CATIA V5-6R2018

Added Values (Detailed)
CATIA V5-6R2018 Release Value Synthesis – Key Messages

1. **Global collaborative innovation**
   - Improved 3D Master & Drafting Solutions
   - Improved Usability & Productivity, Support of ISO 2553:2013 standard allowing to define two different systems for welding symbols
   - 2D Layout for 3D Design
   - Callout size and position enhancement
   - Position and orientation view links
   - 3D Functional Tolerancing
   - Copy/Paste as result with link of FTA features
   - Annotation sketch

2. **Lifelike experience**
   - CATIA Imagine & Shape leap forward
   - New capabilities: Manipulation Laws, Matching, Selection

3. **Single PLM platform for IP management**
   - V5-6R2018 and 3DEXPERIENCE R2018x compatibility
   - Enable co-design from interactive session between V5 and 3DEXPERIENCE across the supply chain

4. **Ready to use PLM business processes**
   - Mechanical Domain:
     - Improved Part Design capabilities: Chamfer, Hole, Sketcher, User Pattern
   - New Product: Stamping Die Face Design
   - Design die surfaces in the context of stamping die tooling design process
   - Shape Domain:
     - Improved Generative Shape Design Capabilities: Diablo, Silhouette, Healing, Remove Face

5. **Lower Cost of ownership**
   - Composite Engineering extended capabilities to support composite processes: Limit Contour, Stacking Management, Producability...
   - New Product: Composites Forming Part

Enhanced Products: 21 | Extended Highlights: 91

Manufacturing process: Adaptive Concentric Milling

New Product: Stamping Die Face Design

Accelerates the design of formed Composites parts with scalable validation levels
Portfolios evolution | CATIA V5 & CATIA V5 PLM Express

Summary of targeted highlights

- # Enhanced Products: 21
- # GA External Highlights: 91

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<thead>
<tr>
<th>Domain</th>
<th>Number of Products enhanced per Domain</th>
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<td>IMA - CATIA - Imagine &amp; Shape 2 Product</td>
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Portfolios evolution | CATIA V5 & CATIA V5 PLM Express

► Packaging update

► New CATIA V5 Products
  ▶ TDF - CATIA - STAMPING DIE FACE DESIGN 2 Product – GA (General Availability)
  ▶ CCF - CATIA - COMPOSITES FORMING PART 2 Product – CA (Controlled Availability)

► Product added to CATIA V5 Configuration
  ▶ TDF - CATIA - STAMPING DIE FACE DESIGN 2 Product added to XL2 - CATIA - All-in-One Marketing EAR 2 Configuration
  ▶ CCF - CATIA - COMPOSITES FORMING PART 2 Product added to XL2 - CATIA - All-in-One Marketing EAR 2 Configuration

► Product / Package withdrawn
  ▶ CAT3DX-P - CATIA V5 PLM Express on 3D Experience Extended
Specify the product
2D Master product definition
Drafting – ID1, GDR

Define the product
3D Master product definition
→ 2D Layout for 3D Design – LO1
→ 3D Functional Tolerancing & Annotation – FTA

Product 2D presentation
→ Drafting – ID1, GDR
→ 2D Layout Browser – LO0

Present/Review the product
Product 3D review
→ Power by 3DExperience review
→ DMU Tolerancing review – DT1
→ 3DVIA & CATIA (free) Players
→ 3D Insight – I3D

Conceptual & preliminary design
2D Layout for 3D Design – LO1

Share/Document/Consume…
Capitalise/Exchange/Archive

CATIA 3D Master & Drafting Solutions overview
CATIA 3D Master & Drafting Solutions

- V5-6R2018 – 28 Added Values

- Interactive Drafting
  - 5 Added Values

- 2D Layout for 3D Design
  - 14 Added Values

- 3D Functional Tolerancing & Annotation
  - 9 Added Values
Interactive Drafting

V5-6R2018 Added Values
Interactive Drafting | V5-6R2018 Added Values

▲ Annotation move Undo-Redo log

▲ Additional support of Undo/Redo steps creation during particular annotation manipulations:

▲ Actions on annotation:
  ▶ Moving annotation,
  ▶ Stretching text,
  ▶ Resizing text,
  ▶ Sliding text,
  ▶ Indicating a movable datum.

▲ Actions on leader:
  ▶ Moving leader,
  ▶ Resizing leader attachment,
  ▶ Moving leader anchor point.

▲ Display of the action in “Undo with history” dedicated dialog box.
Interactive Drafting | V5-6R2018 Added Values

► Annotation placement

► Capability to avoid Annotation placement controlled by hot points (default attachment points).
► New dedicated option proposed in following tab-pages:
  ▶ Mechanical/Drafting/Annotation and Dress-Up,
  ▶ Mechanical/3D Functional Tolerancing and Annotations/Annotation,
  ▶ Infrastructure/3D Annotations Infrastructure/Annotation.
Interactive Drafting | V5-6R2018 Added Values

- Table auto-flip

  - Support of Table mirroring for automatic Table text content flip:
    - New Table “Mirroring” property in “Text” tab-page,
    - Default property behavior managed by standard style new entry
    - Styles > Table > Default > Mirroring.
Interactive Drafting | V5-6R2018 Added Values

Scaling of arrows

- Support of Arrow extremities scaling:
  - Arrow extremities can now be scaled according to View scale,
  - New Arrow “Apply scale” property,
  - Default property behavior managed by standard style new entry.
    - Styles > Arrow > Default > Apply Scale On Extremities.
Support of ISO 2553:2013 (welding symbols) - 1/2

- Support of ISO 2553:2013 standard allowing to define two different systems for welding symbols:
  - System A,
  - System B.
- New standard entry (with illustration) to specify the system to display welding symbols.
- Full support of standard switch.
Interactive Drafting | V5-6R2018 Added Values

Support of ISO 2553:2013 (welding symbols) - 2/2

- Support of ISO 2553:2013 standard defining new presentation of welding symbols:
  - New Weld size and Weld length text editors,
  - New welding symbols representations,
  - Possible second pair of elementary/supplementary welding symbols,
  - Text inside Plug welding symbol,
  - Text upper elementary/supplementary/finishing welding symbols,
  - New staggered intermittent welding symbol representation,
  - Engineering symbols in welding symbol text zones.

- New UI for easy and quick definition of welding symbols:
  - Dynamic definition inside a new panel
2D Layout for 3D Design

V5-6R2018 Added Values
Annotation move Undo-Redo log

- Additional support of Undo/Redo steps creation during particular annotation manipulations:
  - Actions on annotation:
    - Moving annotation,
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2D Layout for 3D Design | V5-6R2018 Added Values

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- New UI for easy and quick definition of welding symbols:
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3DTags integration

2D Layout new support of 3DTag feature:

- 2D Layout visualization filters:
  - Ability to select a 3DTag,
  - Management of 3DTag visibility in 2D Layout view background (visibility, low intensify, pick properties).

- 2D Layout view generation in a drawing:
  - New dedicated Generative View Style to support 3DTag generation from 2D Layout to a drawing,
  - Similar 3DTag representation as in 2D Layout view background when the 2D Layout view is generated in a drawing.
Rendering style per view

- Homogeneous customize rendering style panel with direct apply (no more Apply button)
- New visual feedback to warn User that a rendering style defined on the view cannot be visualized (view feature mask).
Callout size and position

- Better callout size default computation: independent from the resulting View support plane axis system.

- Better callout default position:
  - New offset considered from initial callout position in case of non-perpendicular callout, initialized according to the resulting View bounding box,
  - Dedicated manipulators to modify default offset proposal:
    - Translation mode:
      - Callout extremities position modification along callout direction,
      - Shift key to snap a callout extremity position on the grid of its view,
      - Ctrl key to move a callout extremity and the second callout extremity in the opposite direction.
    - Free manipulation mode:
      - Callout extremities position modification along the callout direction, and callout offset modification,
      - Callout position modification: overall callout translation in a direction perpendicular to the callout
      - Shift key to snap a callout extremity position or callout position on the grid of its view.
  - New dedicated option to control automatic default positioning of non-perpendicular callout.
2D Layout for 3D Design | V5-6R2018 Added Values

- View from planar face reference
  - Creation of a view with associative behavior to a planar face (View from reference command),
  - Set an existing view with associative behavior to a planar face (change view support command),
  - Dedicated “Reference” editor to specify and change the planar face element,
  - Reference element canonicity check and User status provided if reference is no longer planar.
2D Layout for 3D Design | V5-6R2018 Added Values

▶ View from profile

▶ Auxiliary and Section Views from Profile creation enhancement,
▶ Selection of existing profile to initialize the profile of the view,
▶ New entries in the dedicated tools palette:
  ▶ View type of the view (Auxiliary, Section View or Section Cut),
  ▶ Profile type of the view (Offset or Aligned),
  ▶ Creation mode:
    ▶ Sketch mode,
    ▶ Selection mode: select an existing profile which belong to the reference view (activate view).
▶ Automatic recognition of profile type when profile selection.
▶ Capability to memorize last configuration (View type, Profile type, Creation mode) for next view creations.
Position and orientation view links

- Full alignment and orientation management of a view from reference according to its reference view after creation,
- New persistent position link to manage alignment along or perpendicular to folding direction,
- New persistent angle applied between the frame around the view background and the horizontal direction when orientation required,
- Positioning link management after view creation thanks to new view contextual menu entries:
  - Create a Positioning link,
  - Modify an existing Positioning link,
  - Replace Reference View,
  - Delete a Positioning link.
2D Layout for 3D Design | V5-6R2018 Added Values

**View scaling**

- Initialization of view from reference scale, regards to active view scale:
  - Automatic scale propagation according to active view,
  - Dedicated option to control default scale propagation for created view.
2D Layout for 3D Design | V5-6R2018 Added V5

▶ Clipping outline edition

▶ Additional capabilities during view profile edition:
  ▶ Visualization of clipping profile outline during modification,
  ▶ Removal of unwanted control points of polygonal profile,
  ▶ Addition of new control points to polygonal profile.
2D Layout for 3D Design | V5-6R2018 Added Values

► 3D Grid association

► Associate in one shot sub 2D Working Support of 3D Grid for a multiple 2D Layout views selection,
► Capability to set the global 3D Grid as a property for multiple 2D Layout views selection,
► Automatic distribution of compatible sub 2D Working Support of 3D Grid for each 2D Layout view of multiple selections,
► Memorization of last assigned 2D Working Support of 3D Grid to 2D Layout view when display property is de-activated.
3D Functional Tolerancing & Annotation
V5-6R2018 Added Values
Semantic dimension between 2 geometrical elements of same canonicity (prismatic and complex cases)

The Tolerancing Advisor proposes Linear Size and Oriented Linear dimension creation between two prismatic or complex faces with constant distance:

- The faces must have opposite material senses for the Linear Size creation.
Semantic support of ISO 1101:2012 Collection plane, Intersection plane and Orientation plane features (1/3)

- **Collection plane** is proposed for “Profile any line” and “Profile any surface” tolerances when applied in common zone to several features:
  - New “All around” area in the “Toleranced Feature” tab page.
  - The validity of the orientation plane definition is managed each time the toleranced geometry or the used datum feature is modified.
  - When a collection plane is defined, the all around symbol is added to the geometrical tolerance.
Semantic support of ISO 1101:2012 Collection plane, Intersection plane and Orientation plane features (2/3)

**Intersection plane** is proposed for the “Toleranced Lines Definition”:

- The validity of the intersection plane definition is managed each time the toleranced geometry or the used datum feature is modified.
- When the user begins by choosing the symbol, only the datum features that make sense for the tolerance he’s defining are proposed.
- When the user begins by choosing the datum feature, only the symbols that make sense for the tolerance he’s defining are proposed.
Semantic support of ISO 1101:2012 Collection plane, Intersection plane and Orientation plane features (3/3)

- **Orientation plane** is proposed for the “Two Parallel Planes Tolerance Zone” definition:
  - The validity of the orientation plane definition is managed each time the tolerated geometry or the used datum feature is modified.
  - The angle value field in the panel is selectable only when using . It automatically creates a Constructed Geometry and a framed dimension.
Numerical format management of datum target

- The default numerical display parameters for datum target creation are defined in the "Tolerances" tab page of the FTA settings.
- The numerical display parameters of each annotation can be modified by Properties.
Customization of size general tolerances

- The size general tolerances are defined in the dedicated excel sheet (`win_b64\resources\graphic\CATTPSGeneralTolerance.xlsx`).
- The Administrator is able to customize them by:
  - Adding new tolerance classes.
  - Modifying the tolerance values of existing ones.

Added tolerance class:

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<th>Customized Class-1</th>
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Support of ISO 2553:2013 (welding symbols) - 1/2

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- Full support of standard switch.
Support of ISO 2553:2013 (welding symbols) - 2/2

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- New UI for easy and quick definition of welding symbols:
  - Dynamic definition inside a new panel
Annotation placement

- Capability to avoid Annotation placement controlled by hot points (default attachment points).
- New dedicated option proposed in following tab-pages:
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  - Mechanical/3D Functional Tolerancing and Annotations/Annotation,
  - Infrastructure/3D Annotations Infrastructure/Annotation.
Annotation sketch

- Add sketches to annotations, so as to make design intent crisper.
- Creation from any view:
  - Dedicated layout sheet/view is created & edited.
  - Annotation sketch visibility managed along with the visibility of its view for display capture.
- Benefit from layout dress-up features:
  - Dedicated contextual workbench.

2D geometry sketching:

- Sketcher-like User Experience.
- 2D geometry not mixed-up with actual 3D geometry.
Copy/Paste as result with link of FTA features

- A FTA annotation can be copied-pasted as result annotation with the link to the original one:
  - The result and original annotations can be in the same part or in two different parts.
  - The result annotation can not be edited. But its graphical representation can be modified using the Properties panel and manipulators.
  - It has the dedicated mask on the icon in the specification tree for synchronized 🔄 / unsynchronized ⚫ status.
  - The changes made on the original annotation are propagated to the result annotation on synchronization and update.
CAA API enhancements (C++ and Automation)

➤ Openness

➤ C++

➤ CATITPSManageAssociativity : CreateAssociativePosition, CreateAssociativeOrientation, CreateLeaderAssociativePosition, CreateLeaderAssociativeOrientation, ...

➤ CATITPSGeometry : SetRepresentedTTRS

➤ Automation

➤ CATIAAnnotationFactory2 : CreateGDT, CreateCoordDimension

➤ CATIAAnnotation2 : Coordinatedimension

➤ CATIANonSemanticGDT : Get2dAnnot

➤ CATIACoordDim : Get2dAnnot
Mechanical Domain

V5-6R2018 Added Values
Feature Recognition

V5-6R2018 Added Values
Features Recognition | V5-6R2018 Added Values

**Defeature**

- Remove logos from existing parts using automatic defeating feature.
Part Design | V5-6R2018 Added Values

**Sketcher**

- Curvilinear constraint on periodic curve.
  - Improve feedback on the side of the periodic curves that is constrained using curvilinear constraint
  
New « toggle » button in the contextual menu to swap side
Part Design | V5-6R2018 Added Values

▶ Sketcher

▶ Boolean Operations.

▷ Propose boolean operations of closed profiles selected against the other profiles from the sketch
Part Design | V5-6R2018 Added Values

▶ Sketcher

▶ Line normal to curve constraint.

▶ When creating perpendicular constraint between a line and a curve, there are no construction elements created anymore. Only one constraint is created between the curve and the line.
Part Design | V5-6R2018 Added Values

▶ Chamfer

▶ Chamfer with hold curve.

▶ Two new modes available:

▶ Chamfer with hold curve and length

▶ Chamfer with hold curve and angle

▶ These modes allow to match exactly a « target » limiting curve on one side of the chamfer. This hold curve needs to share an adjacent face with the edge to chamfer.
Part Design | V5-6R2018 Added Values

Hole

- Counterdrilled hole with countersunk diameter.
  - New mode for counterdrilled hole defining a specific hole combining counterdrilled shape with counterbored.
  - The new parameters are included into standard files for this type of holes
User Pattern

User pattern associative to Geometrical Set.

- Changing the number of axis systems in a geometrical set impacts user patterns relying on this geometrical set. This way, it is possible to create a user pattern with a variable number of instances depending on the content of a geometrical set.

Axis can be created using Repetition command in Generative Wireframe & Surface, or through knowledge script.
Design die surfaces in the context of stamping die tooling design process

Key Values:
- Early evaluation of stamped part KPIs
- High Productivity with specialized wizard & features
- Easy to learn process oriented application (no need to be a CATIA Surface expert)
- Method planning definition & evaluation based on part geometry
- Conceptual die surface design to enable quick validation based on simulation
- High quality Surface design (from GSD)
- Reduce cost of design change (highly automated design change management)

Key Functions:
- Stamping Direction evaluation
- Process definition
- Geometry generator for each process operation
- Automated Hole Filling
- Addenda Surface concept & detailed design
- Trim line
- Pivot flange
- Binder Design (command from core apps)
- Fillet Radius management
- Key GSD features

Competition:
- NX Stamping die design
- Omnicad Die Surface Design
- Delcam Powershape Pro
- Autoform process designer (stand alone)
Stamping Die Face Design | V5-6R2018 Added Values

Key Added value

What

Delivery of a new workbench, dedicated to the Stamping Die Industry, including new surface design and analysis tools, as well as a conceptual process definition command.
Stamping Die Face Design | V5-6R2018 Added Values

- Reduce lead time, cost and increase quality
  - Up to 50% gain in geometries design time
    - Very productive from concept to detailed design of additional geometries (draw direction analysis, addenda, unfold trim lines…)
  - Quick ramp-up
    - Fully compatible solution with CATIA Generative Shape Design - GSD, easy to learn
  - Reduce cost of design changes
    - Full associativity with design part
  - Proven high quality surface result (CATIA Modeler)
    - Model can be directly used for CAM
  - Stamping Process engineering driven tool
    - CATIA from early cost estimation to design for manufacturing
"Stamping Die Designer" process workflow

0. **Sheet metal part**
   - Start with a Stamped sheet metal design
   - Mechanical & Shape Designer (MES)

1. **Sheet metal checkers**
   - Geometric & Formability analysis
   - Formability
   - Geometry
   - Cost

2. **Conceptual process & Die additional geometries**
   - Operations
   - Addendum surfaces & trim lines

3. **Forming simulation**
   - Compute spring back

4. **Compensate spring back**
   - Apply inverse spring back deformation to the tooling part

5. **Stamping die Tool Structure**
   - Generate tool assembly structure with standard components & templates

6. **Validate the Tooling press line**
   - kinematic & clash validation
   - Structural analysis

7. **Production try out**
   - produce prototype parts
   - Reverse Shape Optimizer (VSO)

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**Roles and Partners**

- Sheetmetal designer
- Tooling face designer
- Simulation expert
- Tooling structure designer

**Cost**

- TG1
- SIMULIA
- MES
- VSO
“Stamping Die Designer” targeted process workflow

- Facilitate design for manufacturing collaboration with early process engineering
- Design 50% faster
- High quality surfaces for manufacturing
- Specialized & global deformation features
- Reduce cost of design change from part to stamping tool structure
Define a conceptual stamping process and generate associated geometries.

0. **Sheet metal surface**
   Start with a Stamped sheet metal design.

1. **Geometry Check**
   Draft, Depth and Cut analysis.

2. **Blank estimation**
   Unfold flanges.

3. **Fill Holes**
   Automated & manual hole filling.

4. **Binder**
   Design binder with embedded surface tools.

5. **Clean Contour**
   Enhanced Untrim & Extrapolate capabilities.

6. **Addenda Design**
   Profile based wizard.

7. **Trim lines**
   Generate & check trim lines.

8. **Compensate spring back**
   Pivot Flange, Fillet reduction.
Mold Tooling Design | V5-6R2018 Added Values

▲ Update content of Meusburger supplier

▲ What
  ▶ Update Meusburger components and Moldbase delivered with Mold and Tooling Design workbench.

▲ Meusburger Content Update:
  ▶ 1 new Moldbase F-Vertical Normal with OverL and OverW
  ▶ 118 new references of component
  ▶ 172 components updated
Shape Domain

V5-6R2018 Added Values
Automotive Body In white Templates
V5-6R2018 Added Values
Automotive Body In white Templates | V5-6R2018 Added Values

- **Diabolo with faces to keep**
  - Only selected faces are modified by diabolo creation.
  - Other faces are kept untouched by the operation
Generative Shape Design
V5-6R2018 Added Values
Generative Shape Design | V5-6R2018 Added Values

► Plane - Planes Between

► New mode to create a plane between two elements.

► Inputs are:

► Two geometrical elements
► One ratio between the elements
Generative Shape Design | V5-6R2018 Added Values

Points Repetition

Repetitions of Axis with specified direction

- Creation of axis systems within « repetition » command with a specified fixed direction
- Inputs are:
  - Two geometrical elements
  - One ratio between the elements
Generative Shape Design | V5-6R2018 Added Values

- **Extrapolate**

- **Option “Assemble Result” kept as default**
  - Option is kept as a default value at part level when creating multiple Extrapolate features.
Generative Shape Design | V5-6R2018 Added Values

Porcupine curvature analysis

Optimal screen display using variable auto scaling

- Allows to scale automatically comb spikes edge per edge
Generative Shape Design | V5-6R2018 Added Values

**Silhouette**

- Silhouette can now select multiple inputs:
  - Surfaces
  - Volumes
  - Part design bodies

- With multiple inputs, « Inside » traces are not available
  (see result filters option)
**Generative Shape Design | V5-6R2018 Added Values**

**Healing**

- Specify either frozen or free elements in Healing definition.
  - Easier specification of free elements that might be modified when running healing feature update.

**Details:**
- Default merging distance value is read from setting path at Tools->Standards->3DModelingDefaultValues->Transform->Healing->Merging distance
- Freeze check buttons, freeze plane elements and freeze canonic elements are moved to front tab.
- Under freeze tab, two radio buttons are added for freeze and unfreeze options selection.
Generative Shape Design | V5-6R2018 Added Values

▲ Remove Face

▲ Remove Edge is integrated in the « Remove Face » command

▲ Remove edges in a surface feature, with automatic extrapolation of adjacent faces to fill the gap.
FreeStyle | V5-6R2018 Added Values

**Unified Patch**

**Multiple improvements**

- Edit U, V Orders & Dimensions labels now offer contextual menu when selection is finished
- Preserve selection when reference support is modified.
- Options "3-Point Patch" and "4-Point Patch" now support manipulators at selection points
- Option “4-point patch” now supports ability to select reference support.
FreeStyle | V5-6R2018 Added Values

- **Matching Constraint**

- **Match 2 Edges**
  - New capability to match a single source to 2 target edges
    - Improved user workflow

2 selected Target edges

2 selected Target edges with Limits
FreeStyle | V5-6R2018 Added Values

- Matching Constraint

- UI Improvements
  - Improved graphical feedback when inverting Source & Target
FreeStyle | V5-6R2018 Added Values

- Control Points

- Diffusion Law

  - New Me/Preferences option “Delay Law Application”
    - Checked: Diffusion Law is applied immediately to selected points/rows
    - Unchecked: Diffusion law is applied only after the change of law, with the last move taken into consideration while applying the current law.

[Diagram showing diffusion applied immediately and after law change]
**FreeStyle | V5-6R2018 Added Values**

- **Styling Fillet**
  - **Approximation**
    - Improved default settings
      - Max order = 6
      - Independent Approximation = ON
    - Improves user workflow to avoid additional clicks for more standard values
  - Improved control point distribution
FreeStyle | V5-6R2018 Added Values

- Porcupine curvature analysis
  - Optimal screen display using variable auto scaling
    - Allows to scale automatically comb spikes edge per edge

New option

Global auto scaling mode

Variable auto scaling mode
FreeStyle | V5-6R2018 Added Values

Parallel Command

- Native CATIA ICEM Shape Design capability **PCS – Parallel Command Session** added

  - The **Parallel Command Session** command provides a way to launch several commands at the same time.
    - Avoids selection issues where only one command can handle the selection.
    - Improves user workflow

The following shortcut keys are also supported:

- F10: Hides/shows Parallel Command Session dialog box
- F11: Switches to next command
- F12: Enables only focus command’s manipulators
FreeStyle | V5-6R2018 Added Values

▶ Display Lists

▶ Native CATIA ICEM Shape Design capability Display Lists added

▶ The Display Lists command offers the capability to manage the visibility of objects of the scene without the need to interact with the Specification tree.

▶ Improves user workflow

▶ Command & attributes renamed from “Display Sets” to “Display Lists” to support imported data from ICEM Surf.
CATIA Icem
V5-6R2018 Added Values
CATIA Icem | V5-6R2018 Added Values

- Unified Patch
- New command
  - Native Freestyle Shape Design command “Unified Patch” added to ICEM Shape Design
CATIA Icem | V5-6R2018 Added Values

- Matching Constraint
  - Match 2 Edges
    - New capability to match a single source to 2 target edges
    - Improved user workflow

2 selected Target edges

2 selected Target edges with Limits
CATIA Icem | V5-6R2018 Added Values

- Matching Constraint
  - UI Improvements
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CATIA Icem | V5-6R2018 Added Values

► Styling Fillet

► Approximation

▷ Improved default settings
  ► Max order = 6
  ► Independent Approximation = ON

▷ Improves user workflow to avoid additional clicks for more standard values

Improved control point distribution
CATIA Icem | V5-6R2018 Added Values

- Patch from Curves
  - Symmetry Plane
    - New ability to select a Plane as support to simulate a symmetry plane
**Porcupine curvature analysis**

- Optimal screen display using variable auto scaling
  - Allows to scale automatically comb spikes edge per edge

New option

Global auto scaling mode

Variable auto scaling mode
CATIA Icem | V5-6R2018 Added Values

- **Parallel command**
  - **3D Curve**
    - 3D Curve is now supported in Parallel Command Sessions (PCS)
Replace behavior

Aggregated Inverse

Replace behavior in case of inversion of direction for CATIA ICEM features is simplified with respect to complexity of user spec tree view by using local inverse.

V5-6R2017 behavior

V5-6R2018 behavior
**Ruler**

**Workflow Improvements**

- The Ruler manipulator stores its current position.
  - On exit and restart of the Ruler command the manipulator will be displayed at the previously stored position.

Activate command and attach ruler

Ruler retains its original position

De-Activate command

Rotate model and Re-Activate command
Imagine & Shape

V5-6R2018 Added Values
Imagine & Shape | V5-6R2018 Added Values

New functionalities

Manipulation Laws

- Provide a new widget in the modification tool to propagate the deformation to direct neighbouring
- Customize the law thanks to an immersive widget
- Set the propagation rank in the same widget
Imagine & Shape | V5-6R2018 Added Values

► New functionalities

► Matching Constraints
  ▶ Match a SubD surface on any kind of surface or curve
  ▶ Tune the connexion continuity, the propagation, …
  ▶ Modify surfaces and see the match recomputed as a constraint

► Face selection feedbacks
  ▶ Better feedback on faces for a better ease of use
Composites Forming Part
(Controlled Availability)
V5-6R2018 Added Values
CATIA Portfolio | Composites Forming Part

**Item ID**

CATIA – COMPOSITES FORMING PART 2 Product (CCF)

**Item Type**

Product

**Prerequisite (s)**

CPE OR CPM OR CPD

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**Item Description**

Accelerates the design of formed Composites parts with scalable validation levels

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**Key Values:**

- Accelerate the design of composites parts by validating the forming manufacturing process
- Usable by designers new to composites
- Support typical processes (matched die & diaphragm) and typical materials (directional & general)
- Provide a scalable simulation, quick in real time, detailed within hours, with comparison capabilities
- Unmatched integration with Structural Analysis

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**Key Functions:**

- Quick forming simulation to verify producibility in principle
- Detailed simulation to optimize forming process using blank holders, punches, grippers and membranes

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**Competition**

- NX Stamping die design
- Omnicad Die Surface Design
- Delcam Powershape Pro
- Autoform process designer (stand alone)
Composites Forming Part (CCF)

Accelerates the design of formed Composites parts with scalable validation

- Accelerates the design of Composites parts
  - By validating the forming manufacturing process
  - Editable associative parts with GSD tools
- Usable by designers new to composites
  - Fully integrated in CATIA
  - Extremely efficient workflow
- Supports typical processes and materials
  - Matched Die & Diaphragm
  - Directional & General materials
- Provides Scaled Quick and Detailed Validations
  - Quick in real time, Detailed within hours
  - Comparison between results
- Unmatched Integration with Structural Analysis
Composites Design
Composites Engineering
Composites Design for Manufacturing
V5-6R2018 Added Values

3DEXPERIENCE®
Composites | V5-6R2018 Added Values

- **Limit Contour** - Improved ease of use and efficiency in both creation and edition

- Modify orientation of input curves
  - Ability to invert and store the orientation of input curves in each Limit contour feature.

- Click: To invert curve orientation
**Composites | V5-6R2018 Added Values**

► **Limit Contour** - Improved ease of use and efficiency in both creation and edition

► **Create user defined assemble curve**

▷ User can create their own assemble curve or edit geometry of existing assemble curve by inverting its curves orientation or reordering its input curves.

![Diagram of Limit Contour and create user defined assemble curve](image-url)
Composites | V5-6R2018 Added Values

**Limit Contour** - Improved ease of use and efficiency in both creation and edition

- Ability to modify relimiting side of user assemble curves
  - User definition of mater side of assemble curves is to be stored in each Limit contour feature.
  - Allows persistent and stable solution at update.
  - Existing data model is enhanced with this new inputs

![Limit Contour Interface](image)
Composites | V5-6R2018 Added Values

- **Limit Contour** - Improved ease of use and efficiency in both creation and edition

- User defined matter area
  - User can create their own matter area by creating an assemble curve with valid matter side.
  - Now user can create various pattern of matter area.
Composites | V5-6R2018 Added Values

- **Limit Contour** - Improved ease of use and efficiency in both creation and edition
- **Gap Fill / Extrapolation Mode**
  - Assemble curve can be defined by joining Relimiting curves in below two ways
  - **Gap Fill Join Type** (Default option)
  - **Extrapolation Join Type**

![Diagram](https://via.placeholder.com/150)

- **Relimiting curves** (Input)
- **User Assemble curve** (Output)
Composites | V5-6R2018 Added Values

- **Limit Contour** - Improved ease of use and efficiency in both creation and edition

  - Upgrade legacy Limit contour
    - Upgrade has been integrated for legacy Limit contour.
Composites | V5-6R2018 Added Values

- **Limit Contour** - Improved ease of use and efficiency in both creation and edition

- **Additional functionalities**
  - New staggering manipulator for better UI feedback
  - Ability to select same curve again for assemble curve creation

-JPEG-
Stacking Management

- New user interface

- Row Filters management is now in a separate Dialog panel

- Table is located now in a 2D viewer as done in Virtual Stacking Management

- Entity 3D Preview is now in a separate Dialog panel

- Display Columns has been modified

- New Review functionality

- New Check Contours functionality
Composites Engineering | V5-6R2018 Added Values

- **Stacking Management**

  - **Review Mode**
    - New Attribute: Part Number
    - Ability to Filter & multi-Select
    - Add attribute to Export file
Composites Engineering | V5-6R2018 Added Values

- Custom Grid Ramp

- Improvements
  - Direct selection of curves.
  - Multi selection of curves.
  - Reorder of curves.
  - Name of origin curve.
  - Warning panel is shown if bottom ramp support is not selected.
**Sub Staggering**

**Improvements**

- Structural Reference Element’s (SRE) dialog box has been fully reviewed to provide a simpler and more efficient GUI
  - Slope parameters displayed only once when same for both sides
  - Sub-staggering accessible via drop-down list (instead of tab pages)
  - Slope given by “angle” removed

- Grid panel now takes advantage of new 3D preview of ramp.
  - Textual description of slope is displayed for selected Structural Reference Element (SRE) in panel.
  - Current Reference Element can be selected using simple click on 3D representation of ramp
  - Staggering of a Reference Element can be selected using double click on 3D representation of ramp
Multi Parallel Command

Improvements

- Curve Multi-Selection in list and in 3D
  - Multi-selection of curves list in Multi Parallel Definition Dialog, in Specification tree and in 3D

- Multi-edition of offset value
  - All selected Curves having same offset value but different than Global offset can have a common offset value changed in one entry
Composites Design | V5-6R2018 Added Values

经营管理 for Hand Layup

Improvements

Two new producibility simulations are added

- **Geodesic**: Extends “principal” geodesic lines from the Seed Point to the ply boundary in the initial directions of warp and weft yarns. This uniquely constrains the fabric bounded by the principal fibres. If draping continues beyond this region, geodesic fibres are extended as required in warp and weft directions from points closest to the principal fibres until the surface is covered.

- **Energy**: The Energy (Frictionless) algorithm models the case of no friction between fabric and mould and allows the fabric to move such as to minimise the overall shear strain energy in the material. This yields a unique solution for a particular Seed Point (which should instead be considered as a Reference Point) and initial warp direction and indicates idealised fiber paths that minimise overall ply deformation.
Composites Design | V5-6R2018 Added Values

➡ Producing for Hand Layup

➡ Improvements

➡ Smooth Regions

➡ A new field available for propagation types that implement CATICciFiberMeshOperator2::IsSeedAreasSmoothSelectionNeeded

➡ Allows for the selection of either:

➡ A closed curve lying on the ply surface.

➡ A surface which intersects the ply surface.
Composites Design for Manufacturing | V5-6R2018 Added Values

► Producibility for Hand Layup

► Improvements
  ► Animate Producibility
    ► A new option, animate producibility results available in the Results Tab if the propagation type supports the interface CATICoProdAnimation or CATICoProdResultError.
Composites Design for Manufacturing | V5-6R2018 Added Values

- Producibility for Hand Layup

- Improvements
  - Edit Producibility UI
    - New option “Show multiple results”
      - Allows displaying simultaneously several Producibility results.
    - New icons depicting the maximum shearing value and the maximum deviation value.
      - “!” is replaced by a yellow point ◆ if the value is between the limit and the maximum angle supported by the material;
      - “!!” is replaced by a Red point ● if the value is bigger than the angle supported by the material.
Composites Design for Manufacturing | V5-6R2018 Added Values

- Splicing plies based on producibility results

- MultiSplice From Producibility
  - New command for the construction of cut-pieces from the fiber mesh propagation simulation
Composites Design for Manufacturing | V5-6R2018 Added Values

► Edit Producibility and Multi-Producibility Transfers categorization

► Producibility

▷ New ability to choose a category in Producibility “geometry Creation” tab

► Multi Producibility edition

▷ New ability to set up default category that will be applied to all new created transfer added.
Manufacturing Domain

V5-6R2018 Added Values
Adaptive concentric milling

What

- Several enhancements in pocketing operation with concentric strategy:
  - Capacity to mix closed and open pocket strategy with a slider (reduce machining time)
  - Specific linking motions to avoid retract and plunge: Stay on bottom
  - Add UI parameters to drive the strategy (Auto, Circular, Dynamic) and the max discretization step

Expected benefit

- Reduce programming time thanks to more flexibility with new parameters & reduce machining time thanks to the capacity to mix strategies
Infrastructure Domain

V5-6R2018 Added Values
Infrastructure Domain | V5-6R2018 Added Values

- Key Added value – GDR
  - Import DXF with new PDELIB7.x
  - user can export with 2 new setting in upper level of Autocad