Cimatron 14
What’s New
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What’s New in Cimatron 14

With over 200 new features and enhancements, Cimatron 14 offers major benefits and more user value to manufacturing shops across their entire range of operations. From Tool design through NC programming, this latest version helps to ensure that our customers are more productive than ever, in their competitive marketplace.

**NC** – New cutting strategies allow faster machining time with the use of cutters such as Barrel, Oval, Lens Tools. Overall reducing Rough machining time by an average of 20%. A new Guided Cleanup process allows controlling the milling direction of each region in Cleanup. Improved 5-axis Tilting toolpath and guiding curves for user controlled tilting.

A large number of new and improved capabilities that boost programming automation for plate machining, such as the Facing procedure, automatic Chamfer Recognition, wider Pocket Recognition capabilities and more functionality of Automated Drill as well as using templates for pocket automation.

**Die** – New integrated incremental analysis options enable the user to accurately simulate the design, and find design flaws before the manufacturing process. Users can see whether they have carrier or material problems, non-desirable punch results or SpringBack, as well as incorrect press force definitions.

New advanced Die design geometry tools such as Addendum faces and Fuzzy Offset were added to extend the application capabilities.

**CAD for Tooling** – Many long requested enhancements in Part, Assembly and Drafting were implemented for faster and easier design. New enhancements to the Mesh kernel of Cimatron with operations such as a new robust selection tool that was introduce for Split/Copy and Add/Remove commands. In addition, Advance 3D Text capability is now available for mesh and solid.

**Mold & Electrode** – New functions and enhancements for faster and safer Mold design such as Hole (Cooling) verification, Hole (Library) recognition and handling of ejectors with different diameters.

**PLM Integration with Windchill** – An integration with PTC Windchill PLM system. This Integration allows Windchill users who work with Cimatron files to perform Windchill operations (such as sync, check out/in, etc.) from within a Cimatron assembly or from the Cimatron browser.
Faster Re-machining Programming

- **Guided Cleanup Application**

Guided cleanup is a new application to better control and manage the re-machining process resulting in faster, safer and more efficient re-machining programing.

This application is best utilized for large components where it is required to approach from different orientations in order to complete the re-machine operation. For example: in large cavities for automotive parts, it is impossible to find a single direction for all Cleanup operations. The guided cleanup process allows the user to define different directions for each cleanup region.

The application consists of two main stages:

- **Re-machine segment table**

  In this tool, the system finds all un-machined regions (segments) for a previously used reference cutter.

  All data is displayed in the graphic area and controlled via a re-machine Segments Table.

  This table is the user interface environment to manage handle and create the various cleanup segments. The segments are an input for a new Guided Cleanup procedure that will be described below.

To better manage the cleanup definition process, the user can hide segments already milled. The user can then set a preferred milling direction for each segment, as well as edit segments.

Numerous options are available for controlling the segment definition to obtain the best results and fastest machining time. For example, the user can Trim, Split and Delete segments.

- **Guided Cleanup Procedure** – A new procedure that uses the segments and direction previously defined. Each segment or group of segments are milled by the direction associated with it, making it a 3+2 Axis procedure.

  The advantage is better control of the machining direction of each region in deep complex cavities.
Enhanced Finishing and Roughing Strategies

Cimatron 14 includes enhanced finishing and roughing strategies to reduce machining time by up to a factor of three, utilizing the new barrel tool. This is achieved due the large curvature of these tools allowing the use of a larger step over, yet keeping the same surface quality.

- **Support Barrel type tools** – Barrel tools is a general name for a family of tools that have variable radii along their profile. Cimatron roughing, finishing procedures now support Barrel, Lens and Oval tools in Rough and Finish procedures as well as 5-axis production.

- **Improve Rough Time** - Roughing machining time can be reduced by up to 40%, with an average of 20% in complex ReRough operations.

5-Axis Tilting

5-axis tilting is used in order to create a toolpath where the holder is collision free with the part, the stock and the fixtures. The 5-Axis Tilting capabilities were significantly improved in this version, utilizing shorter tools for better surface quality and faster machining time.

- **Improve 5-Axis tilting quality** – The 5-axis tilting algorithm was greatly improved. It is now smoother, giving better surface quality with more user control.

- **New ‘Convert to 5-Axis’ procedure** - The user can now modify the ‘convert to 5-axis’ procedure with no need to re-calculate the 3-axis procedure, saving calculation time.

There are two options to define the tilting operation:

- **Automatic** – The system will automatically calculate the 5-axis toolpath as in previous versions.

- **Guided** - The user can use guiding geometries (contours and points) as a reference to direct the tilting method and achieve better control and faster toolpath calculation.

This option is more suitable for simple parts such as pockets and electrodes.
Cimatron 14 includes several significant enhancements for easier, faster and safer plate machining.

**Feature Based Machining** – this new set of capabilities allows users to program plates with many features faster, easier and safer than ever. It includes feature recognition and management tools for holes, pockets and chamfers, use of sequences and pocket templates for greater automation and shop floor standardization.

**The Plate Machining Seat** now includes the new Feature based machining capabilities to simplify the programming, using dedicated automation tools. These new capabilities include: pockets, holes and chamfers powerful recognition and management tools, pocket templates for milling automation including capping of sockets, automated facing procedures and chamfering procedures including detection of imaginary chamfers and holder collision prevention.

- **Robust Pocket recognition**

  Successful recognition of all pockets in a Plate is a preliminary step for automating the process of pocket machining.

  Cimatron 14 includes enhancements and new functionalities that significantly improve the reliability of the Feature Based Machining tool.

  In addition, more pocket types are now successfully recognized and can be used in downstream procedures and templates.

- **Applying NC procedures on Pocket features**

  Recognized Pocket features can now be used as input objects for 2.5-Axis and 3-Axis procedures. The relevant geometry from the Pocket feature is automatically selected by the procedure.

  For example, the ‘Volume Pocket’ procedure takes the pocket’s volume, the ‘Finish’ procedure takes the surfaces and the ‘Profile’ procedure takes the contours and Z values.

An open Pocket with different draft angled walls is recognized by the system

The surfaces associated with selected pocket(s) are selected automatically
- **Templates for Automation of Pocket milling**

  Templates for Pocket is a new method that offers faster, safer and simpler programing for milling multiple pockets.

  In many cases, pocket milling requires similar sequences of NC procedures. These sequences contain for example, roughing, walls finishing, floor finishing, cleanup, chamfering etc.

  When a job requires the milling of multiple pockets, the programming may take a long time and increase the possibility of human errors.

  User can now define and create a Template by selecting any sequence of procedures. Once a Template is defined, the user can apply it on any pocket feature.

  Usage of Templates enables the user to apply a complete milling process to all pocket features in a single operation, saving programing time and preventing user errors.

  Of course, there is a smart option in post processing to sort the operations by tool order for minimizing tool changes.

- **Facing procedure Automation**

  The user can now use a new Facing procedure for easier, faster and safer facing operations.

  Facing is normally the first operation that prepares the initial stock for milling.

  The new Facing procedure has a smart automatic algorithm to support open regions, optimized milling directions, round corners and round connections.

  The new Facing procedure also supports flexible input methods - using Part and Stock geometry or the user's manual input.
■ Chamfering procedure Automation

A new automatic functionality to detect chamfered edges is now available in the Chamfering procedure for faster and safer programing. The automatic functionality detects both “Imaginary” chamfers (sharp edges) as well as “designed” chamfers, based on the user’s request and uses them automatically as input for the chamfering toolpath.

The Chamfers were detected automatically for machining

■ Automatic capping of socket areas in Pocket

Pockets, created by the Pocket Manager, may contain socket areas (e.g. holes, slots) at their bottom floor. Applying automatic milling procedures on those pockets as-is, may result in an inefficient toolpath, which requires the user to apply extra effort in workarounds.

Now, pockets with socket areas at the bottom will be milled correctly using one of the following options:

- Automatically, for the general case, by letting the system cap all sockets in the pocket’s floor
- Manually, for more user control, by capping specific sockets.

Holes in Pocket Floors can be automatically capped as part of feature based machining
**Safe Milling**

- **Enhanced Thread Milling** – A new thread tool can now be defined with the real threading characteristics. Thread tools can now be used for internal and external threading, created by the Automated Drill procedure. The thread milling creates a toolpath based on the real threading motions that updates the remaining stock accordingly. The motions can be viewed in the Navigator and in the Simulator, preventing possible collisions and giving the user better control over the threading process.

- **Multi Stage Spindle** – Users can now define spindles with more than one stage for safer toolpath creation without spindle collision.

- **Finish Quality**

  Finishing procedures now consider the Stock for approach and retract motions, creating safer a toolpath and preventing tool breakage.
Machining Automation

- **New Template Selection Tool**
  
  A new tool that provides the user better control and flexibility in the selection, order and assignment of templates, resulting in more efficient and faster programming time.

  In many cases, the user needs to select and apply a series of templates repeatedly. This may be a tedious process and is prone to user errors.

  Now, the user has a new ‘Template Selection’ tool, which supports the operations: adding and removing templates, setting the order of templates in the series and one click selection for applying the templates series.

  The ability to select multiple templates.

- **Automated Drill Enhancements**

  A list of enhancements and limitation removals for the Automated Drill that were highly requested by Cimatron customers.

  - **Helical Profile** – In the past, the user was able to define the Ramp angle, but not the down step (or the Pitch). The down step had to be calculated by the user. With the new ‘Helical Profile’, it is possible to define both the down step and the ramp angle, eliminating user calculation and possible human errors.

  - **Sequences Hole Breaches** - Sequences that were defined on holes that have breaches, could not be used for holes that don’t have breaches and vice versa. This limitation increased the number of sequences the user had to define and manage.

    This limitation is now removed, reducing the number of required sequences, saving preparation, programming and maintenance time.
- **Manual Sequence** - When loading a sequence manually to a hole defined manually on a point, the system now keeps global anchor points (ST, DP, MD, BT, SB, SH). It eliminates the need to redefine those anchor points again.
General

- **Interface to Makino CSG**
  Cimatron now offers an interface to share data with Makino CSG (Collision Safe Guard) application. The Makino CSG interface is an addition to the current supported interfaces to Vericut and Eureka.

  The interface between Cimatron and Makino CSG enables a real time collision prevention while checking Cimatron’s 3-Axis and 5-Axis program against stock part, fixture and table.

- **Skeleton Curve**
  Creating a mid-curve of any given 2D wireframe. This function is useful for text engraving.

- **Round Corners in Finish 3D Step**
  Finish 3D Step now allows round corners, giving better surface quality, smoother machining and longer tool life.

- **5 Axis milling on mesh**
  Milling a mesh geometry using a new projection 5-Axis NC procedure.

- **Mesh operations available in CAD mode in NC**
  With the ability to apply a NC procedure on Mesh geometry, new Mesh create and edit functionalities are now available for the NC application. See more details in the Part section. The following mesh commands improve the usability of CAD tools for NC applications: Remesh, Volume Text, Split and Copy, Offset and Shell and Mesh diagnostic.
New Simulation and Strip Design capabilities with AutoForm for Die Validation

One of the main challenges faced by die makers is the need to validate the strip design before actually producing the die. A typical die design process consists of several iterations of actual production of the tool or the blank in order to analyze the SpringBack effect, trim optimization and other aspects that might change the desired result. In order to reduce the number of iterations and allow the designer to validate the design before manufacturing the die, Cimatron 14 introduces a new solution for Strip design Simulation. The solution is available through an integration between Cimatron and a new ProgSim Simulation product from AutoForm. The integration supports a three-phase process, which includes a semi-automatic data preparation for AutoForm, running the simulation in the AutoForm environment and displaying the simulation results back in Cimatron for design adjustments.

Data Preparation

- **Guided Data Preparation** – Once a strip was designed in Cimatron and is ready for simulation in AutoForm, a semi-automatic guided tool is available for fast and seamless data sharing between Cimatron and AutoForm.

  Data preparation for Autoform is an important step for executing a successful and accurate simulation. The ‘Export to AutoForm’ function is an interactive guided dialog tool that helps the user in selecting the required information (e.g. Pilot, Trimming, Forming, Cams) for AutoForm integration directly from the Strip. For a safer and faster selection, some entities are automatically selected by the tool and some require manual selection.

- **Kinematic Animation Display** – A new visual tool available to assist the user in preparing the data for AutoForm and making sure that all required objects are indeed selected and defined correctly.
The Integration with AutoForm ProgSim allows the user to load a complete strip design created in Cimatron directly into the AutoForm working environment and run the simulation in one continuous process.

AutoForm ProgSim enables Cimatron users the following functionalities:

- Formability analysis visualization
- Carrier simulation
- Extended material properties
- Define forces per each tool
- Springback compensation
- Trim line optimization
- Collaborate simulation result data with Cimatron

Running the Die Strip simulation in the Autoform environment
Read the Simulation Results and make any required changes in Cimatron

- **Importing data from AutoForm**

  Once the simulation results are available, the user is able to read, manage and display the results in Cimatron. This powerful capability allows Cimatron users to fix issues that were found in the simulation and optimize their Die design in a seamless continuous process.

  The ‘Import to AutoForm’ dialog enables complete control over the entities to be open and displayed in Cimatron. This tool provides better visualization of the different stations and saves time in hiding/showing selections of non-required entities.

  With this tool the user can:
  - Read Simulation results per station
  - Display a Mesh of Springback results
  - Import deviation data for Springback compensation
  - Trim Line optimization to correct punches.
Additional Die Design Capabilities for Faster and Streamlined Design

- **Addendum Faces** – A new capability to define and create addendum faces between the blank faces and the binder in a fast and easy way.

  A predefine list of Cross Section shapes are available for the definition of the Addendum surfaces.

  This capability is mostly required in Transfer and Regular Die design, where large-medium parts are transferred between multiple presses to complete them, such as shells, tube applications, frames and structural components.

- **Fuzzy Offset** – A robust surface offset feature enables the user to easily create an approximate offset surface out of a complex skin surface. The Fuzzy Offset feature is mostly required for Die Casting and Mold use cases.
- **Copy & Mirror of Springback points** – A new option in Springback (and Advance Warping) tool that allows Copy/Mirror of selected points from a SpringBack point table.

- **Correction factor for Forces in “forming” tools** – A new parameter is available in the Preference Editor that controls the ‘Correction Factor’ of FTI forces in a forming operation.

- **Add Forces to Die Quote (report)** – Detailed information about the applied Forces: X, Y, Z, Type, Value is now displayed in the Die Quote report.
New Capabilities for faster and safer Mold Design

- **Hole (Cooling) Verification** – A new tool identifies design errors that may lead to leakage in molds before manufacturing. Leakage may be caused by corrosion in cooling channels that are not far enough from other cooling channels and holes. This new tool allows the user to easily analyse the distance between all hole types (cooling holes, non-cooling holes, and the distance between holes and part faces) to ensure it is above a threshold defined by the user. It helps users to quickly detect design mistakes and fix the design before actually producing the mold. The analysis is much faster than any similar analysis tools we have had before.

- **Hole Recognition** – It is common that mold makers get mold design jobs from sub-contractors or receive maintenance jobs on molds they did not design. In such cases, there is a possibility that some of the holes will not follow the shop floor standards and the mold makers will need to change them in order to fit their drilling standards. This new tool allows the mold maker to quickly and easily analyse the holes used in the design and to ensure all of them adhere to the standards. Different hole libraries may be applied for the analysis.

- **Add Ejectors with different diameters** – The Add Ejector tool in version 14 enables the user to easily and quickly add ejectors with different diameters in a single operation. Users can now add different sized ejectors simply by changing the diameter of the circle used to position them. Varying ejector heights can still be adjusted automatically by the system. This functionality can also be used for other catalog parts.
- **Ejector & Baffle Table Improvements** – The texts created in the Table of Ejectors and Table of Baffles tools can now have prefix and postfix added to them, both on the plate or on the ejector/baffle. In addition, they can now be created as a standard wireframe Curve Text or as a PMI Curve Text (as they were before).

- **Direction Analysis Improvements** – The direction analysis tool now has a new powerful option that allows analyzing the part from 2 pull directions. This allows you to quickly see faces assigned to the core and cavity as well as undercuts. Furthermore, for parts that are hard to position in the mold, you can enter a special positioning mode, where you rotate the part freely and get an instant analysis of the core, cavity and undercuts, helping you to find the right orientation. It is worth mentioning that the direction analysis can now work with object selection as opposed to faces selection before.
Assembly Design

- **BOM Improvements** - Several improvements to BOM capabilities were added to enhance ease of use and efficiency.

  Users can now:
  - Create BOM reports for a specific portion of the assembly, such as a sub-assembly or a folder or a selected group of components or visible parts only, with or without catalog parts.
  - Easily mark assembly components that will require machining.
  - Populate component quantity into the part documents so they can later be presented in part drawings.
  - Better control of ID Numbers by allowing to assign new ID numbers only instead of overwriting ID numbers of deleted parts.
  - A new option to Hide all components except for selected BOM rows
  - See how many objects exist in each part with the new Quantity of Objects column

- **Multi Instance Performance** - significant performance improvement when working with big assemblies (multiple instances). The improvement is mainly visible in ZPR and hide/show operations.

- **Assembly Improvements** - Several improvements to Assembly design capabilities were done to enhance ease of use and efficiency.

  Users can now:
  - Drag a component or external data transfer files from any browser and drop it in the assembly and have it added to the assembly (rather than just opened).
  - Move components to a different sub-assembly using the Rearrange Assembly tool simply by dragging them. Multiple components can now be moved together and they can be moved to different sub-assemblies in the same operation.
  - Use the Cut manager option also in Assembly Copy.
More Assembly Improvements – These are smaller improvements to the assembly

Users can now:

- Add components on cone faces that are located in a higher nesting level (higher level sub-assembly).
- Use Add Folder on selected components and have them assigned to the folder automatically.
- Select and unselect all items in the Cut Manager with a single click.
- Delete a component with greater ease. A feature guide was added to the tool (when used from the menu or toolbars).
- Add a “fresh” part from the catalog when adding parts using Edit Add. This is especially useful when adding ejectors. Existing ejectors often already have features in them that should not appear in the newly added ejectors.
- Avoid creation of duplicate components – when adding a catalog component for which there is a similar single part in the assembly, the system will offer to use the existing part.
- Add a new part or a new sub-assembly directly from the context menu (the old Add New Component command was removed and replaced with Add New Part and Add New Sub-Assembly).
- Lock groups in motion simulation using a pin icon (in addition to the existing functionality of locking using the context menu).
- ECO colors can now be determined in the preferences.
- ECO 3DPDF reports are now generated with a color legend for additional clarity.
- Control visibility of entities in an assembly M-Views (see Drafting section).

Open Setup Wizards at any time – Setup Wizards are used to create new project assemblies for Mold Design, Die and Electrode. They can now be invoked at any time, even when another document is open.
**Curve Text PMI improvements** – Curve Text PMI were introduced in Cimatron 13 to allow users to create texts on parts, as PMI, allowing them to edit them without causing a regeneration of the assembly. More capabilities were added after the official release of Cimatron 13.

Users can now:
- Replace multiple texts throughout the assembly. Users can add generic text such as “Project Number” to all the relevant catalog parts and then replace all instances of text throughout the assembly with the actual project number using the Replace Text tool. This functionality was added in one of the service packs of Cimatron 13.
- Convert a PMI Curve Text to a standard wireframe Curve Text. This is useful when you want to use the text wireframe for operations such as extrude. Note that the text is normally converted to wireframe when the part is imported to NC, Drafting or to another part.
Drafting

- **Drafting Improvements** - Several improvements to Drafting were done to enhance ease of use and efficiency. Users can now:
  
  - Order and move symbols (such as dimensions and tolerances) more easily and quickly.
  - Manually define the origin point for Table of Holes and Label of Holes.
  - Control visibility of entities in an assembly M-View within a part (rather than control the visibility of the entire part). This is controlled in the assembly environment and affects the drafting (when M-Views are used).
  - Enjoy faster creation of section views.
  - Automatically get diameter signs on symmetrical ordinate dimensions.
  - Add a rounded rectangle around dimensions to mark them (this is usually used to mark dimensions that require inspection. Mostly in German companies).
  - Add a symbolic text to multi-views. The text can be saved and reused with multi-view templates.

- **Array of Frames** - Create an array of drawings – usually for each one of the parts in a die sub-assembly. This tool can automate the creation of dozens of drawings all at once saving a lot of time and effort. This kind of drawing is very common in the Die industry where a huge drawing of all the parts is created, printed and hung on the wall where the Die is manufactured.
Part Design Improvements

- **Scale improvements** – Many improvements were done to the Scale tool as well as to the Shrinkage Compensation tool. Users can now:
  - Define an associative center point for the scale that is always at the center of the selected geometry and updates on regeneration.
  - Have easier control of all parameters at the last stage of the feature guide including the Uniform / Non-Uniform parameter that used to reside in the first stage.
  - Keep a list of predefined scale factors (both uniform and non-uniform).
  - View scale factors as percentage.
  - See the scale factor or percentage they have used in the name of the scale feature created on the feature tree.
  - See the last used scale in each part as an attribute in the Cimatron Browser.
  - Use the Shrinkage Compensation tool while the assembly is active (auto activation).

- **Reorder Feature in the Tree** – Feature Reordering in the ‘Feature Tree’ is now supported. The user can select a feature in the Tree and drag & drop the feature in a new location. A red line indicates the position where the feature shall be reordered.

- **Edit Suppressed Feature** – It is now possible to Edit a feature while the feature is suppressed. When confirming the Edit operation a new message is displayed asking the user whether to ‘Unsuppress’ the feature.
- **Pipe - multi selection** – It is now possible to select several input entities such as: wires, edges, curves.

- **Shell - multi lumps** – it is now possible to Shell in a single feature operation multi-lumps objects.

- **Manufacturing Technique Attribute** – ‘Manufacturing Technique’ is now displayed as a separate category (section), allowing to define multiple Manufacturing Techniques attributes.

- ‘Remove & Extend’ enhancement – Better success rate when removing round faces with free edges at the ends of edge sequence.
• **Copy/Move commands – keep topology** – it is now possible to Copy/Move adjacent faces from a solid body and keeping the resulted geometry stitched.

• **Body Integrity Analysis** – A single unified Body Integrity tool to execute all checks in one shot.
  The user has control what checks to execute and what checks to ignore.

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**Robust Selection Capabilities**

• **New Selection Tool** - A new tool of selection allows to select faces and datum as well as facets (mesh) geometry by:
  - **Polygon** – pick points that form a polygon that defines the selection area.
  - **Brush** – a circle with a user defined diameter is dragged and the mark it leaves behind defines the selection area.
  - **Lasso** – freehand draw of the area of selection.
  - **Box** – box selection by dragging from on corner of the box to another. This is the selection method that existed until now.

  • **Smooth + Angle** – this is only applicable for facet selection. A single facet is selected and the selection spreads to neighboring facets as long as the angle between the adjacent facets is lower than a user defined angle.

Using these new selection methods, users can also select “Visible Only” entities as before (select only entities that are seen in the foreground) and unselect by “Shift” key.
Parametric Mesh Design

- **Offset/Shell** – A new option allows to create mesh object(s) (shell or offset) from different types of:
  - Object (solid or Mesh)
  - Faces
  - Facets (including smooth selection)

- **Volumetric Text** – The user can now add 3D text as a volume directly on the model geometry.
  - Engraved – the text is subtracted from the object.
  - Protrusion – the text is added, protruding from the object.
  - Sunken Protrusion – a rectangle around the text is subtracted from the object and the text protrudes in it.
  - The text can also be created as a separate object.
  - Text can be created both on mesh and solid objects.
- **Split/Copy** – The new selection tools are available for Split/Copy and Remove commands. These capabilities can help to fix or improve mesh objects or create new object from existing one.

- **Re-mesh** – A new capability to re-mesh an existing mesh is now available in order to achieve more equilateral triangulation. This can help to fix or improve mesh objects and also makes them ready to be used in FEA calculations.
- **Copy to Mesh – Pick through Points** – A new option that allows pre-selection of geometry points and creation of mesh facets that go through these points. This new option enables selection of geometry points after creation of a mesh body.

- **Mesh diagnostic Knife Edges** – New diagnostic criteria for detecting Knife Edges is now available in the Mesh diagnostic tool.

- **Convert point cloud to mesh** – In version 14, users can read in point cloud objects (scanned model) and convert them to a mesh object. The user has the option to manually set the parameters of the resulting mesh object. This is mostly used for when using scanned data for comparing and running deviation analysis with the design model targeting design deformation.
- **Volumetric Drive** - This new tool creates the volume formed by a 3D object (NC cutter for example) which is driven along a 3D path (3-axes).

  The selected object may be either mesh or solid. The created object is a mesh object.

- **Parameterization to mending Mesh feature** - Add/Remove, Split/Copy, Close Mesh Gaps and Local Decimate features are now created as parametric and editable features in the Tree.

**Data Interface**

- **STEP AP 242 support** – This version supports Export and Import of STEP 242 file format. The support is limited to geometry only.

**New 'Write Step as 242' option**
- **STEP AP 242 support Import of PMI entities** — Import of STEP 242 files support also PMI data.

- **Import Mesh – Separate Shell** - This new setting, in the 'Import' dialog, improves the resulted quality and integrity of imported mesh files that contains overlap and/or intersect lumps.

- **General Enhancements** - The following general enhancements were made:
  - Direct import of SolidWorks files with PMI data (DimXpert type only)
  - Export Mesh and PMI data to JT file format.
  - InterOp 2017 – update to latest CAD releases.
  - Add a new preference parameter that Open DI dialog when importing by Drag & Drop.
  - Resize Import/Export dialog window is now possible
Electrode

- **Electrode Improvements** - Several improvements to Electrode were done to enhance ease of use and efficiency.

  Users can now:
  
  - Rotate Cylindrical blanks – this is useful when you want your electrode UCS or holder to be rotated.
  - Create a chamfer or round on the step only and not on the entire blank – this prevents unnecessary machining on the Base of the blank.
  - Get notified when an empty electrode is about to be created by mistake.

*Round marking, done only on the step of the electrode’s base*
User Experience

- **Customize ZPR Control** - Users can now determine what mouse buttons and/or keyboard keys will be used for the Zoom, Pan and Rotate operations. This will make transition from other software easier as users can configure the system to work the way they are used to.

- **Graphic Performance** - The graphic engine has been improved to enable better performance and memory-usage working with multi instances in ZPR and performing hide/show operations, making full use of the computer’s GPU (video card). The shading technology is also improved, using Open GL advanced version.

- **UX Improvements** - Several improvements to the user interface were done to enhance ease of use and efficiency. Users can now:
  - Enjoy better visibility of file status notification icons with animation added to them.
  - Have a notification as soon as a file has changed and requires saving (including information on what kind of change occurred).
  - Easier work with keyboard customization with a new list of all existing customized keys, as well as the ability to attach keyboard shortcuts to all the buttons on the lower row of the feature guide (OK, APPLY, CANCEL, etc.)
  - Control the position (right or left) of the Dynamic UCS or turn it off completely.

Additional smaller improvements include:
- The look of dialogs and floating messages was improved.
- The look of Parameter buttons was improved.
- Sketcher dimensions are now easy to read in any orientation. No matter how you rotate the model – the dimension text stays readable.
- The last position of dialogs is now kept and used the next time the same dialog is opened.
- Tooltips are now available on prompt cells (at the bottom of the screen) in case the text in them is too long and therefore truncated.
- When clicking the prompt cell showing the current active component, that component is highlighted
- The florescent highlight added in Cimatron 13 can now be disabled using the preferences.
• Users who opt not to have menus opened when hovering over them (but rather only when clicking them) can now choose that behavior in the preferences.

• Users who use the option to always have the scroll Users wheel behave as a zoom in/out control, can now also use it as the Select Other control, by waiting for the Select Other arrow to appear. At that point the Select Other operation will be preferred to the Zoom In/Out operation.
General

Reporting Tools

- **Use Excel with Macro in reports** – In all relevant report tools we have added the option to use Excel templates with macros in them (XLSM file).

Data Management

- **Unlimited file size** – files of unlimited size can now be saved with Cimatron E14.
- **Find File location from sub-documents in the browser** – when documents have sub-documents under them (seen in an area colored in light blue), you can now right-click them and use the option Show Location On Disk to find where those sub-documents are (as you can from within an assembly).
Cimatron 14.0 introduces an integration with the PTC Windchill PLM system. This Integration provides, for the first time, a seamless display and synchronization of data between Cimatron and Windchill and enables Cimatron users to view the Windchill status of Cimatron files in the Cimatron working environment.

Moreover, Cimatron users can now select Cimatron files and execute PLM commands such as Check In, Check Out, Update Status, etc. directly from the Cimatron working environment.

In the Windchill environment, users can now benefit from a better display and management of Cimatron files. In addition, internal and external links between files are recognized and displayed at all levels.

Assembly Tree – each Component has a PLM status icon

Cimatron Explorer – PLM optional commands