WHAT'S NEW IN ARES MECHANICAL

PROFESSIONAL CAD SOFTWARE FOR 2D MECHANICAL DESIGN IN DWG

Version 2027

Gräbert

ARES® Mechanical
MECHANICAL CAD

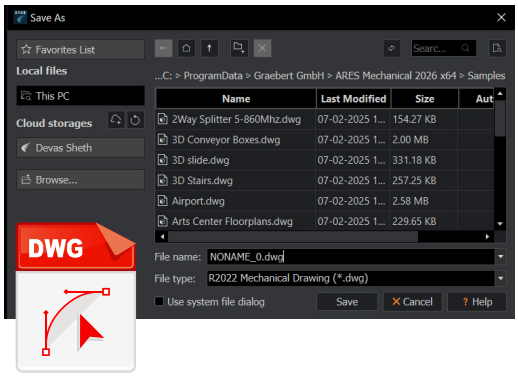
ARES Mechanical is a powerful CAD solution for Mechanical Designers and Engineers

ARES Mechanical CAD software incorporates tools for creating, editing, and publishing mechanical drawings productively and efficiently. It is an intuitive, customizable, comprehensive CAD solution for mechanical CAD designers and engineers. If you are searching for modern 2D Mechanical CAD software for your daily production needs, ARES Mechanical is the right solution for you.

In this brochure we highlight some of the new features in ARES Mechanical 2027 and recent service packs.

1 Support for Mechanical 2022 DWG format (2026 SP3)

ARES Mechanical now supports reading and modifying mechanical data in drawings saved in the DWG 2022 file format. This includes part references, part lists, and bills of materials (BOMs), and all the Mechanical annotations allowing users to work with newer DWG files, while preserving mechanical intelligence and compatibility.



USE CASE EXAMPLE:

A manufacturer opens an older mechanical drawing that has been updated and saved in DWG 2022 format by a supplier using AutoCAD Mechanical. Benefit: Allows seamless updates to parts, annotations, and BOM data without breaking mechanical intelligence.

2 Generate Fits List table automatically (2027 SP0)

Manually tracking and documenting fit tolerances across multiple dimensions can be time-consuming. ARES Mechanical 2027 gives users an automatic way to extract, organize, and maintain fit data directly from the drawing. The new AMFITSLIST command generates a Fits List table from fit dimensions present in the drawing, then inserts it at a user-defined location. The table can be updated or reordered when dimensions change, ensuring the fit data always stays synchronized with the drawing.

USE CASE EXAMPLE:

Manufacturing Documentation Preparation

A designer finishes detailing a shaft-and-hole assembly and generates a fits list to summarize all tolerance fits used in the drawing.

Benefit: *Provides manufacturing teams with a clear, consolidated view of all fit requirements without manual table creation.*

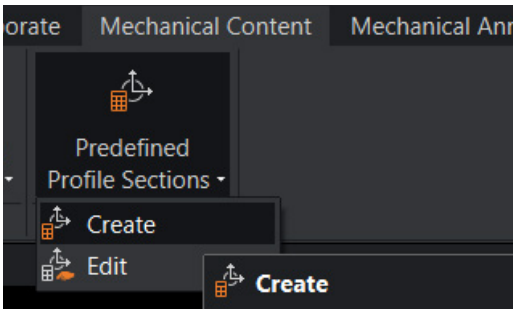
Design Change Management

After adjusting several fit tolerances during a design review, the user updates the existing fits list with a single command.

Benefit: *Ensures documentation stays consistent with the latest design changes, reducing rework and miscommunication.*

3 Calculate inertia for predefined profile sections (2026 SP2)

ARES Mechanical users no longer need to switch to external tools for inertia calculations. The new AM_INERTIAPROF command enables users to calculate and insert the moment of inertia properties for common cross-section profiles directly into their drawings. Users can choose from four predefined profile types: Rectangular, Tube Rectangular, Circular, and Tube Circular.



Command name: AM_INERTIAPROF (AMINERTIAPROF)
AM_INERTIA_RECTANGULAR (AMINERTIA-P1)
AM_INERTIA_TUBERECTANGULAR (AMINERTIA-P2)
AM_INERTIA_CIRCULAR (AMINERTIA-P3)
AM_INERTIA_TUBECIRCULAR (AMINERTIA-P4)
AM_INERTIAPROFEDIT (AMINERTIAPROFEDIT)

The command generates a block containing calculated values, such as:

- I1 – Moment of inertia along the first axis
- I2 – Moment of inertia along the second axis
- Sc – Distance of the outermost fiber in compression from the neutral axis
- St – Distance of the outermost fiber in tension from the neutral axis
- A – Cross-sectional area

The block is placed on the AM_6 layer and integrates seamlessly with the Properties palette. Values are calculated automatically based on user inputs, such as dimensions and thickness.

USE CASE EXAMPLE:

Mechanical Component Optimization

A designer working on a tubular shaft uses the Tube Circular option to check the moment of inertia along the first main axis and the second main axis.

Benefit: *Helps quickly verify that dimensions and wall thickness meet strength requirements before sending to simulation or production.*

Structural Design Validation

A civil engineer designing a support beam inserts a Rectangular profile and immediately obtains I1, I2, and cross-sectional area values.

Benefit: Saves time by getting all relevant section properties directly in the drawing.

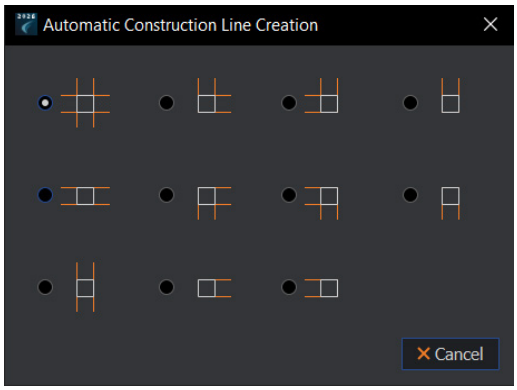
Command name: AM_INERTIAPROF (AMINERTIAPROF)

Ribbon: Mechanical Content > Calculations (new panel) > Predefined Profile Sections

Menu: Mechanical Content > Calculations (new panel) > Predefined Profile Sections

4 Automatic construction lines (2025 SP2)

ARES Mechanical users now have a faster, smarter way to generate accurate reference lines. AM_AUTOCLINES automatically creates vertical and/or horizontal construction lines at the endpoints of selected entities such as lines, polylines, and circles. The new command intelligently generates construction lines only at the outermost X and Y extents of the selection, avoiding overlapping or redundant construction lines.



USE CASE EXAMPLE:

Reference Geometry Creation

A mechanical designer selects multiple lines and circles defining a part outline and automatically generates horizontal and vertical construction lines at their outermost endpoints.

Benefit: *Eliminates manual creation of reference lines and ensures clean, non-overlapping construction geometry for further drafting.*

Layout Preparation and Alignment

While preparing multiple views of a component, the user generates construction lines from existing geometry to align features consistently across views.

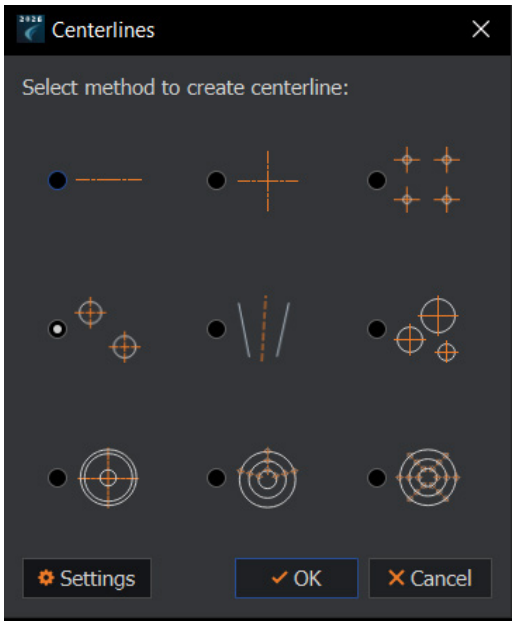
Benefit: *Improves accuracy and speeds up alignment of features without introducing redundant reference lines.*

Command name: AM_AUTOCONSTLINES

Ribbon: Home> Construction Lines > Construction Lines > Automatic Construction Lines

5 Commands for drawing centerlines and centercrosses (2026 SP3)

ARES Mechanical is now equipped with a comprehensive set of commands for creating and editing centerlines and centercrosses. These commands support multiple creation workflows such as centerlines between lines, center crosses in holes, corners, plates, full circles, and angular distributions, while automatically applying drafting-standard–based layers, line types, overshoots, and scaling behavior.



USE CASE EXAMPLE:

Hole and Feature Annotation

A designer places centercrosses in drilled holes using dedicated commands to mark hole centers and diameters in a plate or bracket.

Benefit: *Ensures clear, standards-compliant hole annotations without manually drawing and adjusting centerlines.*

Symmetry and Feature Alignment

A user creates a centerline between two parallel or angled edges to indicate symmetry and align additional features.

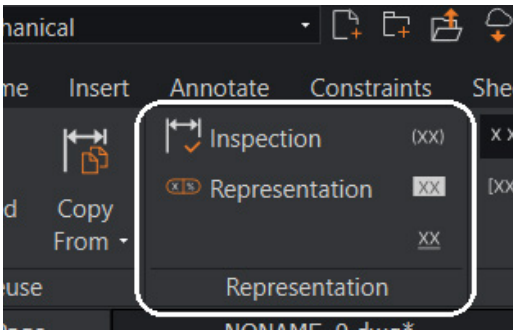
Benefit: Makes symmetry relationships explicit and simplifies positioning of related geometry.

Ribbon: Home > Centerlines > Drop-down - All commands

Menu: Draw > Centerline > Drop-down - All commands

Command name: AM_CENTERCROSS (AMCENCROSS)
AM_CENCRPLATE (AMCENCRPLATE)
AM_CENCRCORNER (AMCENCRCORNER)
AM_CENCRHOLE (AMCENCRHOLE)
AM_CENCRFULLCIRCLE (AMCENCRFULLCIRCLE)
AM_CENCRANGLE (AMCENCRANGLE)
AM_CENCRINHOLE (AMCENCRINHOLE)
AM_CENINBET (AMCENINBET)
AM_CENEDIT (AMCENEDIT)

6 Power Dimensioning contextual ribbon - "Representation" Panel (2026 SP1)



The new Representation panel in the Power Dimensioning contextual ribbon introduces a faster and more intuitive way to apply visual annotations to dimensions such as reference, theoretically exact, not to scale, and inspection indicators without the need for manual text overrides or complex property adjustments.

This enhancement brings clarity and compliance to mechanical drawings by offering quick visual tags for special dimension types, helping users better communicate the design intent, inspection frequency, and precision of measurements. With built-in formatting options and dedicated support for inspection dimensions, users gain both precision and flexibility directly within their active workflow.

USE CASE EXAMPLE:

Tagging Reference Dimensions in Assembly Drawings

An engineer needs to indicate that certain dimensions are for reference only and not subject to manufacturing or inspection.

Workflow: *The engineer selects the relevant dimension and clicks the Reference option from the Representation panel.*

Benefit: *The dimension is visually marked as a reference, helping clarify intent to both design reviewers and manufacturing teams without adding unnecessary clutter.*

Applying Theoretical Dimensions in a Tolerance Stack-Up

A mechanical designer is working on a part where some dimensions must be theoretically exact for a GD&T evaluation.

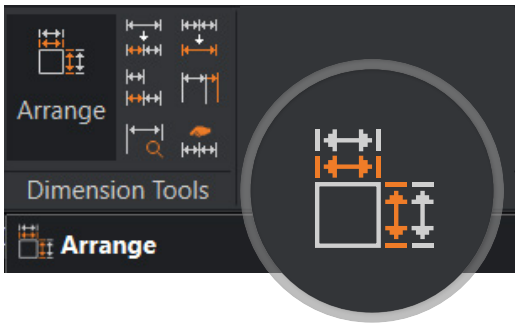
Workflow: *The designer uses the Theoretically Exact option to box selected dimensions that define exact feature locations.*

Benefit: *The boxed format clearly identifies these dimensions as exact, improving communication and compliance with industry standards.*

Command name: Power Dimensioning Contextual Ribbon - "Representation" Panel

Ribbon: Contextual tab will be displayed after selecting the dimension

7 Arrange dimensions manually or automatically (2026 SP2)



Dimension-heavy drawings often become cluttered with overlapping texts or uneven spacing, which reduce readability. Manual repositioning, however, can be time-consuming and prone to error. The new AM_DIMARRANGE command provides users with the ability to automatically or manually rearrange existing dimensions, with consistent spacing relative to each other and to surrounding geometry. This command supports linear, angular, radial, and ordinate dimensions, applying default snap distances and alignment rules to improve clarity. In manual mode, users can fine-tune parameters such as side, spacing, or orientation with immediate visual feedback, making it easier to control the layout of complex drawings.

USE CASE EXAMPLE:

Rearranging Linear Dimensions on a Part Drawing

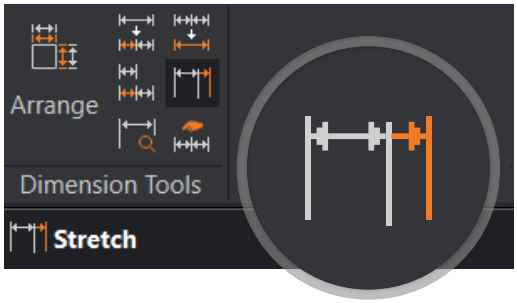
A mechanical engineer has several baseline dimensions for holes and edges placed too close together, making them hard to read. Using AM_DIMARRANGE in _Auto mode, the software redistributes them with consistent spacing along the bounding box. This improves clarity and avoids overlaps without extra manual steps.

Optimizing Radial Dimensions on a Shaft Design

A designer has multiple radial dimensions crowding one side of a circular part. By switching to _Manual mode, the designer selects a start and end angle to spread them evenly across a clean arc segment. The resulting layout is visually balanced, professional, and easier to interpret during manufacturing.

Command name:	AM_DIMARRANGE
Ribbon:	Mechanical Annotate > Dimension Tools > Arrange
Menu:	Mechanical Annotate > Dimension Tools > Arrange

8 Stretch dimension (2026 SP2)



The AM_DIMSTRETCH command allows users to resize geometry by stretching or shrinking it through an associated linear, diametric, radial, or symmetric dimension. Instead of using manual edits, users can interactively update the size of selected geometry simply by entering a new dimension value. Both automatic and centerline-based symmetrical stretching are supported, ensuring flexible and precise resizing operations.

USE CASE EXAMPLE:

Updating a Shaft Length

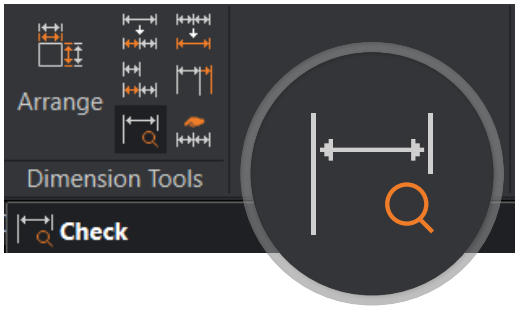
A mechanical engineer edits the linear dimension of a shaft from 100 mm to 120 mm. The geometry and associated dimensions update automatically, ensuring accuracy and saving time.

Symmetrical Stretching for Hole Patterns

A designer modifies the spacing of symmetrical holes across a centerline. Using the Symmetrical option, the holes are resized equally on both sides, preserving balance in the part design.

Command name:	AM_DIMSTRETCH
Ribbon:	Mechanical Annotate > Dimension Tools > Stretch
Menu:	Mechanical Annotate > Dimension Tools > Stretch

9 Check dimension text overrides (2026 SP2)



Until now, users had no quick way to identify where dimension values were manually altered. The AM_DIMCHECK command detects dimensions with overridden text values, and either highlights them or allows step-by-step editing. Users can quickly identify all dimensions that do not display the true measurement (for example, those where the "<>" placeholder has been replaced with custom text).

USE CASE EXAMPLE:

Design Review

Before releasing a drawing, a checker runs AM_DIMCHECK and instantly finds dimensions where "<>" was replaced with fi xed values. The dimensions are highlighted for immediate correction.

Editing Overrides

An engineer inherits a drawing with multiple dimensions showing incorrect text values. By running AM_DIMCHECK in Edit mode, the system walks through each overridden dimension, opening the Note editor for corrections one by one.

Command name:	AM_DIMCHECK
Ribbon:	Mechanical Annotate > Dimension Tools > Check
Menu:	Mechanical Annotate > Dimension Tools > Check

10 Combine multiple dimensions into one (2026 SP2)

The AM_DIMJOIN command allows users to combine two or more dimensions of the same type into a single, consolidated dimension. It supports Linear (Aligned, Rotated), Angular (2-Line, 3-Point), and Arc Length dimensions. (Unsupported dimension types include Radius, Diameter, Jogged, and Ordinate.) When dimensions are joined, the system validates compatibility, merges extension lines, and outputs a single dimension that spans the full range of the selected dimensions.

USE CASE EXAMPLE:

Merging Linear Dimensions

A user selects two aligned linear dimensions marking the same edge of a part. AM_DIMJOIN merges them into one dimension with a clean span, eliminating duplicates.

Combining Angular Dimensions

An engineer joins a 2-Line Angular dimension with a 3-Point Angular dimension representing the same corner. The result is a single angular dimension with correct span.

Command name:	AM_DIMJOIN
Ribbon:	Mechanical Annotate > Dimension Tools > Join
Menu:	Mechanical Annotate > Dimension Tools > Join

11 Align dimensions with a common reference (2026 SP1)

Align dimensions more precisely with the new AM_DIMALIGN command, which allows users to line up linear, ordinate, or angular dimensions with a selected base dimension. By aligning dimensions in relation to a common reference, users can create cleaner and more readable technical drawings with minimal manual adjustment.

The command validates and aligns dimensions of the same type (e.g., linear to linear, angular to angular), ensuring consistent orientation and spacing. It also provides detailed feedback when dimensions cannot be aligned due to mismatch in type, orientation, or geometry. AM_DIMALIGN enhances both visual clarity and drawing accuracy, especially in complex mechanical layouts.

USE CASE EXAMPLE:

Aligning Baseline Dimensions in a Mechanical Part Drawing

A mechanical engineer is dimensioning a side view of a bracket with multiple hole centers and edge distances.

Workflow: *The engineer uses AM_DIMALIGN to select one properly placed dimension as the base, then selects several others (e.g., hole center distances) to align them neatly in a row.*

Benefit: *This results in a clean, aligned series of dimensions that improves drawing readability and saves time compared to manually adjusting each dimension's position.*

Standardizing Angular Dimensions in a Shaft Design

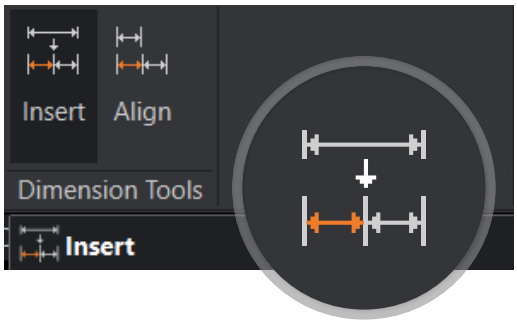
A designer is working on a shaft with multiple chamfers and angled cuts, each measured with angular dimensions.

Workflow: *The designer selects a key angular dimension as the base and uses AMDIMALIGN to align other angular dimensions to the same vertex center.*

Benefit: *Ensures all angles are referenced uniformly, reduces confusion, and maintains consistency in complex geometry.*

Command name:	AM_DIMALIGN
Ribbon:	Mechanical Annotate > Dimension Tools > Align
Menu:	Mechanical Annotate > Dimension Tools > Align

12 Split dimensions (2026 SP1)



The new AM_DIMINSERT command allows users to split a linear or angular dimension into two separate dimensions at a specified point. This capability is especially useful for breaking long dimension lines or angle spans into more readable segments without manually redrawing them.

When splitting, the new dimension is automatically created from the second extension line of the original and inherits properties such as tolerance, fit, and layer assignment.

USE CASE EXAMPLE:

Splitting a Long Linear Dimension in a Structural Design

A structural engineer is documenting the total length of a beam but wants to show it broken into two segments due to layout constraints.

Workflow: *The engineer selects the beam's linear dimension and uses AM_DIMINSERT to split it at a midpoint.*

Benefit: *The resulting dimensions clearly show the individual lengths while maintaining alignment and original tolerance settings—improving clarity and avoiding overlapping annotations.*

Splitting an Angular Dimension in a Mechanical Part Drawing

A designer is documenting a chamfered edge defined by a large angular span.

Workflow: *The designer uses AM_DIMINSERT to split the angular dimension at a significant feature point on the part.*

Workflow: *The designer selects a key angular dimension as the base and uses AMDIMALIGN to align other angular dimensions to the same vertex center.*

Benefit: *This allows the design to reflect more specific angular sub-segments, making the drawing easier to interpret during manufacturing or inspection.*

Command name:	AM_DIMINSERT
Ribbon:	Mechanical Annotate > Dimension Tools > Insert
Menu:	Mechanical Annotate > Dimension Tools > Insert

13 Dimension Overdrawn functionality (2027 SP0)

A Dimension Overdrawn dialog box now appears when a newly placed linear dimension overlaps an existing one.

It enables users to resolve overlaps during dimension creation without deleting and recreating dimensions manually. Using the new dialog, users can replace, break up, or move dimensions, automatically arranging them into chain or baseline layouts while preserving existing dimension properties.

USE CASE EXAMPLE:

Dense Dimensioning Cleanup

A designer adds a new linear dimension between two closely spaced features, overlapping an existing one.

Benefit: *The system breaks the existing dimension into a clean chain, keeping the drawing readable without manual edits.*

14 Power Dimensioning dialog box (2027 SP0)

The new Power Dimensioning dialog box in ARES Mechanical 2027 can be used instead of the contextual ribbon when creating or editing dimensions, giving users more choice in workflows.

Controlled by the AMPOWERDIMEDITOR system variable, this optional dialog provides a centralized, tab-based interface for editing dimension text, representation, units, fit & tolerance, inspection data, and geometry settings.

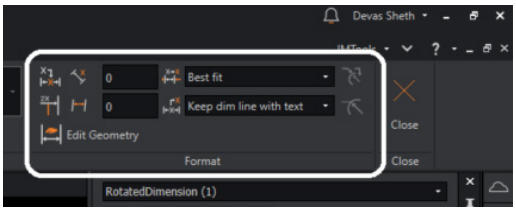
USE CASE EXAMPLE:

Detailed Dimension Editing

A designer working with fits, tolerances, and inspection dimensions enables the Power Dimensioning Dialog Box for easier access to all related settings.

Benefit: *Speeds up complex dimension edits without switching between multiple UI elements.*

15 Power Dimensioning ribbon contextual tab - "Format" panel (2026 SP2)



The new Format panel provides controls to edit and control the geometry and presentation of selected dimensions directly from the Power Dimensioning contextual ribbon. These controls include:

Home Text: Restore rotated text to its default position.

Rotate Text: Rotate dimension text by a user-entered angle.

Symmetry Dimension: Convert a linear/rotated dimension into a symmetry indicator (removes one extension line and arrowhead, doubles the value display). Applicable to aligned/rotated/arc-length/ordinate types (not angular/radial/diametric).

Oblique Angle: Slant extension lines by a given angle for linear aligned/rotated dimensions.

Edit Geometry: Open a dialog showing a graphical representation of dimension components; toggle visibility of text, arrows, extension lines, and dimension lines; choose arrowheads; preview per-dimension changes (does not change DIMSTYLE).

Text–Arrow Fit: Select how text and arrows are arranged when space is limited (options include Text & arrow outside / Arrows outside / Text outside / Bestfit / Text always inside).

Text Movement: Choose behavior when moving text (options include Keep dimension line with text / Move text, add leader / Move text, no leader). Arrow Inside / Landing Line: Control arrow placement and landing line for radial/diametric dimensions.

USE CASE EXAMPLE:

Fixing crowded dimensions in a tight layout

An engineer has dimension text overlapping extension lines on a compact detail. They select the dimension, use Text-Arrow Fit → Bestfit, or Move text, add leader, to immediately produce a readable layout without changing DIMSTYLE.

Benefit: Faster correction, improved clarity, no global style changes.

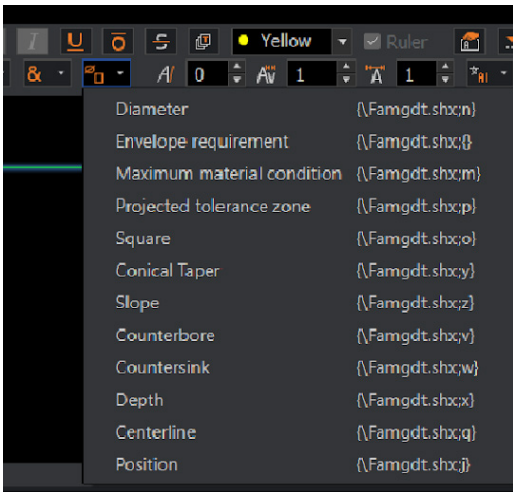
Rotating annotation for clarity on slanted parts

A drafter needs the dimension text rotated 30° to match a slanted feature. They open the Format panel, type 30 in Rotate Text, and the text rotates in place.

Benefit: Precise placement matching part geometry, faster than manual grips and nudging.

Command name:	Power Dimensioning Ribbon Contextual Tab - "Format" Panel
Ribbon:	Contextual tab will be displayed after selecting the dimension

16 “Mechanical symbol” dropdown in the Edit Note dialog box (2026 SP2)



A new Mechanical symbol dropdown has been added to the Edit Note dialog box in ARES Mechanical. This feature provides a curated list of mechanical-specific symbols used in dimensioning and mechanical drawings, expanding symbol insertion beyond the standard “Insert Symbol” gallery. Users can insert the selected symbol at the current cursor position in the dimension text. Available symbols include:

- Position
- Diameter
- Projected Tolerance Zone
- Maximum Material Condition
- Envelope Requirement
- Plus/Minus
- Degree
- Square
- Taper
- Slope
- Counterbore
- Countersink
- Depth
- Centerline.

USE CASE EXAMPLE:

Using Position or Projected Tolerance Zone symbols

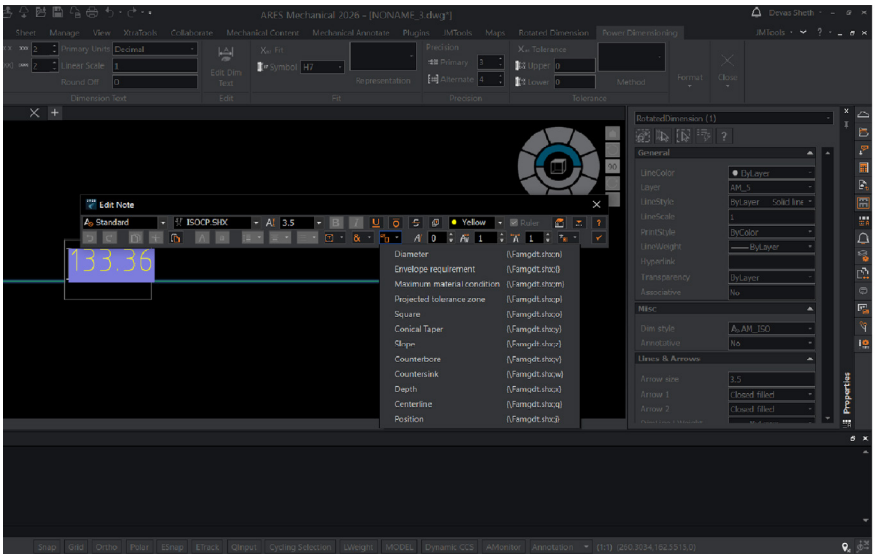
In a GD&T callout, the engineer selects Position and Projected Tolerance Zone symbols directly from the dropdown.

Benefit: *Standards-based feature controls frame annotation in seconds.*

Adding Counterbore and Countersink symbols for fastener holes

A drafter editing a note inserts Counterbore (⌴) or Countersink (⌵) symbols from the dropdown.

Benefit: *Clear manufacturing instruction, no errors from manual text entry.*



New Mechanical symbol dropdown added after the existing Insert Symbol dropdown (only in ARES Mechanical)

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Are you new to ARES Mechanical? Join Graebert's CTO Robert Gräbert and Marketing Manager Dani Ramiro for an introduction to the world of 2D mechanical drawings in DWG format, including an overview of the features and enhancements that streamline mechanical drafting processes.

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