` command. These elements are removed when you exit the command.

Breaking Surfaces

You can now instantly create an isoparametric curve as a limiting element in the context of `Break` command by clicking on the surface to break.

Keeping the Initial Element

The commands `Keep`, `No Keep` and `Keep original` in Generative Shape Design (GSD) workbench and FreeStyle Shaper, Optimizer and Profiler (FSS) workbenches now support each other in activating/deactivating from either of the workbenches.

Creating a Styling Fillet

The Output Result section in the Styling Fillet dialog box now displays the number of domain in a fillet ribbon.

Performing a Surface Curvature Analysis

The `Number of Colors`, `Use Min Max`, `Absolute Extrema`, `Green is Flat` and `Non-linear Ramp` contextual commands are available to customize the color scale for analyzing surfacic curvature.

Enhancements in Dynamic Cutting Plane Analysis

Intrapolation Option

The new interpolation option in the `Cutting Plane Analysis` dialog box computes interpolated NURBS curves when cloud/mesh surfaces are sectioned.

Edge of surface as input for Cutting Plane Analysis

You can now select the edge of the surface as an input curve for Cutting Plane analysis in planes perpendicular to curves mode.

Keep the Cutting Plane analysis result as NURBS curves

You can now choose to keep the Cutting Plane analysis intersection curves as NURBS curves.

Match Constraint Enhancements

Applying Continuity to Opposite Edge

You can now define continuity on the opposite edge for the matching edges for better transition of the surface.

Display of Maximum Deformation Distance and its Position

The `Display deformation distance` option in the `Match Constraint` dialog box displays the maximum deviation value and its position between the original surface and the modified surface. You can also reset the reference surface from where the deformation distance is computed.

Preselection of the Elements for Source Input

The element such a surface or a curve selected before the command is taken as an input for `Source` element.

Distance Analysis Enhancements

The user interface of the `Distance Analysis` dialog box has changed and the new options are added to the dialog box.

Selecting Elements

The `Source` and `Target` element selection options added to `Distance Analysis` dialog box allows you to select the set of element between which the analysis is to be performed.

Surface Edge as analysis input

Now you can select the surface edge as an input to perform distance analysis.

Relimit the input curves being analyzed

The `Relimitation` options added to the `Distance Analysis` dialog box allows you to modify the limits of the curve with the manipulators of analysis command on both the curves being analyzed.

Absolute Value Display

The `User max distance` option in the `Distance Analysis` dialog box displays the absolute value for computation.

Apply Scaling factor

The `x2` and `/2` scaling options added to the `Distance Analysis` dialog box allows you to modify the scale of the comb as in `Porcupine Curvature Analysis`.

Customizing Settings

Tools > Options > Shape > FreeStyle > General tab, Tuning

You can see the attenuation factor values corresponding to the coefficient value set in the tuning settings.

Tools > Options > Shape > FreeStyle > Manipulators tab, Furtive Display Variants

You now have the option to display control points, segmentation, iso-curves and the free edges of the geometry being created while in a `FreeStyle` command. You can specify the default color, line type and the symbol for furtive elements.

Tools > Options > Shape > FreeStyle > Manipulators tab, Dress-up Variants

You can now specify the default color, line type and the symbol for control points, segmentation, iso-curves and the free edges to dress-up the geometry.

Tools > Options > Shape > FreeStyle > General tab, Curve Analysis Result

You can now choose to keep the cutting plane analysis intersection curves as NURBS curves.

Generative Shape Design & Optimizer

New Functionality

Creating a Contour

This new functionality lets you create a contour.

Simplifying Surfaces

This new functionality lets you simplify the topology of surface.

Enhanced Functionality

Creating Multi-Sections Surfaces

You can now define curvature continuity with section supports or guide supports.

A multi-section surface can be created with the **guide curves intersecting at extremity**.

Creating Offset Surfaces

The new `Regularization` option enables you to regularize the offset surface locally.

Creating Swept Surfaces

The new `Compute C0 vertices as twisted areas` check box enables you to fill the C0 vertices areas.

Defining an Axis System

It is now possible to have an associativity between the default computed inputs and user-defined inputs.

Selecting Using Multi-Output

You can now delete one or more elements in one instance while selecting using multi-output.

Global Deformation on Wires

You can now select curves as an element to deform in the **Bump, Shape Morphing, Wrap Curve and Wrap Surface** commands.

Splitting Geometry

The new `Approximation` tab enables you to control the quality of the result of the `Split` through several parameters and modes.

Replacing Elements

A new button enables you to cancel the inversion of the locally inverted feature.

Creating Fill Surfaces

The `Canonical portion detection` check box allows you to compute canonical portions.

Creating the Nearest or the Farthest Entity of a Multiple Element

You can now create the entity that is far from the reference surface.

Creating a Hole

The new `Axis Computation` option enables you to automatically create axis and direction lines passing through the center of the hole.

Creating a Hole Curve

The new `Axis Computation` option enables you to automatically create an axis and direction lines passing through the center of the hole curve.

Creating Multi-Sections Volumes

You can now define curvature continuity with section supports or guide supports.

A multi-section surface can be created with **the guide curves intersecting at extremity**.

Shape Sculptor

New Functionalities

Excavation

In Terrain modeling, this command creates an excavation surface to help you compute the volume of soil to dig out.

Properties

A `Cells` tab has been added, which enables you to **modify the properties** of cells of clouds of points or meshes.

Enhanced Functionalities

Drawing from Meshes

Mesh silhouettes are drawn instead of mesh bounding boxes.

Import

If the dimensions of the data to import exceed those of the work space, or if the data to import would be outside the work space, V5 proposes you a scale factor or a local axis system to fit the data into the work space.

Contour Map

The command `ZLevel Map` has been renamed into `Contour Map` (and `ZLevel` has been renamed `Contour Line`).

It has been enhanced with a better positioning of annotations, and more parameters to manage those annotations.

Customizing Settings

Highlight mode

A new option `Bounding Box` has been added to the `Display Modes` settings, to let you highlight selected clouds of points or meshes using the V5 standard highlight (new) or the highlight by bounding box (used until now).

Imagine & Shape

New Functionality

Moving Along Edges

This new functionality lets you translate vertices along the edges of the base mesh.

Hiding the Mesh

This new functionality lets you hide the mesh to lighten its visualization during manipulation or modification.

Importing Meshes

This new capability lets you import subdivision surfaces from *.obj files.

Modifying the Mesh

This new capability lets you use tools to modify a mesh.

Enhanced Functionality

Applying a Modification

A warning dialog box is displayed if you do not apply the modifications.

Performing a Symmetry

You can now select several subdivision surfaces.

Aligning Vertices

In case of a projection onto a support element, the manipulator is now represented with a sphere in its middle.

Extruding Faces and Edges

Two new buttons appears in the `Tools Palette` when the Fill mode is activated and providing two sets of connected edges are detected with different size in order to let you choose the result.

Previewing the Mesh

When working with the following commands: `Extrude`, `Subdivide` and `Cut`, the mesh being modified is now previewed in lowlight mode.

Note: No specification documentation is associated to this enhancement.

Customizing Settings

General

A new option lets you activate or deactivate the warning message that is displayed when you exit a command without applying the modifications.

Display

You can now visualize a material applied on the sub-elements of a shape on preview.

Digitized Shape Editor

New Functionalities

Properties

A `Cells` tab has been added, which enables you to **modify the properties** of cells of clouds of points or meshes.

Enhanced Functionalities

Drawing from Meshes

Mesh silhouettes are drawn instead of mesh bounding boxes.

Import

If the dimensions of the data to import exceed those of the work space, or if the data to import would be outside the work space, V5 proposes you a scale factor or a local axis system to fit the data into the work space.

Deviation Analysis

When a Realistic Shape Optimizer license is available, you can export the Deviation Analysis data as an ASCII file.

Customizing Settings

Highlight mode

A new option Bounding Box has been added to the `Display Modes` settings, to let you highlight selected clouds of points or meshes using the V5 standard highlight (new) or the highlight by bounding box (used until now).

Quick Surface Reconstruction

New Functionalities

Properties

A `Cells` tab has been added, which enables you to **modify the properties** of cells of clouds of points or meshes.

Enhanced Functionalities

Automatic Surface

Three new options are available:

- Regular stream lines,
- Fill holes,
- Extend surface.

They improve the quality and aspect of the output surface while reducing its weight as well as the computation time.

Drawing from Meshes

Mesh silhouettes are drawn instead of mesh bounding boxes.

Deviation Analysis

When a Realistic Shape Optimizer license is available, you can export the Deviation Analysis data as an ASCII file.

Geometric Tolerances Checker

Defects are identified in the 3D view, via a color map, and can be analyzed.

Dimensioning tolerances are now supported.

Customizing Settings

Highlight mode

A new option Bounding Box has been added to the `Display Modes` settings, to let you highlight selected clouds of points or meshes using the V5 standard highlight (new) or the highlight by bounding box (used until now).

Realistic Shape Optimizer

New Functionalities

Vectors Field Filter

Checks the quality of the vectors field defining the displacement and removes unsuitable vectors.

ICEM Shape Design Center

New Functionality

Managing Multi-Result Operations

This function is now available in the following commands:

- Curve Creation
 - Curve Projection
 - Curve Offset
 - Split Curve
 - Intersection
- Surface Creation
 - Surface of Revolution
 - Flange
 - Styling Fillet
 - Advanced Fillet
 - Tri-Tangent Fillet
- Expert
 - Adjust
 - Helix

Multi-Output capability

This function is now available in the following commands:

- Shape Modification > Refit
- Surface Creation > Surface Offset
- Expert > Adjust

Enhanced Functionality

Approximation tab

New optimized user interface for approximation parameter settings to improve the user workflow during geometry creation.

Patch from Curves

Improved command workflow/UI:

- Improved selection workflow.
- Continuity management options.
- Arbitrary four sides/supports.
- Improved Adapt option.
- Curve projection on Support.

Patch from Patches

New Feature based version of `Patch from Patches` command.

New `Topology` tab to validate geometry inputs.

Advanced Fillet

Improve algorithm.

Domain handling.

Error handling.

Inner qualities.

Flange

Add `Distance`, `Align Opposite`, and `Assign Tangent` options.

Use new parameterizations to get better control points.

G1 tolerance available on Approximation tab.

Improve `Inverse/Reverse` handling.

Improve `Both Side` handling.

Blend Surface

New option `Force G1 in edge direction` improving internal coupling connectivity G1.

New `Alignment` option controlling the distribution of the control points of the blend surface:

- For the modes `Edges Start/End`, `All Edges` and `Auto`, the start and end tangents can be modified via the manipulator.
- As the direction of the cross tangents is fixed when using the alignment modes `Standard` and `Linear`, the start and end tangents cannot be modified.

Loft

Add separate `Spine` option.

New UI with `Support` fields added to `Start and End of Guide` and `Profile Alignment`.

Curve Projection

New method for creating of view direction.

View direction is now stored as a `Line`.

Contextual menu extended in view supported commands.

Styling Corner

Extended stabilization of result:

- Impose reference plane/surface selection to manage orientation during replace/update procedures.
- Projection of result onto support.

Blend Curve

Add `Support` option.

Allow creation of 2D curves on single cells.

Curve Offset

Add `G3 Extrapolation` without an extra segment.

Add `2D Curve` option to get the result as a wire of surface curves.

Change the projection algorithm to nearest projection.

Create an additional result body for the second offset curve if `Both Sides` is selected.

Highlight Analysis

Support of Display Sets.

Split Analysis

Support of Display Sets.

Make **Revert** button available when a Direction is selected.

Rework **Align** mode.

Surface Check

Support of Display Sets

New Functionality

Managing Multi-Result Operations

This function is now available in the following commands:

- Curve Creation
 - Curve Projection
 - Curve Offset
 - Split Curve
 - Intersection
- Surface Creation
 - Surface of Revolution
 - Flange
 - Styling Fillet

Multi-Output capability

This function is now available in the following commands:

- Shape Modification > Refit
- Surface Creation > Surface Offset

Enhanced Functionality

Approximation tab

New optimized user interface for approximation parameter settings to improve the user workflow during geometry creation.

Patch from Curves

Improved command workflow/UI:

- Improved selection workflow.
- Continuity management options.
- Arbitrary four sides/supports.
- Improved Adapt option.
- Curve projection on Support.

Patch from Patches

New Feature based version of `Patch from Patches` command.

New `Topology` tab to validate geometry inputs.

Flange

Add `Distance`, `Align Opposite`, and `Assign Tangent` options.

Use new parameterizations to get better control points.

G1 tolerance available on Approximation tab.

Improve `Inverse/Reverse` handling.

Improve `Both Side` handling.

Blend Surface

New option `Force G1` in edge direction improving internal coupling connectivity G1.

New `Alignment` option controlling the distribution of the control points of the blend surface:

- For the modes `Edges Start/End`, `All Edges` and `Auto`, the start and end tangents can be modified via the manipulator.
- As the direction of the cross tangents is fixed when using the alignment modes `Standard` and `Linear`, the start and end tangents cannot be modified.

Loft

Add separate `Spine` option.

New UI with `Support` fields added to `Start` and `End of Guide` and `Profile Alignment`.

Curve Projection

New method for creating of view direction.

View direction is now stored as a `Line`.

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Blend Curve

Add `Support` option.

Allow creation of 2D curves on single cells.

Curve Offset

Add `G3 Extrapolation` without an extra segment.

Add `2D Curve` option to get the result as a wire of surface curves.

Change the projection algorithm to nearest projection.

Create an additional result body for the second offset curve if `Both Sides` is selected.

Generative Structural Analysis

Enhanced Functionality

Result Visualization

Force Flow 2d and 3d Text Images

Two new images are available in the Generate Images dialog box: Force Flow 2d Text and Force Flow 3d Text.

Export Data from Extrema

You can transfer extrema (coordinates, values, type) into a .txt or .xls file.

Color Map in the Specification Tree

The color map now appears in the Specification Tree under its corresponding image.

Analysis Connections

Creating General Analysis Connections

You can select several mechanical features when defining general analysis connections.

Virtual Parts

Creating Virtual Parts

- Virtual Parts can be applied to Geometrical Groups and Groups by Neighborhood supports.
- When you select a support, the `Select Mesh Part` button is available.

Groups

Grouping Surfaces

You can select surfaces continuously according to the angle between them.

Sensors

Creating Local Sensors

You can select an image as support when creating local sensors in a mono-occurrence or multi-occurrence computed solution. Only entities visualized in the image will be taken into account.

Reference Information

Image Edition

You can select an image as group which allows you to filter entities by values.

Dynamic Response Sets

Defining a Load Excitation Set

Enforced displacement loads can now be selected when defining load excitation sets. The modulation applied to this load can be interpreted as accelerations on the nodes in the load definition.

Nonlinear Structural Analysis and Thermal Analysis

Enhanced Functionality

Geometric Correction for Surface-to-Surface Contact Pairs

In a Nonlinear Structural analysis, you can now specify geometric correction for a surface-to-surface contact pair definition in your model. The contact smoothing definition recognizes cylindrically or spherically shaped surfaces and automatically smooths the surfaces. Using the smoothed surface geometry can greatly improve the results in some cases. For more information about specifying geometric correction in a contact pair, see [Configuring the General Options in a Contact Pair](#).

Saving a Results Image as a New Template

You can now save a results image as the template for other results images you create in your analysis. Nonlinear Structural Analysis and Thermal Analysis make image templates available for selection in the **Abaqus Image Generation** dialog box. For more information about saving results images as templates, see [Generating a Results Image](#).

Job Names Can Start With Numbers

The job name can now start with a number, which provides greater flexibility for naming and sorting the jobs in an analysis. For more information about job definition, see [Creating a New Job](#).

Consistency Check Error Displayed When Input File Writing Errors Occur

Nonlinear Structural Analysis or Thermal Analysis now show a consistency check error at the top of the list of errors in the **Consistency Check Messages** tabbed window if there are one or more input file writing errors in the **Write Input File Messages** tabbed window. This consistency check error does not appear if the input file writing generates only warnings, and only one consistency check error is displayed even if multiple input file writing errors are displayed in the **Write Input File Messages** tabbed window. The **Job Submission** dialog box shows the **Write Input File Messages** tabbed window by default if an input file writing error occurs.

For more information about running consistency checks for an Abaqus job, see [Submitting Jobs](#).

Automatic Creation of Surface Groups During Postprocessing

By default, Nonlinear Structural Analysis and Thermal Analysis now automatically create a display group for every Surface Group definition in your analysis when you run an Abaqus job. This enhancement makes it easier for you to define a results image for selected surfaces in your model, because creating results images on a subset of your model requires the selection of a display group. You can also customize this behavior so that display groups are created for contact surfaces only, or you can disable the new behavior for all surfaces. Control over display group creation for surface groups is available in the **Analysis & Simulation** settings in the **Options** dialog box.

For more information about this enhancement, see [Creating a Display Group](#).

Automatic and Consistent Input File Naming

Nonlinear Structural Analysis and Thermal Analysis now uses the name of the .CATAnalysis file as the basis for the names of all Abaqus jobs you create. This approach makes the management of Abaqus analysis and postprocessing files easier, because their names automatically match those of the CATIA analysis files you create.

For more information about this enhancement, see [Creating a New Job](#).

Extending the Selected Face for Surface Selection

You can now use the **Extend Selected Face** method to select multiple faces as the support for a surface-based feature. For each face you select, the **Extend Selected Face** method selects all faces in the model that are within a user-specified angle from the selected face or faces and connected to them. You can also specify that this method selects only adjacent faces or only meshed surfaces. For more information about this selection method and the features to which it applies, see [Advanced Selection Options for Supports](#).

Abaqus Input File Available After Analysis

When you run an analysis in the Nonlinear Structural Analysis and Thermal Analysis workbenches, the Abaqus input file created by the analysis is now available. This enhancement enables you to make further edits to the input file directly if you want to refine your analysis further or run the analysis elsewhere.

Analyses Can Now Use Up to Four Processors

The Nonlinear Structural Analysis and Thermal Analysis workbenches now enable you to run an Abaqus job using as many as four processors. For more information on parallel processing for Abaqus jobs, see [Controlling Parallel Execution](#).

Support for Rigid and Smooth Couplings

The Nonlinear Structural Analysis workbench now supports the definition of Rigid Couplings and Smooth Couplings. For more information about these features, see [Creating Rigid Couplings](#) and [Creating Smooth Couplings](#) respectively.

Support for General Contact

You can now specify general contact for your analysis in the Nonlinear Structural Analysis workbench, which enables you to prevent surface penetration between multiple bodies in your model. For more information about general contact, see [General Contact](#).

CPD Properties Now Available With ACP License

The ACP license now enables you to use the Imported Composite Property tool and the Mapping Property tool.

Advanced Meshing Tools

Enhanced Functionalities

Surface Meshing

Offset in RBM

You can apply an offset while creating a mesh with the Rule Based Surface Mesher. The mesh is created based on an offset representation of the geometry (the actual geometry is unchanged). This results in a higher quality mesh than if the surface were meshed and then offset.

Curve Projection

The local specification for the projection of curves to constrain a mesh has been enhanced to allow selection of curves from 2D and 3D features as well as the 1D features that were available previously. In addition, if you use automatic mesh capture while creating a surface mesh with RBM, you can also automatically project curves. If you automatically project curves during mesh capture, the same tolerance is used for mesh capture and curve projection.

Mesh Part Selection in RBM

The automatic mesh capture option in the Rules Based Surface Mesh dialog box now includes an option to select all mesh parts. Choosing this option will account for all up to date meshes within the automatic mesh capture tolerance range when you create the surface mesh.

Analysis Tools in RBM

The Free Edges, Cutting Plane, Interferences, and Element Quality Analysis tools are now available in the Rules Based Meshing workbench. These tools are located in the Mesh Analysis Tools toolbar.

Electrical Harness Installation

Customizing Settings

Dump Parameter Settings

You can now generate a report displaying the actual state of settings of all the tab pages.

Electrical Harness Flattening

Enhanced Functionality

Automatically Updating Flatten Links

You can select the desired bundle segment to be mapped by clicking the label. Clicking the label enables you to switch the string to a full path string from the geometrical bundle to the bundle segment name.

Generating the HTML Report during Synchronization

You can now handle the branches whose route orientation has been inverted in harness data during the synchronization process.

Support-Mechanical Parts Management

You can now manage the supports with publications used for branch routing during the **Extract** or **Synchronization** process.

System Routing

New Functionality

Converting Run into Solid Equivalent

You can now create a solid equivalent for run. The CATPart generated using this command can be used to generate the required raster views.

Enhanced Functionality

Routing a Run at an Offset

You can route a run at an offset from a reference surface.

Moving Nodes

You can now move a node to a 3D point.

Translating Run Segment

You can now translate a run segment without moving the free segment. You can also limit the translation of the run segment in Z-direction of compass.

Systems Space Reservation

New Functionality

Converting Run into Solid Equivalent

You can now create a solid equivalent for run. The CATPart generated using this command can be used to generate the required raster views.

Enhanced Functionality

Routing a Run at an Offset

You can route a run at an offset from a reference surface.

Moving Nodes

You can now move a node to a 3D point.

Translating Run Segment

You can now translate a run segment without moving the free segment. You can also limit the translation of the run segment in Z-direction of compass.

Raceway & Conduit Design

New Functionality

Converting Run into Solid Equivalent

You can now create a solid equivalent for run. The CATPart generated using this command can be used to generate the required raster views.

Enhanced Functionality

Routing a Run at an Offset

You can route a run at an offset from a reference surface.

Translating Run Segment

You can now translate a run segment without moving the free segment. You can also limit the translation of the run segment in Z-direction of compass.

Moving Nodes

You can now move a node to a 3D point.

Waveguide Design

New Functionality

Converting Run into Solid Equivalent

You can now create a solid equivalent for run. The CATPart generated using this command can be used to generate the required raster views.

Enhanced Functionality

Routing a Run at an Offset

You can route a run at an offset from a reference surface.

Translating Run Segment

You can now translate a run segment without moving the free segment. You can also limit the translation of the run segment in Z-direction of compass.

Moving Nodes

You can now move a node to a 3D point.

Piping Design

New Functionality

Calculating Weight and Center of Gravity of a Spool

You can now calculate the weight and center of gravity of a spool.

Converting Run into Solid Equivalent

You can now create a solid equivalent for run. The CATPart generated using this command can be used to generate the required raster views.

Enhanced Functionality

Routing a Run at an Offset

You can route a run at an offset from a reference surface.

Translating Run Segment

You can now translate a run segment without moving the free segment. You can also limit the translation of the run segment in Z-direction of compass.

Moving Nodes

You can now move a node to a 3D point.

Display Bend, Flange, Coupling and Clamp in ISOGEN Drawings

Now the ISO symbols for bend, flange, coupling and clamp are displayed in the ISOGEN drawings.

These components are exported to the PCF file. You need to map the ISOGEN SKEY representing the support to the CATIA symbol name.

Note: No specific documentation is associated to this enhancement.

Tubing Design

New Functionality

Calculating Weight and Center of Gravity of a Spool

You can now calculate the weight and center of gravity of a spool.

Converting Run into Solid Equivalent

You can now create a solid equivalent for run. The CATPart generated using this command can be used to generate the required raster views.

Enhanced Functionality

Routing a Run at an Offset

You can route a run at an offset from a reference surface.

Translating Run Segment

You can now translate a run segment without moving the free segment. You can also limit the translation of the run segment in Z-direction of compass.

Moving Nodes

You can now move a node to a 3D point.

Modifying Flexibles

You can now modify a flexible by path point modification by removing or replacing the selected point or adding a point after or before the selected one.

HVAC Design

New Functionality

Calculating Weight and Center of Gravity of a Spool

You can now calculate the weight and center of gravity of a spool.

Converting Run into Solid Equivalent

You can now create a solid equivalent for run. The CATPart generated using this command can be used to generate the required raster views.

Enhanced Functionality

Routing a Run at an Offset

You can route a run at an offset from a reference surface.

Translating Run Segment

You can now translate a run segment without moving the free segment. You can also limit the translation of the run segment in Z-direction of compass.

Moving Nodes

You can now move a node to a 3D point.

Duct Thickness Rule Table

You can now define five new parameters in the HVAC Material Definition Table, viz. Shape, MinHeight, MaxHeight, MinWidth and MaxWidth. In this case, the Material Definition list is filtered by Shape (Round/ Rectangle) and Size (Height and Width for Rectangle and Height only for Round shape.) along with the specification.

Ship Structure Detail Design

New Functionality

Creating a Shell Expansion View

You can now create a shell expansion view of the SDD system.

NC Manufacturing Infrastructure

New Functionalities

ROTABL Output for Machine Instruction

The `Output Axes Moves Only` check box in the `Machine Editor` dialog box helps to generate APT output with some user-defined word in X,Y,Z,I,J,K axis mode without the CATIA0 matrix.

Remove Coincidental Point between Machining Operations

The `Remove coincidental point between operations` check box is available in **Tool motions** tab of `Generate NC output interactively` dialog box. This check box helps to output the coincidental points even if the end point of a machining operation and start point of the subsequent machining operation are exactly the same.

Enhanced Functionalities

Extend Last Selected Edge Command in EdgeSelection Tool bar

You can add an extra element to the last selected edge through the `Extend Last Selected Edge` command in the `Edge Selection` tool bar. The extension can be controlled by a given distance or by a limiting element.

NC Macros

- For the **Ramping up to plane** macro, the slope no longer goes down when the tool path does, thus preventing plunges into the material.
- For **HSM macros**, a new parameter lets you control the Arc Angle value.

Customizing Settings

Lathe deviation (%) stock radius

You can manage the video lathe simulation accuracy through this setting, depending on the size of part.

Display Color for Spindle Rotation Arrow and Arrow Head

You can set the display color for the spindle rotation arrow and arrow head.

Compute Tool Path

Select this check box to manage disable invalid Machining Operations and continue till the end of the Machining Operations list versus stop at first error when computing a tool path.

Prismatic Machining Preparation Assistant

New Functionalities

Create Machinable Axial Feature from Technological Results

A new option has been added in the `Global Feature Recognition` dialog box to create Machinable Axial Feature from the Technological Results information. It strengthens the integration between design and machining by a direct use of Technological Results (including user parameters). The former geometrical recognition of features still exists and could be used on imported models where Technological Results are not defined. A new filter: `only for existing tech.Results/only for non existing tech.Results` eases the hole selection for hybrid models.

Add PP Word in Machining Patterns and Axial Operations

You can insert and display PP Word instructions before any hole in the Machining Pattern and Axial Machining Operations.

Prismatic Machining

New Functionalities

Add PP Word in Machining Patterns and Axial Operations

You can insert and display PP Word instructions before any hole in the Machining Pattern and Axial Machining Operations.

Enhanced Functionalities

Pocketing Machining Operations

Three new strategies Concentric, Inward Spiral Morphing, and Outward Spiral Morphing are added in the Tool Path Style.

These strategies are dedicated to the hard material milling and ensure:

- 1) a constant amount of material removal and load on the tool for Concentric strategy
- 2) a continuous cutting motion without retract and maximum stepover to control the load on the tool for Spiral Morphing.

Profile Contouring Operation

The following enhancements increase automation and productivity:

- 1) **Control of the side to machine:** instead of relying on the orientation of the first curve, you can select a 3D point to indicate the material side without ambiguity.
- 2) **Improved Processing of Join in Profile Contouring:** Chain curves form a join in contiguous sets and process each set as a guide contour.
- 3) Control of the Tool Axis Motion in Profile Contouring: A parameter **Smoothing tool path along Tool Axis (%)** is added in the Stepover tab to smooth the tool path and avoid motions along the tool axis direction.
- 4) **Support of Multiple Contours:** No need to split the Profile Contouring operation into several operations.

Improved Collision Check and Collision Avoidance

Ability to include the Part from Part Operation in the collision check performed on macro for the following Machining Operations: **Profile Contouring, Groove Milling, Pocketing, Facing, Curve Following, 4x Pocketing and Trochoid Milling** Machining Operations. The Part from Part Operation check box is added in the Collision Checking dialog box and the Collision Checking button is added where needed in the Geometry tab.

Improved Intermediate Stock Management in Axial Machining Operations

- A **Drill through stock upto bottom** check box is added in the Geometry tab. This helps to perform the axial operation start from top of the stock up to bottom of the stock in case of Through hole .
- A **Spot drill to part** check box is added in the Geometry tab of the Spot Drilling Machining Operation. It allows the drilling from the intermediate stock up to the part (and ensures the spot-drilling depth/diameter on the part)

Prismatic Roughing

When the tool is in contact with a revolution surface, you can choose to generate an arc interpolation output.

3 Axis Surface Machining

Enhanced Functionalities

Roughing

When the tool is in contact with a revolution surface, you can choose to generate an arc interpolation output.

Rework Area

A new option let you detects bitangency areas between Part and Check Elements

Sweep Roughing, Sweeping, 4-Axis Curve Sweeping, Pencil, Contour-driven, Spiral Milling

The slope in the **Ramping up to plane** macro is no longer allowed to go down when the tool path does, thus preventing plunges into the material.

Sweeping, 4-Axis Curve Sweeping, Pencil, Contour-driven, Spiral Milling

You can control the Transition angle value in **HSM macros**.

Multi-points Probing Machining Operation

Probe along the **normal to the part** for each point is now possible.

All Probing Operations

You can now:

Customize the **NC_PROBING_CYCLE_OFF** for each probing operation.

Set **different values** for each point for a user parameter.

Define the **Compensation** for APT generation.

Multi-Axis Surface Machining

Enhanced Functionalities

Multi-Axis Spiral Milling

The slope in the **Ramping up to plane** macro is no longer allowed to go down when the tool path does, thus preventing plunges into the material.

You can control the Transition angle value in **HSM macros**.

Multi-Pocket Machining

Enhanced Functionalities

Power Machining

When the tool is in contact with a revolution surface, you can choose to generate an **arc interpolation output**.

Multi-Pockets Flank Contouring

You can **reduce feedrate in corners**.

You can switch the **feedrate in linking motions to a rapid one**, based on the length of the passes.

You can allow and control an **overlap distance** for close tool paths.

You can choose to generate an **arc interpolation output** for circular motion in the plane perpendicular to the tool axis.

The slope in the **Ramping up to plane** macro is no longer allowed to go down when the tool path does, thus preventing plunges into the material.

You can control the Transition angle value in **HSM macros**.

Advanced Machining

New Functionalities

Create Machinable Axial Feature from Technological Results

A new option has been added in the `Global Feature Recognition` dialog box to create Machinable Axial Feature from the Technological Results information. It strengthens the integration between design and machining by a direct use of Technological Results (including user parameters). The former geometrical recognition of features still exists and could be used on imported models where Technological Results are not defined. A new filter: `only for existing tech.Results/only for non existing tech.Results` eases the hole selection for hybrid models.

Add PP Word in Machining Patterns and Axial Operations

You can insert and display PP Word instructions before any hole in the Machining Pattern and Axial Machining Operations.

Enhanced Functionalities

Pocketing Machining Operations

Three new strategies `Concentric`, `Inward Spiral Morphing`, and `Outward Spiral Morphing` are added in the `Tool Path Style`.

These strategies are dedicated to the hard material milling and ensure:

- 1) a constant amount of material removal and load on the tool for `Concentric` strategy
- 2) a continuous cutting motion without retract and maximum stepover to control the load on the tool for `Spiral Morphing`.

Profile Contouring Operation

The following enhancements increase automation and productivity:

- 1) **Control of the side to machine:** instead of relying on the orientation of the first curve, you can select a 3D point to indicate the material side without ambiguity.
- 2) **Improved Processing of Join in Profile Contouring:** Chain curves form a join in contiguous sets and process each set as a guide contour.
- 3) **Control of the Tool Axis Motion in Profile Contouring:** A parameter **Smoothing tool path along Tool Axis (%)** is added in the `Stepover` tab to smooth the tool path and avoid motions along the tool axis direction.
- 4) **Support of Multiple Contours:** No need to split the `Profile Contouring` operation into several operations.
- 5) **Improved collision check and collision avoidance:** Ability to include the Part from `Part Operation` in the collision check performed on macro for the following Machining Operations: **Profile Contouring, Groove Milling, Pocketing, Facing, Curve Following, 4x Pocketing** and **Trochoid Milling** Machining Operations. The `Part from Part Operation` check box is added in the `Collision Checking` dialog box and the `Collision Checking` button is added where needed in the `Geometry` tab.

Improved Intermediate Stock Management in Axial Machining Operations

- A **Drill through stock upto bottom** check box is added in the `Geometry` tab. This helps to perform: axial operation start from top of the stock up to bottom of the stock in case of `Through hole`.
- A **Spot drill to part** check box is added in the `Geometry` tab of the `Spot Drilling` Machining Operation. It allows the drilling from the intermediate stock up to the part (and ensures the spot-drilling depth/diameter on the part).

Prismatic Roughing

When the tool is in contact with a revolution surface, you can choose to generate an arc interpolation output.

3-Axis Surface Machining Roughing

When the tool is in contact with a revolution surface, you can choose to generate an arc interpolation output.

Rework Area

A new option lets you detect bitangency areas between Part and Check Elements.

Sweep Roughing, Sweeping, 4-Axis Curve Sweeping, Pencil, Contour-driven, Spiral Milling

The slope in the **Ramping up to plane** macro is no longer allowed to go down when the tool path does, thus preventing plunges into the material.

Sweeping, 4-Axis Curve Sweeping, Pencil, Contour-driven, Spiral Milling

You can control the Transition angle value in **HSM macros**.

Multi-points Probing Machining Operation

This enhancement allows to:

Customize the `NC_PROBING_CYCLE_OFF` for each probing operation.

Set different values for each point for a user parameter via `User parameter` contextual menu.

Probe along the normal to the surface for each point via `Point` contextual menu.

Select output points for APT generation via `Output Point: Tip/Center` option in the `Strategy` tab.

Multi-Axis Spiral Milling

The slope in the **Ramping up to plane** macro is no longer allowed to go down when the tool path does, thus preventing plunges into the material.

You can control the Transition angle value in **HSM macros**.

Lathe Machining

New Functionalities

Start Limit Mode and Offset Support in Rough Turning Machining Operation

Start element selection is provided in Geometry tab to limit input stock in Rough Turning Machining Operation.

Multi-Slide Lathe Machining

Enhanced Functionalities

Define Spindle Way of Rotation during Replay and Simulation

You can get information on spindle associated with a machining operation by placing pointer on machining operation node in PPR tree or on bar (associated to machining operation) in Gantt chart. The Spindle contextual menu helps to activate or deactivate the display of spindle rotation arrow. This helps to give a quick understanding of the spindle rotation.

STL Rapid Prototyping

New Functionalities

Properties

A `Cells` tab has been added, which enables you to **modify the properties** of cells of clouds of points or meshes.

Enhanced Functionalities

Drawing from Meshes

Mesh silhouettes are drawn instead of mesh bounding boxes.

Import

If the dimensions of the data to import exceed those of the work space, or if the data to import would be outside the work space, V5 proposes you a scale factor or a local axis system to fit the data into the work space.

Customizing Settings

Highlight mode

A new option Bounding Box has been added to the `Display Modes` settings, to let you highlight selected clouds of points or meshes using the V5 standard highlight (new) or the highlight by bounding box (used until now).

DPM Process and Resource Definition

Enhanced Functionalities

Displaying Selected Action and Modification Statement in DPM Title Bar

You can display the name of selected action and modification statement in the DPM title bar. When there is a change in the action and mod statement during re-load of project; then corresponding name gets updated in the title bar. This helps to see the current action and modification statement names without any additional action in the title bar of loaded project.

Simulation Positions in the Manufacturing Context

You can load the manufacturing context and get the products/resources loaded in the last assembled positions. This allows to see the current state of the assembly before that particular process. The 3D States and Positions are also supported in the Manufacturing Context.

Enhancements in DPM Search Dialog Box

You can show/hide selected attributes and search by specifying advanced search criteria in the `Search Criteria` dialog box. The ability to see only the needed search attributes helps to save the time as no need to scroll and sort through the non-needed search attributes.

Insert TSAs under Resource without behavior

You can now use the `Insert template` command to insert TSAs under resource if it does not already have behavior under it. You can also specify the number of TSAs to be inserted in the **Quantity of Objects**. This helps to make changes to process and resource planning on the fly without reloading the project or closing the `Insert template` command.

Knowledge Advisor

Enhanced Functionality

Design Table is now Microsoft Office 2010 compatible.

Note: No specific documentation is associated to this enhancement.

Knowledge Expert

Enhanced Functionality

Design Table is now Microsoft Office 2010 compatible.

Note: No specific documentation is associated to this enhancement.

DMU Engineering Analysis Review

Enhanced Functionality

Color Map in the Specification Tree

The color map now appears in the Specification Tree under its corresponding image.

Image Edition

You can select an image as group, which allows you to filter entities by values.

DMU Composites Review

New Functionalities

Interactive Ply Table

Lets you visualize plies interactively

Enhanced Functionalities

Commands

They have been grouped into a single menu and a single toolbar.

Human Builder

New Functionality

XML Settings for VOA

This describes the functionalities for XML Settings for VOA.

Enhanced Functionality

Defining the Occupant Posture Prediction Dialog box

The Method section of the Occupant Posture Prediction Definition dialog box has been enhanced to reflect postures according to the J4004 reference.