DELMIA Solutions Version 5 Release 18
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DPM Process and Resource Definition

New Functionalities

PRD

Process and Resource Definition

Use products of a Black Box for Manufacturing Assemblies and Manufacturing Kits
> A Black Box is a CATIA product which can contain as many as desired products (parts). Starting from release R18 you can use products of a Black Box in DPM for Manufacturing Assemblies and Manufacturing Kits.

Creating 3D States
> Starting with R18 the position of the 3D State can be synchronized and directly displayed while defining a new state with the help of the Option Synchronize to States Definition in the Dialog Create Named State.

Creating a Start Condition and 3D Context
> In Release R18, all characters occurring in the name of the resource or product can be used as searching criteria. The Search algorithm does not differentiate between lower-case or upper-case letters.

Synchronize Position and States, Modify Position and Redefine Reference Object
> Starting with release R18 you can find the three new commands in the State Management Tools toolbar:
> Synchronize Position, Modify Position and Redefine Reference Object.

Create Positions
> In new dialog "Create Position" you can create an absolute and a relative Position. It is possible that you can create states for absolute and relative position directly in the dialog.

Cross-Highlighting during Process Verification
> There is the new option Child processes in the Cross-Highlight Options dialog box.

Cross-Highlight Assigned Objects
> Starting with R18 there is the new option Child processes in the dialog Cross highlight options: within cross highlight of assigned objects command a possibility is given to highlight products, resources and Manufacturing Assemblies/ Manufacturing Kits to assign to children of the current process

About the Process Verification Options
> Starting with R18 there is a new option Keep Assembly State after PV in the dialog Process Verification.

Restoring the Original Design Position
> Starting with R18 the user has got the possibility to display the see the displaced positions in a list when using the option Restore Selected Design States.

Resource attributes
> Sub Type, Manufacturer, Time and Cost attributes have been added to the Properties dialog (Resource tab) for resources.

Save selected product data as a CGR file
> A new command allows for product data inside a CATProcess document to be saved into a CGR file.

Save selected product data as a CATProduct
> A new command allows for product data inside a CATProcess document to be saved into a CATProduct document.

Manufacturing Hub

Quick access data
> Quick access functionality allows specific datasets to be quickly created and shared between users, and provides a means of automatically loading a particular dataset into DPM.

Error logging
> A log file is created for each V5 session to document any errors or warnings that are encountered when data is exchanged between the Manufacturing Hub and DPM.

Accessing non-geometric data in VPM and ENOVIA
> Non-geometric data (e.g., spreadsheets and other documents) from VPM referenced by products and resources in the Manufacturing Hub is accessible.

Reference-based volumetric filtering
> Volumetric filtering allows for a filter to be based on a user-defined 3D bounding box. This bounding box can now be associated to a specific reference product.

Collaborative simulation
> Publishing and subscribing mechanisms are provided to make simulation data visible to others (publishing) and to access simulation data that others have published (subscribing).

Verifying detailing data integrity
> The CATDUAV5 tool helps in solving inconsistencies when opening a detailing with updated product geometry or assembly data.

Enhanced Insert from Template command
    User interface enhancements (the addition of Select Parent and Update Resource Tree commands) assist in providing improved performance of Instantiate from Template.
Volume filtering for product load
    When loading a product from the Manufacturing Hub, a volume of interest can be specified to limit the product data to be loaded.

Enhanced Functionalities

Engineering Requirements Planning

Automatic Re-planning
    Support for Manufacturing Assemblies and Manufacturing Kits is added to the Automatically Re-plan unchanged Engineering Requirements command.
Editing relation attributes
    User attributes defined for the Process Implements Requirement relation can be edited and saved when assigning Engineering Requirements.
Multiple selection of Engineering Requirements
    Support for Manufacturing Assemblies and Manufacturing Kits is added to the Automatically Re-plan unchanged Engineering Requirements command.

Customizing Settings

Process and Resource Definition

Verification
    The options Resource Display and Display Parent Activity's Resource are deleted from Verification tab.
Tree
    Option to display the E5 (Manufacturing Hub) configured name of processes objects in DPM process tree.
Tree
    Perspective and Parallel options to display data with or without a vanishing point, respectively.
Tree (R17 SP2)
    Use Condition, States and Position - Use this option to hide states, conditions and positions in the PPR tree.
Gantt Chart (R17 SP2)
    On Opening Gantt order rows according - When enabled the option Begin times all rows of the same sibling hierarchy inside the Gantt should be sorted according their Begin Times - Children rows will stay beneath their parents. When selected PPR Tree, rows will be sorted as same in the PPR Tree.
State Management (R17 SP2)
    You can alter the state management options to see or hide conditions, positions, or states in the PPR tree.

Manufacturing Hub

Quick access
    New options for Quick Access functionality allow for the configuration of mail options and the shared repository folder.

Equipment Arrangement

New Functionality

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

The Copy/Paste function, together with the Translate function, lets you copy and paste elements in the same design document. You can also copy from one work package and paste into another.

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

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

Plant Layout

New Functionality

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

DMU Navigator

Enhanced Functionalities

Inserting Components
> You can now interrupt the activate terminal node process.

Running the CATDMUUtility Batch Process
> 3D XML format now supported as an output option for CATParts.

Running the CATDMUUtility2D Batch Process
> 3D XML format now supported as an output option for CATDrawings.

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

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

DMU Optimizer

This section identifies what new or improved capabilities have been documented in the Version 5 Release 18 of DMU Optimizer User's Guide.

Enhanced Functionalities

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

More about the Spatial Split option
  Spatial split now uses a new algorithm which improves the quality of the envelope.

DMU Space Analysis

This section identifies what new or improved capabilities have been documented in the Version 5 Release 18 of DMU Space Analysis User's Guide.

Enhanced Functionalities

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

Measure Tools

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

Knowledgeware Capabilities

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

Customizing Settings

XML Settings
> To improve the visualization of clash results in XML files: XML settings have been enriched with two new options (Zoom on interferences (Second XML type) only and Transparent background available in **Tools > Options > Digital Mockup > DMU Space Analysis > DMU - Clash Process**)
> To classify better XML result files, the Aggregate files in folders option has been added in **Tools > Options > Digital Mockup > DMU Space Analysis > DMU - Clash Process**.

First XML Type
> Two new options are available: Transparent background and Aggregate files in folders.

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

DMU Space Analysis Interoperability

Exporting Clash Results Using First XML- Products from ENOVIA V5 database
> A new scenario has been added to show how to export clash results in first XML format.
Exporting Clash Results Using Second XML - Products from ENOVIA V5 database
> A new scenario has been added to show how to export clash results in second XML format.

New Functionalities

Penetration Management Tools
> Are now available within DMU Space Analysis (in User Tasks section).

DMU Composites Review

Welcome to the DMU Composites Review User's Guide!

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

This overview provides the following information:
- DMU Composites Review in a Nutshell
- Before Reading this Guide
- Getting the Most Out of this Guide
- Accessing Sample Documents
- Conventions Used in this Guide

DMU Composites Review in a Nutshell

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

Before Reading this Guide

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

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

Getting the Most Out of this Guide

To get the most out of this guide, we suggest that you start reading the User Tasks section, which deals with handling all the product functions.
The Workbench Description section, which describes the DMU Composites Review workbench, will also certainly prove useful to find your way around the DMU Composites Review workbench.
Navigating in the Split View mode is recommended. This mode offers a framed layout allowing direct access from the table of contents to the information.

Accessing Sample Documents

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

Conventions Used in this Guide

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

Human Activity Analysis

Enhanced Functionality

Display additional NIOSH Information

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

Human Builder

New Functionality

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

Vehicle Occupant Accommodation (license required)

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

Human Data Catalogs (licenses required)

Human Anthropometry Catalog
Human Posture Catalog
Human Preferred Angles Catalog
> Manikin simulation and/or posture-creation can be time-consuming. The human catalogs improve the user experience and reduce simulation creation time.

Enhanced Functionality

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

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

Enhanced Management of Look At behavior
> You can explicitly choose which hand is to be the target of the Look At, the left, right, or either hand.
> That way, you will be able to activate the right option from the start, and will not have to switch back and forth between the IK Mode (or Reach Mode) and the general options page.

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

Manikin Constraints dialog box - Priority
> Specific explanation of the priority settings in the dialog box.

NC Machine Tool Simulation

New Functionalities

Multi-resource Machine Simulation command
 This command enables users to simulate operations on lathe and mill-turning machines.
Generate Simulations in Batch Mode
 This command enables users to select aspects of a simulation to run in batch mode so that the user can save the time required to visualize each simulation.

Enhanced Functionality

The Fault List command works for mill-turn machines
 The Fault List command was called the Collision List command in previous releases. It has been renamed because it now lists faults that are not collisions, such as violation of travel limitations. The Fault List command has been enhanced so that collisions can be detected after Multi-resource Machine Simulations have been run.
Subprograms can be generated in NC Code
 This is handled by the postprocessor and can be taken into account during simulation and collision checking.

NC Manufacturing Infrastructure

Enhanced Functionalities

Support chamfers on T-Slotter tool
> It is now possible to define four additional parameters on a T-slotter tool:
> Top and Bottom Chamfer angles
> Diameters at the top and bottom.

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

Support tool assignments to Turret-Stations for a Manufacturing Program
> Capability to add tools/tool assemblies present in the Resource list to a turret and also view the tools and their stations for a specific turret using a Turret Station editor.

Ability to select/modify a station for a tool on Tool-change activity
> Capability to select or modify a station for a tool/tool assembly on a Tool Change activity.
> This change will be reflected in the Turret Station editor of the corresponding Manufacturing Program.

Capability to associate Video Result CATProduct to a machining operation
> Possibility to associate Video Result CATProduct to an operation:

> ○ by using the Associate Video Result to Machining Operation command after a material removal simulation using the Full Video command or Video from last saved result command

> ○ by using Associate a Video Result to the last MO of the program option in the Generate NC Output Interactively or Generate NC Output in Batch Mode dialog box.

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

Sub-program support for NC code based simulation
> NC code based simulation of ISO file containing subprograms or ISO file split into several smaller files.

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

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

Identical Parts Machining
> New functionalities allow methodology for defining the process for machining an assembly of identical or similar parts.
> Copy Program: extends the Copy Transformation function to Manufacturing Program.
> Merge Program: allows merging machining programs.

Improved management of NURBS output for machining operations
> Capability to choose which operations will be output in APT with NURBS format and which ones will not contain NURBS.

Change feedrate and tool gage without recomputing the toolpath
> Capability to modify feedrates, spindle speeds and tool gage G1 without breaking a toolpath.
> The toolpath is updated with the new values.

Improved Performance of Toolpath Analysis
> Subsequent polygonal rendering for the users are now significantly faster or instantaneous when compared to the first time polygonal rendering.

Reference compensation point for the gauges on Lathe tool assembly
> Capability to define the reference tool compensation point on which the gauges values are defined.
> It provides new output parameters to define in lathe tool change and lathe reference point modification syntaxes (linked to the PP word table). The syntaxes are generated with parameter values that are automatically computed according to the active compensation point.

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

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

NC Machine Tool Builder

New Functionalities

New MillTurn Machine
> This new functionality is available from the New Creation toolbar.

New Spindle
> This new functionality is available from the New Creation toolbar.

New Turret
> This new functionality is available from the New Creation toolbar.

Insert Spindle
> This new functionality is available from the Component Management toolbar.

Insert Turret
> This new functionality is available from the Component Management toolbar.

Remove Spindle
> This new functionality is available from the Component Management toolbar.

Remove Turret
> This new functionality is available from the Component Management toolbar.

Create Mount Point
> This new functionality is available from the Device Attributes toolbar.

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

Prismatic Machining Preparation Assistant

Enhanced Functionalities

Improved Milling Feature Recognition

When running Feature Recognition, the milling areas (with not planar top element) are now taken into account for feature creation (Prismatic Machining Area). Previously, conditional checking was done on top and bottom face. These conditions were replaced by stringent condition checking on bottom face only, whereas conditions are not applied on top face. The top face can be any surface, it does not need to be planar which is parallel to bottom face.

Feature recognition will now provide a Prismatic Machining Area corresponding to a Complex Pocket. The boundary will be defined by a hard boundary only. An imaginary contour at the intersection of Hole and Slot, is automatically defined to close the hard contour.

Prismatic Machining

New Functionalities

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

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

Enhanced Functionalities

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

Support of Helical strategy in Profile Contouring
> Helix tool path style is now possible in Between two Planes mode.

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

Capability to manage machinable features in Manufacturing View
> Capability allows user to sort and filter Machinable Axial features and Machinable Milling features. Also user can now delete unused Machinable Area features.

Lathe Machining

Enhanced Functionalities

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

Single center plunge for narrow groove in Groove Turning
> New Single plunge option for center plunge (suitable for machining a narrow groove).

Choice between linear and angular feedrate units
> Feedrates can now be set in linear or angular units.

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

Enhanced behavior of Linking Macro in Groove Turning
> Optimization of interrupted linking macro by returning to next cutting position instead of previous tool path).

New interrupt option in Groove Turning
> Linking macro can now be interrupted after a number of specified levels.

Optimized feedrates in Grooving by level mode in Groove Turning
> Optimization of feedrates during return motion at the end of a level thanks to a Max approach distance parameter.

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

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

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

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

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

Support approach and retract motion for stock computation
> Capability incorporates Approach/Retract motions in the Stock Computations, leading to improved Stock Update.

Reference compensation point for the gauges on Lathe tool assembly
> User can now specify the reference tool compensation point on which the gauge values are defined.
> It provides new output parameters to define in lathe tool change and lathe reference point modification syntaxes (linked to PP word table). The syntaxes are generated with parameter values that are automatically computed according to the active compensation point.

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

Data Exchange Interfaces

Enhanced Functionalities

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

STEP Import
> Faceted_brep are now imported as V5 solids.

STEP Report File
> Geometric Validation Properties: The estimation of computation error on GVP is available in the report file.
> Assemblies: information on assemblies is available in the conversion summary at import and at export.

STEP Import and Export
> STEP support small scales files.

IGES 3D Export
> V4 models referenced in a CATProduct are exported to .igs files.

DXF/DWG Import
> Proxy graphics are supported.
> Autocad2007 is supported.

Infrastructure User Guide

New Functionality

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

Enhanced Functionalities

Capturing Simple Images
> In non-standard scale mode, you can only capture pixel images.

Creating, Opening, and Saving Documents
> For dialog boxes allowing you to select a file or a directory, the last accessed path is now stored in preference files.

Customizing Fonts for Displaying Geometry Area Texts
> TrueType font versioning: the display of a text for a particular font in a particular version remains unchanged in future Version 5 releases or service packs.

Customizing Print Settings Before Printing Documents
> In non-standard scale mode, only the Rasterization rendering mode is supported.

Managing Document Save
> The Apply Save to editor scope only option now applies to the Save Management command.
> When you resize the Save Management dialog box, the new size is kept.

Running Batches
> The Generic Batch lets you run executable files in batch mode.
> When running the Print Batch in non-standard scale mode, only the Shading rendering mode is supported.

Selecting Objects Using a Filter
> The Tangent Intersection Edges activation mode is now available.

Selecting Objects Using the Search… Command
> The Pre-highlight option lets you activate or deactivate pre-highlighting of elements found by the search.
> Navigation options are now provided to display the search results.

Using the Paste Special… Command
> Ability to edit the link created when pasting a body As Result With Link in the same document.

Customizing

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

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

Managing User-Defined Toolbars
> Toolbar positioning has been modified when using the Restore position capability.

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

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

MULTICAx AD Plug-in

Enhanced Functionalities

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

DELMIA Solutions Version 5 Release 17
Product Enhancement Overview

This document lists both new and enhanced product functionalities for the current release.
Click Product Enhancement Overview to view this document in PDF format. Warning ! please note that this PDF document does not include the pages related to DELMIA Automation.

Click the following links to view the Product Enhancement Overview document from the previous two releases:

- V5R17 Product Enhancement Overview
- V5R16 Product Enhancement Overview

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- Plant Layout

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- MULTICAx Unigraphics Plug-in

MULTICAx ID Plug-in

Enhanced Functionalities

IDI MS12 is now supported.

MULTICAx Pro/Engineer Plug-in

Enhanced Functionalities

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

MULTICAx Pro/ENGINEER P Plug-in (Indirect mode) now supports Pro/ENGINEER Wildfire 3.0 on Windows 32 and 64 bits.

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

MULTICAx SolidWorks Plug-in

Enhanced Functionalities

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

MULTICAx Unigraphics Plug-in

Enhanced Functionalities

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