

# IronCAD 2027 New Features

## Optimized for Industrial Equipment & Machinery Manufacturing, Metal Fabrication and Tooling Manufacturing, Assembly Layout and Design of Industrial Equipment and Facilities, and Modular Product Manufacturers

IronCAD 2027 delivers powerful, targeted enhancements that directly address the challenges of designing heavy-duty machinery, precision tooling, complex piping systems, large-scale assemblies, and modular equipment. These updates accelerate sketching and surfacing for custom metal fabrication, simplify detailed production drawings for fabrication shops, enable faster assembly layouts for industrial facilities, and support scalable modular designs with seamless data flow between 3D and 2D environments—ultimately reducing design cycles, minimizing errors, and improving downstream manufacturing efficiency.

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## Key Improvements

### Enhanced Performance and Productivity

IronCAD 2027 delivers substantial performance gains in load/save operations and model interaction. Large-scale industrial assemblies, complex facility layouts, and heavy designs now open and respond significantly faster, reducing downtime and improving daily modeling efficiency. This is accomplished with improvements in the Graphical Interface and internal improvements to Internal Link creation, along with the Limited Link usage enhancements.

### Improved Linking and Editing Capabilities

The enhanced linking architecture provides better internal data management and new editing options for limited external links. The **“Set as Active”** command allows loading limited links to fully editable states, complemented by a **“Deactivate All”** option to return to the Limited Link state. This offers manufacturers and teams working with many external linked files greater flexibility to modify components while gaining significant performance with direct editing of the full model structure.

## Flexible Assembly Support

Assemblies can now be designated as “flexible,” enabling child component positions to vary across different instances while preserving original design intent. This is particularly valuable for industrial equipment and facility layout projects where the same sub-assembly (such as robotic arms, conveyors, or modular frames) must appear in multiple configurations within one project with different mechanism behaviors.

## Smart Annotation Enhancements in ICD

IronCAD 2027 introduces several intelligent annotation improvements:

- Remember simple modification to Attachment Point Smart Annotations during update
- Improvements on location of text and arrows during creation
- Improvements on Automatic Item Bubble Placement and positioning

These features produce more concise, standards-compliant drawings with reduced manual cleanup.

## AI Chatbot, AI Design Assistant, AI Drawing Assistant, and Python Integration

- **IronCAD AI Chat Bot** - Real-time chat about IronCAD-related questions and technical questions that is directly within IronCAD’s interface, providing instant answers and also proactively prompting users with helpful questions when certain commands are used, guiding them toward IronCAD’s innovative methods of design directly within their workflow, is now available free to all users (Trial and Commercial users) through December 31<sup>st</sup>, 2026.
- **IronCAD AI Design Assistant** - The new IronCAD AI Design Assistant revolutionizes how you design by combining powerful artificial intelligence with your real IronCAD catalog data. It leverages not only the extensive library of existing catalog components but also rich added metadata—including detailed descriptions, engineering parameters, materials, and Smart Attachment information - to understand and interpret your design intent. This represents a major industry leap: moving beyond generic AI suggestions to constructing real, manufacturable designs grounded in your actual catalog data and proven engineering intelligence.
- **IronCAD AI Drawing Assistant** - The IronCAD AI Drawing Assistant learns directly from your company’s existing technical drawings and cataloged 3D components to dramatically accelerate and standardize the creation of 2D manufacturing drawings. The IronCAD AI Drawing Assistant turns your existing 2D and 3D data into a powerful intelligence layer that makes detailing faster, more accurate, and far more

consistent — helping your team deliver high-quality drawings with significantly less effort.

- **IronCAD Python Integration** - IronCAD now includes native support for the official Python Package ([python-ironcad](#)), seamlessly integrated into IronCAD 2027. This powerful package exposes IronCAD's entire API to Python, enabling users and developers to automate, customize, and extend IronCAD directly from Python scripts. Using simple Python code, you can create custom automation tools, batch process designs, generate complex models programmatically, integrate with external systems, or build specialized applications—all while leveraging the complete IronCAD API. The IronCAD Python Package transforms IronCAD into a highly programmable 3D CAD platform, giving technical users and developers unprecedented control and flexibility to innovate faster inside of IronCAD.

## Usability and Workflow Improvements

A range of enhancements streamlines daily tasks:

- One-handed camera controls and visual orbit pivot for intuitive navigation of large assemblies
- Freely movable TriBall into open space without a geometry snap point
- Movable quick parameter edit dialog
- Perspective with Ortho on orthographic viewing orientation

## Drawing and Documentation Enhancements

Key ICD improvements include:

- Bulk Drawing Template accessible for Create Drawing Process
- Export of multiple sheets to a single DWG with multiple layouts
- Faster geometry hit-testing and full 3D Isometric dimensioning support
- Text Box supporting Vertical Alignment along with Horizontal Alignment
- Item Bubble Support for Prefix and Postfix options

Together, these updates enable faster creation of accurate, production-ready drawings while maintaining live links to the 3D model.

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**IronCAD 2027. Optimized for complex design challenges.**

# AI Chatbot, AI Design Assistant, AI Drawing Assistant, and Python Integration

## IronCAD AI Chat Bot

Real-time chat about IronCAD-related questions and technical questions that is directly within IronCAD's interface, providing instant answers, is now available free to all users (Trial and Commercial users) through December 31<sup>st</sup>, 2026. Visit [www.ironcad.com/xxx](http://www.ironcad.com/xxx) to access these time-limited free license to run IronCAD's AI Chat Bot.

- **Key Benefits to Users:**

- Ask IronCAD related questions to receive answers from IronCAD generated databased of information for more accurate results than traditional search and AI queries.
- The AI Chat Bot proactively prompts users with helpful questions when certain commands are used, guiding them toward IronCAD's innovative methods of design directly within their workflow eliminating trial-and-error and shortening the learning curve.

## IronCAD AI Design Assistant

The new IronCAD AI Design Assistant revolutionizes how you design by combining powerful artificial intelligence with your real IronCAD catalog data. It leverages not only the extensive library of existing catalog components but also rich added metadata—including detailed descriptions, engineering parameters, materials, and Smart Attachment information - to effectively help AI systems better understand and interpret your design intent.

Simply describe what you need in natural language. For example: "Create a stainless steel water tank that holds 500 gallons, with a flat bottom, side access port, and standard fittings."

The AI Design Assistant analyzes your request, searches the enriched catalog, and intelligently proposes one or more complete solutions. It automatically configures components with the correct parameters and uses Smart Attachments to position everything accurately in 3D space. With one click, you can accept and generate the fully built model directly in your scene—ready for further refinement, simulation, or manufacturing. In addition, you can modify the configured modeling steps to accurately

refine the results or ask follow-up questions to allow the AI to continue to update your designs.

This represents a major industry leap: moving beyond generic AI suggestions to constructing real, manufacturable designs grounded in your actual catalog data and proven engineering intelligence.

- **Key Benefits to Users:**
  - **Dramatically Faster Concept-to-Model:** Go from idea to complete 3D assembly in seconds instead of hours by describing what you want rather than manually searching and building.
  - **Accurate & Intelligent Configuration:** The AI respects real engineering parameters, materials, and Smart Attachment rules, reducing errors and ensuring designs are practical and catalog-compliant.
  - **Context-Aware Positioning:** Components are automatically placed with correct spatial relationships and attachments—no more tedious manual alignment.
  - **Leverages Your Existing Catalog Investment:** Makes your entire library of parts and metadata significantly more powerful and accessible.
  - **Supports Complex Requirements:** Handles multi-part systems with material, size, capacity, performance, and interface specifications in a single request.
  - **Creative Exploration & Iteration:** Quickly generate and compare design alternatives, then refine them with precise control.
  - **Ideal for Both New and Experienced Users:** Lowers the barrier for quick ideation while accelerating detailed work for power users.

The IronCAD AI Design Assistant transforms your catalog from a simple parts library into an intelligent design partner—delivering real engineering value, unprecedented speed, and higher design quality.

**Availability:** Trial usage and Based on Custom Customer Projects. IronCAD is actively accepting and working with customers on real-world projects using the IronCAD AI Design Assistant. Contact IronCAD for details.

## IronCAD AI Drawing Assistant: Smart & Consistent 2D Detailing

The IronCAD AI Drawing Assistant learns directly from your company's existing technical drawings and cataloged 3D components to dramatically accelerate and standardize the creation of 2D manufacturing drawings.

Once trained on your historical drawings and reused 3D catalog parts, the AI understands your specific annotation styles, dimensioning practices, tolerances, notes, symbols, and view preferences. When you generate a new drawing from 3D components (using IronCAD reusable catalog items), the AI Drawing Assistant automatically applies the appropriate annotations across views — replicating your established standards with high accuracy.

It intelligently recognizes components and features, then places dimensions, GD&T, surface finishes, weld symbols, bill of materials callouts, and other annotations exactly where your team would expect them.

**Future Capabilities** The assistant will extend to imported 3D models, legacy drawings, and visual representation data, enabling rapid recreation of fully associated IronCAD drawings that maintain live links to the imported 3D geometry.

- **Key Benefits to Users**

- **Massive Time Savings:** Reduce drawing detailing time by 50-80% by automating repetitive annotation work.
- **Consistent & Professional Output:** Ensure every drawing follows your company standards without manual effort or variation between team members.
- **Reuses Knowledge:** Captures and applies best practices from your best drawings and most-used components.
- **Fewer Errors:** AI applies correct annotations based on learned patterns, reducing missed dimensions and inconsistent detailing.
- **Faster Drawing Creation:** Go from 3D model to fully annotated drawing in minutes instead of hours.
- **Scalable Across Projects:** Works seamlessly with both native IronCAD catalogs and future imported data workflows.
- **Continuous Improvement:** The more you use it, the smarter it becomes as it learns from your latest drawings.
- **Faster Migration of Legacy Design Data:** Rapidly convert legacy drawings and imported CAD data into fully detailed IronCAD drawings by automatically applying learned company practices and visual representation - preserving design intent, reducing manual rework, and accelerating transition from other systems.

The IronCAD AI Drawing Assistant turns your existing 2D and 3D data into a powerful intelligence layer that makes detailing faster, more accurate, and far more consistent — helping your team deliver high-quality drawings with significantly less effort.

**Availability:** Based on Custom Customer Projects. IronCAD is actively accepting and working with customers on real-world projects using the IronCAD AI Drawing Assistant. Contact IronCAD for details.

## IronCAD Python Integration

IronCAD now includes native support for the official Python Package ([python-ironcad](#)), seamlessly integrated into IronCAD 2027. This powerful package exposes IronCAD's entire API to Python, enabling users and developers to automate, customize, and extend IronCAD directly from Python scripts.

When installed with IronCAD 2027, the package allows you to:

- Connect to a running IronCAD session
- Launch new instances
- Drive the full user interface and document operations
- Access and manipulate geometry, features, assemblies, drawings, parameters, and more

Using simple Python code, you can create custom automation tools, batch process designs, generate complex models programmatically, integrate with external systems, or build specialized applications—all while leveraging the complete IronCAD API.

- **Key Benefits to Users:**

- **Powerful Automation:** Streamline repetitive tasks, create design configurations, and automate complex workflows with clean, modern Python scripting.
- **Full API Access:** Harness the complete power of IronCAD's API in a familiar, industry-standard Python environment.
- **Easy Integration:** Combine IronCAD with data science tools, PLM systems, ERP, machine learning, and other Python-based applications.
- **Custom Tool Development:** Build your own add-ons, design wizards, and specialized extensions without needing traditional COM/.NET development.

- **Enhanced Productivity for Power Users:** Write scripts for parametric design, bulk modifications, design validation, and drawing automation.
- **Future-Ready Foundation:** Opens the door to tighter integration with AI tools and external automation platforms.
- **AI Connected:** Provides a convenient process to connect AI systems to interact directly in IronCAD with IronCAD's functions.

The IronCAD Python Package transforms IronCAD into a highly programmable 3D CAD platform, giving technical users and developers unprecedented control and flexibility to innovate faster.

#### **Project Links:**

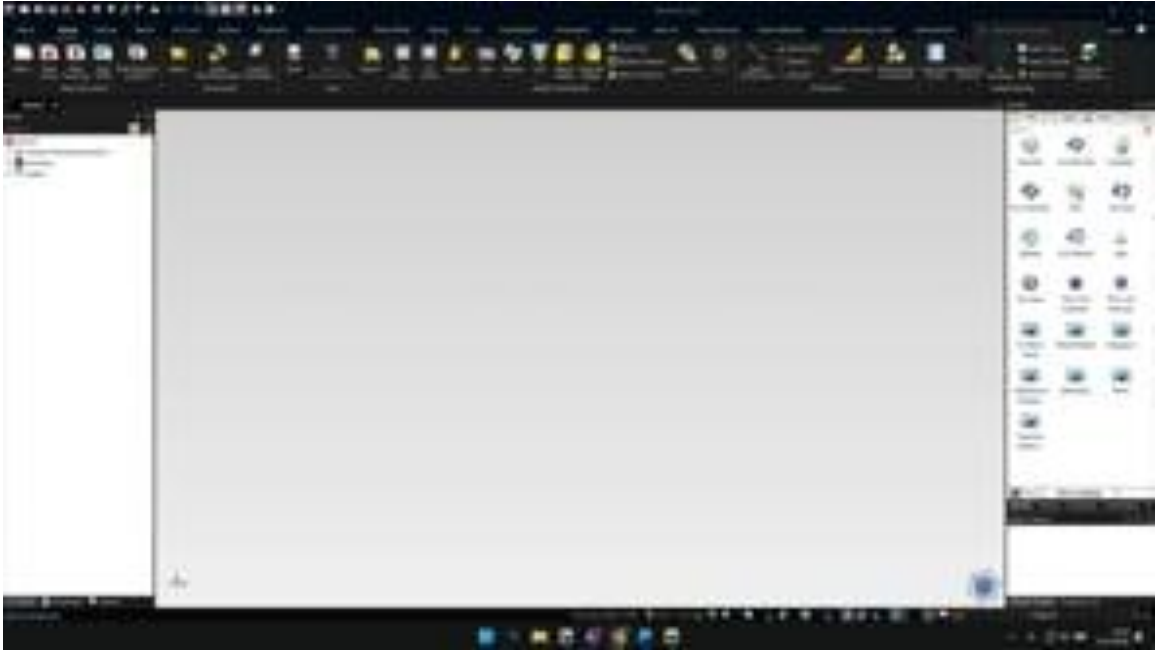
- <https://pypi.org/project/python-ironcad/>
- <https://github.com/IronCAD/python-ironcad>

## General / Modeling Enhancements

### Direct Edit for Limited Link Parts

Limited external links can now be converted to fully editable links via “Set as Active” in the context menu, with “Deactivate All” for quick management.

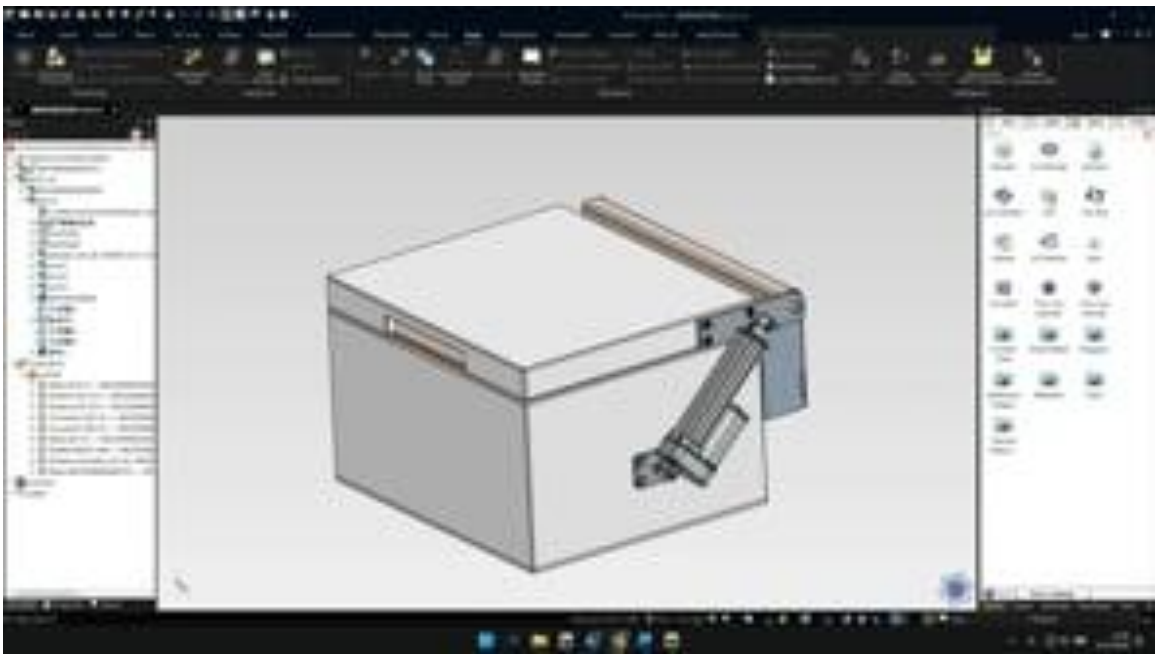
- **Key Details/Parameters:**
  - “Set as Active” and “Deactivate All” added to limited link context menu.
- **Benefits:** Manufacturers working with many external linked components can quickly activate and edit components while gaining significant performance with the limited link capability in the assembly design.



## Flexible Assembly Support

Assemblies can now be marked “flexible,” allowing child component positions to vary across different instances while preserving design intent.

- **Benefits:** Ideal for industrial equipment and facility layouts where the same sub-assembly (e.g., robotic arms or conveyor modules) must appear in multiple configurations within one project and have different mechanism behaviors.



## Skeleton Design: Powerful Top-Down Design Control

Skeleton Design brings robust top-down modeling capabilities to your workflow. Create a dedicated skeleton model (typically a structured part file) that serves as the central "source of truth" for your assembly. Define and organize all critical reference geometry—including sketches, planes, axes, points, curves, surfaces, and bodies—within this single, lightweight model.

Mark geometry as public (published) for controlled sharing. Then, use intuitive copy operations – Copy Public Geometry, Copy Body, Copy Faces, Copy Curves/Sketches, Copy Vertex, and Copy Reference Geometry - to propagate this geometry into other parts across the assembly. Changes made to the skeleton can update all dependent parts (using Regenerate commands), ensuring consistent design intent and parametric control. This approach is especially effective for Structured Parts designs today, enabling hierarchical, assembly-driven modeling without excessive in-context references.

- **Key Benefits to Users**

- **Faster Design Iteration:** Modify the skeleton once and updates propagate reliably to multiple parts—ideal for complex assemblies, layouts, or families of products.
- **Improved Design Intent & Consistency:** Centralize critical dimensions, positions, and relationships in one place, reducing errors from duplicated or conflicting geometry.
- **Lightweight & Stable Models:** Skeleton parts remain simple and efficient, avoiding heavy assembly-level sketches or excessive external references that can slow performance or cause rebuild issues.
- **Better Team Collaboration:** Multiple users can reference the shared skeleton without constant access to the full top-level assembly, supporting PDM workflows and parallel design.
- **Scalable for Large Projects:** Perfect for structured, hierarchical designs (machines, frames, mechanisms, etc.) where overall layout drives component details.
- **Reduced Maintenance:** Easier to manage updates, configurations, and variants while maintaining full control over how geometry is shared and consumed.

Skeleton Design empowers you to work smarter—define the big picture first, then let the details follow with confidence and efficiency. Whether you're building intricate mechanisms or large structural assemblies, this feature streamlines top-down design

for higher productivity and fewer headaches. Currently available in Structured Parts and will extend to Innovative Parts in future versions.

## Camera Enhancements for One-Handed Camera & Visual Pivot

New one-handed camera controls and a visual orbit pivot (colored sphere) provide intuitive navigation for large, complex assemblies and facility layouts.

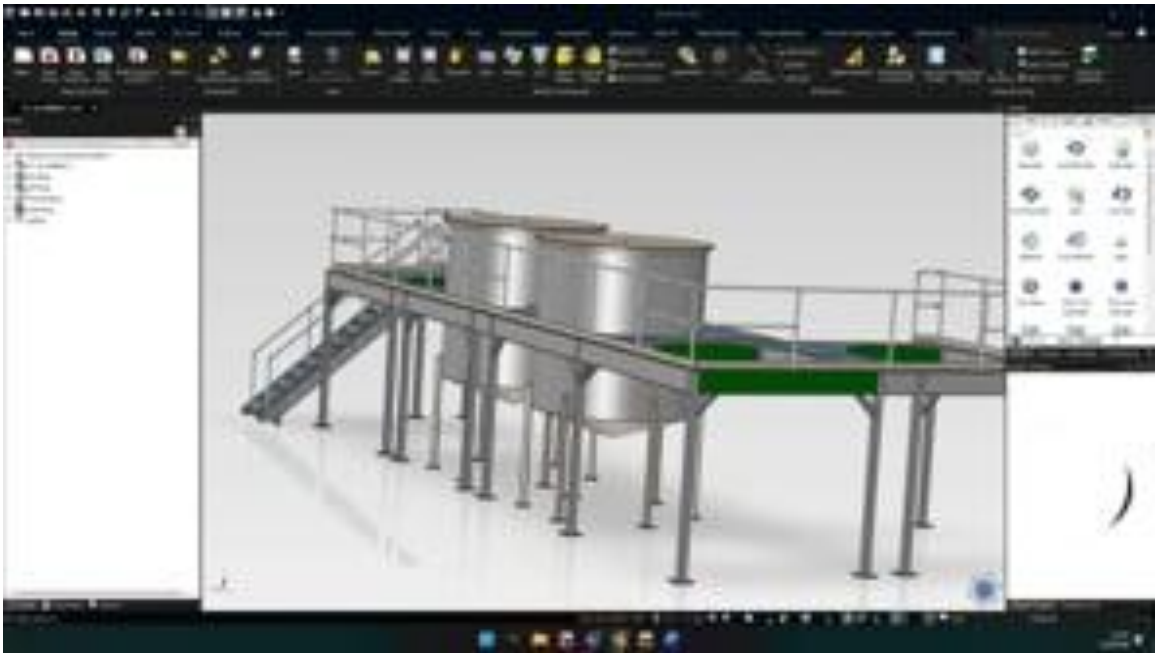
- **Key Details/Parameters:**
  - [MMB]+[LMB] = Pan
  - [MMB]+[RMB] = Zoom
  - [MMB]+Click+[LMB] = Walk (and additional one-handed commands)
  - Notes: These commands can be customized in Tools/Options/Interaction/Mouse
  - Pivot visualized as orange (background or scene rotation center) or cyan (geometry center location) for instant feedback.
- **Benefits:** Assembly and facility designers can navigate massive industrial equipment models faster and more ergonomically, reducing fatigue during long layout sessions.



## Camera Setting - Perspective with Ortho

New Camera setting to enable Perspective when the current view is not directly looking at a face or Top, Front, Right views. In those views, the camera will be set to Orthographic.

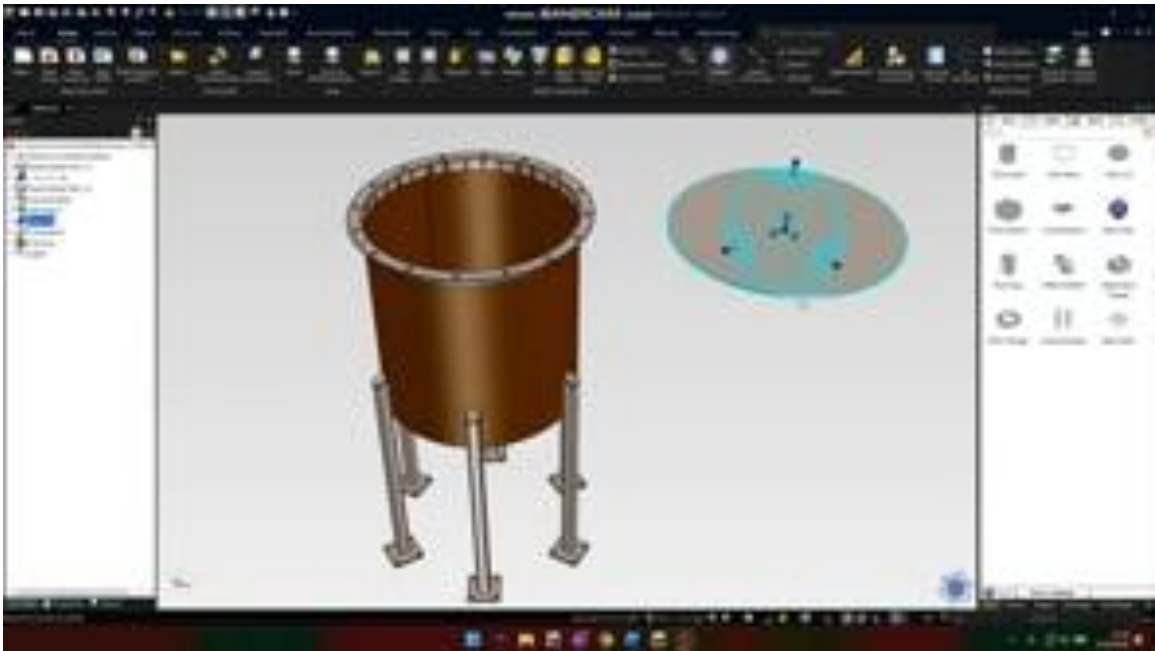
- **Benefits:** Users can effectively use perspective in non-orthographic viewing directions, and the system will automatically convert orthographic when the viewing direction is set to an orthographic view, such as Front, Top, Right, or when using the look at tool.



## Move TriBall Freely in Space

Right-click toggle allows free movement of the TriBall center in space without needing to be on geometry.

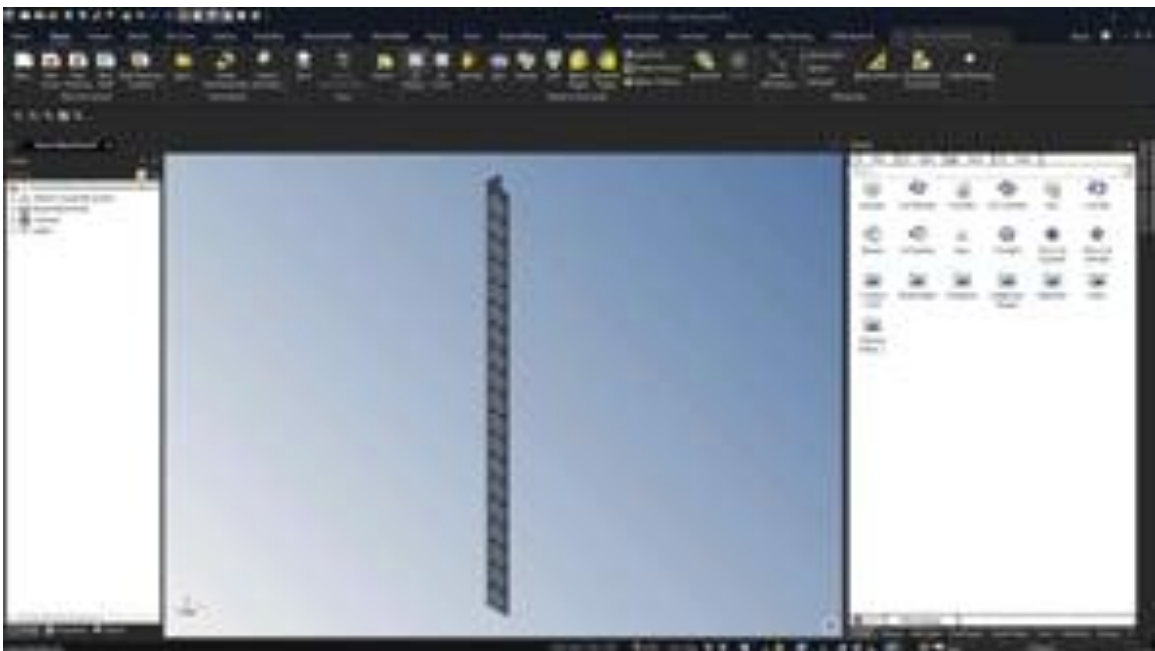
- **Benefits:** Faster positioning of large components during facility and equipment layout.



### Quick Parameter Edit Dialog – Move and Reset

The quick parameter edit popup is now movable, with position persisted across sessions.

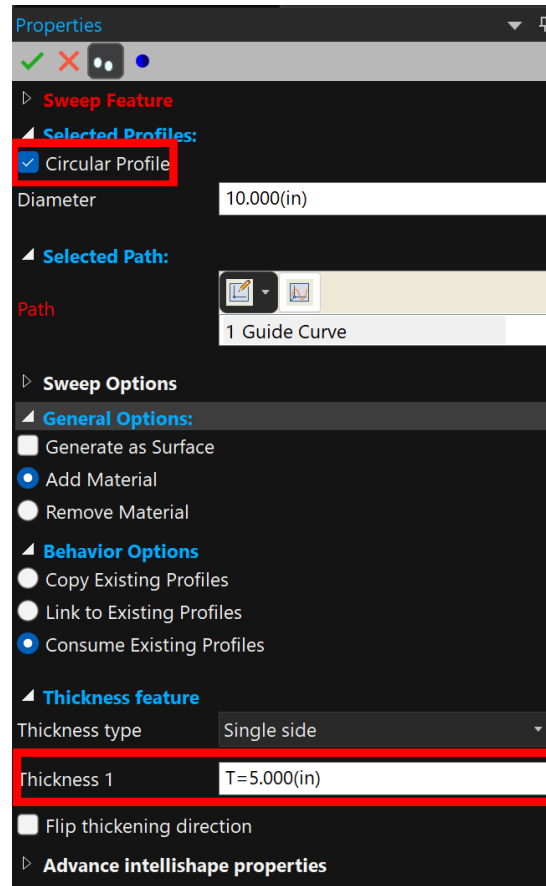
- **Benefits:** Streamlined parametric editing for repetitive machinery design tasks.



## Parameter for Thicken Thickness in Sweep (and Extrude/Spin)

Thickness values for circular sweeps, extrude, and spin are now fully parameterizable and auto-named.

- **Benefits:** Parametric control of wall thickness in piping, ducts, and thin-walled components ensures design intent is maintained across revisions.



## Sweep and Extrusion Enhancements

Improved sweep section when placed on the curve for direction (up/down/all), plus initial offset (pre-extrude) and target-to-surface controls for extrusions.

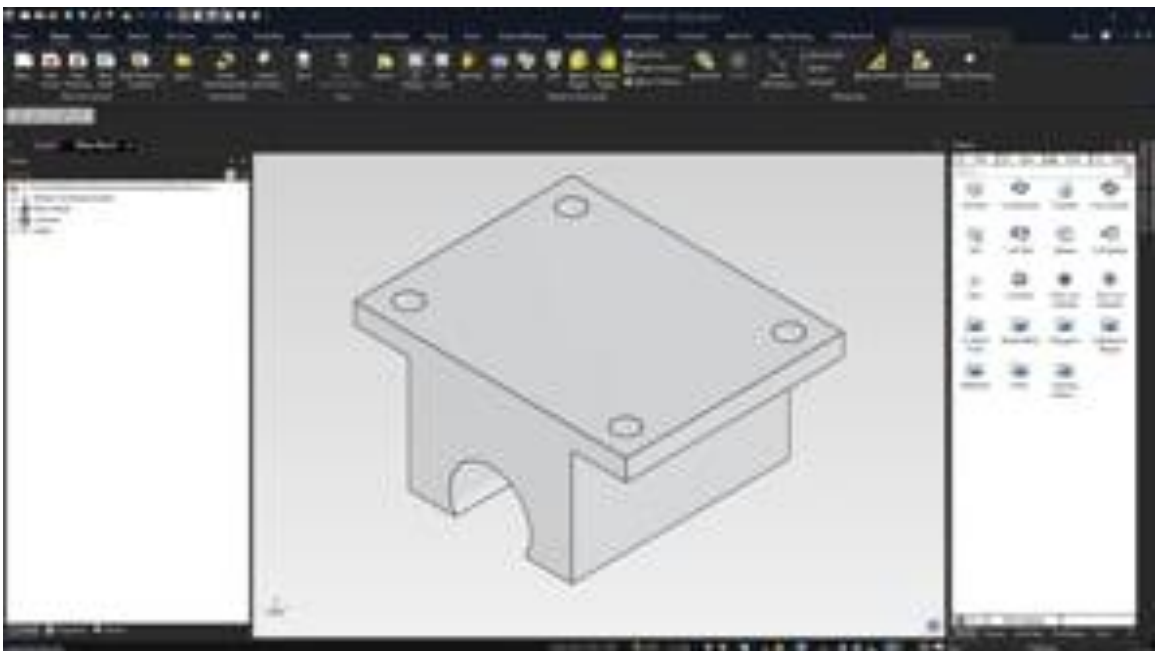
- **Benefits:** Faster creation of complex structural members and custom extrusions used throughout industrial equipment.



## Fillet/Chamfer Upgrades

Asymmetric tangency blends (different radii top/bottom) and new edge overflow options (“scroll over,” “trim at edge”) prevent failures.

- **Benefits:** Reliable blending on heavy, multi-thickness fabrications and castings.



## Blend/Chamfer Edge Delete Key Behavior

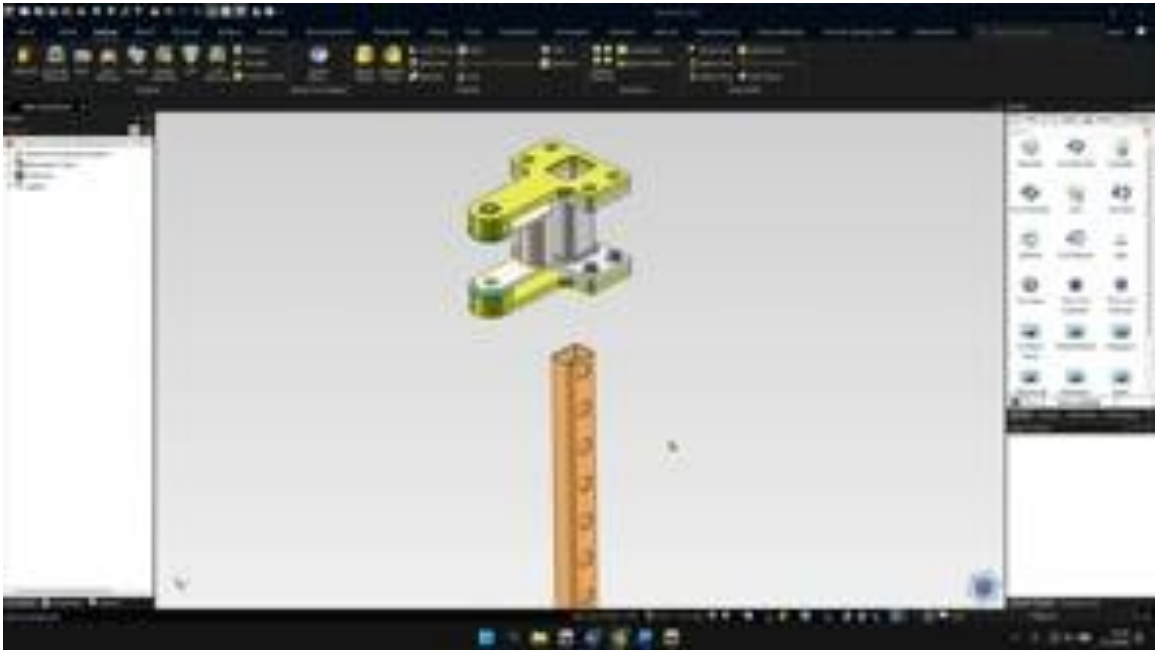
Pressing Delete on a selected chamfer or blend edge/face now deletes the geometry itself, not just the feature value.

- **Benefits:** Reduce time correcting incorrect delete key actions when wanting to delete features.

## Feature & Wireframe External Reference Enhancements

Features (extrude, array, mirror, etc.) and 3D wireframe/piping tools now support external references from other parts, with full update tracking and projection-like constraints.

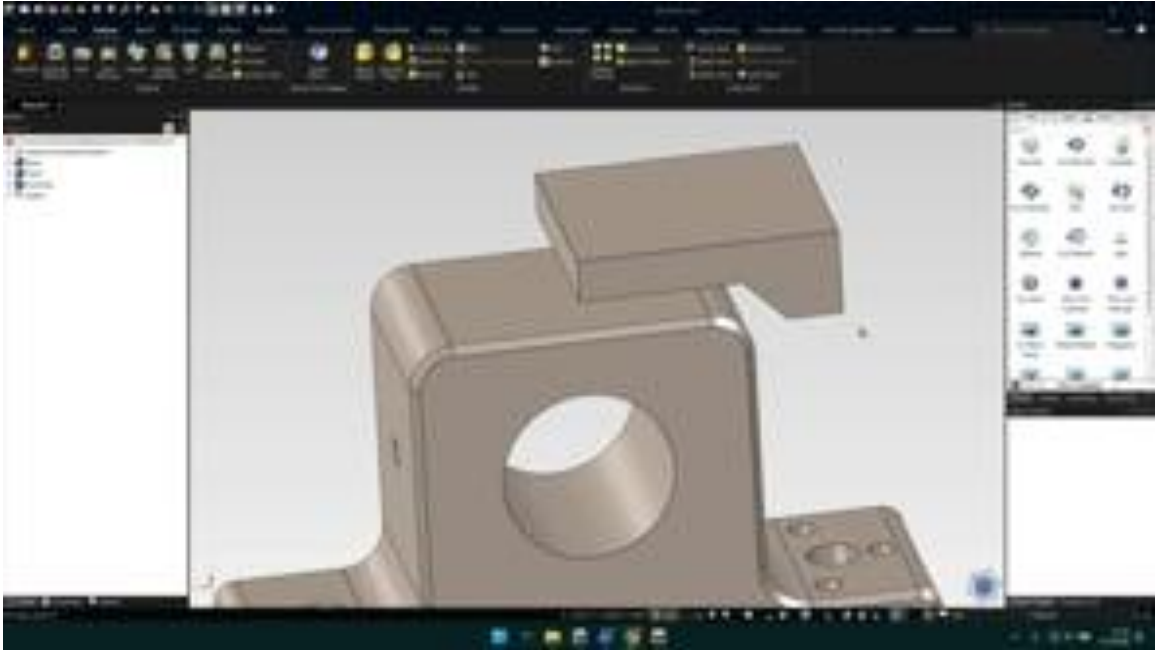
- **Benefits:** Enables advanced design capabilities by allowing components to reference a shared master geometry across the entire product family.



## Extrude Offset from External Reference

The extrude command has been enhanced to support and extrude to a face on an external part with a set offset distance.

- **Benefits:** Allows more complex designs where features on different parts can update with changes on another while maintaining a desired offset.

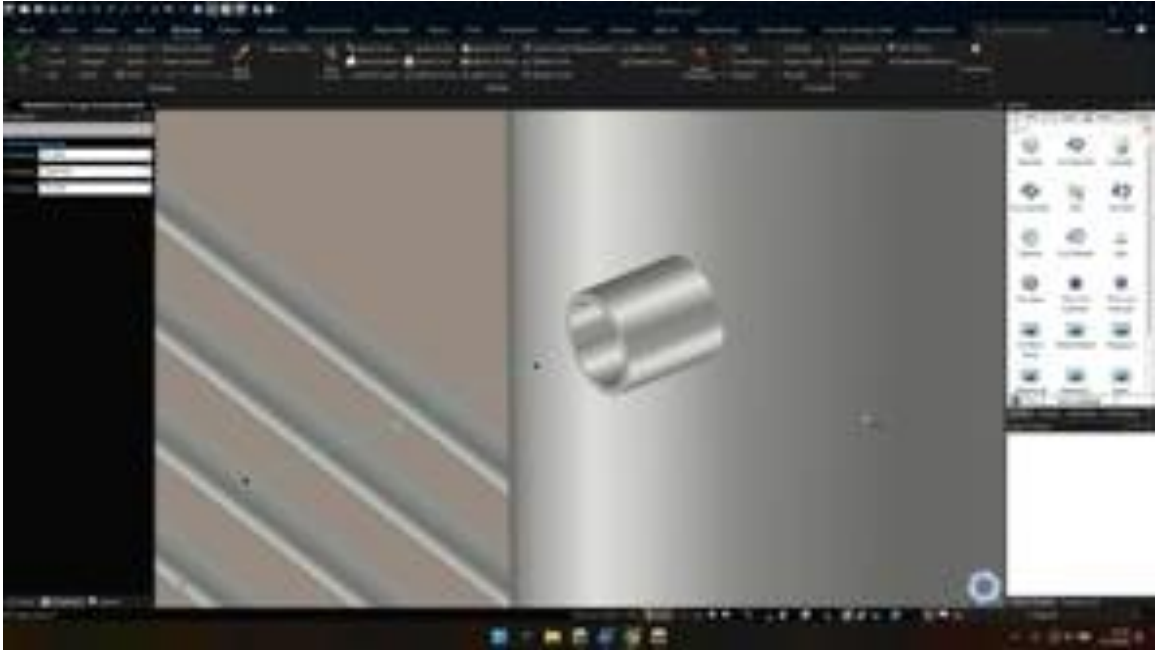


### 3D Curve External Reference

New command to create a 3D Curve External reference constraint to Point/Lines has been added to aid in the design and modification updates when utilizing 3D Curves.

- **Benefits:**

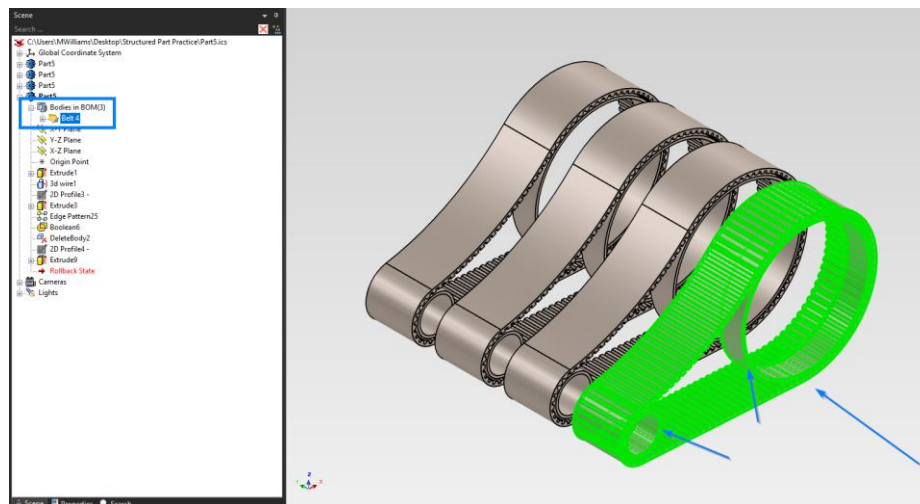
- Enables reliable design intent when working with 3D curves by keeping them associatively linked to reference points or lines.
- Automatically updates the 3D curve when the referenced geometry is modified, reducing manual rework.
- Improves design accuracy and consistency in complex models that rely heavily on 3D curves (such as sweeps, lofts, rails, and path-based features).
- Makes 3D curve-based modeling more robust and easier to modify throughout the product development process.



## Body Group Highlighting

Selecting a group in the BOM tree instantly highlights all associated bodies in the viewport.

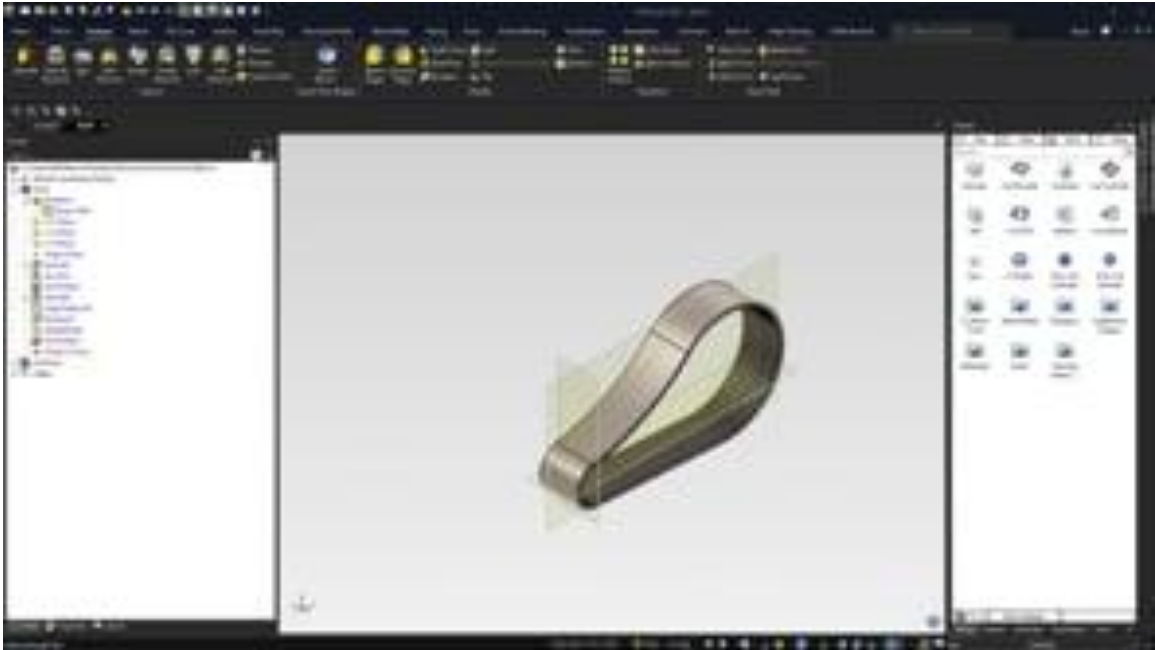
- **Benefits:** Rapid visual verification of complex multi-body industrial equipment designs.



## Multibody Extrusion/Stretch

Extrude or stretch commands can now generate one new body per closed sketch region, with improved region selection.

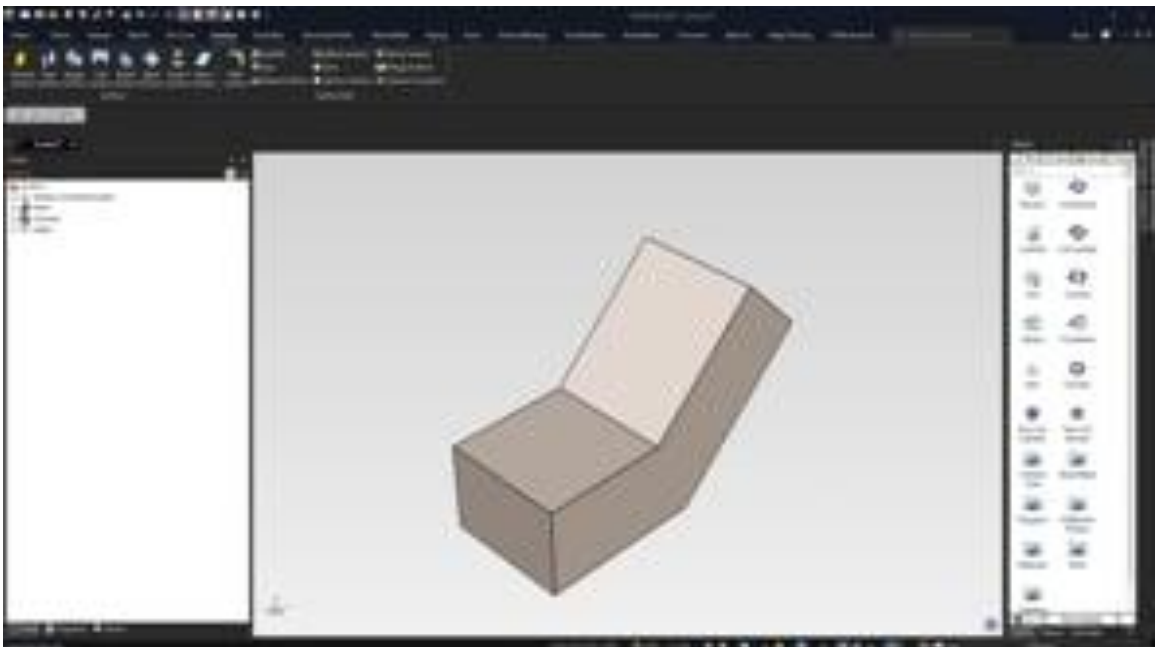
- **Benefits:** Dramatically speeds up multi-part fabrication from a single sketch—ideal for tooling plates and machinery frames.



## Offset Surface Enhancements

Offset surface tool now includes “reverse all” direction control, full support for stitched/volumetric surfaces, and the ability to omit individual faces.

- **Benefits:** Precise surface modeling for castings, enclosures, and heavy machinery fairings.



## Parallel Curve Operations

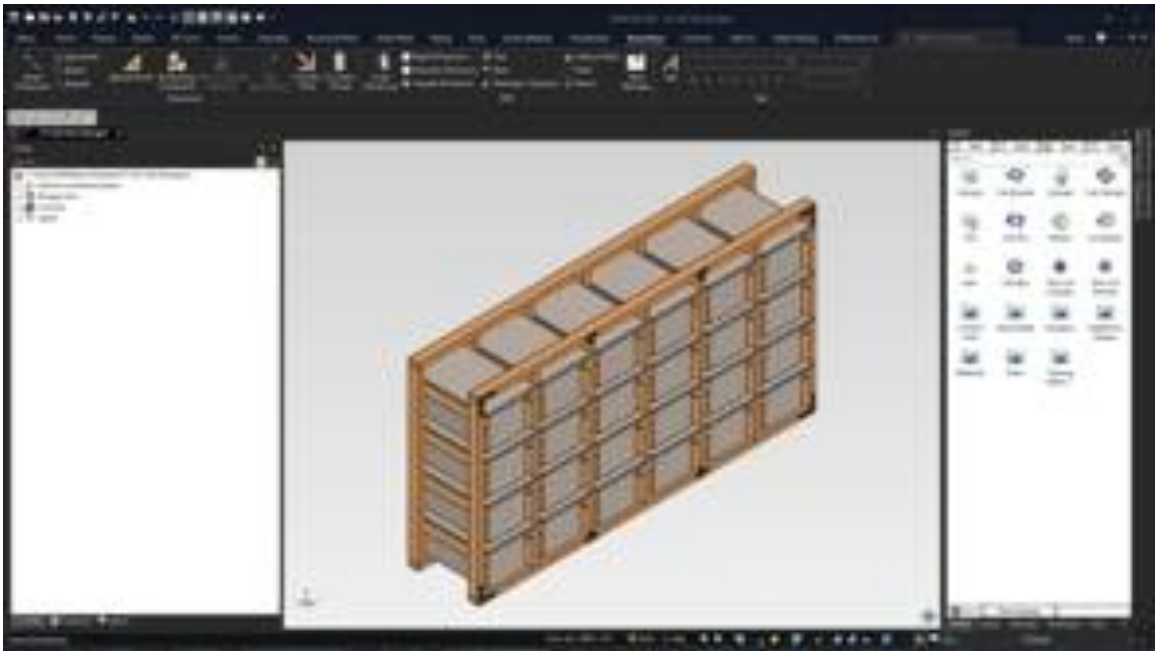
Full-featured parallel curve generation from surface edges and intersections, including repeated and batch operations.

- **Benefits:** Essential for aircraft-style fairings, automotive tooling, and industrial conveyor surfaces.

## Measurement and Analysis

Surface curvature tool with color mapping and on-click evaluation; Center of Gravity is now visual, allows output, and usable in measurement commands.

- **Benefits:** Critical mass and balance analysis for heavy industrial machinery and modular assemblies.

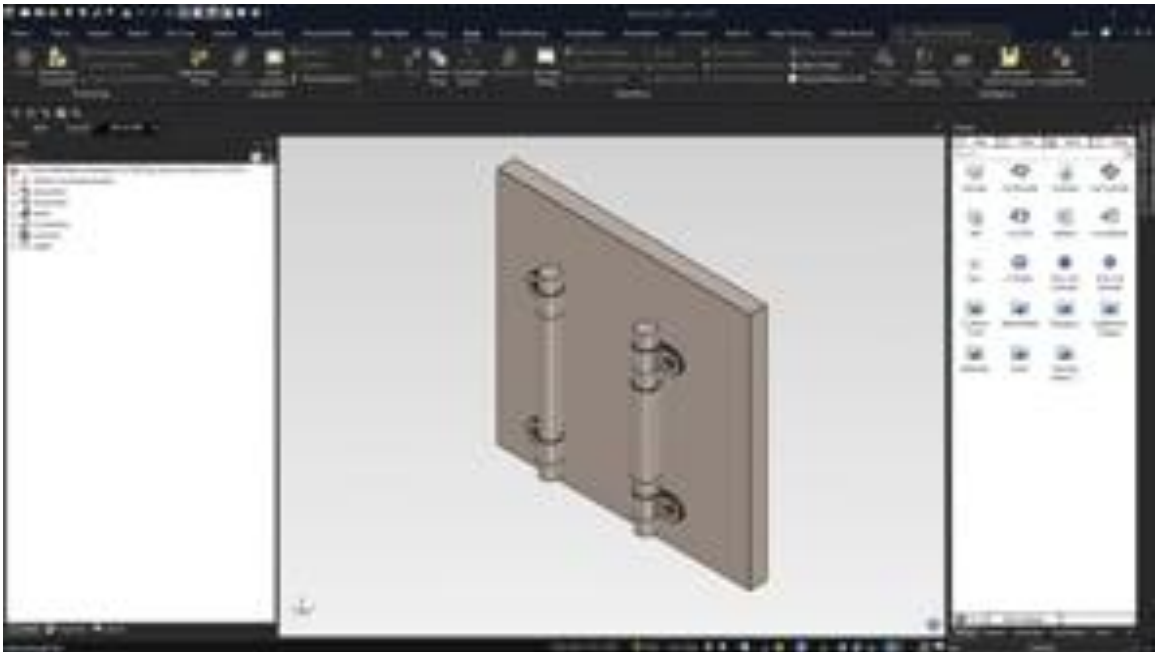


## Distance Constraint on Circular Shapes

New intelligent options have been added to the Distance Constraint tool specifically for cylindrical and circular shapes. Users can now quickly set Center-to-Center, Minimum, and Maximum distance constraints between cylinders, holes, arcs, or circular edges.

- **Benefits:**

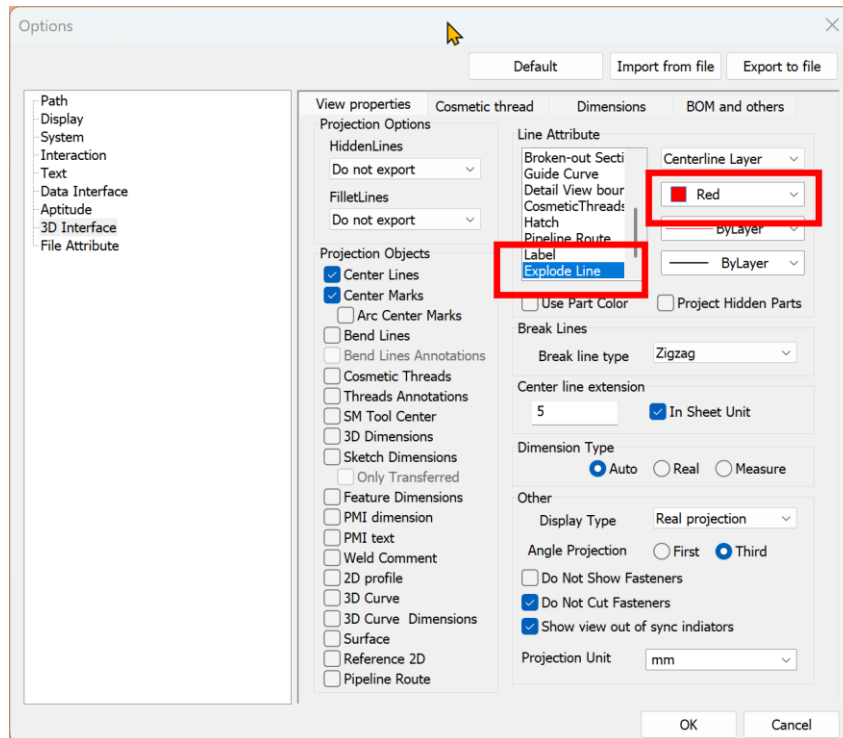
- Provides precise and meaningful control when positioning or constraining cylindrical features (such as holes, bosses, pipes, or shafts).
- Center-to-Center option simplifies common tasks like aligning hole patterns or component centers.
- Minimum and Maximum options are ideal for clearance checks, fit tolerances, and design envelope constraints.
- Significantly speeds up assembly constraint creation involving round geometry.
- Improves design intent capture and makes models more stable and predictable during updates or modifications.
- Reduces the need for complex workarounds or additional reference geometry when working with cylindrical parts.



## Exploded Line Projection & 2D Export Improvements

Exploded lines project into 2D views with independent styles (DRAFT); DWG export now preserves exact broken-view break line lengths.

- **Benefits:** Professional assembly instructions and service drawings for industrial equipment.



## Piping Improvements

Refined deletion rules, intelligent flange/connector placement (back-to-back, flip, snap), enhanced preview with endpoint switching and 3D sphere adjustment.

- **Benefits:** Faster, error-free piping design for industrial facilities and machinery skids.

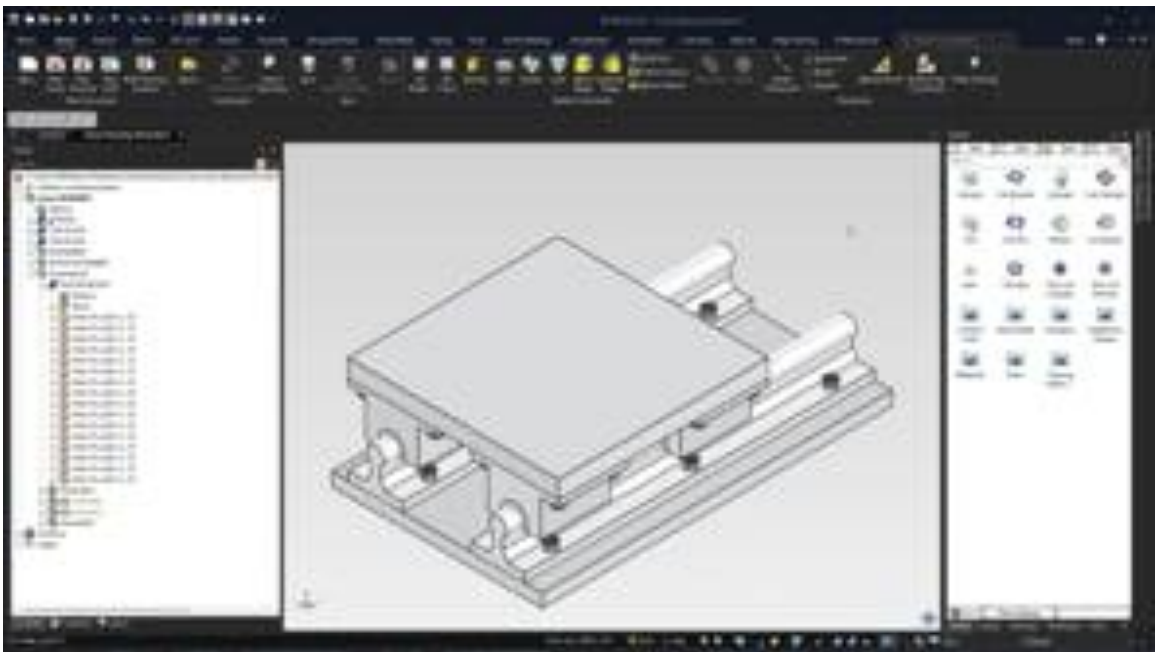




## Transparency for Assemblies

Temporarily make any selected surface/face transparent to select behind it; single command to revert all.

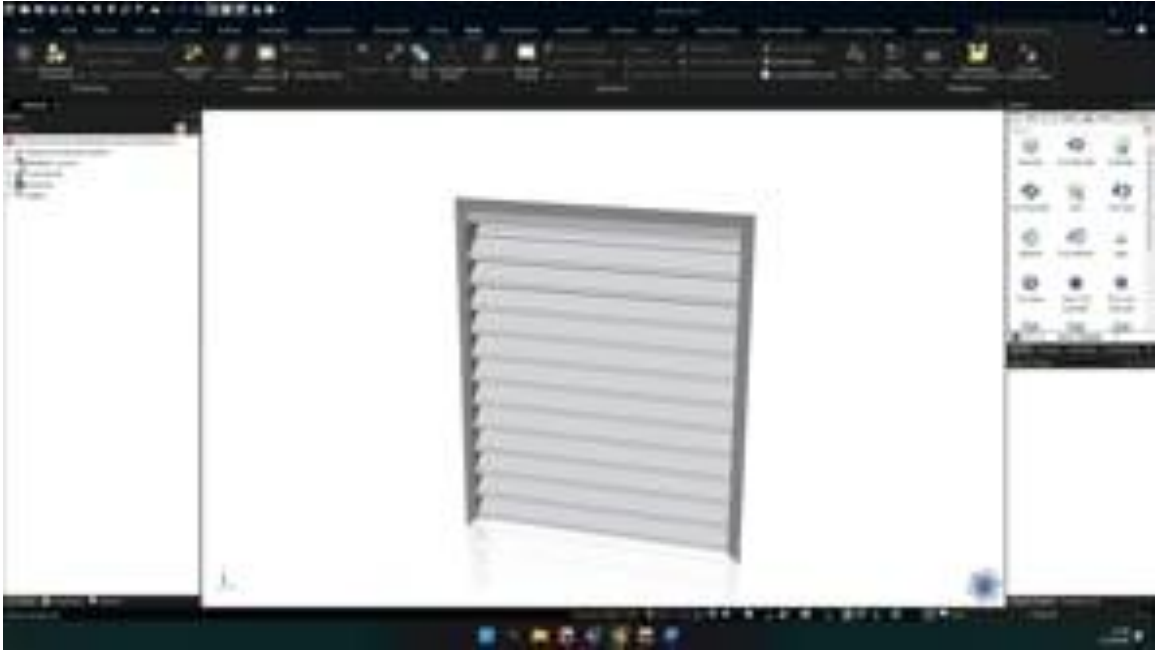
- **Benefits:** Essential when working inside dense, large-scale industrial assemblies.



## Replication (“Copy with Mates”)

Copy entire components—including constraints—“Copy with Mate.”

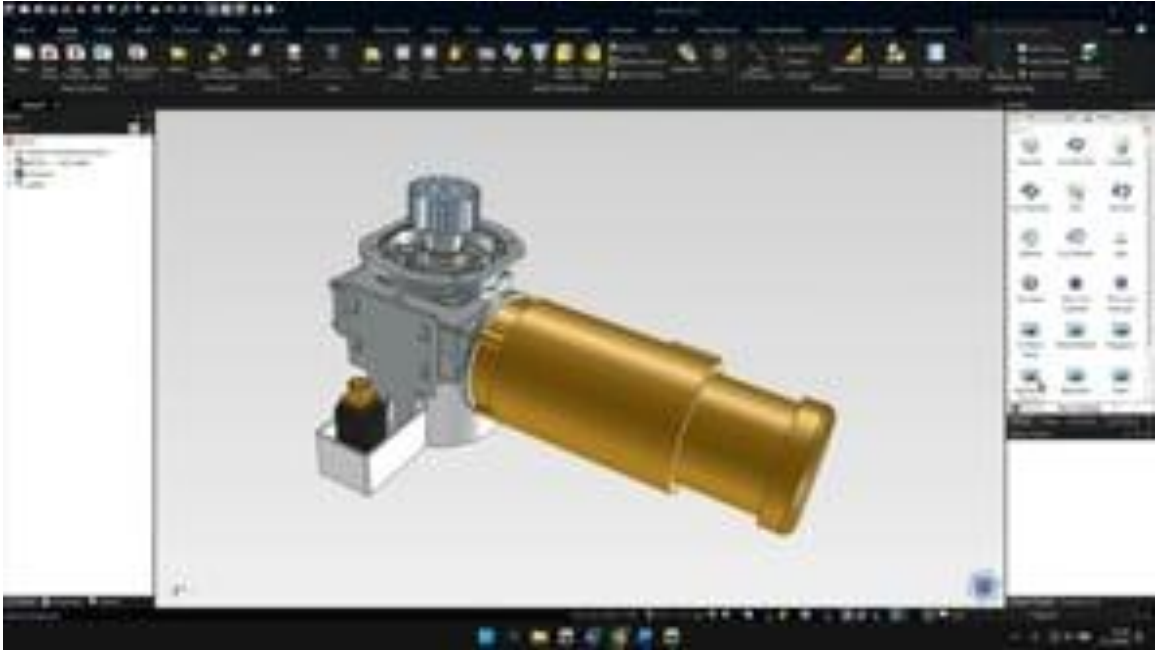
- **Benefits:** Massive productivity gain when duplicating repetitive sub-assemblies in equipment and facility layouts.



## Import/Export BOM/Custom Property

Custom attributes, geometry, mass/volume properties, and BOM rows are fully preserved between all exports/imports (.x\_t, .asat) and between 3D and 2D environments.

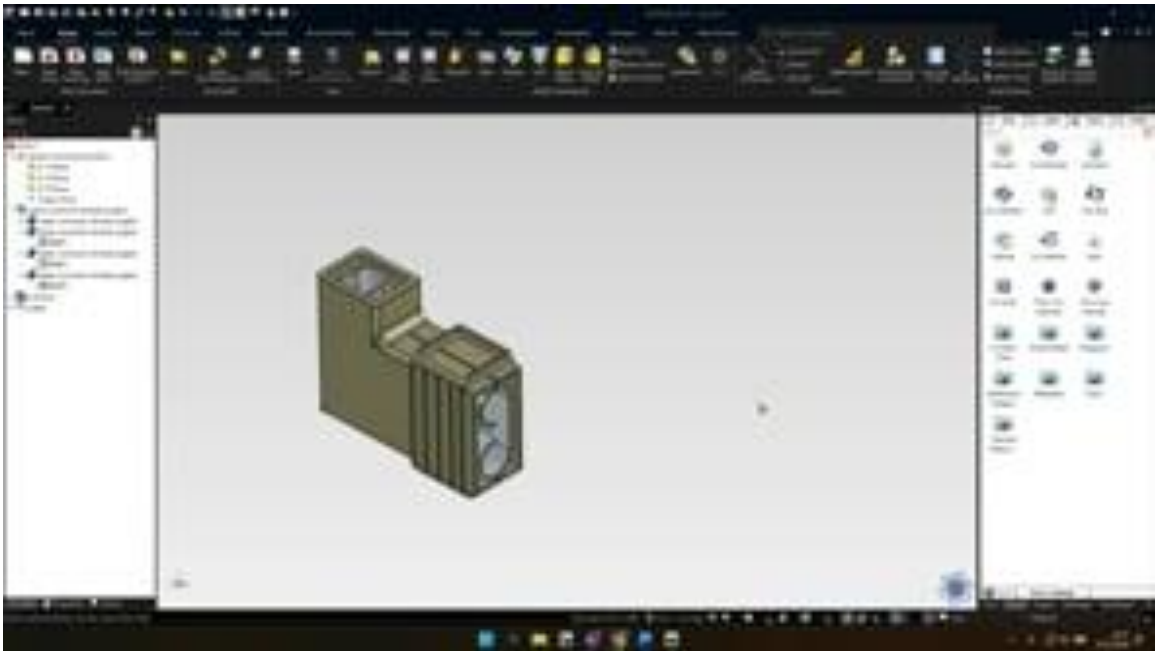
- **Benefits:** Seamless data continuity across the entire design-to-manufacture workflow for industrial equipment and modular manufacturers.



## Feature Recognition Improvement

New options have been added to run Feature Recognition at both the part level and assembly level. This enhancement significantly speeds up the process of analyzing and converting imported geometry (such as STEP, IGES, or neutral files) into editable IronCAD features.

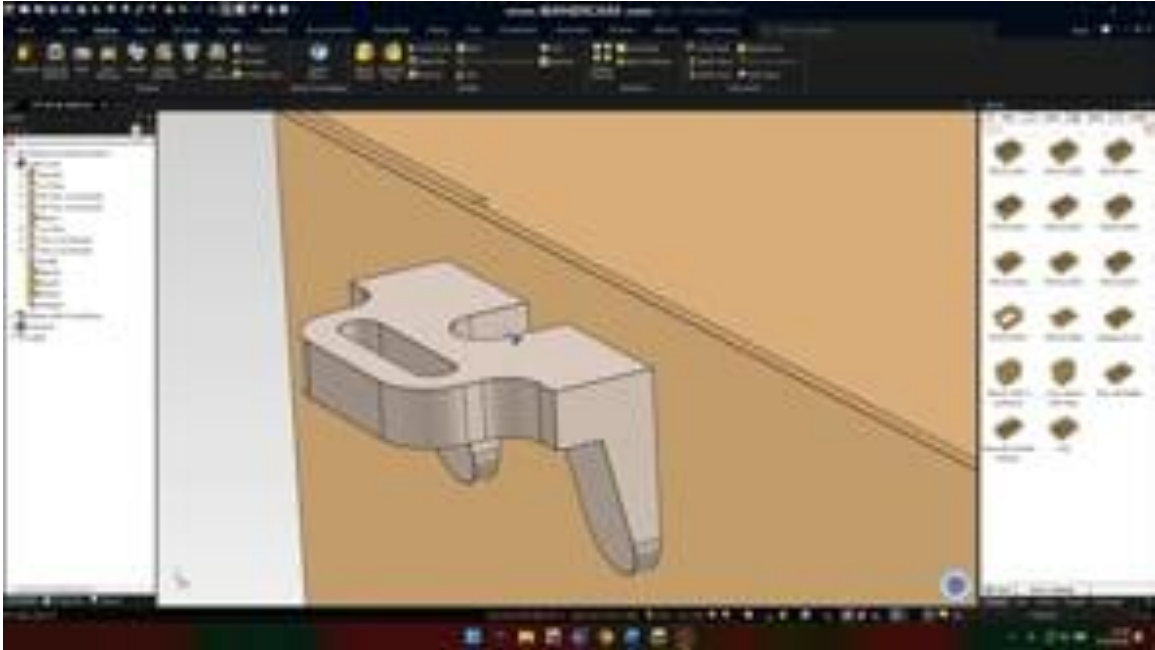
- **Benefits:**
  - Much faster Feature Recognition on large imported parts and complex assemblies.
  - Enables users to efficiently recreate intelligent, history-based features from imported data.
  - Reduces the time required to make imported models fully editable.
    - Improves productivity when working with customer-supplied models or legacy data.
  - Provides greater flexibility by allowing users to choose between part-level (focused) or assembly-level (comprehensive) recognition.
  - Helps bridge the gap between imported “dumb” geometry and native IronCAD smart models.



## Stretch Part/Assembly Based on Sketch

New options have been added to the Stretch command, allowing users to stretch a part or entire assembly based on an underlying sketch definition instead of the traditional sizebox.

- **Benefits:**
  - Significantly improved stretching accuracy on parts with angled, non-orthogonal, or complex geometry that previously didn't stretch correctly using the sizebox method.
  - Provides more predictable and controllable results when modifying angled walls, drafted faces, or irregular shapes.
  - Greater design flexibility by allowing users to drive the stretch behavior directly from the original sketch intent.



## Stretch Part/Assembly Include/Ignore Suppressed Features

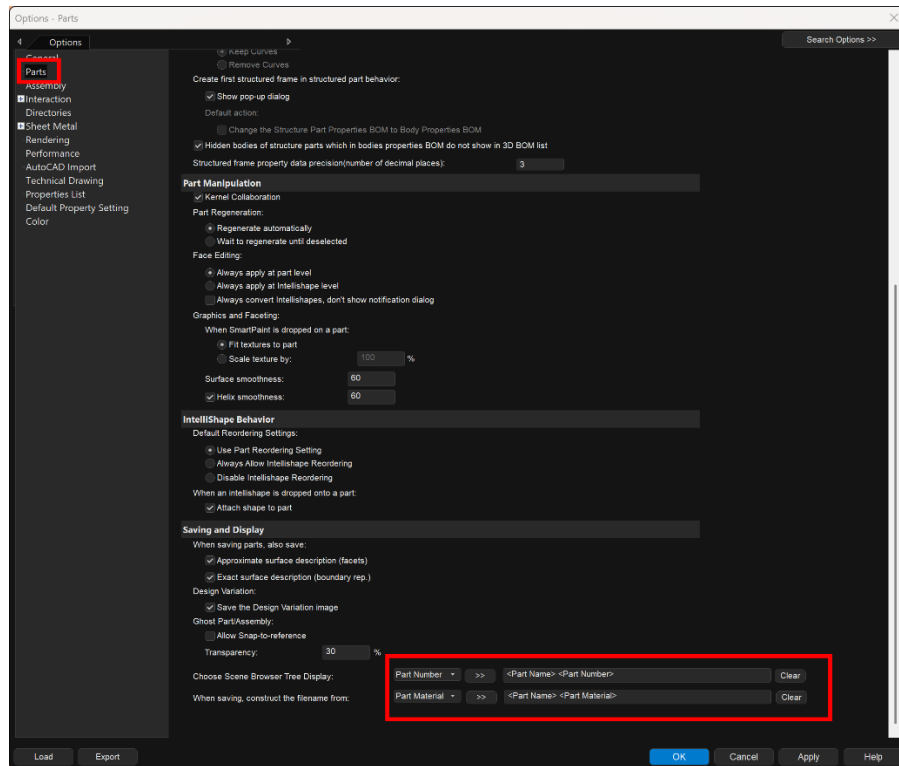
New improvements in the Stretch Part/Assembly to Include or Ignore suppressed features has been added giving more control and predictability of the stretched results.

- **Benefits:**
  - The new “include/exclude suppressed features” option gives users precise control over which features are affected during stretching operations.
  - Reduces the need for workarounds and manual adjustments when resizing or modifying complex parts and assemblies.

## Custom Part Name and File Name Formatting

Completely redesigned interface for part name and saved file name formats with larger edit boxes, clear/reset buttons, and removal of “NULL” from dropdowns.

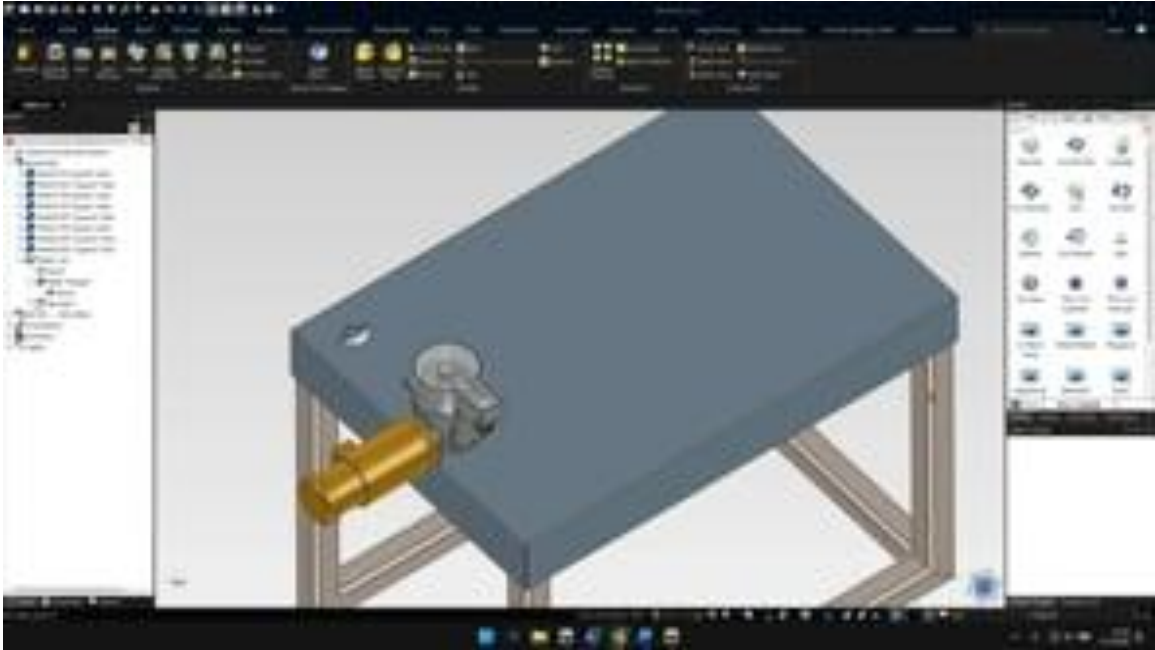
- **Benefits:** Consistent, professional naming display and export.



## Pattern Feature Node Box Selection

The Pattern Feature tool has been enhanced with a new box selection (window selection) capability inside the Pattern Node selection mode. Users can now drag a box to quickly select multiple nodes instead of picking them individually.

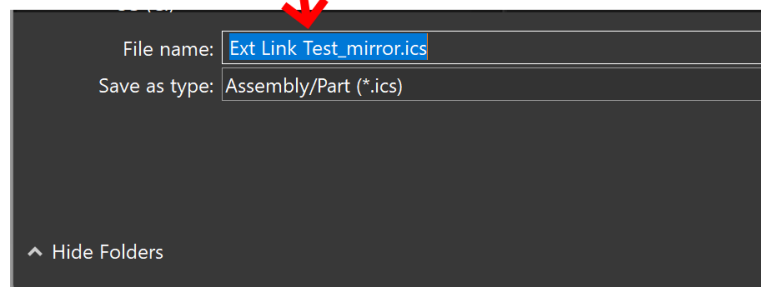
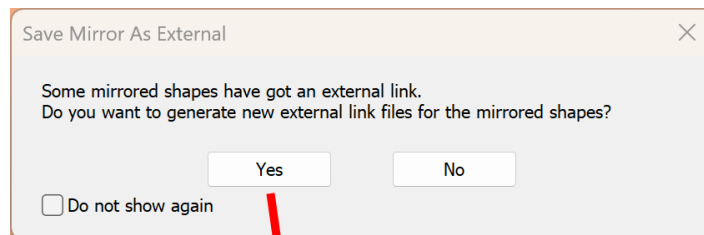
- **Benefits:**
  - Dramatically speeds up the selection of large numbers of pattern nodes or instances.
  - Makes editing and managing complex patterns (especially with many elements) significantly faster and more efficient.



## Save-As for Mirrored Externally Linked Part

Save-As dialog now prompts for custom file names when saving mirrored external parts, giving full control over mirrored component libraries.

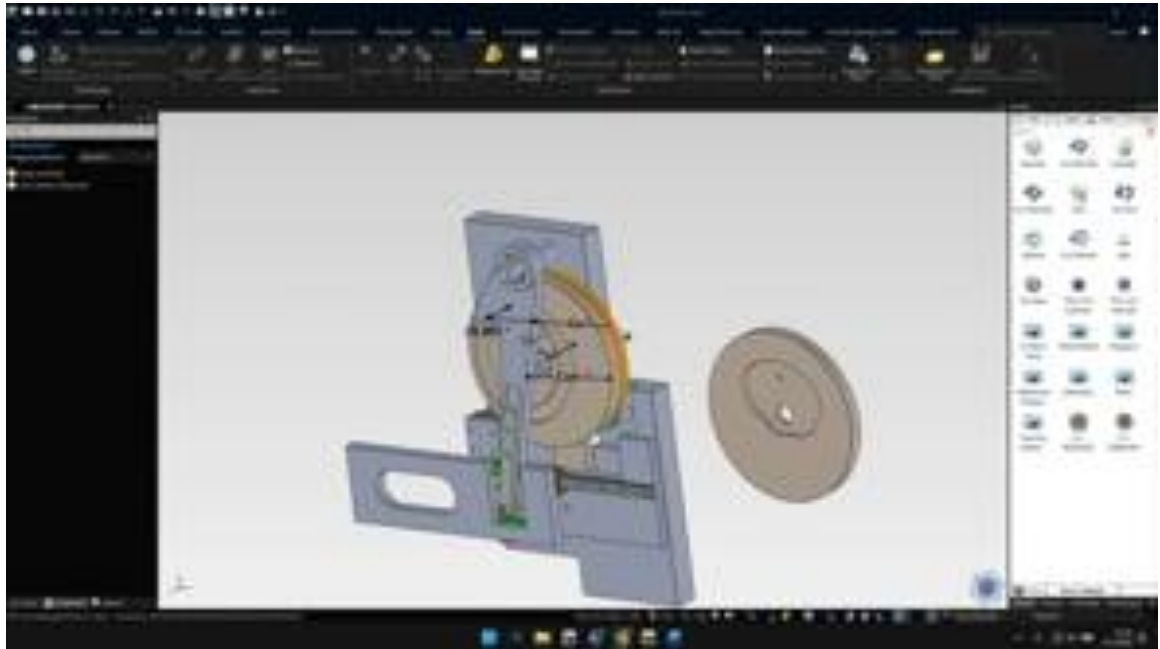
- **Benefits:** Tooling and modular manufacturers can efficiently manage left/right hand variants without file-naming conflicts, streamlining large product families.



## Cam Path Editing

Users can now directly edit Cam Paths that are generated from Motion Paths. This new capability allows designers to modify the shape, control points, and parameters of automatically created cam profiles after the motion simulation has been performed.

- **Benefits:**
  - Greater flexibility to refine and optimize cam profiles after initial motion analysis.
  - Saves significant time by allowing direct editing of generated cam paths instead of starting over.



## Alt Key to Add to Assembly

A new productivity enhancement has been added to part/assembly insertion workflows. When the “Automatic Add to Selected Assembly” option is disabled, users can now hold the Alt key while dropping a part or sub-assembly to selectively add it to the currently active assembly.

- **Benefits:**
  - Provides quick, on-demand control over whether a component is added to an assembly during drag-and-drop operations.
  - Eliminates the need to constantly toggle the automatic add setting on and off.
  - Significantly improves workflow efficiency when building large assemblies with a mix of top-level and nested components.



## Scene Browser Filters Remove External Linked Icons on Link Path

Removes excess custom icons for linked item paths in the scene browser for better clarity. This results in a much cleaner and more streamlined tree view, especially in large assemblies with many external links.

- **Benefits:**
  - Significantly improved readability and reduced visual clutter in the Scene Browser.
  - A more professional and organized workspace appearance.

## Improved Multiple Feature Part Performance in Linking

Significant performance optimizations have been implemented for assemblies containing large numbers of Feature Part links. This results in faster create, open, close, save, and update operations in deep assemblies.

- **Benefits:**
  - Dramatically improved speed and responsiveness when working with complex, heavily linked assemblies.
  - More reliable real-time updates across linked parts.
  - Reduced wait times, enabling smoother workflows on large-scale projects.

## Scalable Point Clouds

Point cloud support has been enhanced with new scaling tools. Users can now scale individual point clouds, selected groups, or all point clouds in the scene, with a one-click reset option.

- **Benefits:**
  - Easier integration and alignment of large scan data or reference point clouds into IronCAD scenes.
  - Improved usability when working with real-world captured data for reverse engineering or facility layout projects.
  - Greater flexibility to match point cloud scale with the design environment.

## Pattern Feature Tangent to Face

The Pattern Feature tool with the Edge Pattern now automatically detects and aligns patterned features to tangent faces. This intelligent behavior ensures correct orientation when patterning along edges.

- **Benefits:**
  - More accurate and predictable patterning results with minimal manual correction.
  - Better performance on both Innovative and Structured parts.
  - Saves time and reduces errors when creating patterned features on curved or complex geometry.

## Reverse Guide Curve

A new “Reverse Guide Curve” option has been added, giving users instant control over the direction of guide curves during feature creation (sweeps, lofts, etc.).

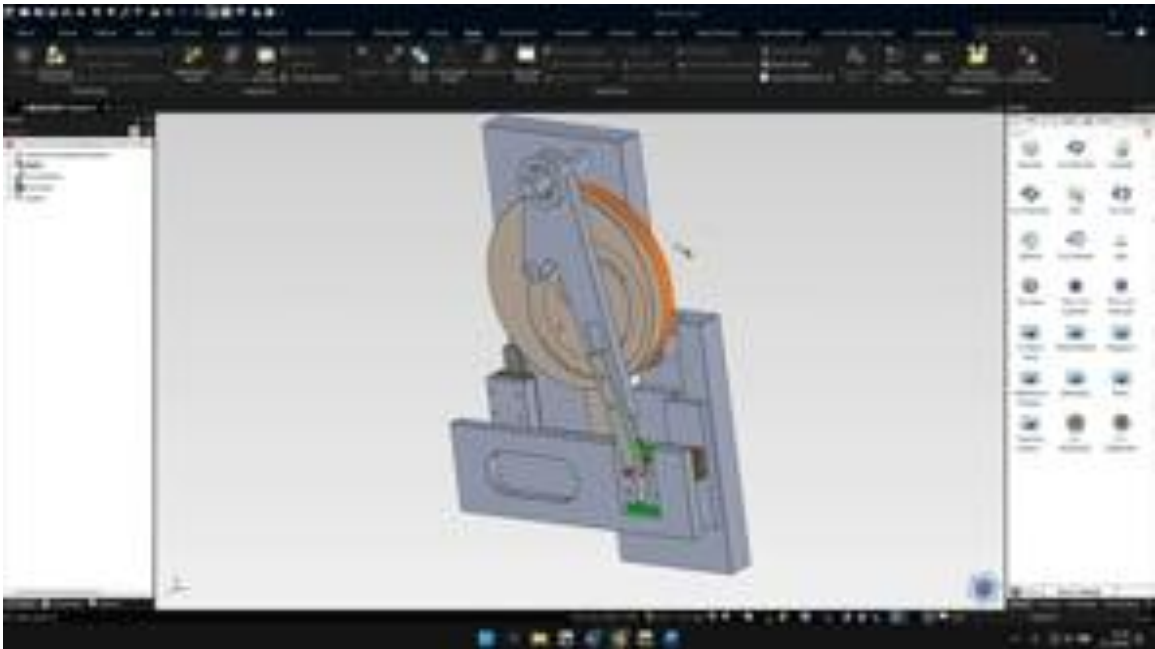
- **Benefits:**
  - Improved control and flexibility during advanced feature creation.
  - Eliminates time-consuming workarounds to achieve the correct feature orientation.
  - Faster and more intuitive creation of complex swept or lofted geometry.

## Sketch Enhancements

### Center Line Sketch Dimension

IronCAD 2027 introduces robust dimensioning tools for center lines directly in sketches. Users can now annotate diameters across a center line with full parametric control, dramatically streamlining workflows for spin features, lathe-turned components, and symmetrical parts common in industrial machinery shafts, tooling fixtures, and modular equipment frames.

- **Key Details/Parameters:**
  - Supports length constraints across the center line or Y-axis.
  - When the constraint crosses a center line, the displayed/parameter value is automatically doubled for true-diameter representation.
  - Choice of constraint type is selected at creation and locked thereafter for design intent preservation.
  - Fully applies to sketch “center lines” or the Y-axis.
- **Benefits:** This eliminates manual double-entry and calculation errors in symmetrical designs, ensuring precise, manufacturing-ready sketches that integrate directly into spin operations and large-scale assembly layouts.



## Offset Fill Gap by Extending Arc and Line

The offset command now automatically fills gaps by intelligently extending lines and arcs to intersect naturally (beyond simple arc solutions), ensuring clean, closed loops even with mixed geometry.

- **Key Details/Parameters:**
  - New “Extend lines and arcs at corners” option in Advanced Settings.
  - Handles arcs, lines, and mixed-geometry loops seamlessly.
- **Benefits:** Metal fabrication and modular product designers save hours of manual cleanup when creating offset profiles for sheet-metal enclosures, tooling plates, and structural frames.

## Sketch Text Improvements

New Place Text on Curve functionality has been added to the Sketch Text tool. Users can now select a curve, arc, circle, or spline in a sketch and place text that follows the path of the selected curve.

- **Benefits:**
  - Enables the creation of professional path-based text directly in sketches (great for labels, engravings, serial numbers, logos, and decorative elements).
  - Produces accurate, editable text that conforms to curved geometry without manual workarounds.
  - Text remains associative to the curve — when the underlying curve is modified, the text updates automatically.



### Offset Constraint Preference (Persist Value)

Offset constraint in Edit Cross Section now persists the last-used value as a user setting.

- **Benefits:** Repetitive workflows become faster and more consistent.

### External Reference Geometry for Sketches

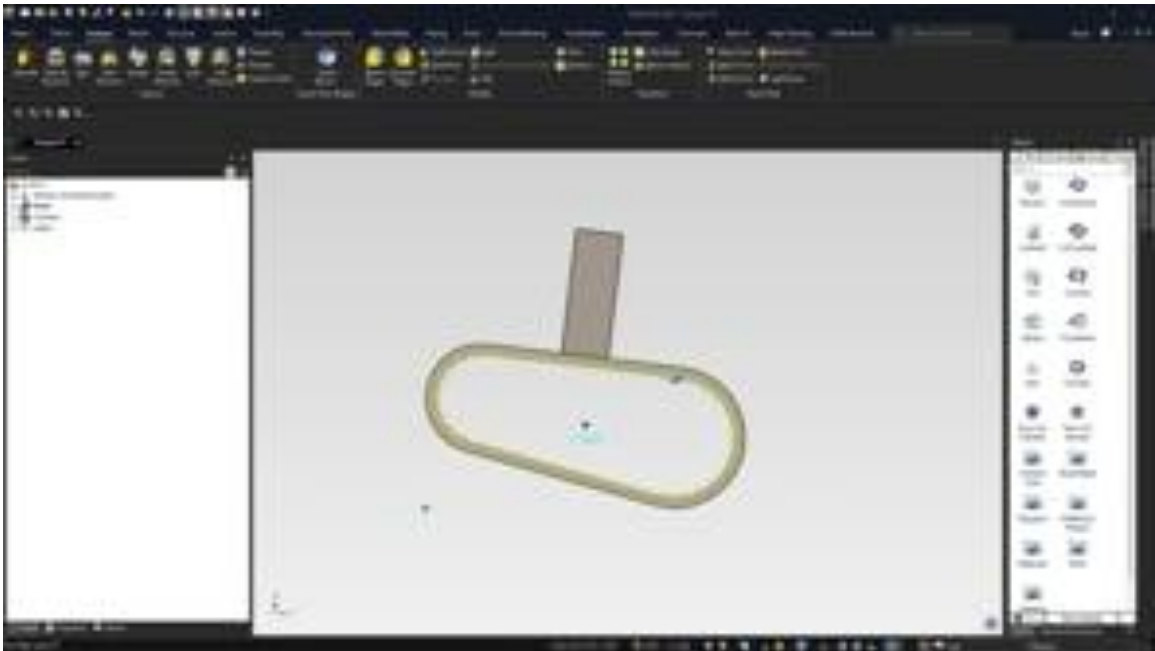
Sketches can now reference geometry from other parts (innovative or structured engineering mode).

- **Key Details/Parameters:**
  - Warnings for missing references or required open files.
- **Benefits:** Enables shared reference geometry.

### Mirror Constraint Interaction

Mirror axis now highlighted in magenta during sketching for immediate visual feedback.

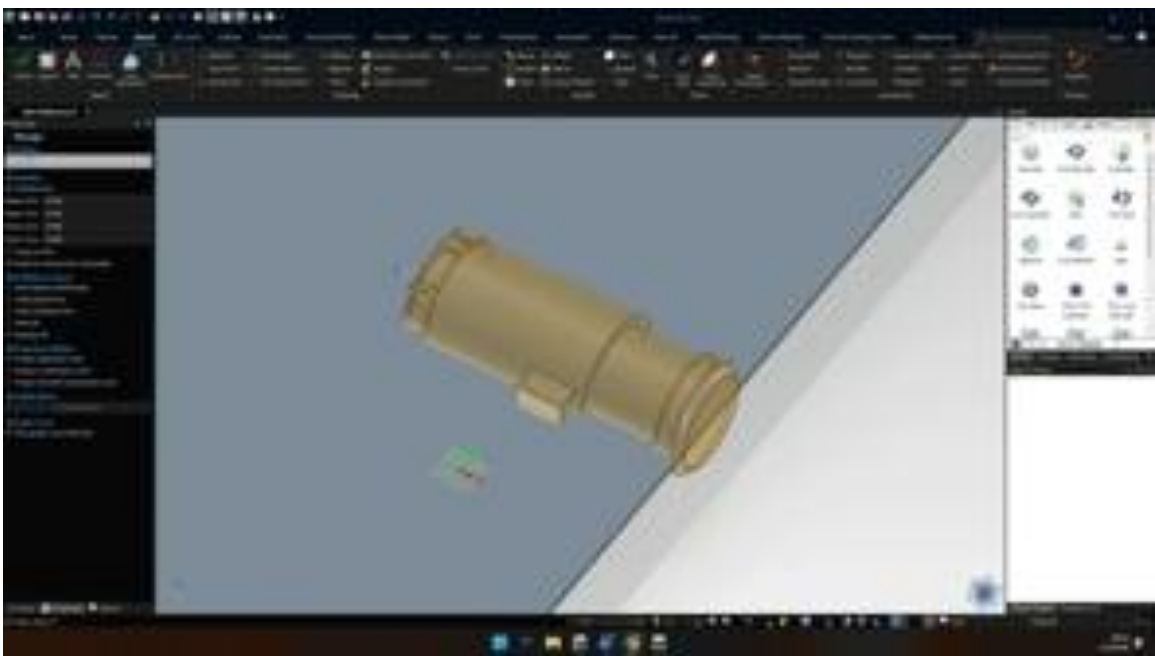
- **Benefits:** Faster, error-free creation of symmetrical machinery components.



## Penetrating Face Projection Tools

New projection tools for penetrating faces has been implemented. The Penetrating Face Project tool creates live intersection curves between solids/surfaces and the sketch plane.

- **Benefits:** Real-time design updates for complex tooling and fabricated structures.



## Spline Curve & Constraint Expansion

Fillet/radius now works on splines and intersections; new control-point constraints (coincide, fix, horizontal/vertical, midpoint); spline endpoint extension to lines/arcs.

- **Benefits:** Extended capabilities when working with Spline curves.



## Equidistant Point Geometric Constraint

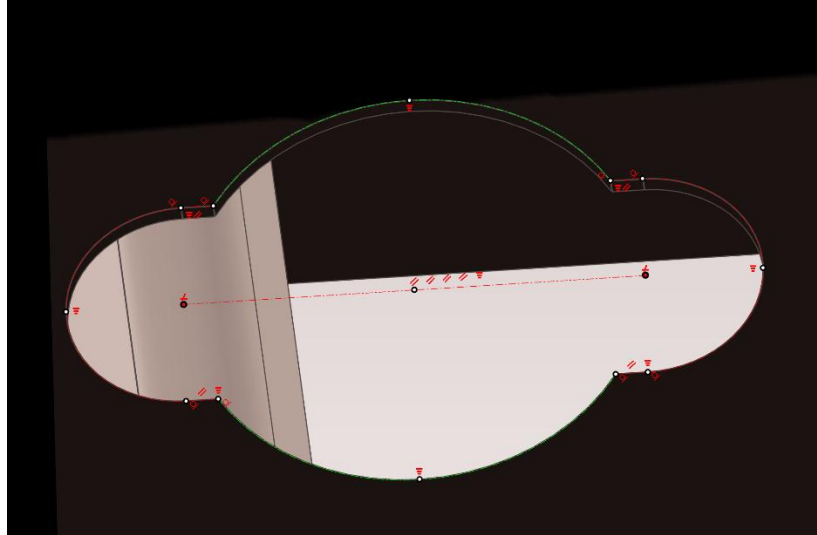
New constraint keeps multiple sketch points equally spaced along a line or direction.

- **Benefits:** Perfect for bolt-hole patterns, mounting arrays, and grid layouts.

## Construction and Center Point Display

Construction lines show constraint status by color; through-hole centers use prominent snap markers matching endpoint handles.

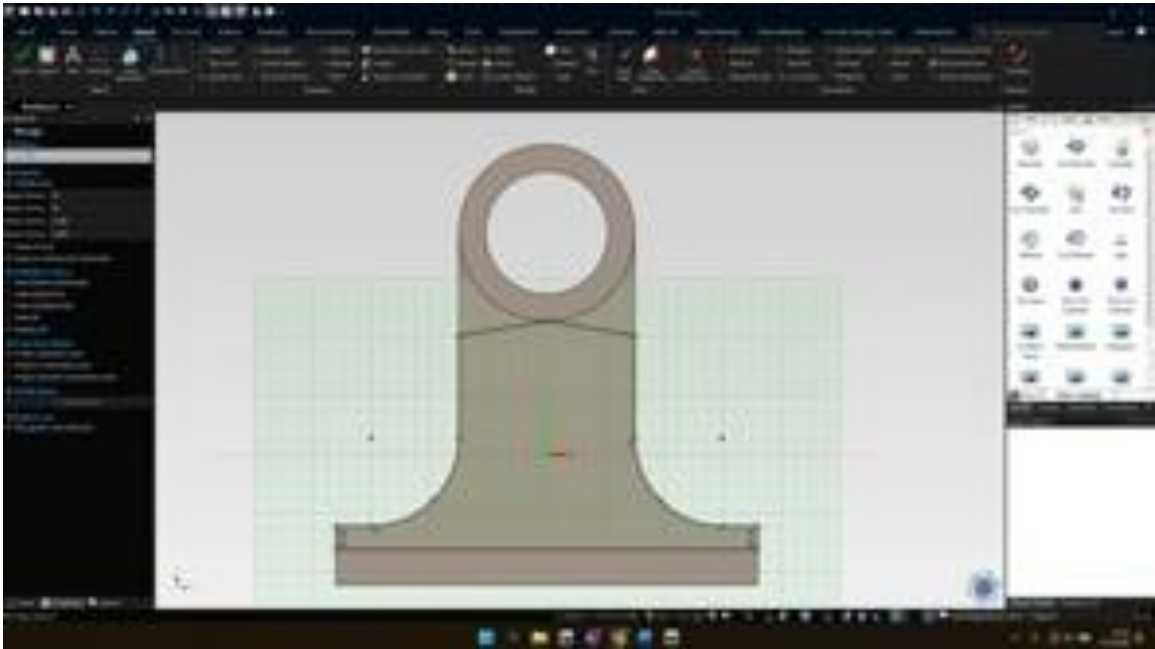
- **Benefits:** Clearer visual feedback during precision tooling and fabrication sketching.



## Sketch Constraint Interaction

Left panel automatically lists selected geometry and constraints for easy editing/deletion.

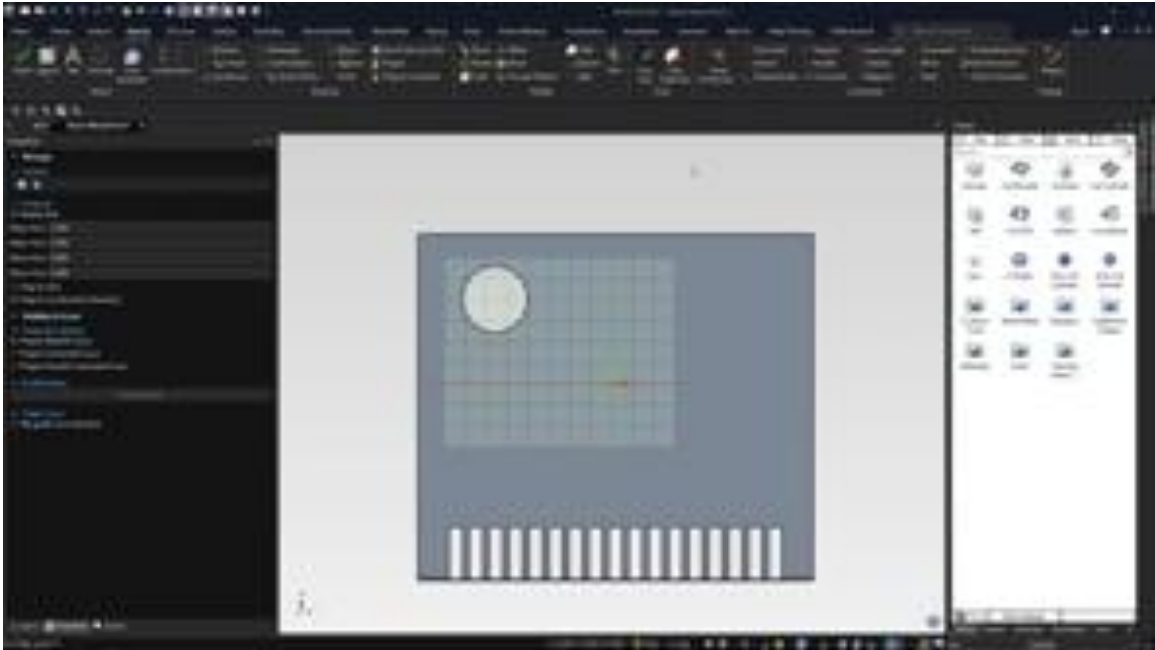
- **Benefits:** Accelerated constraint management in large, feature-rich sketches.



## Sketch Snap Circle Center Optimization

The sketch snapping behavior for circular edges has been significantly optimized. When selecting or hovering over circles, arcs, or cylindrical edges in sketch mode, the center point snap is now much more visible and responsive, with improved highlight and feedback.

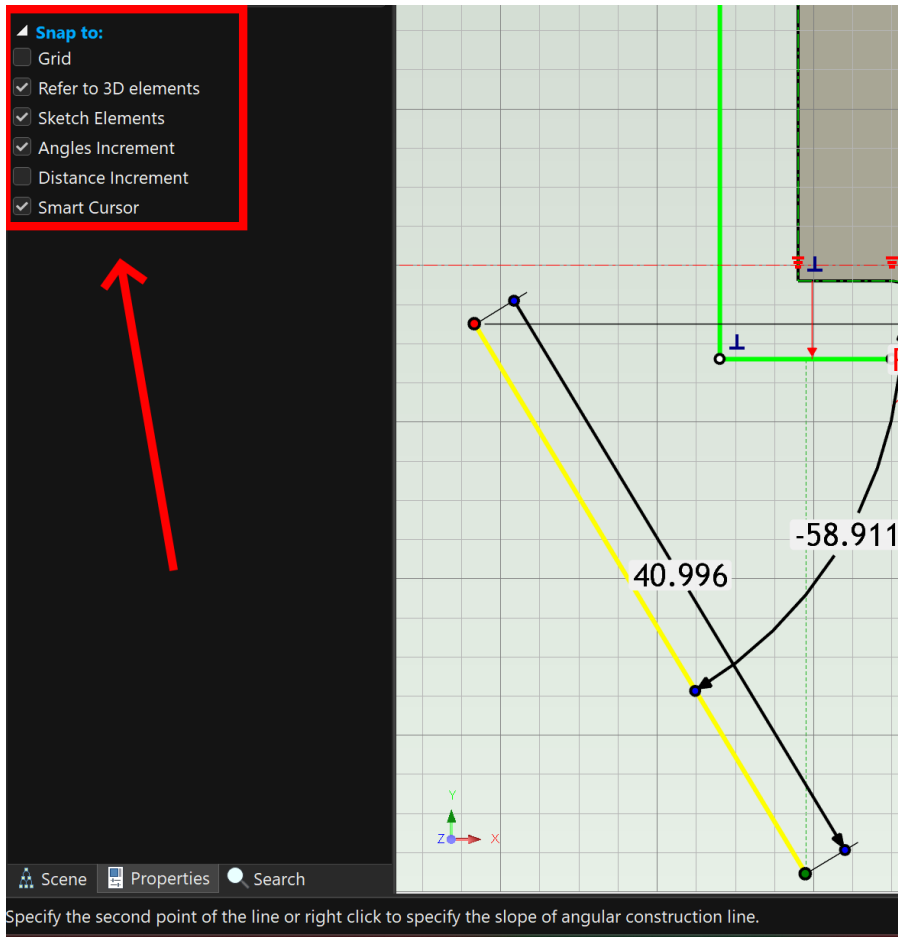
- **Benefits:**
  - Easier and more accurate snapping to circle centers during sketch creation and editing.
  - Better visual feedback reduces guesswork and eye strain when working with round geometry.
  - Faster sketch creation, especially when adding dimensions, constraints, or relations to hole centers and arc midpoints.



## Snap Options Side Panel

Snap settings now exposed in the sketch side panel for instant access and control.

- **Benefits:** Quicker, more precise sketching for fabrication and tooling profiles.



## Offset Command Remembers Constraint Setting:

The Offset Curve command has been improved to remember if the Offset Constraint option was selected. This helps reduce the repetitive task to uncheck this option if a user did not want to create a constraint during the offset.

- **Benefits:** Eliminates the repetitive need to uncheck the “Create Constraint” option every time a user wants a non-associative offset.

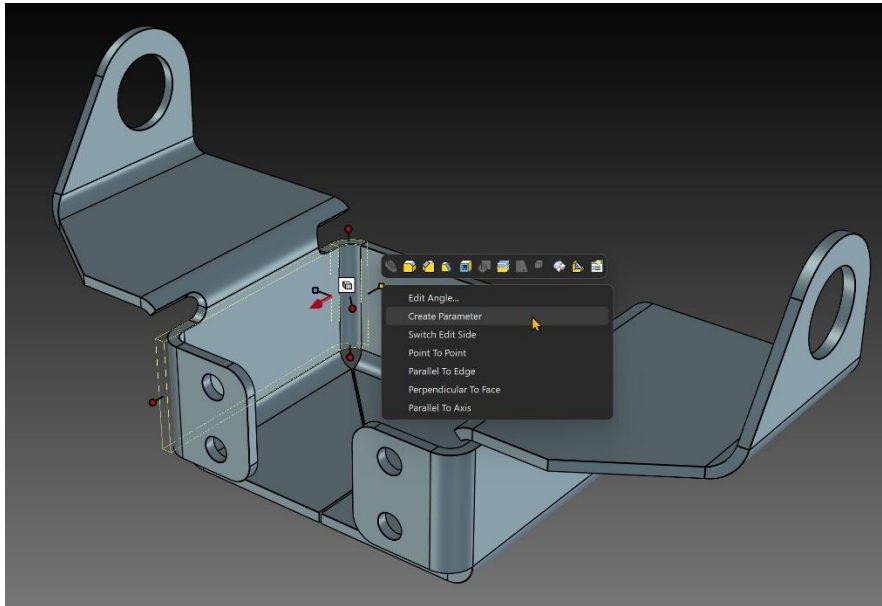
## Sheet Metal Enhancements

### Bend Angle Parameters:

New right-click functionality has been added to sheet metal bends, allowing users to quickly create parameters for the Bend Angle. This makes the bend angle fully parametric and editable through the parameter table or design variations.

- **Benefits:**

- Enables powerful parametric control over bend angles in sheet metal parts.
- Allows easy creation of design variations, configurations, and families of parts by simply changing parameter values.



## Bend Table Improvements

The use Bend Table in Sheet metal support Radius and now Angle to set the Bending Parameters of Bend Deduction, K-Factor and Unfold length. Users can dynamically set the Bending Parameters based on the Radius or based on an Angle value.

- **Benefits:**
  - Greater flexibility and accuracy when working with varying bend angles and radii in sheet metal designs.
  - Enables more realistic manufacturing-based calculations by selecting the most appropriate parameter (Radius or Angle) for different bends.

## Support for Unfold of External Mirror Sheet Metal Parts

Externally linked and mirrored sheet metal parts now fully support the Unfold operation, maintaining proper associativity with the original part.

- **Benefits:**
  - Allows users to create flat patterns from mirrored sheet metal components without breaking external links.
  - Maintains full design intent and associativity between mirrored parts and their flat patterns.

## Closed Corner Support on Mirrored Parts

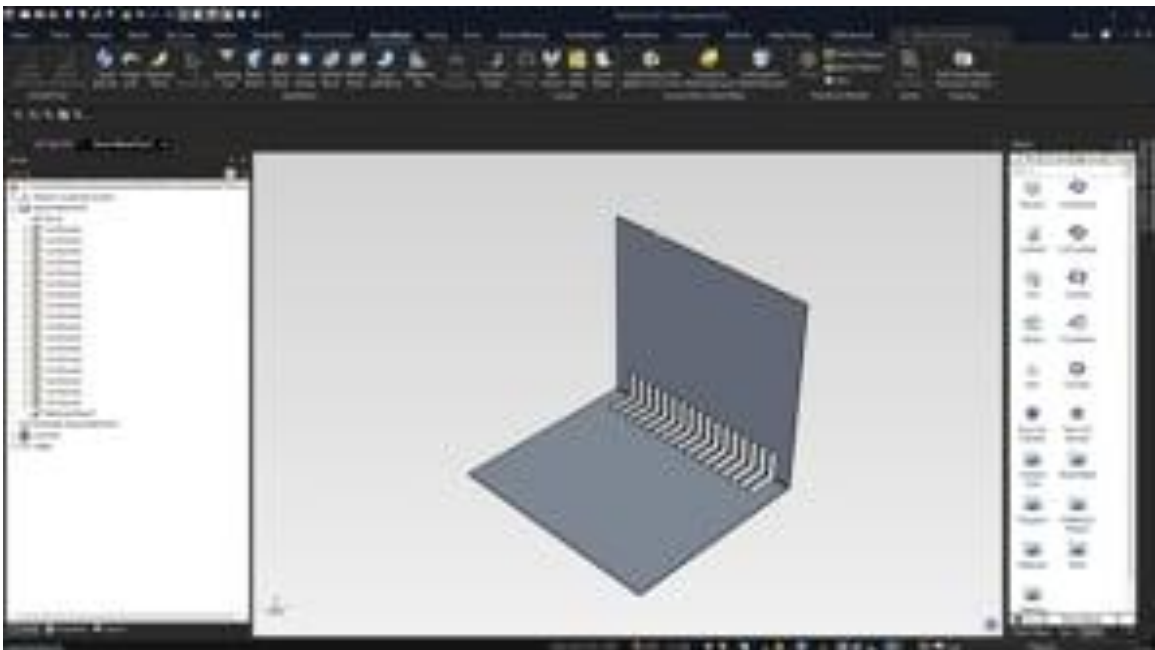
The Closed Corner tool now works seamlessly on both regular and mirrored sheet metal parts.

- **Benefits:**
  - Enables consistent use of the Closed Corner feature regardless of whether the part is in its original or mirrored state.
  - Improves design flexibility and usability when working with symmetrical or mirrored sheet metal designs.

## Bend Line Callout in IronCAD Drawing Consolidation

When creating bend lines with multiple segments in drawings, IronCAD now generates a single bend line callout instead of individual callouts for each segment.

- **Benefits:**
  - Significantly reduces drawing clutter and improves readability of sheet metal drawings.
  - Creates cleaner, more professional manufacturing drawings.
  - Saves time on manual cleanup and callout management.

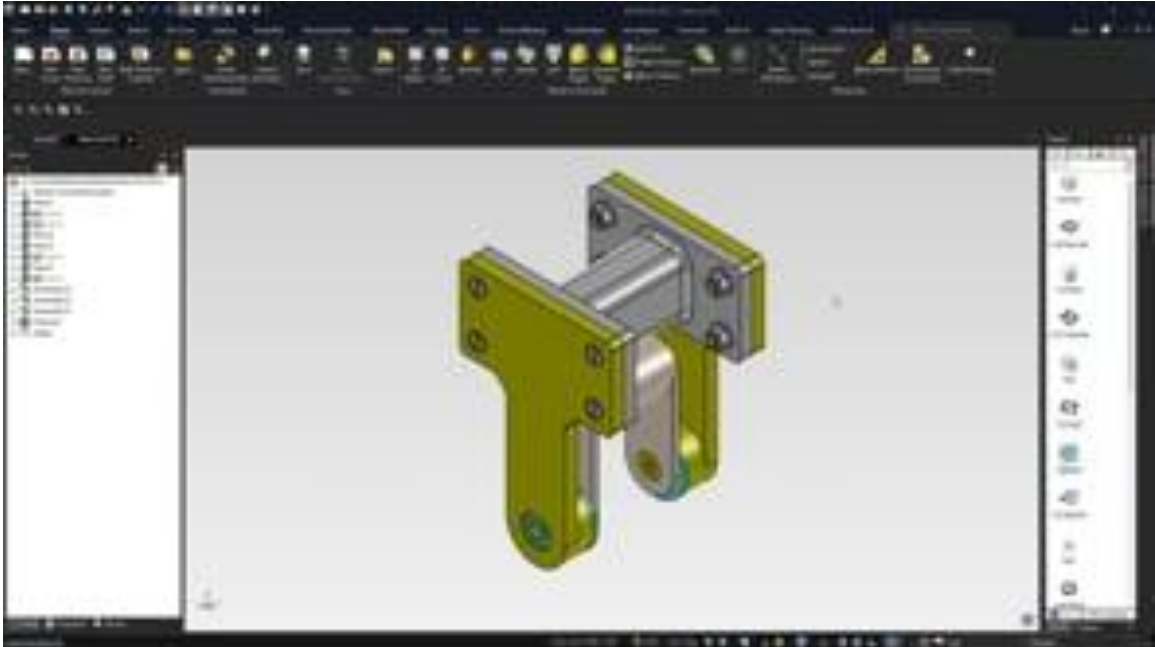


## ICD (IronCAD Drafting) Enhancements

### Use Bulk View Template in Create Drawing from Selected

New ability to use pre-defined bulk view templates to quickly layout a selected part/assembly has been added.

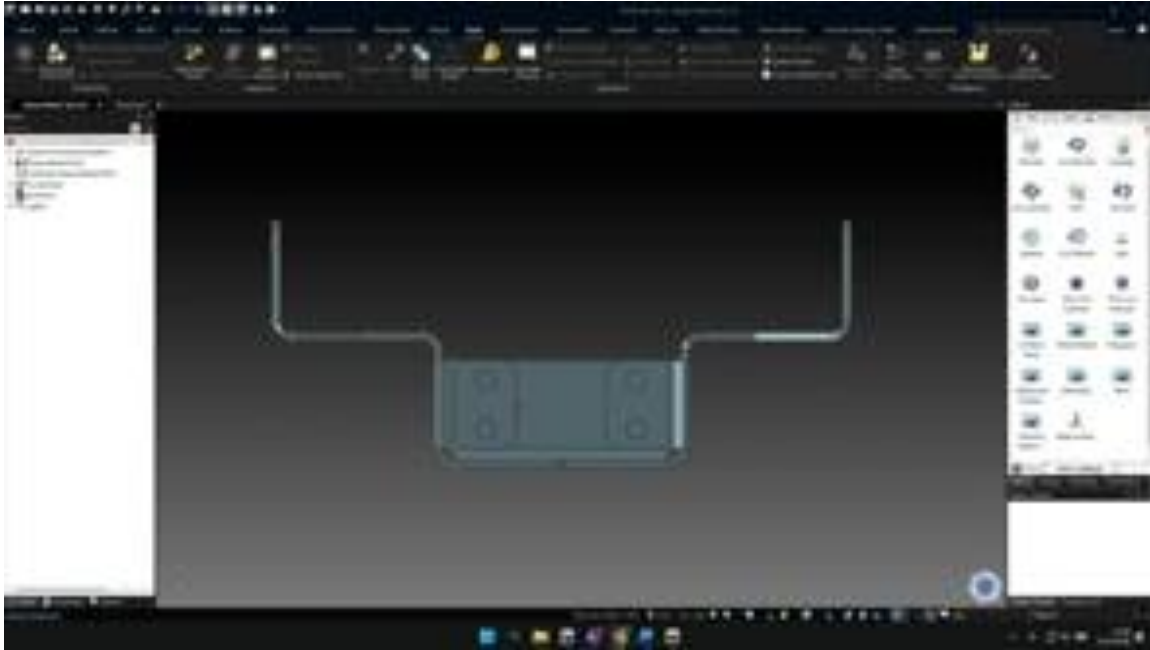
- **Benefits:** Reduce the manual view layout process to quickly create a new single selected template.



### Smart Annotation Improvements for Dimension Placement

Significant improvements have been made to the Smart Annotation system regarding automatic dimension and annotation placement. When the software detects that generated dimensions or callouts would appear in very tight or crowded spaces, it now intelligently moves them to more appropriate, readable locations.

- **Benefits:**
  - Cleaner and more professional-looking drawings by automatically avoiding overcrowded or overlapping annotations.
  - Reduced manual repositioning of dimensions after running Smart Annotation or Auto Dimension tools.
  - Better readability and clarity of drawings, especially on complex parts with many features in a limited space.



## Smart Annotation Allow Automatic Dimensions to Update

Automatic (Smart) dimensions can now be updated while preserving user-made adjustments. Users can make basic placement modifications — such as flipping arrow directions, changing inside/outside dimension placement, or adjusting text position — and these changes will be maintained when the dimensions automatically update due to model or geometry changes.

- **Benefits:**
  - Combines the power of smart/associative dimensions with the flexibility of manual placement control.
  - Eliminates the need to repeatedly reposition or fix dimensions after design changes.
  - Saves significant time during drawing updates while ensuring dimensions remain accurate and professionally placed.

## Support Hole Callout in Auto Dimension Tool

The Auto Dimension tool now includes full support for hole callouts. When using automatic dimensioning on drawing views, hole callouts are generated intelligently alongside standard dimensions.

- **Benefits:**

- Faster creation of complete, professional manufacturing drawings with automatic hole callouts.
- Reduced manual work when detailing parts with multiple holes.
- More consistent and standards-compliant drawings with less effort.

## Hatch - Solid Color Improvements

Users can now apply solid color fills to hatched faces in addition to traditional hatch patterns. The solid colors are exported as true solid fills in PDF/DWG export.

- **Benefits:**
  - Enhanced visual clarity and presentation quality in technical drawings.
  - Better ability to highlight specific materials, sections, or features.
  - More professional and readable drawings for customer reviews and manufacturing communication.

## Move Users Of Style To Other Style

A new utility allows easy reassignment or migration of users from one dimension/style setting to another.

- **Benefits:**
  - Streamlines drawing standard updates and style management across large drawing sets.
  - Saves significant time when company standards change or when cleaning up legacy drawings.
  - Improves consistency when transitioning between different drawing styles or templates.

## Dash Linestyle Improvement

Dashed line styles have been enhanced for better rendering, consistency, and visual quality across technical drawings and annotations. Specifically the dash linestyle can appropriately scale when the view scale is not 1:1.

- **Benefits:**
  - Cleaner, more professional appearance of hidden lines, centerlines, and other dashed elements.
  - Improved readability and print quality of drawings.
  - More consistent behavior across different view types and zoom levels.

## Display Thread and Hole Callout Independently

Enhanced callout flexibility in IronCAD Drafting (ICD) now lets users display thread and hole callouts independently via a simple checkbox. This removes previous rigid dependencies and provides cleaner, purpose-specific drawings for fabrication and assembly documentation.

- **Key Details/Parameters:**
  - New checkbox “Show Hole Data” on the Callouts page.
  - Unchecking it displays only the thread callout (ideal for threaded holes), regardless of underlying hole data.
- **Benefits:** In metal fabrication and tooling shops, this produces concise, shop-floor-ready drawings that focus exactly on threading requirements without visual clutter—speeding up quoting, CNC programming, and quality checks for industrial equipment components.

## Isometric View Linear Dimension for Piping

Users can now create linear dimensions directly on piping in ICD isometric views, including dimensions from 3D wire geometry. This adds full annotation power to complex piping layouts in industrial facilities and machinery.

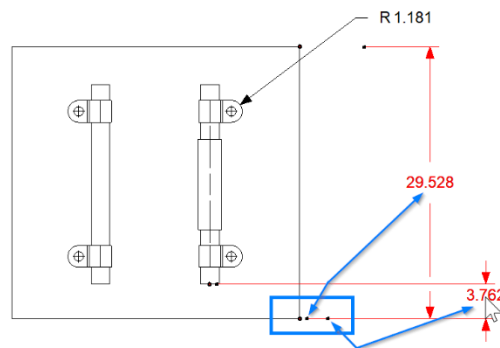
- **Key Details/Parameters:**
  - Supports multiple geometry pairings (edge, point, wire).
  - Multiple plane selection via Tab key for precise placement.
  - Full extension line and real-time preview support.
  - Dimensions annotate 3D wires, persist through model changes, and update automatically.
  - Shift Tab to select the line length or length to Virtual sharp of the connected bends.
- **Benefits:** Critical for assembly layout and facility design teams, this ensures accurate pipe routing documentation that remains live-linked to the 3D model—reducing field errors and rework in heavy industrial equipment installations.



## Mark Dimension Ends Independently

Linear and angular dimensions now allow independent selection and editing of start/end symbols (including dots), with per-symbol size control and full support in styles and layers.

- **Key Details/Parameters:**
  - Dedicated symbol pickers for start and end in property dialogs.
  - Individual size adjustment per symbol end.
  - Fully supported in the styles/layers manager and dimension dialog.
- **Benefits:** Fabrication and tooling manufacturers gain precise control over drawing clarity, enabling custom dimension styles that match company standards and improve readability on complex machinery drawings.



## Cosmetic Thread – Always Use Primary Line Type for Through Holes

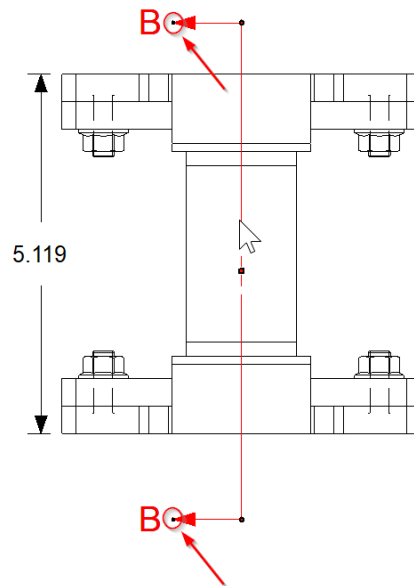
New option in cosmetic thread styles: “Always use primary line type for through holes.”

- **Benefits:** Produces consistent, standards-compliant drawings for threaded holes.

## Section Line Arrow Length Handle

New draggable handle on section lines for manual arrow-length adjustment.

- **Benefits:** Cleaner, standards-compliant section views for large machinery drawings.



## Export ICD with Multiple Sheets as Single DWG

New checkbox exports all sheets to a single DWG file containing multiple layouts.

- **Benefits:** Simplifies data exchange with fabrication partners and facility contractors who require consolidated drawing packages.



## Circular Hole Pattern Automatic Detection

New intelligence has been added to the Circular Hole Pattern annotation tool. The system now automatically detects and selects circular hole patterns (bolt circles, hole arrays, etc.) when creating annotations, greatly reducing the number of manual selections required.

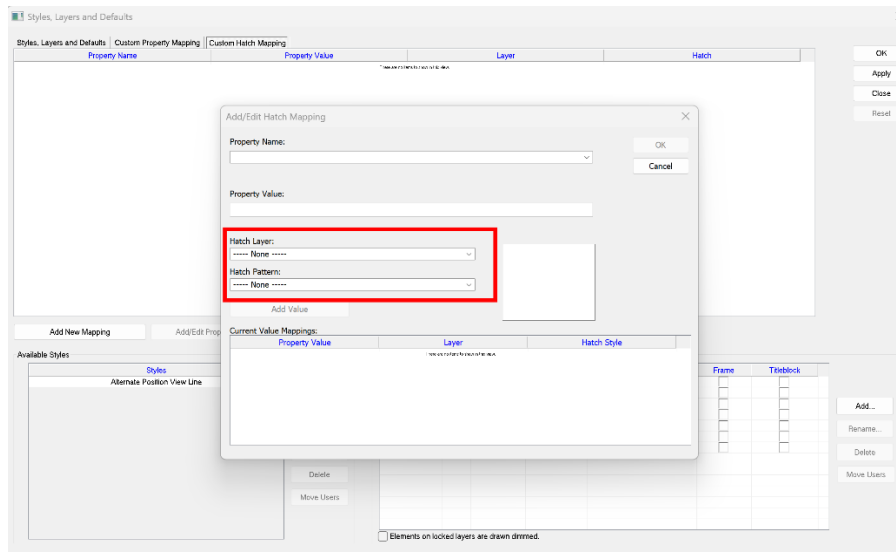
- **Benefits:**
  - Significantly faster creation of circular hole pattern callouts and dimensions with fewer clicks.
  - More intuitive and efficient annotation workflow when detailing parts with bolt circles or circular hole arrangements.



## Add Layer to Custom Hatch Mapping

Layer mapping now included in custom hatch mappings (fully backward-compatible with older templates).

- **Benefits:** Greater control over drawing standards for metal fabrication and industrial documentation.



## Remove Linked Scene from Drawing

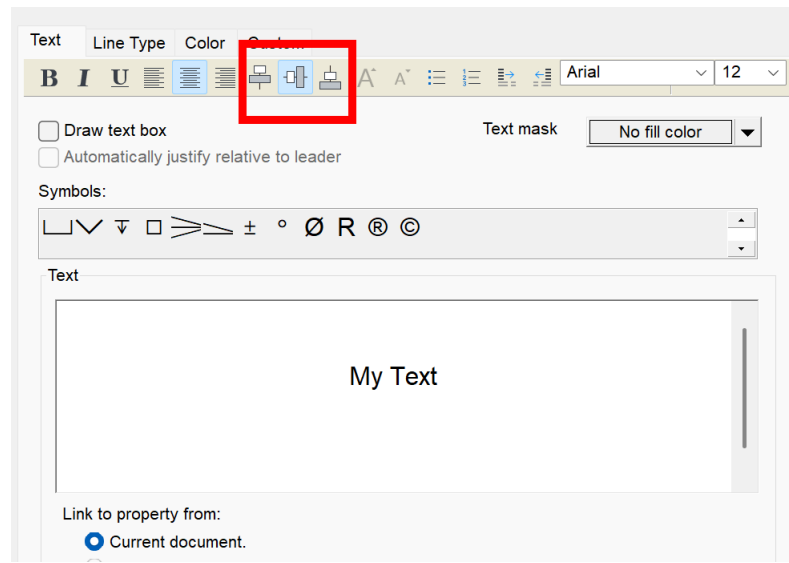
“Change Source” for scene links intelligently merges multiple uses onto a single file link when already referenced.

- **Benefits:** Reduces drawing bloat and link management overhead in large facility layouts.

## Text Box Vertical Alignment

Top, middle, and bottom alignment options now available in the ICD text box dialog.

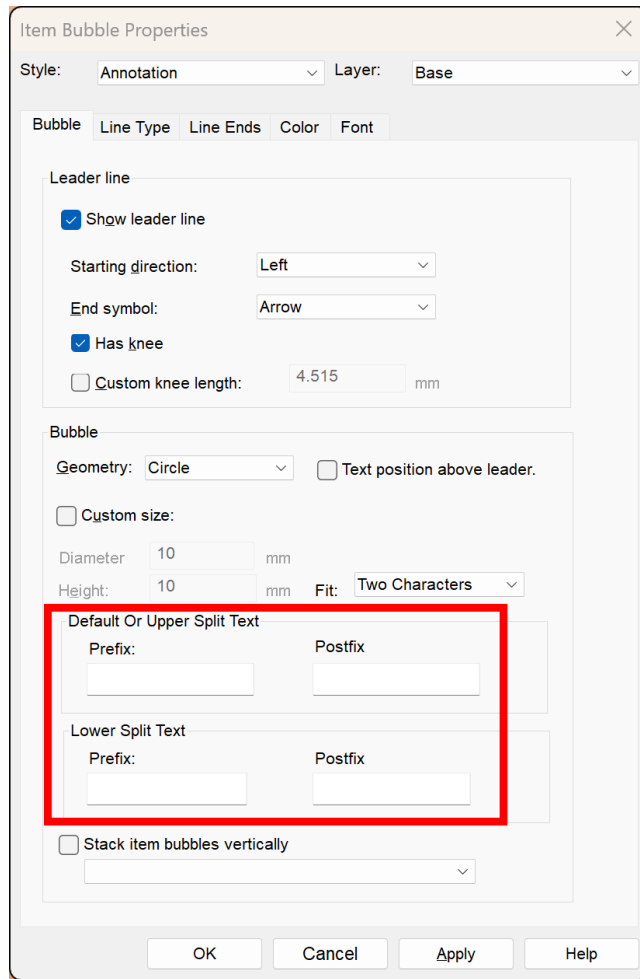
- **Benefits:** Professional, readable annotations in crowded machinery drawings.



## Item Bubble Prefix and Postfix

Prefix/postfix support for item bubbles in BOM/balloons (e.g., “x3”, “qty:”).

- **Benefits:** Clearer bill-of-materials callouts for modular product and assembly documentation.



## Technical Drawing Browser – Hide Icon

Hidden items/assemblies now show grayed-out icons in the TD browser; icons reset when unhidden.

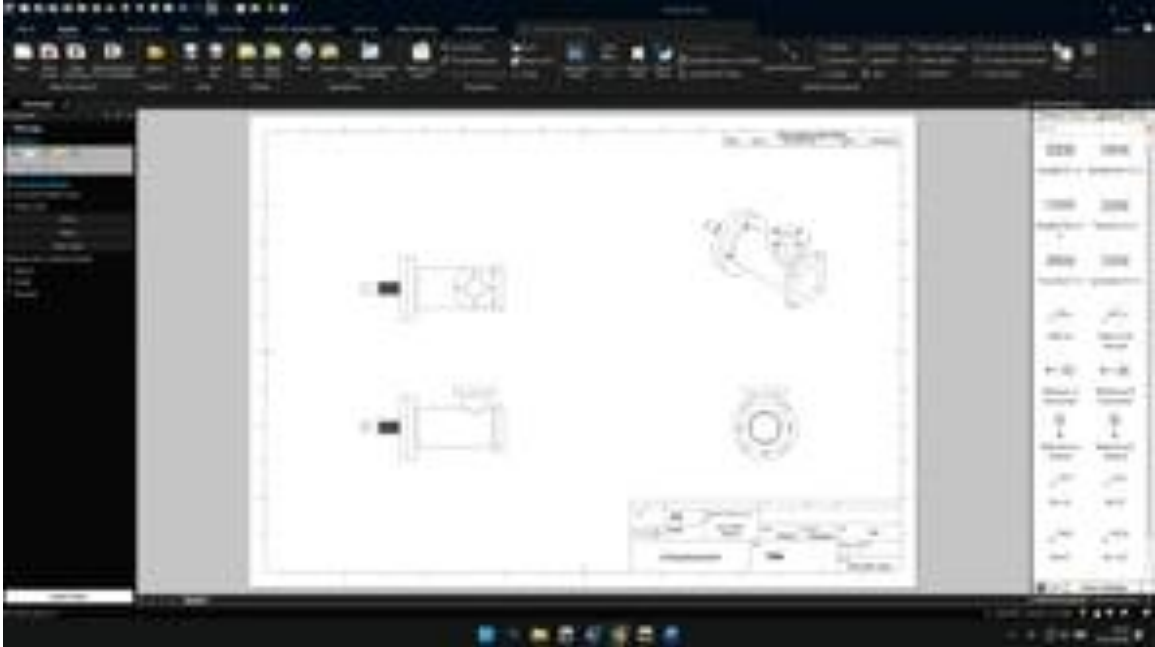
- **Benefits:** Improved visibility management in complex multi-level industrial assemblies.

## Copy View Maintains Alignment

When copying drawing views that have dependent or aligned views (such as auxiliary, detail, or section views), the alignment relationships are now automatically maintained on the copied views.

- **Benefits:**
  - Saves significant time when duplicating sets of aligned or related drawing views.

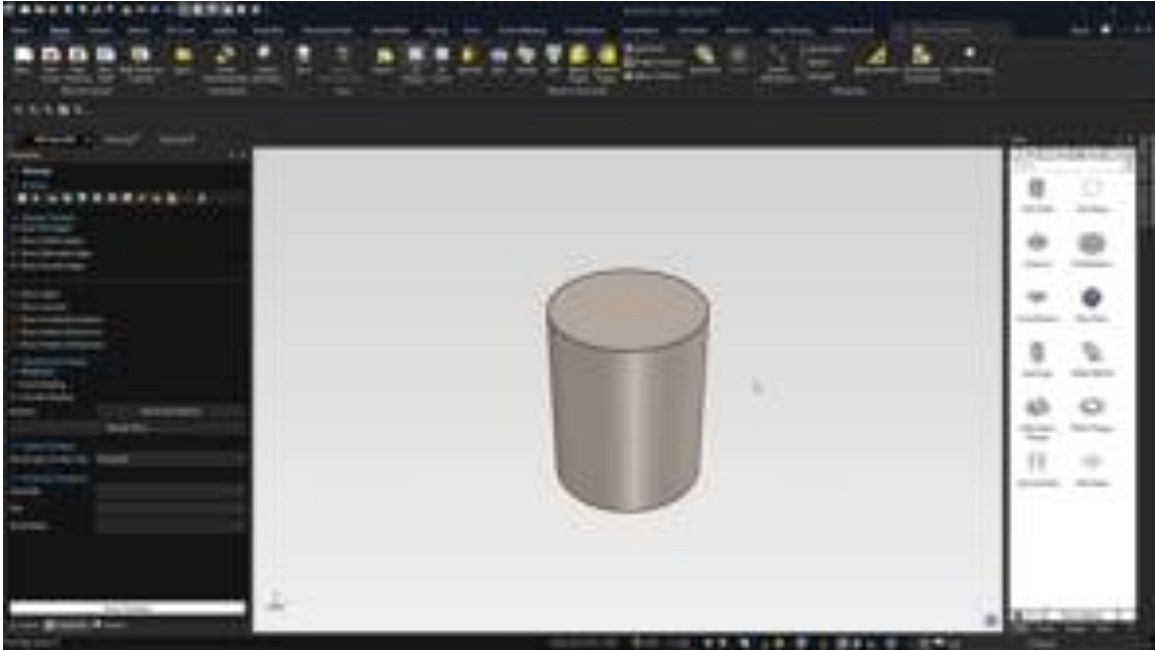
- Maintains proper orthographic alignment and projection relationships automatically after copying.
- Eliminates the need to manually realign copied views, which was previously a tedious and error-prone task.



## Geometry HitTest Improvement & 3D Wire Dimensioning

Faster hit-testing and full support for dimensioning 3D wires (pipe centerlines) in ICD.

- **Benefits:** Accelerated dimensioning of piping and wireframe elements in industrial facility drawings.



## Dimension Arrow Types

New expanded settings have been added to give users greater control over dimension arrow styles. Users can now select from a wider variety of arrow types and configurations for linear, radial, angular, and ordinate dimensions.

- **Benefits:**
  - Increased flexibility to match different company standards, customer requirements, or industry-specific drawing conventions.
  - Ability to use more appropriate and professional arrow styles (e.g., filled, open, dot, slash, architectural ticks, etc.).

## 3D BOM Enhancements

### Expression Bar & Format Precision

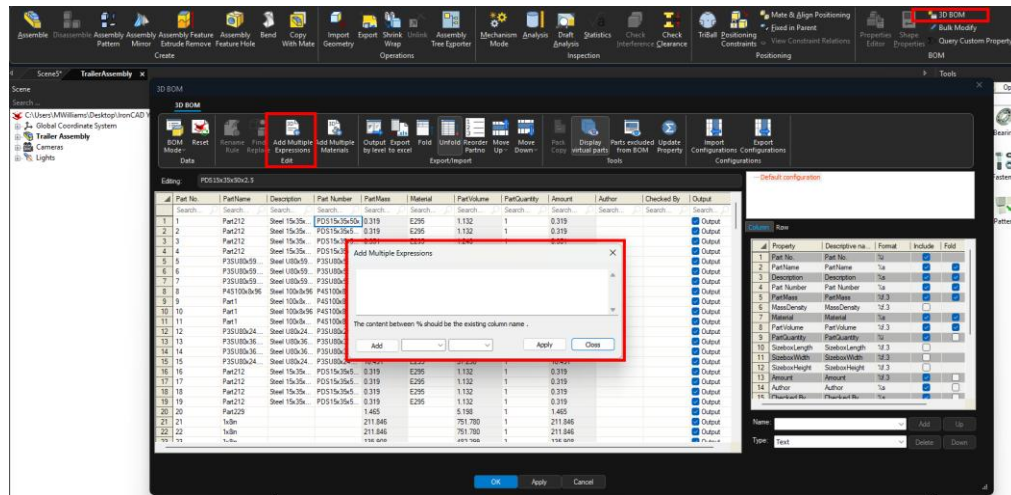
New horizontal expression edit bar above the BOM table, plus per-column display precision formatting (%s, %i, %f.N).

- **Benefits:** Rapid, accurate BOM customization for cost estimation and manufacturing in industrial equipment projects.

### Batch Add Expressions in 3DBOM

Batch-apply expressions to entire columns via a dedicated dialog; changes shown in red until saved.

- **Benefits:** Massive time savings when updating large BOMs for modular machinery families.



### 3DBOM and 2DBOM Bidirectional Association

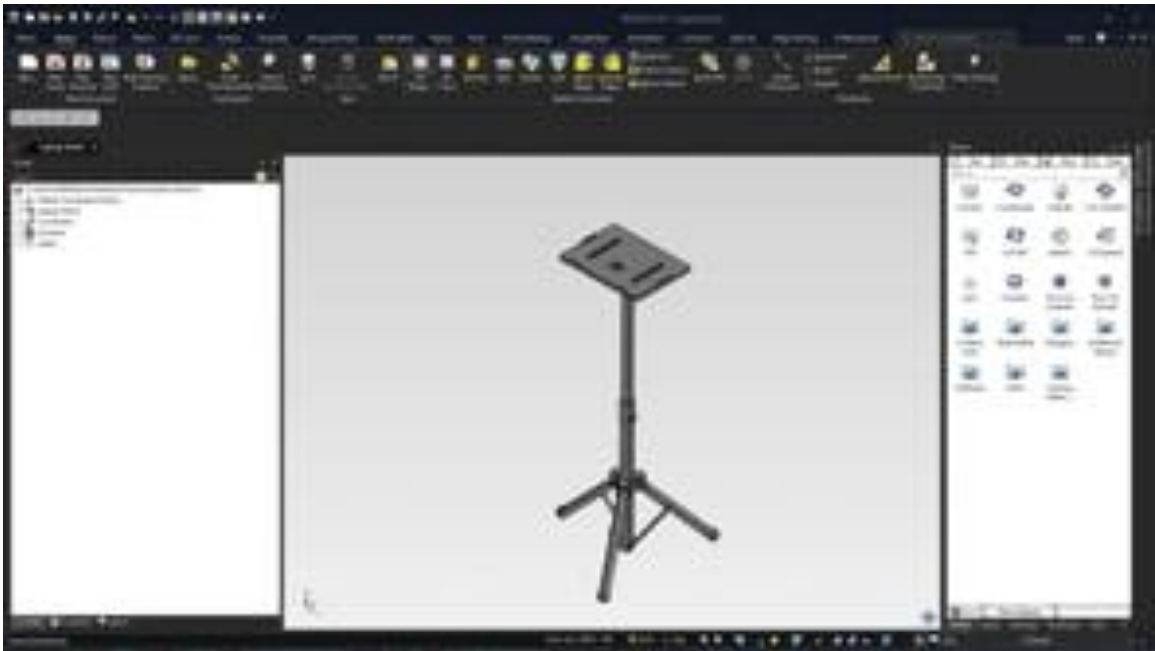
Full two-way synchronization between 3DBOM and 2DBOM for attributes, order, folding, hierarchy, and mirror status.

- **Benefits:** Eliminates data discrepancies between 3D and drawing environments—critical for accurate fabrication and assembly documentation.

### Batch Renaming in Save

External filenames automatically match tree/name display; prefix/suffix options added during packaging. New option in Tools/Options/General to enable the automatic sync.

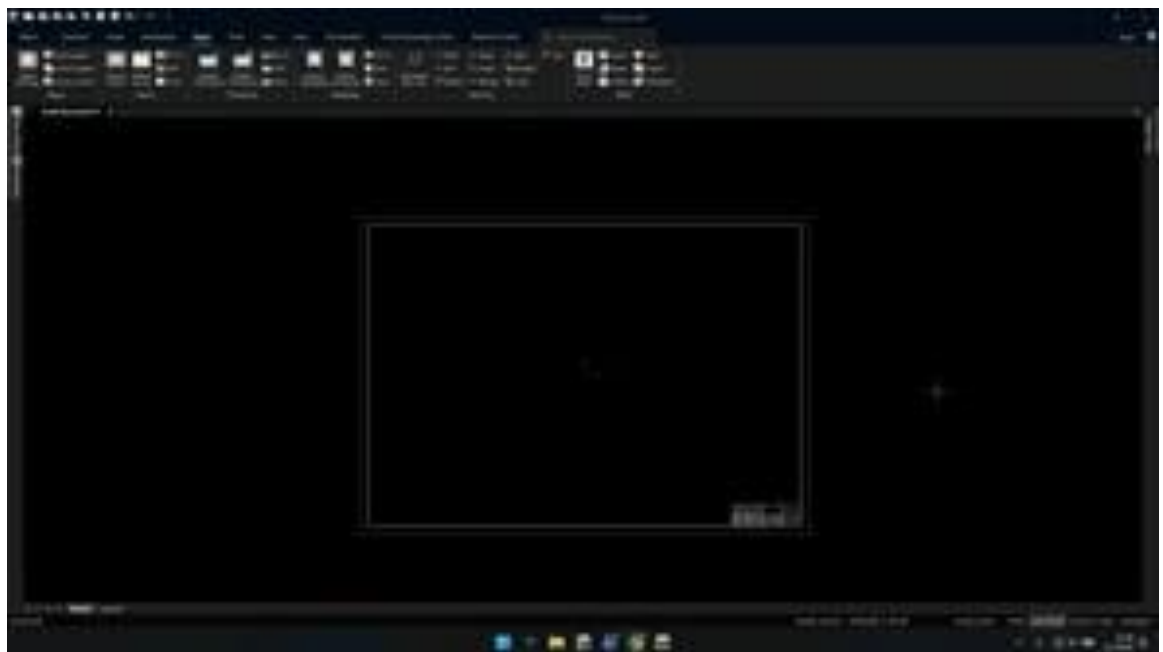
- **Benefits:** Organized file management for large modular product releases.



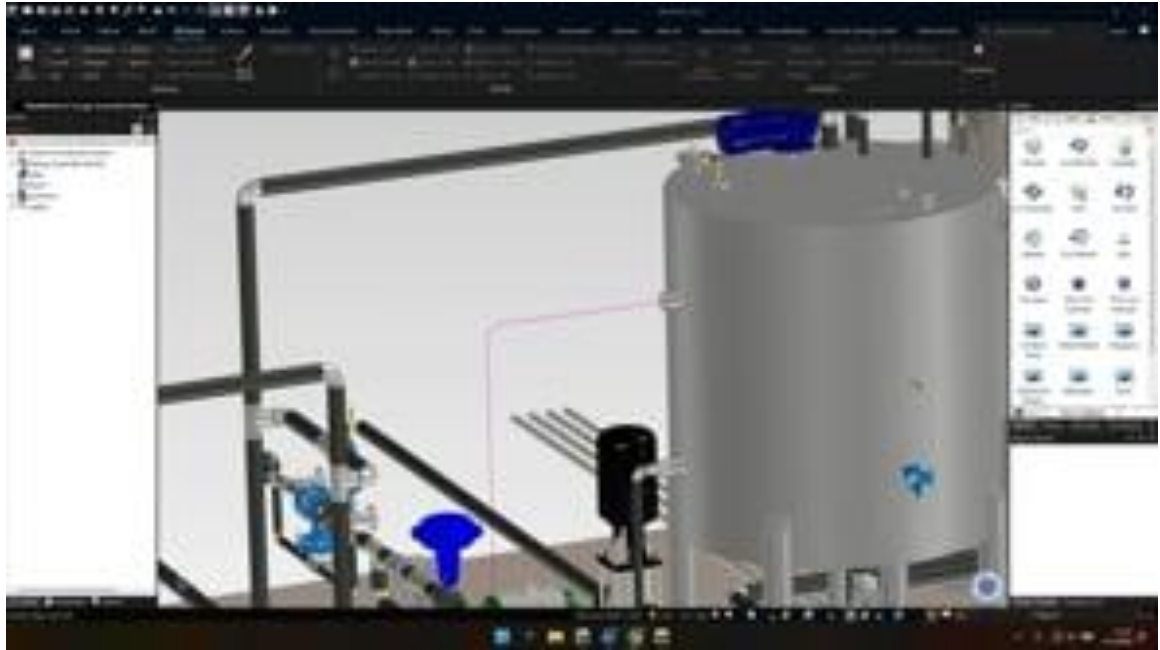
## IronCAD DRAFT Environment

### Projection Enhancements

- **Projection Status Notification** New icons show if a part has already been projected in the current drawing to help reduce replication.

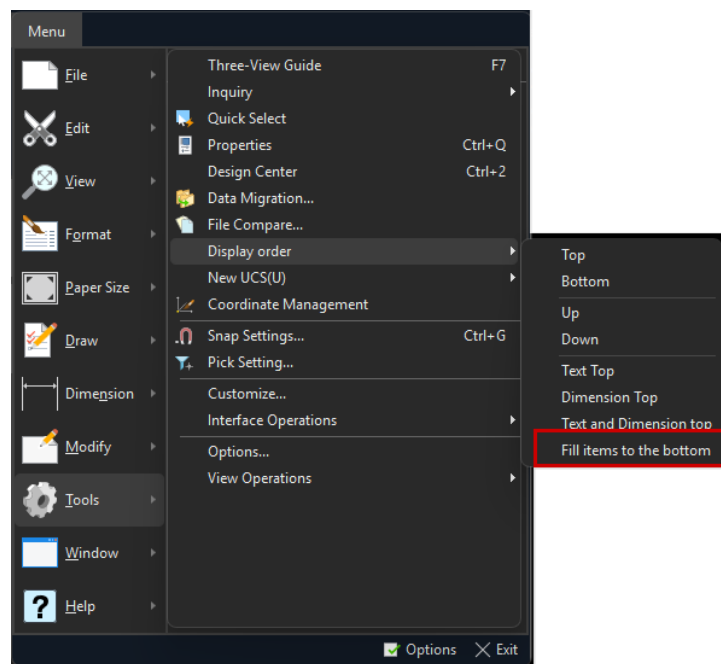


- **3D Curve Dimensions Projection** 3D Curve Dimensions can now be projected to the DRAFT environment.

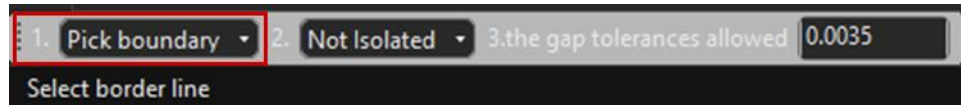


## Fill and Hatch Enhancements

- **Fill Items to Bottom:** New command that quickly moves all fill-related objects (including section lines, hatches, gradients, and similar elements) to the bottom of the drawing order.

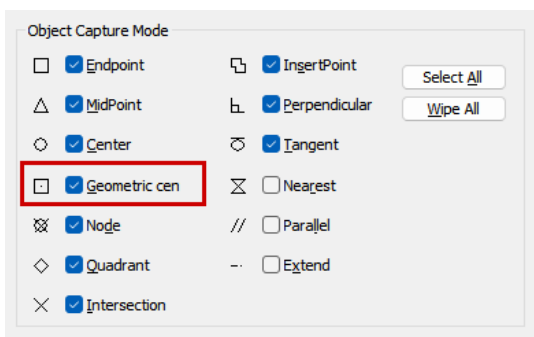


- **Enhanced Fill Functionality:** A new “Pick Boundary” method has been added. Users can now select one or more closed entities to generate fills, providing greater flexibility and speed.



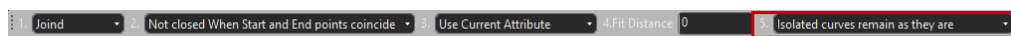
## Drawing and Editing Tools

- **Rounded Corners:** Now supports rounding corners between rays, construction lines, and regular lines.
- **Object Snapping:** Added “Geometric Center” (centroid) snap option, which works on closed splines and polylines. The maximum number of segments considered can be controlled by adjusting the system variable PLINEGCENMAX (range: 0 to 10,000,000).



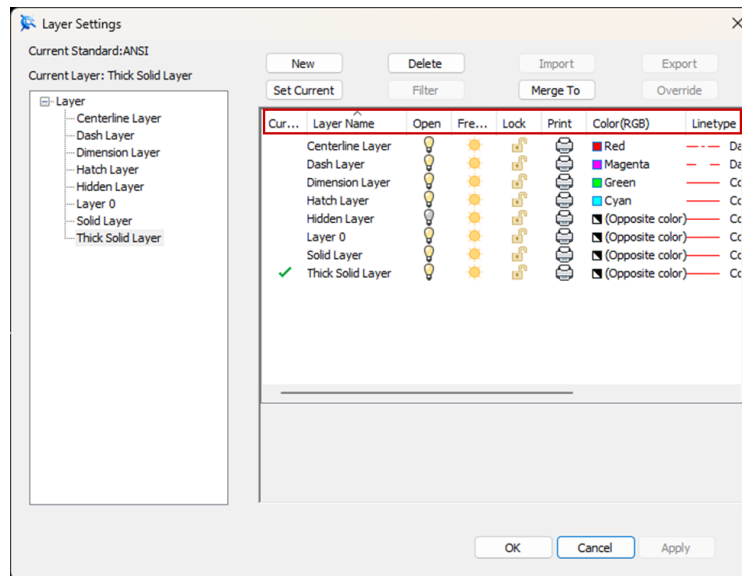
## Polyline Improvements:

- Polyline creation from data now supports polar coordinate input. Example formats: 1,1 (absolute), @2.5,1 (relative Cartesian), @3<45 (relative polar).
- New keyboard commands PEdit and PE for polyline editing.
- Enhanced merge functionality with an option to automatically convert isolated curves into polylines.



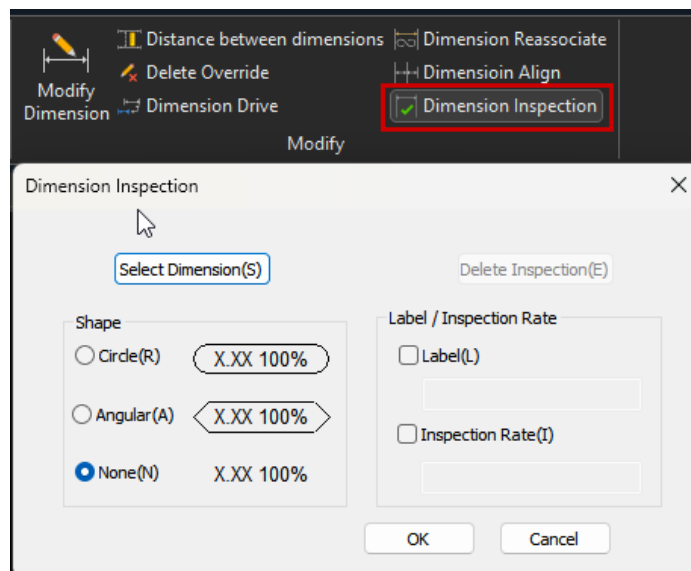
## Layer Management

- The Layer dialog has been improved with clickable column headers, allowing sorting by Current, Layer Name, On/Off, Freeze, Lock, Print, Color, Linetype, Lineweight, and Description.

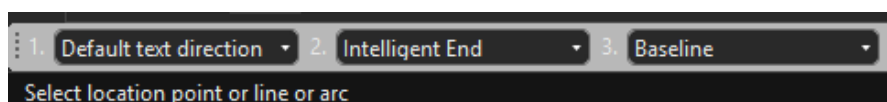


## Annotation and Dimensioning

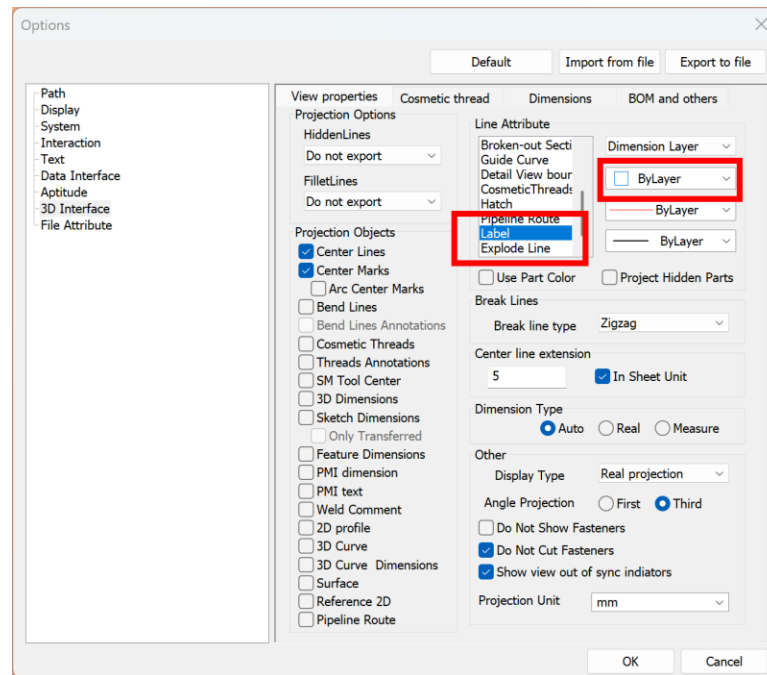
- Annotation Verification (Inspection Symbols):** Supports adding inspection symbols to selected dimensions, including inspection labels and rates. This can be done via a dedicated tool or directly through the Properties panel.



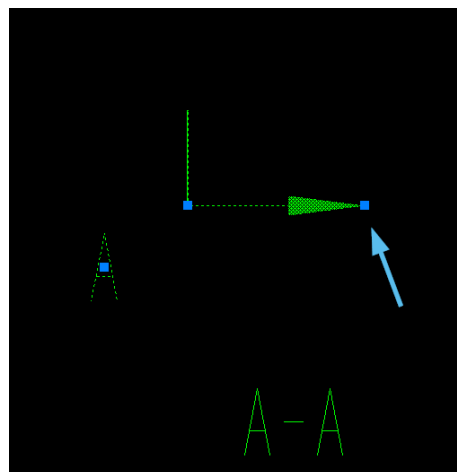
- Plate Thickness Annotation:** New tool that lets users pick two points to automatically measure and create a plate thickness callout. Supports both “t” and “δ” symbols. Text direction and leader line style can be set from the contextual menu.



- **Geometric Tolerance (GD&T):** Optimized vertical alignment of multi-row tolerance frames, including better handling when tolerance 2 is enabled or when multiple datum references are used.
- **View Label Layer Setting:** View labels can be set to specific layers to control color and visibility.



- **Section Symbol Improvements:** Added new clamp-style grips for easier adjustment of leader length. Vertical label position now adjusts automatically.

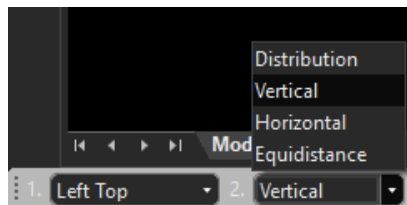
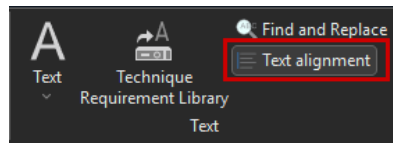


- **Smart Deviation Adjustment:** Dimensions (linearity, radius, diameter, and angle) now support automatic text spacing adjustment based on upper and

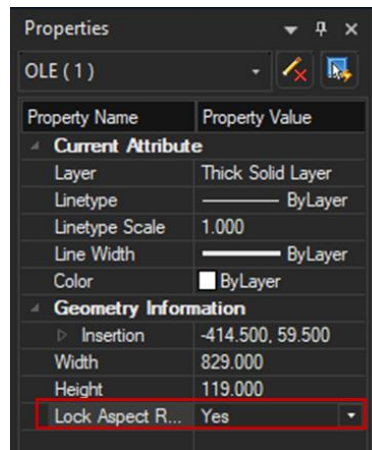
lower deviation values. This behavior can be toggled using the system variable AutoAdjustDev (0 = disabled, 1 = enabled).

## Text and Object Tools

- **Text Alignment:** New TextAlign command and menu option. Allows quick alignment of multiple selected text objects according to chosen alignment methods.

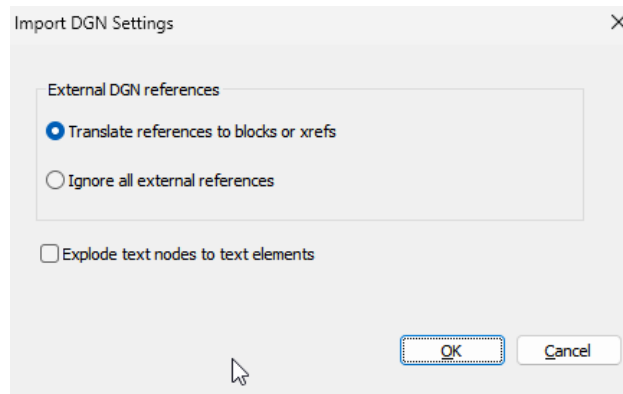


- **Text Columns:** Improved compatibility with DWG files and expanded support for multiple image formats in text columns.
- **OLE Object Enhancement:** When an inserted OLE object is selected, users can now lock the aspect ratio directly in the Properties panel.

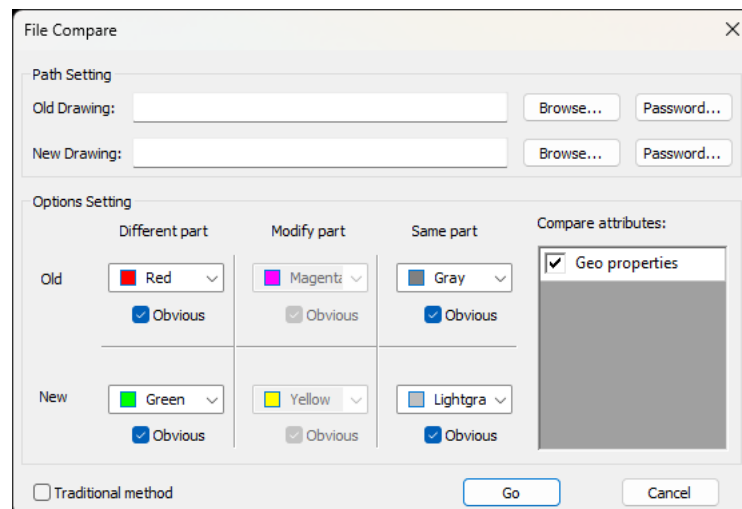


## File Handling and Compatibility

- **DGN Format Support:** Added ability to export files in DGN V8 format. Import now supports external references and improved text handling.



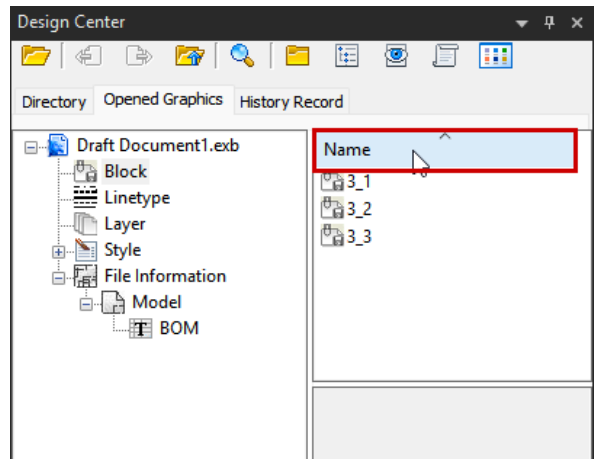
- **Image Insertion:**
  - Direct paste support for common image formats (JPG, JPEG, PNG, BMP, GIF, ICO, TIF, TIFF, DIB).
  - Drag-and-drop functionality for both images and PDFs.
  - Controlled by system variables: ImgDropMode (0/1) for images and PDFDropMode (0/1/2) for PDFs.
- **Enhanced File Comparison:** The comparison details table now supports clear row-by-row comparison.



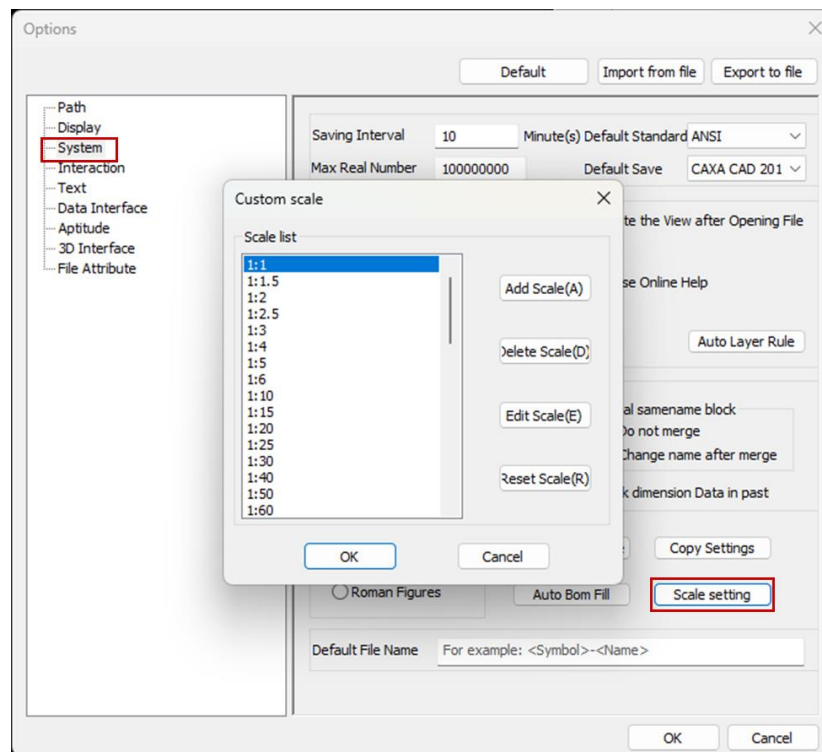
- **Selective Paste:** Excel spreadsheets can now be selectively pasted directly as native DRAFT table objects.

## User Interface and Workflow Improvements

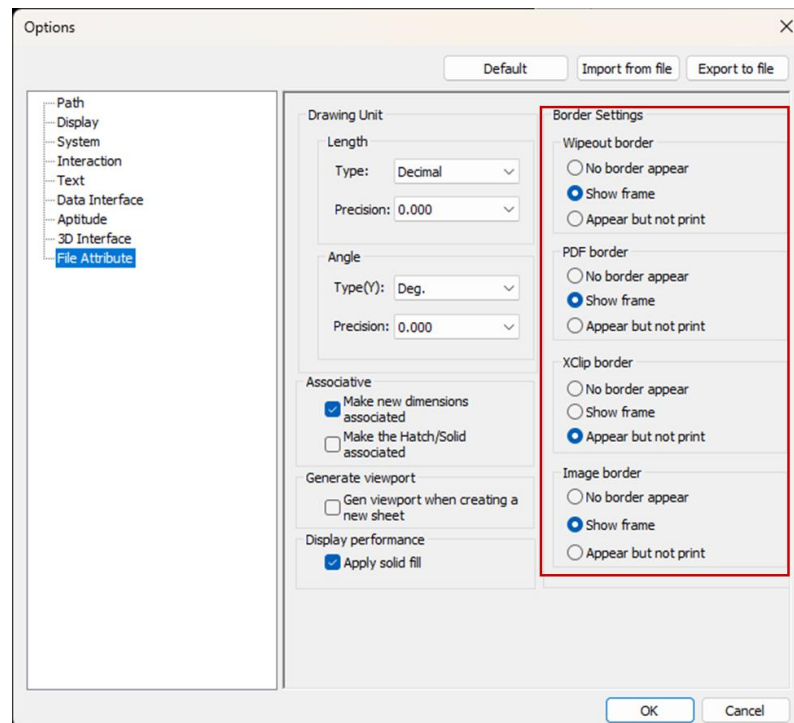
- **Design Center:** Object display order changed to dictionary-based sorting. In list mode, users can now sort by clicking column headers.



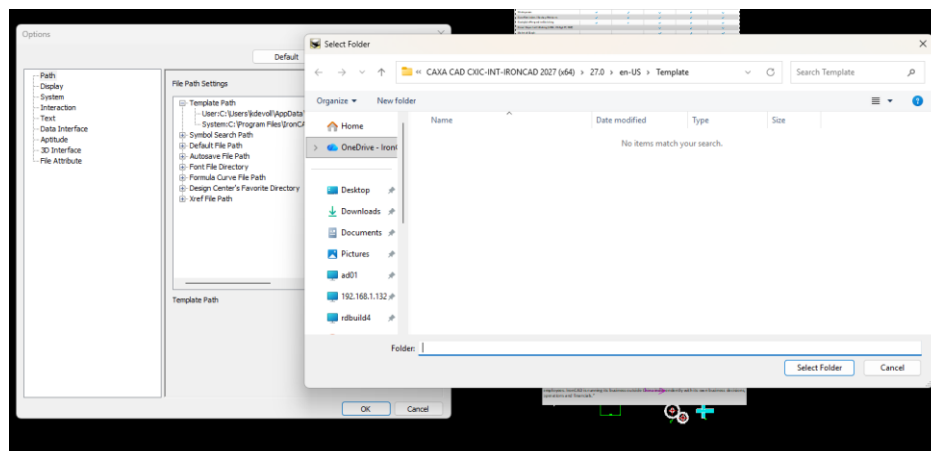
- **New File Behavior:** New system variable NewMDITabDefault (default = 0) controls behavior when clicking the “+” tab. Users can choose default template or template selection with Shift key.
- **Scale List:** New scaling settings interface in Options, allowing custom scale lists that apply to sheet setup, printing, viewports, and transfer tools.



- **Border Display Control:** System options now allow control over border visibility and printing for regions, PDF underlays, external references, and images (adjustable via FRAME system variables).



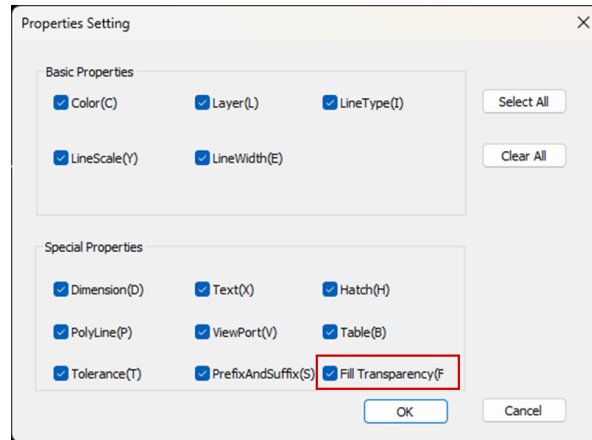
- **System Path Selection:** Improved and more convenient folder and network shared path selection in options.



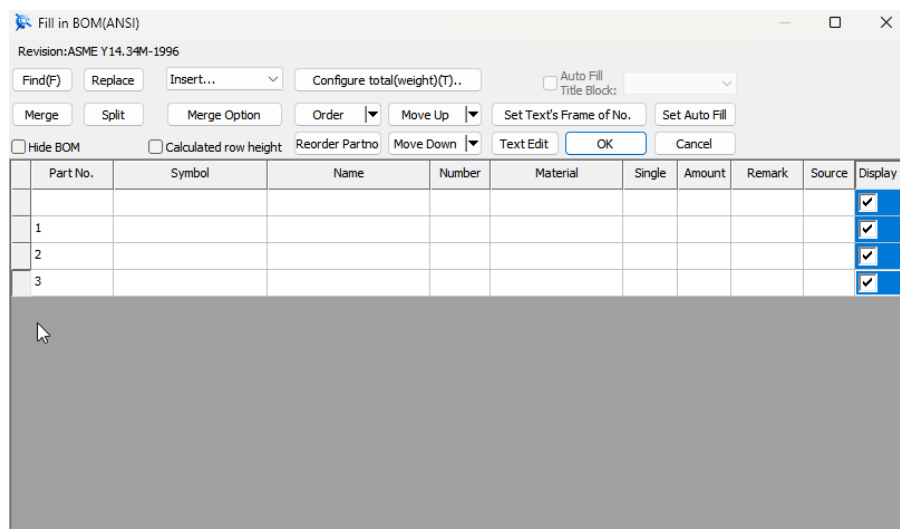
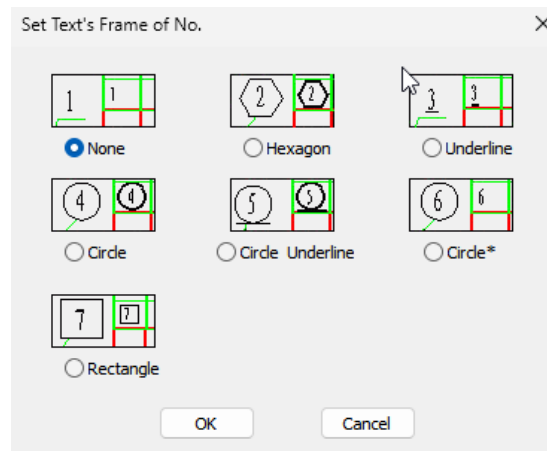
- **Document Switching Experience:** Dialogs and toolbars now automatically hide when working with detail tables or editing text for a cleaner workspace.

## Other Enhancements

- Feature matching now supports fill transparency.



- Optimized detail table filling in Item Bubble and BOM, including improved serial number text border display (graphic and text description formats) and batch selection in the Display column.



- Title blocks, drawing frames, and detail tables can now be inserted using block template methods.
- Improved compatibility with Microsoft Pinyin input method for Chinese Input (cursor following supported).

## Collaboration and Communication Improvements

### 3MF Export

IronCAD 2027 now supports exporting 3D models directly to the 3MF (3D Manufacturing Format) file type. This modern format is specifically designed for additive manufacturing and provides a more robust, accurate, and rich alternative to traditional STL files.

- **Benefits:**
  - Seamless export to the latest 3D printing and additive manufacturing workflows.
  - Better preservation of model data including colors, textures, materials, and multiple objects in a single file.
  - More accurate geometry representation compared to STL, resulting in higher quality 3D printed parts.
  - Improved compatibility with modern 3D printers, slicers, and additive manufacturing software.
  - Reduced file size and better handling of complex assemblies and multi-material designs.
  - Future-proof support for next-generation manufacturing and digital sharing workflows.

This new export capability makes IronCAD even more versatile for users involved in rapid prototyping, 3D printing, and digital manufacturing.

### IronCAD Web View Improvements

The IronCAD Web Viewer (ICWebViewer) has been significantly enhanced with improved performance, usability, and compatibility updates. These improvements deliver a smoother, more responsive, and reliable experience when sharing and viewing IronCAD models in web browsers.

- **Benefits:**
  - Faster loading and smoother navigation of large and complex 3D models in the web viewer.

- Improved overall responsiveness and reduced lag during model rotation, zooming, and sectioning.
- Better compatibility across modern web browsers and devices.

## NestIt for IronCAD

IronCAD is excited to announce the launch of NestIt for IronCAD, a powerful new add-on that streamlines the workflow from design to optimized fabrication for sheet metal and flat pattern parts.

Developed in partnership with NestIt Software, this utility enables IronCAD users to prepare and transfer flat patterns directly to NestIt Pro—a user-friendly, high-performance nesting solution known for true-shape automatic nesting, minimal waste, and efficient material utilization.

### Key Features and Benefits

- **One-Click Export to NestIt Pro:** Simply select your parts and choose "Send to NestIt". The add-on automatically collects flat patterns, quantities, and material selections, generating DWX data that passes seamlessly into NestIt Pro.
- **Material-Aware Grouping:** Parts are intelligently grouped by material, allowing users to nest each set independently for precise optimization.
- **Rapid Material Savings:** Quickly generate optimized nests to reduce scrap, lower material costs, and improve quoting accuracy with detailed reports.
- **Smooth Production Workflow:** Export nesting results for manufacturing, including support for common cuts, remnants, and machine-ready outputs available in NestIt Pro.

Whether you're creating quotes, preparing production runs, or optimizing inventory, NestIt for IronCAD eliminates manual data entry and file handling, saving time and reducing errors.

## IronCAD Native Translator

The IronCAD Native Translator Add-on extension has been updated to support the latest file formats from several of the most common 3D CAD systems on the market.



Import now supports:

- CATIA V5: V5R8 – V5-6R2026
- Pro/E (CREO): 16 – Creo 12.0
- UG NX: 11 – NX 2512
- Inventor: V11 – V2026
- SolidWorks: 98 – 2026
- SolidEdge: V18 – SE 2026
- JT Import: 8 – 10.2, 10.3, 10.5 -10.11
- IFC: IFC 2×3, IFC 4×2, IFC 4×2, and IFC 4×3
- Rhino: V2-8
- DXF/DWG: 2.5 – 2026

### **IronCAD Architectural & Viewing Translator**

The Architectural & Viewing Translator add-on extension has been updated to support the latest file.



These include:

- IFC Reader/Writer: IFC 2×3, IFC 4×1, IFC 4×2, and IFC 4×3
- Revit Reader/Writer: 2015 to 2025 (write to 2025 only)
- SolidWorks Drawing Reader: 2013-2026
- GLTF Writer: 2.0