



GSTARCAD MECHANICAL 2017

User Guide

Chapter 1 GstarCAD Mechanical 2017 Introduction

Abstract

GstarCAD Mechanical 2017 drafting/design software, covers all fields of mechanical design. It supplies the latest standard parts library, symbols and dimensioning tools in line with standards in different countries being compatible with AutoCAD Mechanical drawings. The standardized and intelligent mechanical design greatly enhances the efficiency of designers and helps them complete accurate product ideas.

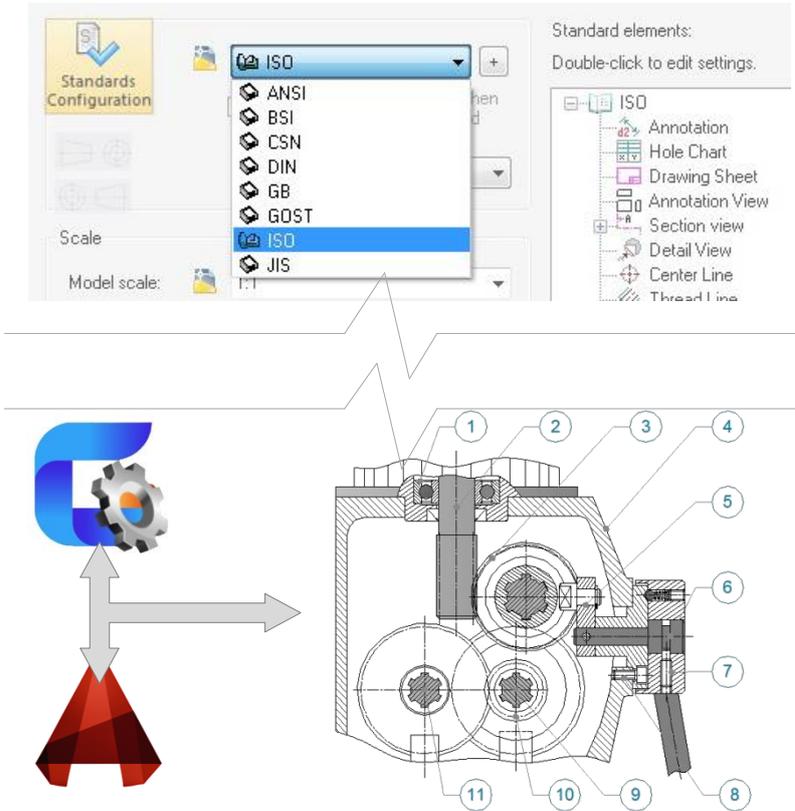
International Drafting Standards and Compatibility with AutoCAD Mechanical Drawings

Standards-Based

GstarCAD Mechanical 2017 supplies international drafting standards like ANSI/BSI/CSN/DIN/GB/GOST/ISO/JIS that are easily customizable, helping engineers to meet design requirements by simplifying mechanical design process in the global marketplace consistently.

Seamlessly Data Displaying

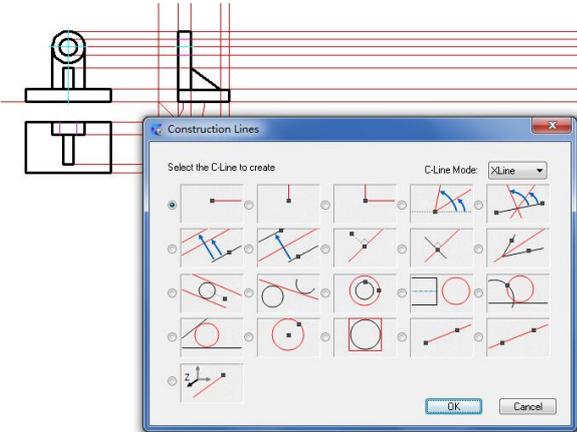
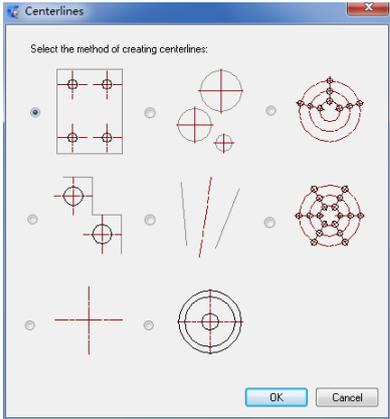
GstarCAD Mechanical 2017 could accurately read and display the complete underlying data of AutoCAD Mechanical drawings and vice versa. Seamlessly data display including title blocks, balloons, part list, BOM, surface texture, feature control frame, welding symbols and so on.



Professional Tools to Simplify 2D Mechanical Design

Center Line

GstarCAD Mechanical 2017 enriches functionalities of drawing tools based on GstarCAD platform in accordance with special requirements of mechanical design. The center line function could help users accurately and quickly create centerlines on standard layers according to requirements of different mechanisms.

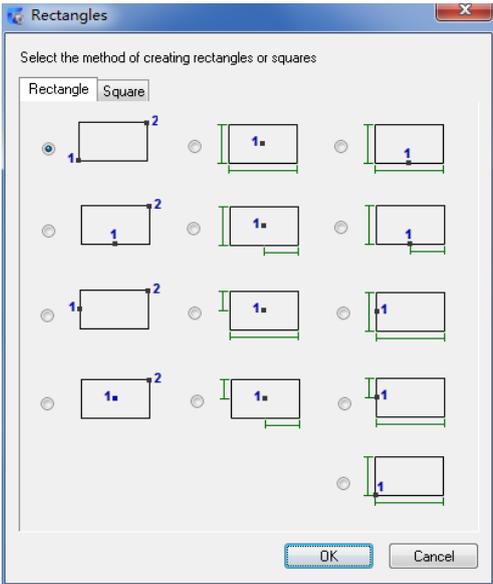


Construction Line

When creating an aligning view for geometry, the construction line helps improve design efficiency. For better distinction and management, the construction line automatically locates in proper layer. It will be invisible when printing.

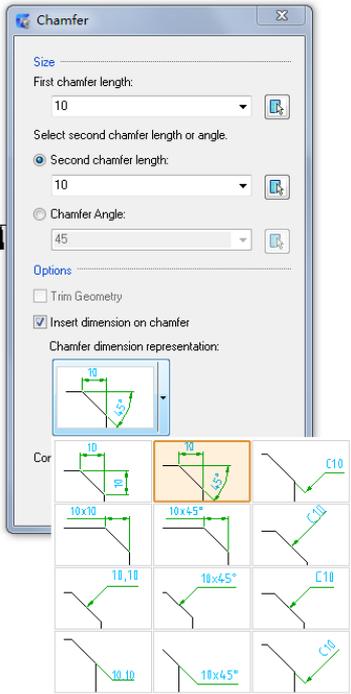
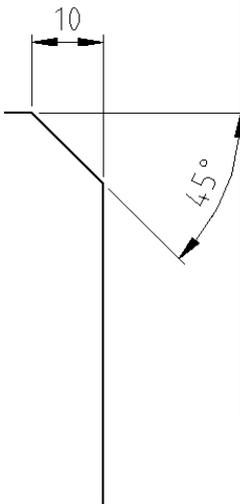
Rectangle

The rectangle command could create various kinds of rectangles and squares and help users conveniently and flexibly place a rectangle at a specified position without further modification and repetitive work.



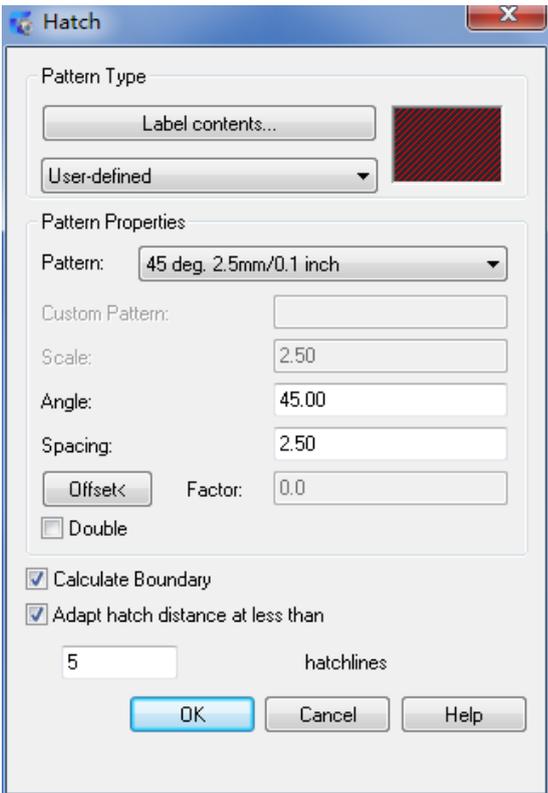
Associate Detailing Tool (Chamfer)

Update drawings quickly with powerful tools that enable users to edit previous operations, saving valuable design time. Easily re-edit features without having to remove and re-create the original feature. For example, resize a chamfer using the original dialog parameters by simply double-clicking the chamfer.



Associate Detailing Tool (Hatch)

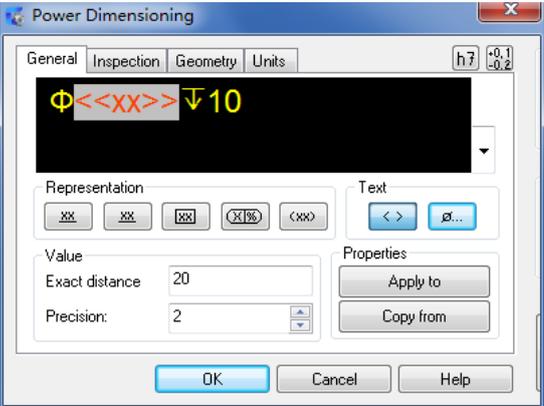
You can easier distinguish parts of a section and detail view by hatch patterns. GstarCAD Mechanical 2017 contains user-defined hatch, associative hatch and predefined hatch.



Easier Mechanical Documentation with Dimensioning Tools and Symbols Annotation

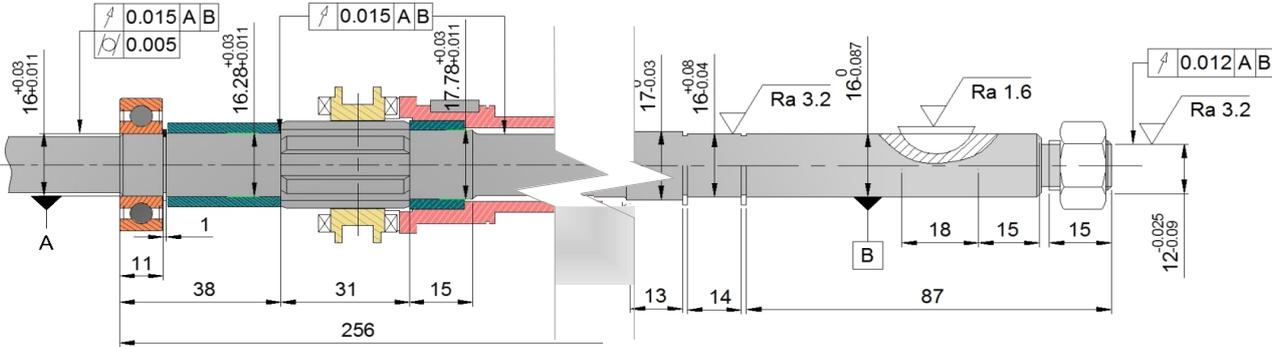
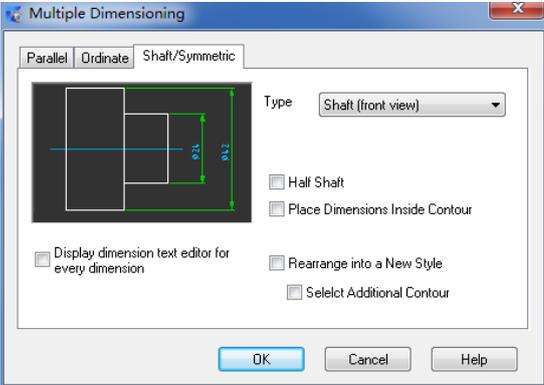
Power Dimensioning Tool

GstarCAD Mechanical power dimensions function greatly reduces the dimensioning work of engineering drawing. Help users create multiple types of dimensions by one command, flexibly define dimension style and content, add tolerance and fitting to dimension, modify representing method to display dimension properties, add inspection content for fully dimension effect.



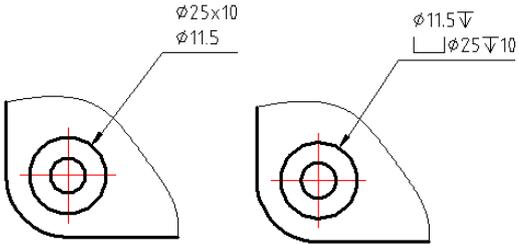
Multiple Dimensioning Tool

Multiple dimensioning function could batch create multiple parallel dimensions, coordinate dimensions, axis dimensions and symmetric dimensions and intelligently recognize the space of dimensions and place them on corresponding layers. Greatly simplify the task of creating and editing dimensions.



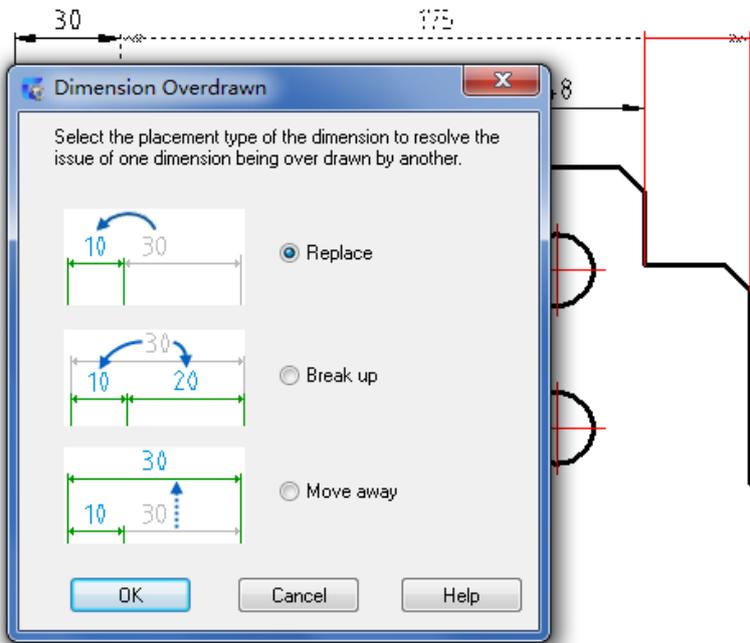
Leader Note Tool

The Leader Note function could create annotation for graphics and add multiple feature information to display real features of specified mechanism. The balloon is linked with selected graphic object, when the object moves, the annotation will change its position accordingly.



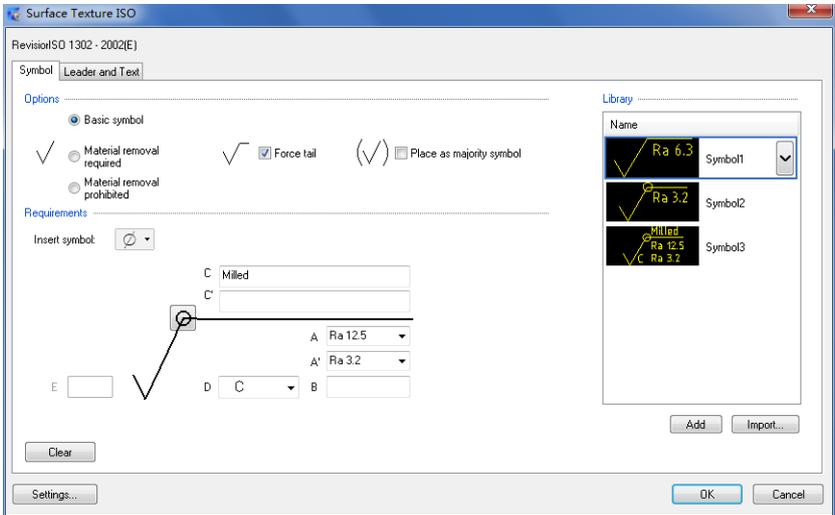
Dimension Edit Tool

The dimension edit tool could quickly stretch, add or combine dimensions and inspect dimensions, as well help users edit dimensions accurately.



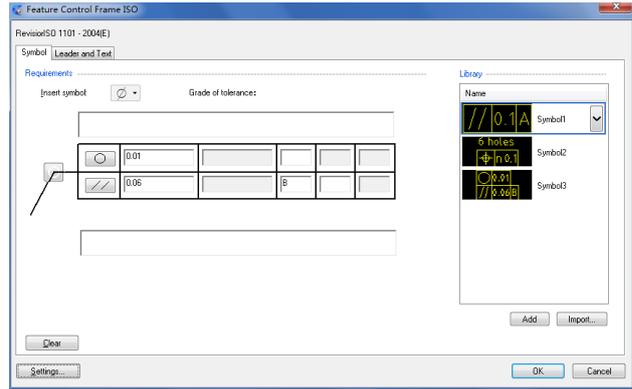
Surface Texture Tool

GstarCAD Mechanical supplies more than 11 kinds of symbol annotations, including Surface texture symbol, feature control frame, welding symbol, datum identifier symbol, datum target, taper&slope,edge symbol and so on. Those symbols could be attached on target objects or dimensions. They could automatically adapt to drawings by moving or scaling and make engineering dimensioning standard. The surface texture symbol is one kind of most popular annotation tool for mechanical design; it meets different requirements of surface treatment annotation during design process.



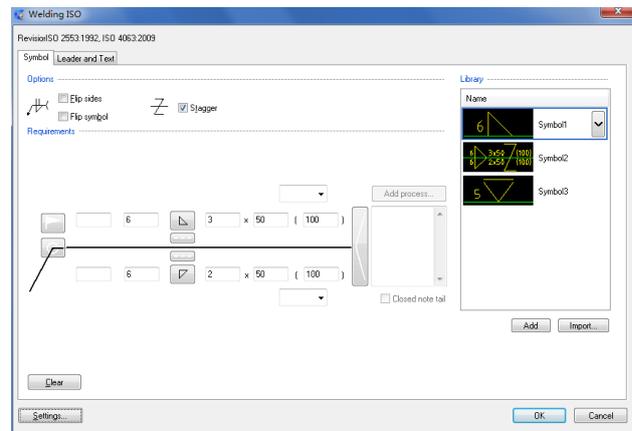
Feature Control Frame Tool

According to the grade of tolerance and standard annotation symbols, feature control frame could be added at any points, lines or planes on a mechanism in a drawing. The tolerance is an error between the ideal element and actual element after machining.



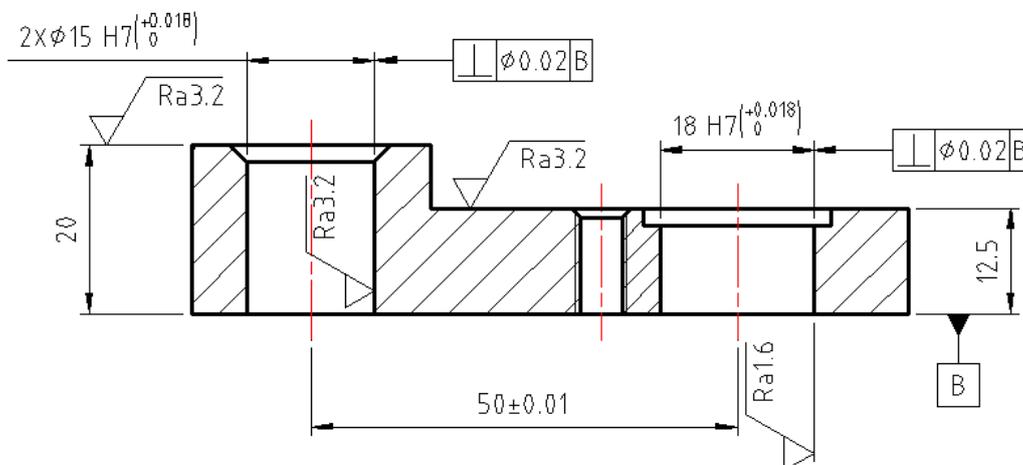
Welding Symbol Tool

The welding symbol annotation function could add welding features to geometry to display comprehensive welding information. It could contain all dimension styles under current drawing standards.



Import to Library and Attachment

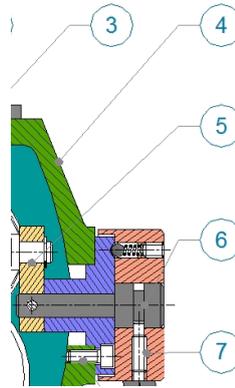
Users could add any new created symbol annotation to the symbol library. Insert common symbols to a drawing and save large of time on dimensioning. Be different from drawing standard, the symbol library is saved in drawing format. The symbol is attached on geometry, when the geometry moves, the symbol moves accordingly.



Accurate Data Management and Reports

Balloon and Part List

According to manual dimensioning, GstarCAD Mechanical 2017 is much simpler and effective. The balloon and part list could be intelligently associated with mechanical parts and keep update when detected parts information changing. Reduce cost of some mistaken information caused by recording, recognition and sorting, as well help users reduce workload and ensure data accurate.



Item	Description	Part Number	Qty	Material	Dimensions
11	Hypoid Gear 1	HG1-007-5	2	Mild Steel	P120x 60 x 5
10	Hypoid Gear 2	HG2-007-6	1	Mild Steel	P40 x 20 x 2
9	Transversal Shaft Geared	TSG-007-4	3	Mild Steel	Ø75 x 75mm
8	Hex Bolt	HBT-007-11	6	Steel Carbon	M26 x 70 x 22.5
7	Outer Cap	OTC-007-10	1	Alloy Cast Iron	Ø200 x 80mm
6	Pin Cap Stuck	PCS-007-8	6	Commercial Steel	Ø60 x 120mm
5	Pin Fastener Radial	PFR-007-9	1	Commercial Steel	Ø40 x 50mm
4	Gear Box Case	GBC-007-1	1	Alloy Cast Iron	1398 x 1270 x 980mm
3	Hypoid Gear 3	HG3-007-7	2	Mild Steel	P80 x 40 x 3
2	Vertical SD Shaft	VSDS-007-2	2	Mild Steel	Ø90 x 950mm
1	Bearing	BRVS-007-3	2	UHMWPE	int. Ø85 x ext. Ø120 x 16mm

PART LIST

BOM Data Extraction

The BOM could display component parts features comprehensively. It coordinates all data information of component parts and control changes of global. GstarCAD Mechanical 2017 supports extraction BOM data from DWG files for better reuse, promoting more efficient and accurate exchange of design data and facilitates earlier collaboration between design and manufacturing workgroups.

A	B	C	D	E	F	G	H	I
BOMKEY	ITEM	NAME	PARTNUMBER	QTY	MATERIAL	DIM	BOM_UNITS	NOTE
2	71B3	1 Bearing	BRVS-007-3	2	UHMWPE	int.Ø85xext.Ø120x16	2	00000
3	71C3	2 Vertical SD Shaft	VSDS-007-2	2	Mild Steel	Ø90 x 950mm	2	00001
4	71E4	3 Hypoid Gear3	HG3-007-7	2	Mild Steel	P80x40x3	2	00002
5	71C6	4 Gear Box Case	GBC-007-1	1	Alloy Cast Iron	1398x1270x980mm	1	00003
6	71C9	5 Pin Fastener Radial	PFR-007-9	1	Commercial Steel	Ø40x50mi	1	00004
7	71CD	6 Pin Cap Stuck	PCS-007-8	6	Commercial Steel	Ø60x120ma	6	00005
8	71D0	7 Outer Cap	OTC-007-10	1	Alloy Cast Iron	Ø200x80mm	1	00006
9	71D3	8 Hex Bolt	HBT-007-11	6	Steel Carbon	M26x70x2.5	6	00007
10	71D9	9 Transversal Shaft Geared	TSG-007-4	3	Mild Steel	Ø75x750me	3	00008
11	71D6	10 Hypoid Gear 2	HG2-007-6	1	Mild Steel	P40x20x2	1	00009
13	71E0	12 Hypoid Gear 1	HG1-007-5	2	Mild Steel	P120x60x5	2	00010

Associative Hole Charts

GstarCAD Mechanical supplies associative hole charts. Accurately create links between model and tables, support for real-time adding or removing holes, as well sort holes according to specified rules. It reduces some mistakes caused by manual operations. The filter function in tables could help users list different holes in different hole charts for optimizing manufacturing process.

Hole Chart (GMHOLECHART)
Creates coordinate dimensions for holes in a work piece, dimensions the size of those holes, and generates a hole chart for that work piece.

1.6	52	62	Ø6	Clearance	ISO273-M10 H13
1.5	52	41	Ø2.55	Thread Blind Hole	ISO26-M10x1.25x30 / 35
1.4	52	29	Ø2.55	Thread Blind Hole	ISO26-M10x1.25x30 / 35
1.3	52	8	Ø6	Clearance	ISO273-M10 H13
1.2	36	62	Ø2.55	Thread Blind Hole	ISO26-M10x1.25x30 / 35
1.1	36	8	Ø2.55	Thread Blind Hole	ISO26-M10x1.25x30 / 35
Hole	X	Y	Ø	Description	Standard
LIST OF COORDINATES					

B	Ø6	62	2
A	Ø2.55	8	4
Hole	Ø	Y	Qty
HOLE TABLE			

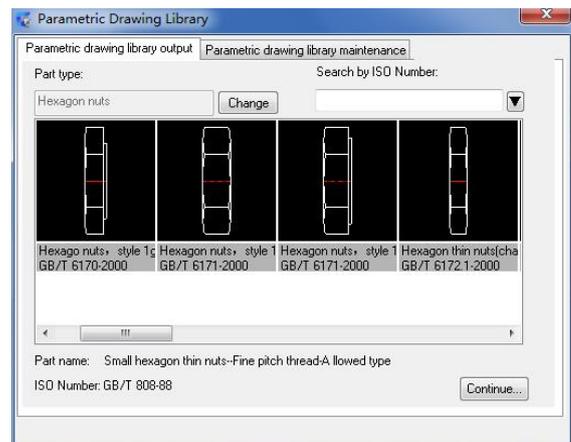
Form Tools

In the daily work of design engineers, they need to process a lot of forms, such as summary form design and processing, processing technology form definition and filling, processing card management and so on. The GstarCAD Mechanical supplies a powerful and practical form function. Users could create various kinds of forms during the product design. It simplifies design process and shortens design stages.

Standard Part and Symbol Libraries

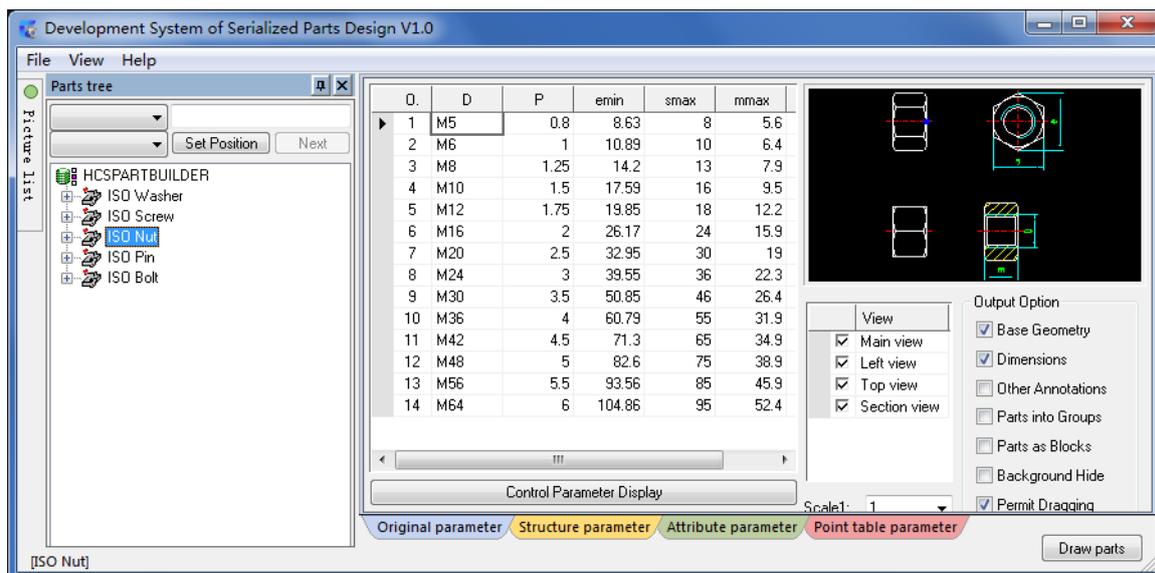
Parametric Graphic Library

In the past the parametric graphic library could only be established by programming and it is difficult to maintain and update. Now GstarCAD Mechanical 2017 supplies an open parametric graphic library. Users could invoke and manage parametric parts as many times as necessary, they could also find a part by part type or query standard number.



Serialization Part Design System

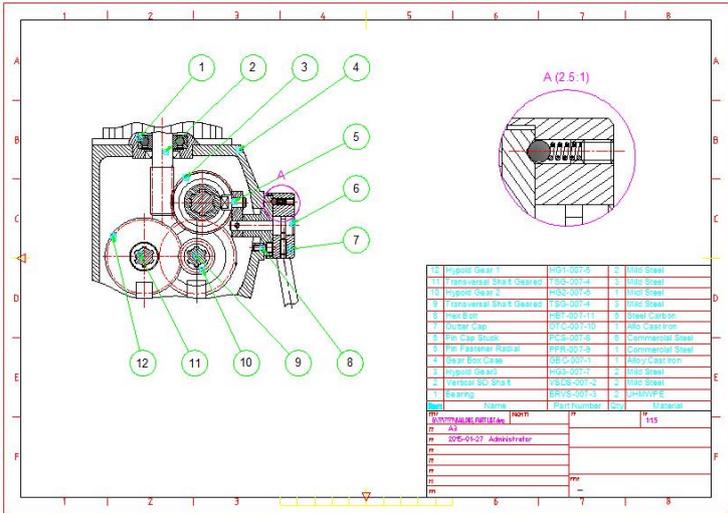
According to requirements of different professions in manufacturing industry, GstarCAD Mechanical 2017 supplies serials of national standard parts libraries and they are open to manage, maintain and customize. Users could easily create, retrieve, modify and expand the company's own serialization standard graphic library.



Plotting Faster by Intelligent Mapsheet, Detail and Scaling Tool

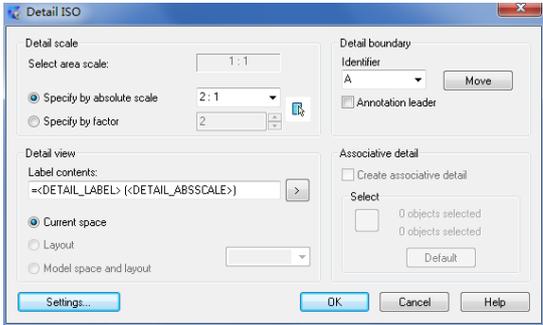
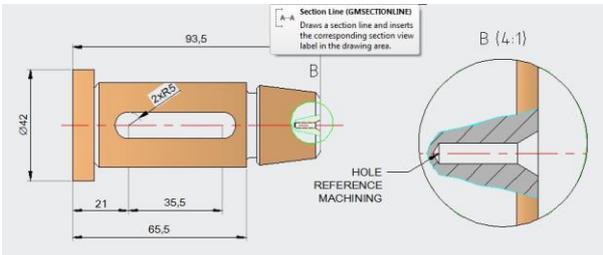
Title, Board and Revision Block

GstarCAD Mechanical 2017 contains a set of predefined titles and boards in line with standards in different countries. It could quickly create a drawing with standard title block and board. Double-clicking the board could modify its size, change the title block style and adjust its scale and other information. Users could customize the tile and board according to their own enterprise standard, as well insert revision table to any position of a drawing including the title.



Detail View

GstarCAD Mechanical supplies intelligent detail view design tools, the drawing will be automatically updated as design changing. It could magnify the partial region by box selection in the drawing and easily create linked partial views with different scales.



Scale Area

In the model space, the drawing could contain different scale area with different scale factors. In the layout, users could create a viewport with a proper length ratio or a proper text ratio for printing.

With the advantage of GstarCAD Platform

Features	GstarCAD	GstarCAD Mechanical
Full CAD Functionality	✓	✓
Familiar CAD Interface	✓	✓
DWG Compatibility	✓	✓
Compatible with AutoCAD Mechanical Drawings		✓
Support for International Standards		✓
Powerful Drafting Tools		✓
Intelligent Dimensioning Tools		✓
Powerful Mechanical Annotation Symbols		✓
Standard part libraries		✓
Associative Balloon and BOM		✓
Integrated Date Management		✓
Completing Drawing and Dimensioning accelerates 60%		✓
Completing Design and Engineering Task accelerates 90%		✓
Number of Required Command reduces 75%		✓

Chapter 2 Create Geometry

Summary

Numerous special requirements are involved due to characteristics of mechanical drawing. For this reason, a number of drawing functions are included in GstarCAD Mechanical 2017 based on GstarCAD platform features, which has greatly improved drawing efficiency, enhanced drawing quality and precision. These features can be found in the 'Drawing tools' option from GstarCAD Mechanical pul-down menu, which may also be accessed by users through a toolbar.

Common drawing commands

Symmetrical Line

Ribbon: Home tab ► Draw panel ► Symmetrical Line.

Keyboard command: GMSYMLINE

Mirror line drawing refers to the case where a graphic is drawn on one side of a line being the axis of symmetry and then a symmetrical graphic is formed on the other side. This is mainly used for drawing a solid of revolution.

To Create Symmetrical Line

1. Click Home tab ► Draw panel ► Symmetrical Line.
 2. Select the axis of symmetry.
 3. Draw the required graphic on one side of the axis of symmetry.
-

List of Prompts

Select Centerline or new starting point: Specifies an existing centerline to use as the line of symmetry, or inserts the starting point for a new a centerline.

New starting point: Specifies the starting point to draw a new centerline.

Centerline endpoint: Specifies the end point of the centerline.

Start point: Begins drawing the polyline that is to be mirrored about the selected centerline. Subsequent prompts are identical to that of the PLINE command. Please refer help on the PLINE command for details.

Center line

Ribbon: Home tab ► Draw panel ► Centerline

Keyboard command: GMCENTERLINE

It draws a centerline at a specified place in the drawing area.

To create a center line

1. Click Home tab ► Draw panel ► Centerline.
 2. Press ENTER to display the Select Options for Centerlines dialog box, or specify a starting point for the centerline.
 3. Specify the endpoint for the centerline.
-

List of Prompts

Dialog box: Displays the centerlines dialog box, using which you can change the centerline creation method.

Starting point: Specifies the start point of the centerline.

Ending point: Specifies the ending point of the centerline.

✧ Centerline Cross

Ribbon: Home tab ► Draw panel ► Centerline drop-down ► Centerline Cross

Keyboard command: GMCENTERLINE

Places a centerline cross at a specified point.

To Create a Centerline Cross

1. Home tab ► Draw panel ► Centerline drop-down ► Centerline Cross.
 2. Press ENTER to display the Select Options for Centerlines dialog box, or specify a starting point for the centerline.
 3. Specify the center point of a hole.
 4. Specify a diameter value for the centerline cross or select a quadrant point.
 5. Press ENTER.
-

List of Prompts

Dialog box: Displays the Centerlines dialog box, using which you can change the centerline creation method.

Center point: Specifies the center of the centerline cross.

Diameter: Specifies the diameter of the centerline cross to place.

✧ Center Mark

Ribbon: Home tab ► Draw panel ► Centerline drop-down ► Center Mark

Keyboard command: GMCEN

Creates center mark or center lines of circles and arcs.

To Create a center mark

1. Click Home tab ► Draw panel ► Centerline drop-down ► Center Mark.
 2. Select an arc or a circle.
-

List of Prompts

Select arc, circle or polyline arc segment: Select the object generating center mark.

✧ **Centerline Cross on a Plate**

Ribbon: Home tab ► Draw panel ► Centerline drop-down ► Centerline Cross on Plate

Keyboard command: GMCENCRPLATE

Places holes at the corners of a closed contour that represents a plate.

To Create a Centerline Cross on a Plate

1. Click Home tab ► Draw panel ► Centerline drop-down ► Centerline Cross on Plate.
 2. Enter a value to specify the offset of the center-cross from the contour.
 3. Select the objects.
 4. Press ENTER.
 5. Use the pointing device to specify where to place the centerline crosses: inside or outside the plate.
 6. Specify a diameter value for the hole to create a cross with a hole, or select N for No Hole to create a centerline cross without a hole, and then specify a diameter value for the cross. You can also select S for Standard Parts to insert a standard hole.
-

List of Prompts

Offset of center-cross: Specifies the distance to a centerline cross from the nearest edge of the closed contour.

Border: Specifies the objects that form the closed contour.

Side to insert: Specifies whether the centerline crosses are placed inside or outside the closed contour. Click inside to draw centerlines within the contour.

Diameter of hole: Specifies the diameter of the holes to draw. Enter S to specify that you want to select the hole from the standard part content library. Specify "No hole" to create centerline crosses without holes.

Centerline cross diameter: Specifies the diameter of the centerline cross to place on the hole.

✧ **Centerline Cross in a Corner****Summary**

Ribbon: Home tab ► Draw panel ► Centerline drop-down ► Centerline Cross in Corner

Keyboard command: GMCENRCORNER

Places a hole at a specified distance from 2 contour lines.

To Create a Centerline Cross in a Corner

1. Click Home tab ► Draw panel ► Centerline drop-down ► Centerline Cross in Corner.
 2. Press ENTER to display the Select Options for Centerlines dialog box, or specify the first contour line.
 3. Specify the second contour line.
 4. Specify the offset of the center-cross from first contour line by using the pointing device or enter a value.
 5. Specify the offset of the center-cross from second contour line by using the pointing device or enter a value.
 6. Press ENTER.
 7. Specify a diameter value for the hole to create a cross with a hole, or press N for No to create a centerline cross without a hole, and then specify a diameter value for the cross, or press S for Standard Parts to insert a standard hole.
-

List of Prompts

Dialog box: Displays the Centerlines dialog box, using which you can change the centerline creation method.

First contour line: Selects the first of two lines required to define a corner.

Second contour line: Selects the second of two lines required to define a corner.

Offset from first contour line: Specifies the distance from the first line to the center of the hole to draw.

Offset from second contour line: Specifies the distance from the second line to the center of the hole to draw.

Diameter of hole: Specifies the diameter of the holes to draw on the pitch circle. Enter S to specify that you want to select the hole from the standard part content library. Specify "No hole" to create centerline crosses without the holes.

Centerline cross diameter: Specify the diameter of the centerline cross to place on the hole.

✧ Centerline In-between

Ribbon: Home tab ► Draw panel ► Centerline drop-down ► Centerline In-between

Keyboard command: GMCENINBET

Draws a centerline in between two lines to mark them as symmetrical.

To Create a Centerline Between Two Lines

1. Click Home tab ► Draw panel ► Centerline drop-down ► Centerline In-between.
 2. Select first contour line (1).
 3. Select second contour line (2).
 4. Press ENTER.
-

List of Prompts

The first contour line: Select first contour line.

The second contour line: Select second contour line.

✧ Centerline Cross in Holes

Ribbon: Home tab ► Draw panel ► Centerline drop-down ► Centerline Cross in Holes

Keyboard command: GMCENCRINHOLE

Places a centerline cross on a circle.

To Create a Centerline Cross in a Hole

1. Click Home tab ► Draw panel ► Centerline drop-down ► Centerline Cross in Holes.
 2. Select the objects to which you want to add a centerline.
 3. Press ENTER.
-

List of Prompts

Select objects: Specifies the holes to draw centerline crosses for. Press ENTER once you have selected the holes.

✧ Centerline Cross With a Hole

Ribbon: Home tab ► Draw panel ► Centerline drop-down ► Centerline Cross with Hole

Keyboard command: GMCENCRHOLE

Places a hole at a specified point.

To Create a Centerline Cross in a Hole

1. Click Home tab ► Draw pane ► Centerline drop-down ► Centerline Cross with Hole.
 2. Press ENTER to display the Select Options for Centerlines dialog box, or specify a starting point for the centerline.
 3. Specify the center point of a hole.
 4. Specify a diameter value for the hole to create a cross with hole. You can enter several values for the diameter separated by the | sign (for example, 1.5|2.5|4|6|12).
 5. Press ENTER.
 6. Specify another location to insert further centerline crosses with holes.
-

List of Prompts

Dialog box: Displays the Centerlines dialog box, using which you can change the centerline creation method.

Specify center point: Specifies the center of the hole.

Specify diameter: Specifies the diameter of the hole. You can specify several values separated by the "|" character, which draws concentric holes for each diameter you specified. Specify "No hole" to create centerline crosses without the holes.

Centerline cross diameter: Specify the diameter of the centerline cross to place on the hole.

✧ Centerline Cross with an Angle

Ribbon: Home tab ► Draw panel ► Centerline drop-down ► Centerline Cross with an Angle

Keyboard command: GMCENCRANGLE

Creates concentric pitch circles and places holes on it at specified angles.

To create a Centerline Cross with an Angle

1. Click Home tab ► Draw panel ► Centerline drop-down ► Centerline Cross with Angles.
2. Press ENTER to select another type of centerline , or specify a center point.
3. Specify a center point for the circle.
4. Specify a diameter value for the circle.
5. Specify a diameter value for the hole if you want to create a cross with hole. Select N for No Hole to create a centerline cross without hole, and then specify a diameter value for cross; or select S for Standard Parts to insert a standard hole.
6. Specify the angle at which the centerline crosses are to be placed on the circle.
7. Press ENTER.

List of Prompts

Dialog box: Displays the Centerlines dialog box, using which you can change the centerline creation method.

Specify center point: Specifies the center of the pitch circle.

Specify diameter: Specifies the diameter of the pitch circle. You can specify several values separated by the "|" character, which draws concentric pitch circles for each diameter you specified.

Diameter of hole: Specifies the diameter of the holes to draw on the pitch circle. Enter S to specify that you want to select the hole from the standard part content library. Specify "No hole" to create centerline crosses without holes.

Centerline cross diameter: Specifies the diameter of the centerline cross to place on the hole.

Angles: Specifies the angle from the X axis at which the centerline cross is to be drawn. You can draw multiple centerline crosses by specifying multiple angles separated by the "|" character.

✧ Centerline Cross on a Full Circle

Ribbon: Home tab ► Draw panel ► Centerline drop-down ► Centerline Cross on Full Circle

Keyboard command: GMCENCRFULLCIRCLE

Creates concentric pitch circles and places a specified number of holes on them, distributed uniformly.

To Create a Centerline Cross on a Full Circle

1. Click Home tab ► Draw panel ► Centerline drop-down ► Centerline Cross on Full Circle.
 2. Specify the center point, or press ENTER to select another type of centerline.
 3. Specify a diameter for the circle by selecting a point on a hole, or enter a value.
 4. Specify a diameter value for the hole to create a cross with a hole, or select N for No Hole to create a centerline cross without a hole, and then specify a diameter value for the cross. You can select S for Standard Parts to insert a standard hole.
 5. Specify the number of centerline crosses you want to draw on the circle's periphery.
 6. Press ENTER.
 7. Specify the rotation angle for the centerline crosses.
 8. Press ENTER.
-

List of Prompts

Dialog box: Displays the Centerlines dialog box, using which you can change the centerline creation method.

Center point: Specifies the center of the pitch circle.

Specify diameter: Specifies the diameter of the pitch circle. You can specify several values separated by the "|" character, which draws concentric pitch circles for each diameter you specified.

Diameter of hole: Specifies the diameter of the holes to draw on the pitch circle. Enter S to specify that you want to select the hole from the standard part content library. Specify "No hole" to create centerline crosses without the holes.

Centerline cross diameter: Specify the diameter of the centerline cross to place on the hole.

Construction line

✧ Construction line

Ribbon: Home tab ► Construction panel ► Construction Lines

Keyboard command: GMCONSTLINES

With the feature of construction line, users may create construction lines extendable infinitely at both ends or ray construction lines at one end according to specific conditions.

To Create a Construction line

To create a horizontal or Vertical or cross construction line

1. Click Home tab ➤ Construction panel ➤ Construction Lines.

2. In the Construction Lines dialog box, select



3. Specify an insertion point for the construction line .

To Create a Construction Line at an Angle Relative to Another Line

1. Click Home tab ➤ Construction panel ➤ Construction Lines.

2. In the Construction Lines dialog box, select

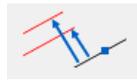


3. Specify a point through which the construction line must pass.
4. Specify a base line by clicking a point.
5. Specify a relative angle by clicking a point or by entering an angle value.

To Create a Construction Line at a Preset Distance From a Selected Line

1. Click Home tab ➤ Construction panel ➤ Construction Lines.

2. In the Construction Lines dialog box, select



3. Select an xline, a ray, or a line.
3. Specify an insertion point, or enter a distance value.
4. If you entered a distance value in step 4, specify the side to offset the construction line from the base line.

To Create a Construction Line Perpendicular to Two Points

1. Click Home tab ➤ Construction panel ➤ Construction Lines.

2. In the Construction Lines dialog box, select



3. Specify a point through which the construction line must pass.
4. Specify a second point, or enter an angle value, to define the direction of the construction line.

To Create a Construction Line Perpendicular to a Line

1. Click Home tab ➤ Construction panel ➤ Construction Lines.

2. In the Construction Lines dialog box, select



3. Select an xline, a ray, or a line.
4. Specify a point through which the construction line must pass.

To Create a Construction Line that Bisects an Angle

1. Click Home tab ► Construction panel ► Construction Lines.
2. In the Construction Lines dialog box, select
3. Select the first, and then the second line of the angle, or press ENTER to define the angle.
4. Select the first, and then the second line of the angle.
5. Note: To define the angle press ENTER, select the vertex of the angle, then select the first and the second endpoint of the vertex.



To Create Two Parallel Construction Lines Tangential to a Circle

1. Click Home tab ► Construction panel ► Construction Lines.
2. In the Construction Lines dialog box, select
3. Select an arc or a circle.
4. Specify a point or enter an angle value to define the direction of the C-lines.



To Create a Construction Line Tangential to two Circles

1. Click Home tab ► Construction panel ► Construction Lines.
2. In the Construction Lines dialog box, select
3. Select the first arc or circle, and then the second arc or circle.



To Create Concentric Circular Construction Lines

1. Click Home tab ► Construction panel ► Construction Lines.
2. In the Construction Lines dialog box, select
3. Select an arc or a circle.
4. Specify a diameter value for the concentric circle. To draw more than one circle, type | (a pipe) between diameter values.



To Create the End View of a Shaft

1. Click Home tab ► Construction panel ► Construction Lines.
2. In the Construction Lines dialog box, select
3. Select the centerline of the shaft.
4. Click a point on the contour side view.
5. Specify an insertion point for the construction circle.



To Circular Construction Line Tangential to two Lines

1. Click Home tab ➤ Construction panel ➤ Construction Lines.
2. In the Construction Lines dialog box, select
3. Select the first line, and then the second line.
4. Specify a diameter value for the circle.



To Circular Construction Line Tangent to a Line

1. Click Home tab ➤ Construction panel ➤ Construction Lines.
2. In the Construction Lines dialog box, select
3. Select an xline, a ray, or a line.
4. Specify a point to define the location of the circle.
5. Specify a radius value.
6. Press ENTER.



List of Prompts

Insertion point: Specifies a point through which the construction line is drawn.

Second point: Specifies the location of the construction line by specifying the second point through which the construction line is drawn.

Angle: Specifies the location of the construction line by specifying the angle the construction line makes with the X axis. You can draw multiple construction lines by specifying multiple angles separated by the "|" character.

Base angle: Creates a construction line with a relative angle from an existing line.

Relative angle: Specifies the location of a construction line by specifying a relative angle from the construction line created before. You can draw multiple construction lines by specifying multiple angles separated by the "|" character.

Select XLINE, RAY or LINE: Creates a construction line parallel to the line you select.

Distance: Draws a construction line at the specified distance from a point selected in a previous prompt. You can draw multiple construction lines by specifying multiple distances separated by the "|" character.

Side to offset: Specifies which side from the previously selected object the command draws the construction line.

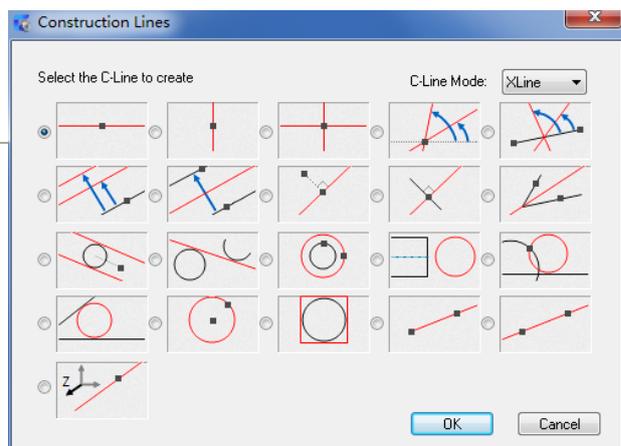
Base point for perpendicular line: Draws a construction line through the point you select. The command draws the construction line perpendicular to a line that you selected in a previous prompt.

First line: Specifies the first of two lines required to draw a construction line that bisects the angle between the two lines

Second line: Specifies the second of two lines required to draw a construction line that bisects the angle between the two lines

Bi-section vertex: Specifies the vertex formed by two lines through which a construction line bisecting it is drawn.

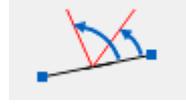
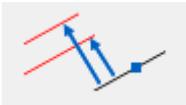
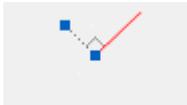
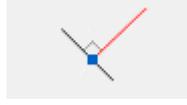
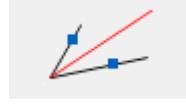
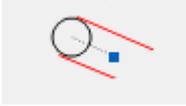
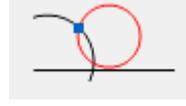
Construction Line Dialog box



Use this dialog box to select from a wide variety of construction line.

Construction Line Mode: Ray

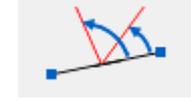
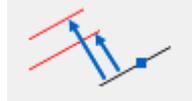
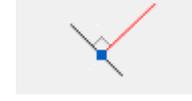
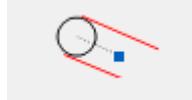
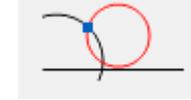
Invokes the GMCONSTSWI command such that construction lines are set to Ray.

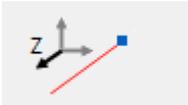
 <p>Draws a horizontal construction line.</p>	 <p>Draws a vertical construction line.</p>	 <p>Draws a construction line cross.</p>	 <p>Draws a construction line when you specify two points or a point and an angle.</p>	 <p>Draws a construction line through a point, by specifying an angle relative to an apparent line that goes through that point.</p>
 <p>Draws a construction line parallel to an existing line at a specified distance.</p>	 <p>Draws a construction line parallel to an existing line and bisects the distance between the selected line and a specified point or a specified distance.</p>	 <p>Draws a construction line that is perpendicular to a specified line.</p>	 <p>Draws a construction line through a specified point that is a perpendicular to the direction specified by another point or angle.</p>	 <p>Draws a construction line that bisects an angle.</p>
 <p>Draws 2 parallel construction lines that are tangential to a specified circle.</p>	 <p>Draws 2 construction lines that are tangential to 2 specified circles.</p>	 <p>Draws a construction line circle that is concentric to a specified circle.</p>	 <p>Draws a construction line circle to represent a top view of a shaft or hole.</p>	 <p>Draws a construction line circle that uses a specified line as a tangent.</p>
				

Draws a construction line that is tangential to two specified circles.	Draws a construction line circle.	Draws a construction line rectangle around a circle.	Creates construction lines starting from one point.	Draws a construction line through a point, which extends to infinity in both directions.
				
Draws a construction line along the z axis.				

Construction Line Mode: Ray

Invokes the GMCONSTSWI command such that construction lines are set to Ray.

				
Draws a horizontal construction line.	Draws a vertical construction line.	Draws a construction line cross.	Draws a construction line when you specify two points or a point and an angle.	Draws a construction line through a point, by specifying an angle relative to an apparent line that goes through that point.
				
Draws a construction line parallel to an existing line at a specified distance.	Draws a construction line parallel to an existing line and bisects the distance between the selected line and a specified point or a specified distance.	Draws a construction line that is perpendicular to a specified line.	Draws a construction line through a specified point that is a perpendicular to the direction specified by another point or angle.	Draws a construction line that bisects an angle.
				
Draws 2 parallel construction lines that are tangential to	Draws 2 construction lines that are tangential to 2	Draws a construction line circle that is concentric to a	Draws a construction line circle to represent a top view	Draws a construction line circle that uses a specified line as a

a specified circle.	specified circles.	specified circle.	of a shaft or hole.	tangent.
				
Draws a construction line that is tangential to two specified circles.	Draws a construction line circle.	Draws a construction line rectangle around a circle.	Creates construction lines starting from one point.	Draws a construction line through a point, which extends to infinity in both directions.
				
Draws a construction line along the z axis.				

❖ **Automatic Construction Lines**

Ribbon: Home tab ➤ Construction panel ➤ Construction Lines drop-down ➤ Automatic Construction Lines

Keyboard command: GMAUTOCLINES

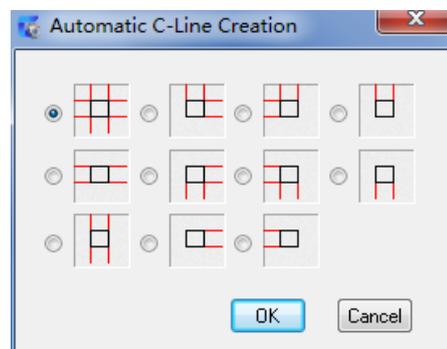
Creates vertical and horizontal construction lines on all endpoints of selected objects.

To Create Construction Lines Automatically

1. Click Home tab ➤ Construction panel ➤ Construction Lines drop-down ➤ Automatic Construction Lines.
2. In the Automatic C-line Creation dialog box, choose the desired icon.
3. Select the object on which you want to insert the construction lines .

Automatic Construction Lines Dialog box

Use this dialog box to create vertical or horizontal construction lines around objects you specify. Click the button corresponding to the construction line style you require. After you select the objects, the program draws the construction lines.



❖ **Erase Construction Lines**

Ribbon: Home tab ➤ Construction panel ➤ Erase Construction Lines

Keyboard command: GMERASECL

Erases selected construction lines.

To Erase Construction Lines

There are two options to delete construction lines.

1. Delete all construction lines from the drawing.
2. Delete only selected construction lines.

Use GMERASEALLCL to delete all construction lines from a current drawing.

Use GMERASECL to delete selected construction lines from a current drawing.

1. First corner of window: Select a point.
 2. Second corner of window: Select a point.
-

List of Prompts

First corner of window: Selects the first corner of a region that contains the construction lines to erase.

Second corner of window: Selects the opposite corner of the region.

✧ Projection On/Off

Ribbon: Home tab ► Construction panel drop-down ► Projection On/Off

Keyboard command: GMPROJO

List of Prompts

Projection: Turns off or on the creation of projection lines.

✧ Tracing Contours

Ribbon: Home tab ► Construction panel drop-down ► Contour Trace

Keyboard command: GMCONTRACE

Creates a closed contour that traces the contour of an area by letting you specify the boundaries segment by segment.

Commonly Steps

To Check Whether a Contour is Closed or not

Click Home tab ► Construction panel drop-down ► Contour Trace.

To check the outer contour of an area:

- 1) At the Command prompt, select Outside.
- 2) Select the objects that form the closed loop.
- 3) Press ENTER.
- 4) To use any entities on underlying layers as additional boundaries, select them.

- 5) Press ENTER again.

To check the inner contour of an area:

- 1) Click in the area.

In the Message dialog box, click No.

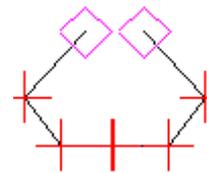
The system traces the contour, pausing at each vertex to mark it. If an open end is found, a small square is drawn around that point and the tracing direction reverses. If another open end is found in the same place when tracing in the reverse direction that is where the contour is open.

Each time the system pauses type Y and press ENTER. Repeat until the entire contour has been traversed and then type N and press ENTER.

Note: To remove the vertex markers, type REDRAW at the command prompt and press ENTER.

To Trace the Outer Contour of an Enclosed Area, Segment by Segment

1. Click Home tab ► Construction panel drop-down ► Contour Trace.
2. At the Command prompt, select Outside.
3. Select the objects that form the closed loop.
4. Press ENTER.
5. If you want to use any entities on underlying layers as additional boundaries, select them.
6. Press ENTER again.
7. In the Message dialog box, click No.



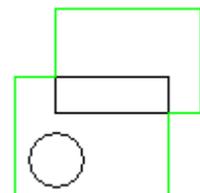
The system traces the contour, pausing at each vertex to mark it.

8. Each time the system pauses type Y and press ENTER. Repeat until the entire contour is drawn.

Note: To remove the vertex markers, type REDRAW at the command prompt and press ENTER.

To Trace the Outer Contour of an Enclosed Area

1. Click Home tab ► Construction panel drop-down ► Contour Outside.
2. Select the objects that bound the area you want to trace.
3. To use any entities on underlying layers as additional boundaries, select them.
4. Press ENTER.



To Trace the Inner Contour of an Enclosed Area, Segment by Segment

1. Click Home tab ► Construction panel drop-down ► Contour Trace.

2. At the Command prompt, select Inside.
3. Click in an area to define the boundaries of the contour you want to trace.
4. In the Message dialog box, click No.

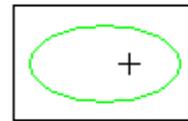
The system traces the contour, pausing at each vertex to mark it.

5. Each time the system pauses type Y and press ENTER. Repeat until the entire contour is drawn.

Note: To remove the vertex markers, type REDRAW at the command prompt and press ENTER.

To Trace the Inner Contour of an Enclosed Area

1. Click Home tab ► Construction panel drop-down ► Contour Inside.
2. To define the area by selecting the objects on the boundary:



Type S and press ENTER.

Select the objects and press ENTER when you finish.

3. If you want to use any entities on underlying layers as additional boundaries, select them.
4. Click inside the enclosed area. A contour is drawn along the boundary.

List of Prompts

Inside: Traces the inner contour of an area.

Point in area: Selects the objects around the point you click.

Select objects: Specifies the area by selecting the objects that bound the area.

Select additional boundary: Specifies additional boundaries such as entities on underlying layers.

Outside: Traces the outer contour of an area.

Select objects: Specifies the area by selecting the objects that bound the area.

Select additional boundary: Specifies additional boundaries such as entities on underlying layers.

Smart Line

Ribbon: Home tab ► Draw panel ► Line drop-down ► Smart Line

Keyboard command: GMSMARTLINE

It is an upgraded version of the traditional Line feature, allowing users to draw a line which is vertical, parallel to, or form any angle with the reference line.

To Create a Smart line

1. Click Home tab ► Draw panel ► Line drop-down ► Smart Line.
2. Specify the first point location .
3. Specify the next point location, or select the target for orthogonal/angle line, and proceed to the next step.
4. Specify the next point location.

Parallel /Perpendicular

Menu: Draw ► Mechanical draw ► Parallel/Perpendicular

Keyboard command: GMPARALLELLINE

This allows drawing a line which is perpendicular \parallel to the reference line.

To create a paralleline

1. Click Menu Draw ► Mechanical draw ► Paralleline/Perpendicular.
 2. Select the target for paralleline.
 3. Specify the start point of paralleline, or a distance which is parallel to the target line, and then select the start point on the side of target line.
 4. Specify the end point of paralleline.
-

List of Prompts

Select line to parallel/perpendicular to: Select the target line in drawing.

Specify start point of parallel line or enter distance: Select one point in drawing as the start point of parallel line

Specify end point or enter length: Select one point from drawing as the end point of parallel line

Tangent Line

Menu: Draw ► Mechanical draw ► Tangent Line

Keyboard command: GMTANGENTLINE

This allows drawing a line which is tangential to a circle or arc, or forms a certain angle with the tangent of the circle or arc.

To Create a Tangent Line

1. Click Menu Draw ► Mechanical draw ► Tangent Line.
2. Select the start point of tangent on the target.
3. Specify the end point of tangent.

List of Prompts

Select point on circle (arc, ellipse): Select a point on the circle or arc in drawing.

Specify end point of tangent or [input Angle]: Select one point in drawing as the end point of tangent.

Input Angle: Specify the relative angle between the line being drawn and the tangent.

Input Angle: Enter the value of angle formed with the tangent.

Specify end point: Specify the end point of angle line in drawing.

Common Tangent Line

Menu: Draw ► Mechanical draw ► Common Tangent Line

Keyboard command: GMCOMMONTANGENT

To Create a Common Tangent Line

1. Click Menu Draw ► Mechanical draw ► Common Tangent Line.
 2. Select the first target for common tangent.
 3. Select the second target for common tangent or select it after selecting the orientation and location.
-

List of Prompts

Specify first point:

Specify next point or [Angle/Length/Undo]:

Select tangent to draw (dashed line means not select): <Not select>

Perpendicular Line

Menu: Draw ► Mechanical draw ► Perpendicular Line

Keyboard command: GMPERPBISECTOR

This allows drawing a line which is perpendicular to and bisects the reference line.

To Create a Perp Bisector Line

1. Click Menu Draw ► Mechanical draw ► Perpendicular Line.
 2. Select the target for perpendicular bisector.
 3. Specify the start point of perpendicular bisector.
 4. Specify the end point of perpendicular bisector or the target for intersection at the end of perpendicular bisector.
-

List of Prompts

Select base line: Select a base line in the area.

Specify starting point of perpendicular line/<Midpoint>/0 Snap type: Select one point from drawing as the start point of perpendicular line.

Enter length/<Second point>/0 Snap type :

Angle Line

Menu: Draw ► Mechanical draw ► Angle Line

Keyboard command: GMANGLELINE

Draw a line which forms a specified angle with reference line.

To Create an Angle Line

1. Click Menu Draw ► Mechanical draw ► Angle Line.
 2. Select the target for angle line.
 3. Specify the start point of angle line.
 4. Specify the end point of angle line or the target for intersection at the end of angle line.
-

List of Prompts

Select base line: Select a base line in the area.

Specify start point: Select one point in the area.

Enter angle (counterclockwise 0—360)/<Specify a point>/0 Snap ON: Input the angle

Angular Bisector Line

Ribbon: Draw ► Mechanical draw ► Angular Bisector Line

Keyboard command: GMBISECTOR

Draw a bisector or average bisector of included angle.

To Create an Angular Bisector Line

1. Click Menu Draw ► Mechanical draw ► Angular Bisector Line.
 2. Select the second side of bisector as target.
 3. Specify bisection number.
 4. Specify the end point of bisector.
-

List of Prompts

Select first line: Select one side of the angle in drawing

Select second line: Select another side of angle in drawing

Input the number of equal portions<2> Enter the bisection number of angular bisector

Length: Select one point from drawing as the end point of angular bisector or enter its length value.

Radiation

Menu: Draw ► Mechanical draw ► Radiation

Keyboard command: GMRADIATION

To Create Radiation

1. Click Menu Draw ► Mechanical draw ► Radiation.
 2. Set the number of rays.
 3. Select the center point.
 4. Select the end point of ray.
-

List of Prompts

Input Rays number<3> Enter the number of rays.

Center point or[select Circle]<C> Select a point or circle in drawing.

Specify end point: Select one point in drawing as the end point of ray or enter its length value.

Wavy line

Ribbon: Home tab ► Detail panel ► Wavy line

Keyboard command: GMWAVILNESSLINE

Select a start point and end point and set the number of segments to draw a waviness line between two points; when multiple points are set, multiple waviness lines can be drawn.

To Create a Wavyline

1. Click Home tab ► Detail panel ► Wavyline.
 2. Specify the start point of waviness line.
 3. Specify the end point of waviness line.
 4. Specify the multi-segment value of line segments.
-

List of Prompts

Specify start point : Select one point from drawing as the start point of waviness line.

Specify end point: Select one point from drawing as the end point of waviness line.

Input the number of bands<2> Enter the number of wave segments for waviness line.

Zigzag Line

Ribbon: Home tab ► Draw panel ► Line drop-down ► Zigzag Line

Keyboard command: GMZIGZAGLINE

To Create a Zigzag Line

1. Click Home tab ► Draw panel ► Line drop-down ► Zigzag Line.
 2. Specify the start point of zigzag line.
 3. Specify the next point of zigzag line.
-

List of Prompts

Input first point: Select one point from drawing as the start point of zigzag line.

Specify next point or[Undo] Select one point from drawing as the next point of zigzag line, and press ENTER to finish drawing.

Undo: Abandon previous operation.

Pipeline

Menu: Draw ► Mechanical draw ► Pipeline

Keyboard command: GMPIPELINE

This allows drawing a pipeline by setting its outer diameter and wall thickness.

To create a Pipeline

1. Click Menu Draw ► Mechanical draw ► Pipeline.
 2. Specify pipeline outer diameter.
 3. Specify pipeline wall thickness.
 4. Choose whether or not to draw a center line.
 5. Specify the start point of pipeline.
 6. Specify the next point of pipeline or specify it after specifying the pipeline graphics, and press ENTER to complete drawing pipeline.
-

List of Prompts

Specify pipe outer Radius<10> Enter pipeline outer diameter.

Specify pipe thickness<0> Enter the wall thickness of pipeline.

Draw centerline [Yes/No]<Y> Choose whether or not to draw a center line.

Specify start point: Select one point in drawing as the start point of pipeline.

Specify next point or [Arc/Diameter/Undo] Select one point in drawing as the end point of pipeline, and complete drawing.

Arc: Specify the pipeline style to be an arc.

Input start angle of pipe<90> The start point of pipeline is the value of start angle to be specified for arc.

Specify end point of arc or [Radius] Specify the end point or radius of arc

Specify Radius of arc: Specify the radius of arc

Specify inclusive angle of arc: Specify the inclusive angle of arc

Diameter: Specify end of pipe diameter

Undo: Abandon the previous operation

Rectangle

Ribbon: Home tab ► Draw panel ► Rectangle

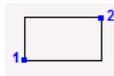
Keyboard command: GMRECTANG

Creates a rectangular polyline.

Commonly Steps

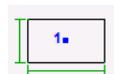
To Create a rectangle starting with the one corner and by specifying the opposite corner

1. Click Home tab ► Draw panel ► Rectangle.
2. In the "Rectangle" dialog box, select .
3. Specify first corner.
4. Specify second point.

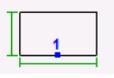


To Create a rectangle by using the center of the rectangle as the start point, and by specifying the full base and full height

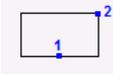
1. Click Home tab ► Draw panel ► Rectangle.
2. In the "Rectangle" dialog box, select .
3. Specified the center point of the rectangle.
4. Specified rectangle hemline or enter the bottom value.
5. Specified rectangle around the high or high value input full.



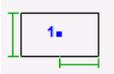
To Create a rectangle by selecting the middle of the base as the start point, and by specifying the full base and full height

1. Click Home tab ➤ Draw panel ➤ Rectangle.
2. In the "Rectangle" dialog box, select. 
3. Specify the center point of the rectangle.
4. Specify rectangle hemline or enter the bottom value.
5. Specify rectangle around the high or high value input full.

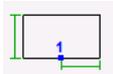
To Create a rectangle by selecting the middle of the base as the start point, and by specifying a corner point

1. Click Home tab ➤ Draw panel ➤ Rectangle.
2. In the "Rectangle" dialog box, select. 
3. Specify the bottom edge of the rectangle midpoint.
4. Specify corner of the rectangle.

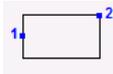
To Create a rectangle by using the center of the rectangle as the start point, and by specifying half of the base and half of the height

1. Click Home tab ➤ Draw panel ➤ Rectangle.
2. In the "Rectangle" dialog box, select. 
3. Specify center point of the rectangle.
4. Specified the bottom half of the half the rectangle hemline or input rectangular value.
5. Specified rectangle full high or input rectangular all high value.

To Create a rectangle by using the middle of the base as the start point, and by specifying the full base and half of the height

1. Click Home tab ➤ Draw panel ➤ Rectangle.
2. In the "Rectangle" dialog box, select. 
3. Specified the rectangle hemline midpoint.
4. Specified the bottom half of the half of the rectangle hemline or input rectangular value.
5. Specified rectangle around the high or high value input full.

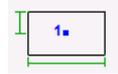
To Create a rectangle by using the midpoint of the height as the start point, and by specifying the opposite corner

1. Click Home tab ➤ Draw panel ➤ Rectangle.
2. In the "Rectangle" dialog box, select. 

3. Specified the midpoint of the rectangle height.
4. Specified diagonal point of the rectangle.

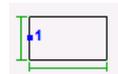
To Create a rectangle by using the center of the rectangle as the start point, and by specifying the full base and half of the height

1. Click Home tab ► Draw panel ► Rectangle.
2. In the "Rectangle" dialog box, select.
3. Specified the center point of the rectangle.
4. Specified rectangle hemline or enter the bottom value.
5. Specified rectangle around the high or high value input full.



To Create a rectangle by using the midpoint of the height as the start point, and by specifying the full base and full height

1. Click Home tab ► Draw panel ► Rectangle.
2. In the "Rectangle" dialog box, select.
3. Specified the center point of the rectangle.
4. Specified rectangle hemline or enter the bottom value.
5. Specified rectangle around the high or high value input full.



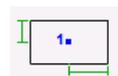
To Create a rectangle by using the center of the rectangle as the start point, and by specifying corner

1. Click Home tab ► Draw panel ► Rectangle.
2. In the "Rectangle" dialog box, select.
3. Specified the midpoint of the rectangle height.
4. Specified diagonal point of the rectangle.



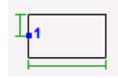
To Create a rectangle by using the center of the rectangle as the start point, and by specifying half of the base and half of the height

1. Click Home tab ► Draw panel ► Rectangle.
2. In the "Rectangle" dialog box, select.
3. Specified the rectangle hemline midpoint.
4. Specified the bottom half of the half of the rectangle hemline or input rectangular value.
5. Specified rectangle around the high or high value input full.



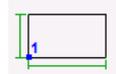
To Create a rectangle by selecting the midpoint of the height as the start point, and by specifying the full base and half of the height

1. Click Home tab ► Draw panel ► Rectangle.
2. In the "Rectangle" dialog box, select.
3. Specified the center point of the rectangle.
4. Specified rectangle hemline or enter the bottom value.
5. Specified rectangle around the high or high value input full.



To Create a rectangle by using a corner of the rectangle as the start point, and by specifying the full base and full height

1. Click Home tab ► Draw panel ► Rectangle.
2. In the "Rectangle" dialog box, select.
3. Specified corner of the specified rectangle.
4. Specified rectangle hemline input value or the bottom edge.
5. Specified rectangle full high or high value input full.



To Create a square using with the midpoint of the base as the start point and by specifying the full base

1. Click Home tab ► Draw panel ► Rectangle.
2. In the "Rectangle" dialog box, select.
3. Specify square hemline midpoint.
4. Specify the base and the bottom edge of square input value.



To Create a square using the midpoint of the height and by specifying the full base

1. Click Home tab ► Draw panel ► Rectangle
2. In the "Rectangle" dialog box, select
3. Specify the midpoint of the designated square height.
4. Specify the base and the bottom edge of square input value.



To Create a square using the midpoint of the base as the starting point and by specifying half of the base (width)

1. Click Home tab ► Draw panel ► Rectangle.
2. In the "Rectangle" dialog box, select.
3. specify square hemline midpoint.
4. specifies the square half the hem or input half bottom value.



To Create a square using the midpoint of the height as the starting point and specifying half of the base (width)

1. Click Home tab ➤ Draw panel ➤ Rectangle
2. In the "Rectangle" dialog box, select
3. Specifies the height of the midpoint of the square.
4. Specifies the square half the hem or input half bottom value.



To Create a square using the center of the square as the starting point and by specifying half of the base (width)

1. Click Home tab ➤ Draw panel ➤ Rectangle.
2. In the "Rectangle" dialog box, select.
3. specify center point of the square.
4. specifies the square half the hem or input to half bottom value.



To Create a square using a corner of the base as the starting point and by specifying the base (width)

1. Click Home tab ➤ Draw panel ➤ Rectangle.
2. In the "Rectangle" dialog box, select.
3. specifies the square bottom corner point.
4. specify the base and the bottom edge of square input value.



To Create a square using the center of the square as the starting point and by specifying the base (width)

1. Click Home tab ➤ Draw panel ➤ Rectangle.
2. In the "Rectangle" dialog box, select.
3. specify center point of the square.
4. specify the base and the bottom edge of square input value.



List of Prompts

First corner point: Defines the starting point to draw the rectangle. Press the SPACEBAR to switch to the starting point position. A glyph at the cursor indicates the current start point.

Other corner point: Defines the second point required to draw the rectangle. When dimensional input is on, tooltips display distance to the second point. You can Press TAB to move to the value you want to change and specify the distance to the second point using the keyboard.

Angle: Rotation angle - Specifies how far to rotate the rectangle. Specify a value for the angle or click two points in the drawing area to define the direction

Base: Draws a rectangle when you specify the midpoint of the base and a corner on the opposite side.

Height: Draws a rectangle when you specify the midpoint of the height and a corner on the opposite side.

Center: Draws a rectangle when you specify the center of the rectangle and a corner.

Dialog: Displays a dialog box with various options to create squares and rectangles..

Hatch

✧ User-Defined Hatch

Ribbon: Home tab ► Draw panel ► Hatch drop-down ► User-Defined Hatch

Keyboard command: GMUSERHATCH

This allows filling certain patterns to represent the feature of relevant region. Its operation is identic with that in GstarCAD, please refer to the part of hatch pattern in GstarCAD.

To Create a Non-Associative Hatch

1. Click Home tab ► Draw panel ► Hatch drop-down ► User-Defined Hatch.
2. From the drop-down, click the desired predefined hatch pattern.
3. Select the area you want to hatch.
4. Click in the area.

List of Prompts

Additional boundary: Defines the area to hatch by selecting its boundaries.

Area to be hatched: Defines the area to hatch by selecting a point within an enclosed area.

Objects: Defines the area to hatch by selecting the object to hatch.

Hatch Dialog box

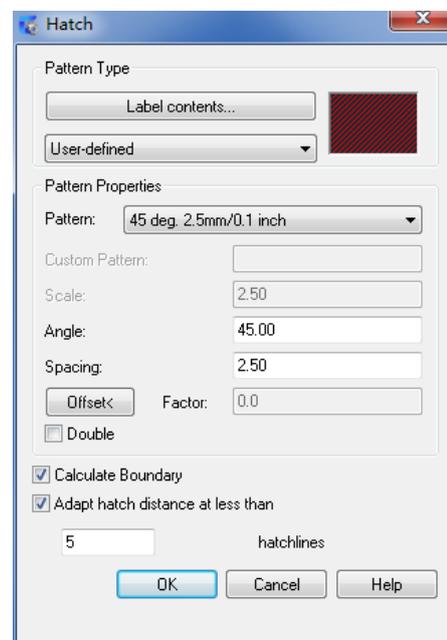
Pattern Type

Selects the type of pattern to use.

Pattern: Shows a palette of preview images of the selected hatch pattern.

Preview window: Shows a preview image of the selected hatch pattern.

Pattern types: Specifies the pattern type. Types of patterns are



User-defined, Custom, and Predefined.

Pattern Properties

Modifies the properties of the selected hatch pattern.

Pattern: Lists available hatch patterns for the selected pattern type.

Custom Pattern: Specifies a predefined pattern in the acad.pat file or a custom pattern in its own PAT file.

Scale: Expands or contracts a predefined or custom pattern. GstarCAD stores the scale in the HPSCALE system variable. This option is available only if you set the pattern type to Predefined or Custom.

Angle: Specifies the angle for the hatch pattern in relation to the X axis of the current UCS . GstarCAD stores the angle in the HPANG system variable.

Spacing: Specifies the spacing of lines in a user-defined pattern. GstarCAD stores the spacing in the HPSPACE system variable. This option is available only if you set the pattern to User-defined.

Offset: Shifts a hatch pattern in relation to an adjacent hatch.

Double: Draws lines at 90-degree angles to the original.

Calculate Boundary

Calculates the new boundaries of a hatch area when editing.

Adapt hatch distance at less than 5 hatch lines

Specifies the number of hatch lines that the program draws when the area to hatch is very small. The availability of this option depends on the setting in the configuration. The number of hatching lines is 5 by default.

❖ **GMHATCH_45_2**

Fills an enclosed area with a 45-degree, 2.5 mm/0.1" non-associative hatch.

❖ **GMHATCH_45_5**

Fills an enclosed area with a 45-degree, 5 mm/0.22" non-associative hatch.

❖ **GMHATCH_45_13**

Fills an enclosed area with a 45-degree, 13 mm/0.5" non-associative hatch.

❖ **GMHATCH_135_2**

Fills an enclosed area with a 135-degree, 2.7 mm/0.12" non-associative hatch.

❖ **GMHATCH_135_4**

Fills an enclosed area with a 135-degree, 4.7 mm/0.19" non-associative hatch.

❖ **GMHATCH_135_11**

Fills an enclosed area with a 135-degree, 11 mm/0.4" non-associative hatch.

❖ **GMHATCH_DBL**

Fills an enclosed area with a 45 and 135 degree, 2.3 mm/0.09 non-associative cross hatch.

Chapter 3 Edit Geometry

Summary

In mechanical drawing, often encounter chamfer, process, such as breaking groove shaft work, also relates to design the hole, hole arrangement problem, auto customization for handling these problems a number of tools, improve the efficiency of these specific parts processing.

Fillet

Ribbon: Home tab ► Modify panel ► Fillet

Keyboard command: GMFILLET2D

Rounds and fillets the edges of objects with an arc of a specified radius.

Commonly Steps

To Fillet two Line Segments

1. Click Home tab ► Modify panel ► Fillet.
2. Select the first object you want to fillet.
3. Select the second object.

2. Press Enter to display the Fillet dialog box.

3. Clear the Trim geometry check box.

4. Click OK.

5. Select the first object.

6. Select the second object.

To Fillet Entire Polyline

1. Click Home tab ► Modify panel ► Fillet.
2. Enter P for polyline.
3. Select the polyline.

To Configure the Default Value for Fillets

1. Click Home tab ► Modify panel ► Fillet.

2. Press Enter to display the Fillet dialog box.

3. In the Fillet size list, select the default value.

4. Click OK.

To Fillet Without Trimming

1. Click Home tab ► Modify panel ► Fillet.

List of Prompts

Object: Fillets the vertex by selecting the objects that form the vertex.

Polyline: Fillets all vertexes of a polyline you select.

Select polyline: Specifies the polyline to fillet.

Setup: Displays the Fillet dialog box, enabling you to specify the fillet radius.

Dimension: Creates a dimension for a filleted edge.

Fillet arc: Select the fillet to dimension.

Dimension line location: Specifies the starting and endpoint of the dimension line.

Options - Displays the Radius Dimension Options dialog box.

Fillet Dialog Box

Size

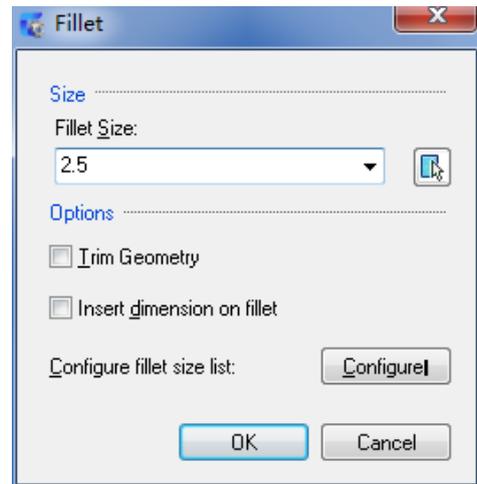
Fillet Size: Displays a drop-down list of common radii to select from. You may also enter a radius value. Reverts to the drawing so you can specify a radius value by selecting an existing arc or circle or by entering a value at the command prompt.

Options

Trim geometry: Trims the intersecting lines to the endpoints of the fillet line. This also sets the TRIMMODE system variable to 1.

Insert dimension on fillet: Inserts dimensions for the fillet created.

Configure: Opens the Fillet List Configuration dialog box where you can customize the fillet size drop-down list.



Chamfer

Ribbon: Home tab ► Modify panel ► Fillet drop-down ► Chamfer

Keyboard command: GMCHAM2D

Connects two non-parallel objects by extending or trimming them to intersect or to join with a beveled line.

Commonly Steps

To Chamfer Objects by Specifying Distances

1. Click Home tab ► Modify panel ► Fillet drop-down ► Chamfer.
2. Press ENTER for Setup.
3. In the Chamfer dialog box:
4. Select a value for the first length from the First chamfer length list or click  to specify the length by clicking two points, or enter a value.
5. Select a value for the second length from the Second chamfer length list or click  to specify the length by clicking two points, or enter a value.
6. Click OK.
7. Select the first object.
8. Select the second object.
9. Press ESC to exit the command.

To Chamfer Entire Polylines

1. Click Home tab ► Modify panel ► Fillet drop-down ► Chamfer.
2. Press ENTER for Setup.
3. In the Chamfer dialog box:

Select a value for the first length from the First chamfer length list or click  to specify the length by clicking two points, or enter a value.

Select a value for the second length from the Second chamfer length list or click  to specify the length by clicking two points, or enter a value.

4. Click OK.
5. Select the polyline.
6. Press ENTER.
7. Press ESC to exit the command.
8. Note: You can't trim trimmed objects, when the trim mode is on.

To Configure the Default Values for Chamfer

1. Click Home tab ► Modify panel ► Fillet drop-down ► Chamfer.
2. Press ENTER for Setup.
3. In the First chamfer length list and the Second chamfer length list, select values for the chamfer.
4. Click OK.
5. Press ESC to terminate the command.

To Chamfer With Dimension

1. Click Home tab ► Modify panel ► Fillet drop-down ► Chamfer.
2. At the command prompt, enter the D .I
3. Choose to chamfer line marking, and then select the first and second dimension line. Will display the Chamfer dimensions
4. Press the ENTER key.

List of Prompts

First object: Specifies the first of two lines

First object: Creates a chamfer by connecting two lines to connect with a beveled line. Select the first of the two lines required to define the chamfer.

Second Object: Specifies the second of the two lines required to define the chamfer.

Polyline: Chamfers all vertexes of a polyline.

Select polyline: Specifies the polyline to modify.

Setup: Shows the Chamfer dialog box, enabling you to specify the chamfer lengths and chamfer dimension representations.

Add Dimension: Creates dimensions for a chamfer.

Chamfer Dialog Box

Size

First Chamfer Length: Displays a list of values that you can use to specify the distance of the first chamfer from the selected edge. The button reverts to the drawing, where you can specify a value for the first chamfer distance. This value is displayed in the Input field.

Second Chamfer Length: Displays a list of common values that you can use to specify the distance of the second chamfer from the selected edge. The button reverts to the drawing, where you can specify a value for the second chamfer distance. This value is displayed in the Input field.

Chamfer Angle: Displays a list of available angle values that you can use to specify the distance of the second chamfer from the selected edge. The button reverts to the drawing, where you can specify a chamfer angle. This value is displayed in the Input box.

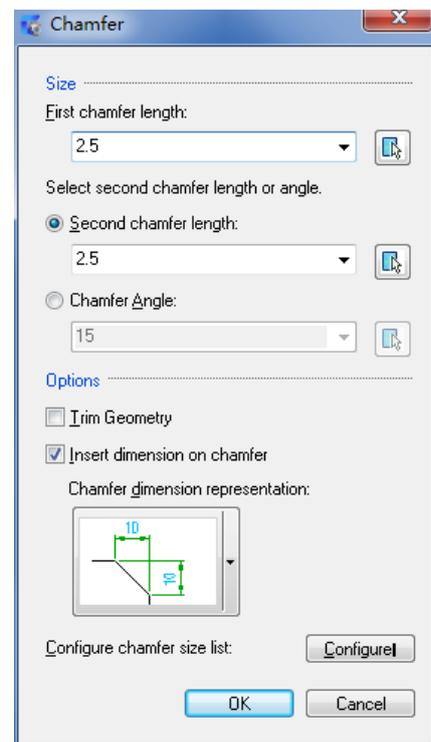
Options

Trim Geometry: Sets the GMCHAM2D command to trim the selected edges to the chamfer line endpoints

Insert dimension on chamfer: Inserts a dimension when you create the chamfer.

Chamfer dimension style: Shows an image of the currently selected chamfer dimension representation. Click the drop-down arrow to select a different chamfer dimension representation.

Configure: Displays the Chamfer List Configuration dialog box that enables you to customize the list of chamfer sizes.



Chamfer List Configuration Dialog Box

Chamfer Sizes: Displays a list of chamfer sizes that are available for selection.

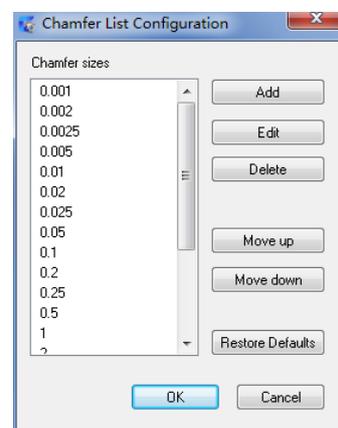
Add: Inserts a blank entry to the Chamfer sizes list and enables you to specify a value.

Edit: Enables you to modify the chamfer size of the selected entry.

Delete: Deletes the selected entry.

Move up: Moves the selected entry one position up.

Move down: Moves the selected entry one position down.



Restore Defaults: Resets the chamfer list to its default settings. Customizations made to this list are lost.

Break

Menu: Modify ► Break ► GM Break

Keyboard command: GMBREAKATPT

Breaks a line, polyline, at a specified point.

To Break an Object at Selected Points

1. At the Command prompt, enter GMBREAKATPT.
 2. Select point marked 1 as the break point.
 3. Select the parts of the lines marked 2. The lines are deleted and the result is shown on the right.
-

List of Prompts

Select line, circle or arc to break

Specify break point(uSe an object to break)

Dynamic Extension

Menu: Modify ► Dynamic Extension

Keyboard command: GMDYNGMICEXTEND

Extends an object to meet another object.

To Dynamic Extension Entity

1. At the Command prompt, enter GMDYNGMICEXTEND.
 2. Select an arc, circle or line which requires dynamic extension as the target
 3. Specify the end point for target extension and complete extension
-

List of Prompts

Select object: Select an arc, circle or line from a drawing as target

Intersect

Menu: Modify ► Dynamic Extension

Keyboard command: GMINTERSECT

Trims line or arc intersecting, and deletes superfluous sections.

To Intersect Objects

1. At the Command prompt, enter GMINTERSECT.
 2. Select the arc, line.
-

List of Prompts

Select the first entity: Select arc, line in engineering drawing as the goal.

Select the second entity: Select the second arc, line.

Truncation Line

Menu: Draw ► Truncation Line

Keyboard command: GMSECTIONSYPBOL

The geometry of the shaft breakage treatment.

List of Prompts

1. At the Command prompt, enter GMSECTIONSYPBOL.
2. Select first generatrix of shaft.
3. Select second generatrix of shaft.
4. Specify cutoff position.

Scale X,Y

Menu: Modify ► Zoom scale ► Scale xy

Keyboard command: GMSCALEXY

Sets scale of X and Y axes independently of each other.

To Scale objects

1. At the Command prompt, enter GMSCALEXY.
 2. Specifies the objects to rescale.
 3. Press ENTER.
 4. Specify a base point for the scale operation.
 5. Determines the X/Y axis scale factor.
-

List of Prompts

Select objects: Specifies the objects to rescale.

Base point: Specify a base point for the scale operation.

The base point you specify identifies the point that remains in the same location as the selected objects change size (and thus move away from the stationary base point).

X-axis scale factor: Specifies the scale factor for the X direction.

Y-axis scale factor: Specifies the scale factor for the Y direction.

Reference

Uses two points to specify the reference length and two points to specify a new length. The relation between the lengths is the scale factor.

- **Reference length** - Specifies the first point for the reference distance.
- **Second point** - Specifies the second point for the reference distance.
- **New length** - Specifies the first point for the new distance.
- **Second point** - Specifies the second point for the new distance.

One-quarter Mirror

Menu: Modify ► Mirror ► One-quarter Mirror

Keyboard command: GMMIRROR4

Creates three mirror image copy of objects.

To Create mirror image copy of objects

1. At the Command prompt, enter GMMIRROR4.
 2. Select objects.
 3. Press ENTER.
 4. Specify a base point for the mirror operation.
-

List of Prompts

Select object

Select (Enter) one-quarter mirror point

Continuous Copy

Keyboard command: GMCONTINUOUSCOPY

It's stronger than copy of CAD, and it can remember the distance and direction.

To Copy Objects Continuously

1. At the Command prompt, enter GMMIRROR4.
 2. Select objects.
 3. Press ENTER.
 4. Specify a base point for the copy operation.
 5. Specify a second point .
 6. Specify the next point, or press ENTER.
 7. Press ESC to finish.
-

List of Prompts

Select object: In engineering drawing geometry selection need continuous replication.

Specify base point: Specify a base point for the copy operation.

Specify second point or < use first point as displacement >

Specify second point. Right click on default distance copy, use the ESC key to exit.

Merge Overlapped

Menu: Modify ► Erase ► Merge Overlapped

Keyboard command: GMERASEREPETITION

Cleans up overlapping geometry by removing duplicated and/or unneeded objects.

To Erase Overlapped Objects

1. At the Command prompt, enter GMERASEREPETITION.
 2. Select Objects.
 3. Press ENTER.
-

List of Prompts

Select objects

Select the objects to erase

Chapter 4 Drawing and Layout

Summary

GstarCAD Mechanical 2017 provides for inserting graphics borders and title bar in the drawing area command. The user can insert a graphic frame at any stage of the design process. The user can specify the title bar, and the proportion of property sheet format, you can also specify the height and width of the graphics area, and let GstarCAD Mechanical to calculate the proportion. If you have been drawing must be located in the graphic frame geometry and annotation, the user can use the command line option to select, re scaling and graphic frame in moving it to the appropriate location.

The use of model space and layout, multiple views of the user can create the same model with different proportion. The main advantage of using a layout is the view relevance. If the user changes in a viewport, all other viewport will occur in the same change, because each viewport is another view of the same model. You can also freeze object new viewport, and will not affect other objects in the view.

Drawing Title/ Borders

Ribbon: Annotate tab ► Sheet panel ► Title Border

Keyboard command: GMTITLE

GstarCAD Mechanical 2017 provides multi standard element set. Completely solve the customer drawings communication standard is inconsistent with foreign elements. For example: GB, ISO, ANSI, DIN, BSI, JIS, CSN and other countries standards. At the same time, completely solve the needs of the many layouts drawing of single DWG designer.

Commonly Steps

To Insert a Drawing Border with Title Block

1. Click Annotate tab ► Sheet panel ► Title Border.
2. In the Drawing Borders with Title Block dialog box, Click OK.
3. Specify the insertion point of the drawing border.

To Edit a Drawing Border

1. Double-click the drawing border to edit.
2. In the Drawing Border with Title Block dialog box, edit the settings.
3. Click OK.
4. Select the objects to move into the changed drawing border.

To Edit a Title Block

1. Double-click the title block to edit.
2. In the Change Title Block Entry dialog box, edit the settings.
3. Click OK.

List of Prompts

Insertion point: Specifies where to insert the bottom left corner of the drawing border.

Select objects: Specifies what objects to rescale and move inside the drawing border.

New location for objects: Specifies where to insert the objects.

Drawing Borders With Title Block Dialog Box

Paper Format

Contains standardized drawing borders, saved in the title border folder. You can use one of the standardized format or select a customer format. To use a custom format, start the GMOPTIONS command and double-click Drawing Sheets in the IM:Standards dialog box. In the Drawing Sheet Settings dialog box, under Default Paper Format, choose Additional to select and add a customer paper format to the existing list.

Title Block

Contains standardized title blocks, which are saved in the title border folder. You can use one of the standardized format or select and use a customer format. To use a custom format, start the GMOPTIONS command and double-click Drawing Sheets in the IM:Standards dialog box. In the Drawing Sheet Settings dialog box, under Default Title Block, choose Additional to select and add a customer paper format to the existing list.

Extension Title Blocks: Opens the Extension Title Blocks dialog box so that you can add title block extensions to the drawing. This option is available only in the GOST standard.

Scale

Displays all standard scale factors. You can also enter your own scale factor.

Calculate: Suggests a scale factor for a drawing section or for the size of the existing objects in the drawing of a pre-selected border. The calculated scale factor appears in the scale box. The Calculate option is not available in layout.

Options

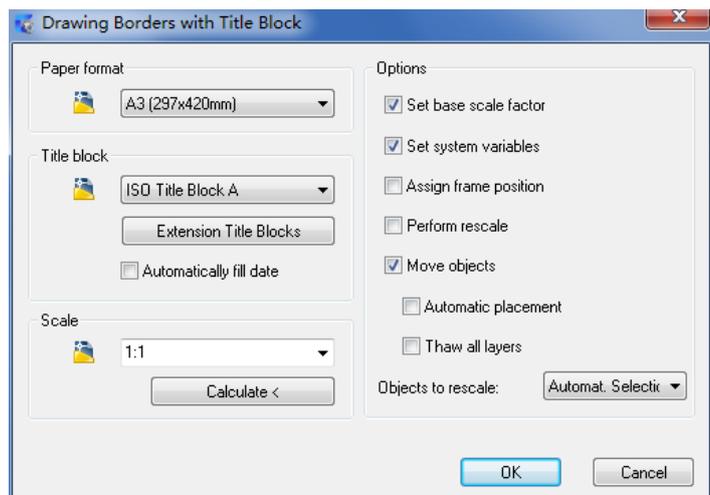
Set Base Scale Factor: When selected, the section scale factor is stored in the configuration as base scale factor (only in Model space).

Set system variables: Sets grid, snap, limits, linetype scale, and other options, to the default values that correspond to the selected scale factor you selected.

Perform rescale: Adjusts text, symbols, dimensions, tables, and other elements to conform to the changed scale factor.

Move Objects: Places existing objects in the drawing editor within the inserted drawing border. If you select the Automatic Placement option as well, selected objects move to the center of the border.

Automatic Placement: Moves selected objects to the center of the border. Selects all objects located within an



inserted drawing border automatically, if the Objects to Rescale: Automatic Selection option is selected in the Drawing Sheet Settings dialog box (accessed through the IM:Standards dialog box). If no border exists, selects all objects. If objects are outside the border, clear the Automatic Selection check box in the Drawing Sheet Settings dialog box (accessed through the IM:Standards dialog box). Otherwise, the objects outside the border are not part of the selected border.

Thaw All Layers: Selects all layers containing objects, moves them into the border, and thaws them. It restores the state of the layers when complete.

Objects to rescale: Specifies whether the drawing objects are to be selected automatically or manually.

Revision

Revisions on /off

Ribbon: Annotate tab ► Sheet panel ► Revisions on /off

Keyboard command: GMREV

When revisions are turned on, GstarCAD Mechanical tracks the date of the block of xrefs inserted in the drawing. If GstarCAD Mechanical detects a change in an xref, it prompts you for confirmation and adds a revision line to the revisions list. If no revisions list exists, it creates one and prompts you to place it in the drawing area. Generally, activate this command after changing your drawing. The first time you launch the command, it inserts a block with a header into the drawing.

Add one Revision Line

Ribbon: Annotate tab ► Sheet panel ► Turns revisions on or off ► Add one Revision Line

Keyboard command: GMREVLIN

Use this command to add a revision line to the revision list manually. If no revisions list exists, it creates a revision list and prompts you to place it in the drawing area. If you insert at least one drawing border/title block, select the drawing border whose objects are included. Otherwise the Edit Revision Attributes dialog box appears immediately. You can change or add entries for the number, the date, the revision description, the name, and the DWG name.

To Insert a Revision List

1. Attach an xref drawing to the assembly drawing.
2. At the Command prompt, enter GMREV.
3. Save and close the assembly drawing.
4. Open the xref drawing and change anything (for example, detailing).
5. Save the xref drawing.
6. Open the assembly drawing again.
7. In the message box, click OK.
8. Specify an insertion point for the revision line.

9. Complete the entries in the Edit Revision Attributes dialog box.
10. Click OK.

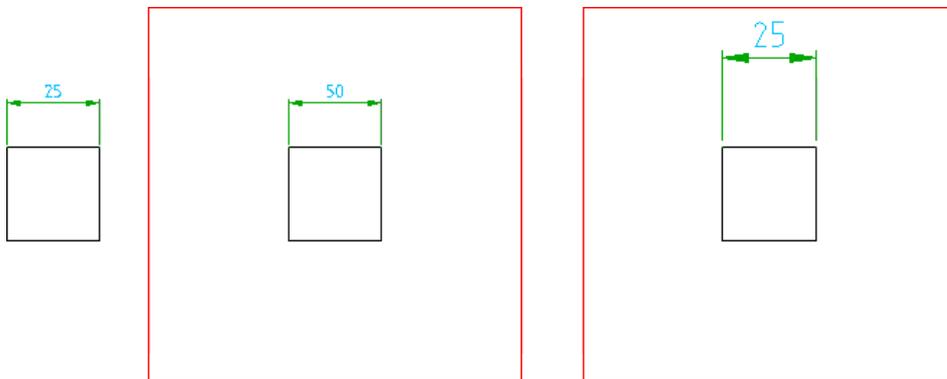
Scale Area/ Viewport

Scale Areas

Ribbon: View tab ➤ Scale Areas panel ➤ Scale Area

Keyboard command: GMSCAREA

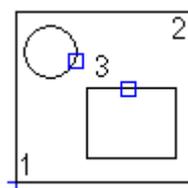
A scale area is a region in model space that has a scale that is different to the scale of model space. Scale areas give you the ability to divide model space into regions with different scales. Scale areas support two independent methods of scaling; length scaling and text scaling.



Commonly Steps

To Define a Scale Area with Length Scaling

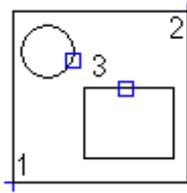
1. Click View tab ➤ Scale Areas panel ➤ Scale Area.
2. Click two points (1 and 2) to define the border of the scale area.
3. In the Scale Area dialog box, in the Scale box, specify the scale factor.
4. Select Length scaling .
5. Select Perform Rescaling
6. Click OK.
7. Select the objects to rescale (3)
8. Press ENTER.



To Define a Scale Area with Text Scaling

1. Click View tab ➤ Scale Areas panel ➤ Scale Area.
2. Click two points (1 and 2) to define the border of the scale area.

3. In the Scale Area dialog box, in the Scale box, specify the scale factor.
4. Select Text scaling .
5. Select Perform Rescaling
6. Click OK.
7. Select the objects to rescale (3)
8. Press ENTER.



List of Prompts

Specify first point or [Circle/Object]: Specifies a point to define a corner of a rectangular area, a circular area or select an object to convert an existing closed contour as the scale area border.

Scale Area Dialog Box

Scale and type of scaling

Scale

Specify the scale for the scale area or viewport. You can select a preset value, or enter any desired scale.

Note: For example, to specify a scale of 3:1, enter 3:1 or 3.0 (which is converted to 3:1).

< This symbol temporarily reverts to the drawing. You can select a scale defining entity (for example, another scale area, a dimension, or a scale text) to set the scale of the area you are editing.

Length Scaling: Sets the length scaling factor. It defines the ratio between drawn length and real length. For example, with a length scaling of 2.0, a line of 100 units is dimensioned as a line of 200 units.

Text Scaling: Sets the text scaling factor: This setting defines the dimension text height, symbol sizes, and table sizes. For example, with a text scaling of 2.0, the dimension text height is doubled in size.

Automatic View Creation in: Creates a viewport using the same scale you used in the scale area. In this viewport, all text heights and symbols sizes appear in the right size.

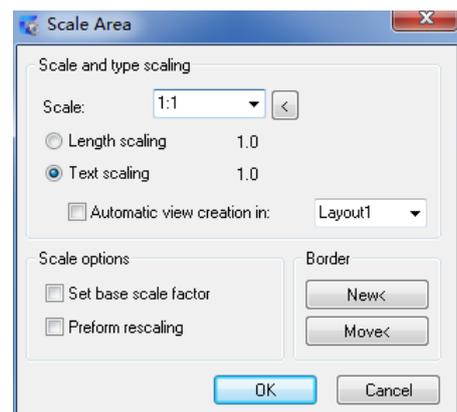
Layout List: Selects a layout for the viewport. You cannot create viewports in more than one layout.

Scale options

Set Basic Scale: Sets the text scaling factor used in the scale area as the basic scale of model space. This option is not available if you length scaling is selected.

Perform Rescaling: Prompts you to select existing objects within the scale area to rescale, once the dialog box closes.

Note: This option is selected by default if you change the scale.



Border

New < Redefines the border of the scale area.

Move < Removes the border of the scale area.

Create a viewport

Ribbon: Layout tab ► Viewports panel ► Viewport

Keyboard command: GMVPORT

Creates a viewport in layout mode or edits an existing viewport.

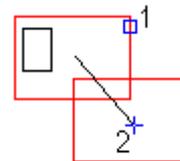
Commonly Steps

To Create a Viewport

1. Switch to layout mode.
2. Click Layout tab ► Viewports panel ► Viewport.
3. Define the border of the viewport .
4. In the View dialog box, specify the scale.
5. Click OK.

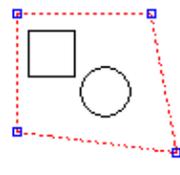
To Move a Scale Area or a Viewport

1. Double-click the viewport or scale area border (1).
2. In model space the Scale Area dialog box appears, in layout the View dialog box appears:
3. Under border, click Move<.
4. Choose OK.
5. Select the target position for the viewport (2).
6. Choose OK.



To Resize a Scale Area or a Viewport

1. Click the viewport or scale area border.
2. Use the grips to resize the viewport or scale area.



List of Prompts

Specify first point or [Circle/Border/Object] Specifies a point to define one corner of a rectangular area for the viewport.

Specify second point: Specifies the diagonally opposite corner of the rectangular area for the viewport.

Circle: Defines a circular area for the viewport.

- **Center of circle** - Specifies the center of the circular area.
- **Radius** - Specifies the radius of the circular area.
- **Diameter** - Specifies the diameter of the rectangular.

Border: Specifies an existing of a scale area in model space for the viewport.

View Dialog Box

Scale

Sets the scale for the viewport. Select a preset value, or enter any desired scale. For example, to specify a scale of 3:1, enter 3.0 or 3.1 (which is converted to 3:1).

< this symbol Temporarily reverts to the drawing. Then you can select a scale-defining entity (for example, another scale area, a dimension, or a scale text) to set the scale of the viewport you are editing.

Model space

border <

Sets a scale area in the drawing. This scale area sets the shape, size, and scale of the viewport. This way, the viewport is attached to the scale area. Changes to this scale area are automatically adapted by the viewport.

Midpoint: Attaches the viewport to a point that you select in your drawing. This point is the view target for the viewport. Midpoint does not specify the shape or the scale of the viewport.

Detach: Detaches the viewport from either the scale area or the midpoint it is attached to.

Scale options

Zoom according to Scale: Sets the viewport zoom factor to reflect the scale you selected. This action is performed after you close the dialog box.

Perform Rescaling: Rescales objects in your drawing. Also affects the scale of dimension text height and symbol sizes.

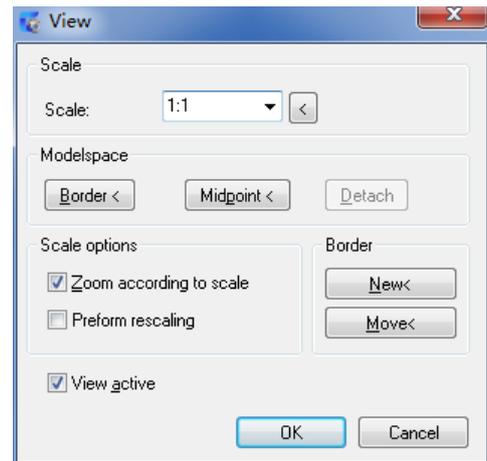
Note: This option is selected automatically if the scale is changed.

Border

New < Redefines the border of the viewport.

Move < Moves the border of the viewport.

View active: Activates the viewport.



Rescale

Ribbon: View tab ► Scale Areas panel ► Rescale

Keyboard command: GMRESCALE

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling

List of Prompts

Enter new symbol scale: Specify scale factor to enlarge or shrink the notes.

Select object for rescaling: The specified annotation to Scaling.

Scale Monitor

Ribbon: View tab ► Scale Areas panel ► Scale Monitor

Keyboard command: GMSCMONITOR

Displays the scale of a scale area or viewport.

List of Prompts

Pick scale area to edit/Pick inside scale area to zoom/<Return> to finish

Specifies a scale area or view port to monitor its scale (The area is automatically zoomed). Click the scale area or viewport border to edit the scale.

View tools

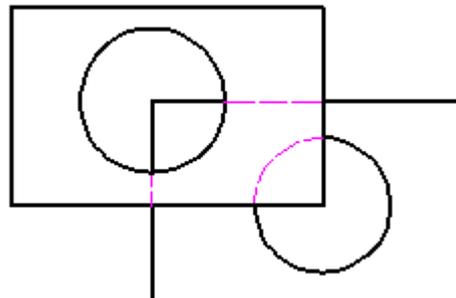
Create and Edit Hide Situation

Ribbon: Home tab ► Detail panel ► Hide Situation

Keyboard command: GMSHIDE GMSHIDEEDIT

Draws hidden lines to represent hidden edges, when you specify what objects lie in front and what objects lie behind.

When you modify or move objects that are part of a “hide situation” created with this command, the hidden lines update automatically. A hide situation can contain several overlapping levels.



Once you select the foreground objects, GMSHIDE automatically chooses objects for the background. If the number of background objects is too large, it prompts you to select the background objects from the Hide Situation dialog box.

The Hide Situation dialog box enables you to:

- Create additional levels for hide situations that are more complex than two-level hide situations (foreground and background only).

- Move objects from one level to another.
- Add/remove objects to/from a level.

Commonly Steps

To Create a Hide Situation

1. Click Home tab ► Detail panel ► Hide Situation drop-down ► Create.
2. Click in the graphics window to select foreground objects.
 - To select a free entity (a line, an arc, a circle, and so on), continue clicking the object until you see its name in the tooltip.
 - To select a block instance, click a geometry that belongs to it.

Selection information is displayed on the command line. Continue to select objects, and then press ENTER to quit.

When foreground objects are selected, objects that are partially or totally within a bounding box surrounding the foreground objects are automatically added to the background selection set.

To Remove Elemental Geometry (Free Objects) from a Level

1. Click Home tab ► Detail panel ► Hide Situation drop-down ► Edit.
2. In model space, select an object that belongs to the hide situation you want to edit.
3. In the tree view, click the level you want to edit.
4. Click  on the vertical toolbar. The dialog box hides. Objects that belong to the selected level are displayed in dotted lines.
5. Click the free object you want to remove.
6. Press ENTER.
7. Click OK.

To Change How Hidden Edges are Displayed

1. Click Home tab ► Detail panel ► Hide Situation drop-down ► Edit.
2. In model space, select an object that belongs to the hide situation you want to edit
3. Click the root node of the tree view in the Hide Situation dialog box.
4. If the dialog box has not expanded and Object Settings is not visible, click .
5. To display hidden lines as dashed lines, select Display hidden lines. If this button is not checked, hidden lines are not visible.
6. Click OK.

To Add a Level to a Hide Situation

1. Click Home tab ► Detail panel ► Hide Situation drop-down ► Edit.
2. In model space, select an object that belongs to the hide situation you want to edit.
3. Click . A new level is added.
4. Use the  buttons to position the level.
5. Click  on the vertical toolbar. The dialog box hides.
6. In model space, click the objects you want to add to this level.
7. Click OK.

To Remove a Level in a Hide Situation

1. Click Home tab ► Detail panel ► Hide Situation drop-down ► Edit.
2. In model space, select an object that belongs to the hide situation you want to edit.
3. In the tree view, click the level you want to remove.
4. Click  on the vertical toolbar.
5. Click OK.

List of Prompts**Select foreground objects**

Specifies the objects to be the in the foreground.

Valid foreground objects include blocks, any closed contour created with lines, arcs. When you confirm the foreground selection set, the program creates the background selection set automatically based on the foreground objects.

Hide Options Dialog Box

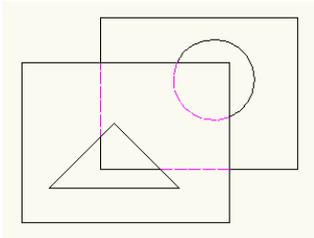
Use this dialog box to set the defaults for the GMSHIDE command.

GMSHIDE dialog box defaults

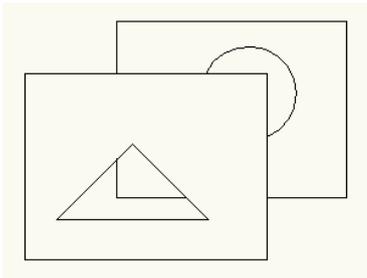
Display hidden lines

Specifies that hidden lines appear as dashed lines by default. If you do not select this check box, hidden lines are not visible by default.

- See an example of the effects of the Display hidden lines check box



In this example, the Display hidden lines check box is selected.

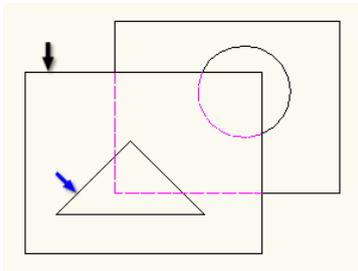


In this example, the Display hidden lines check box is cleared.

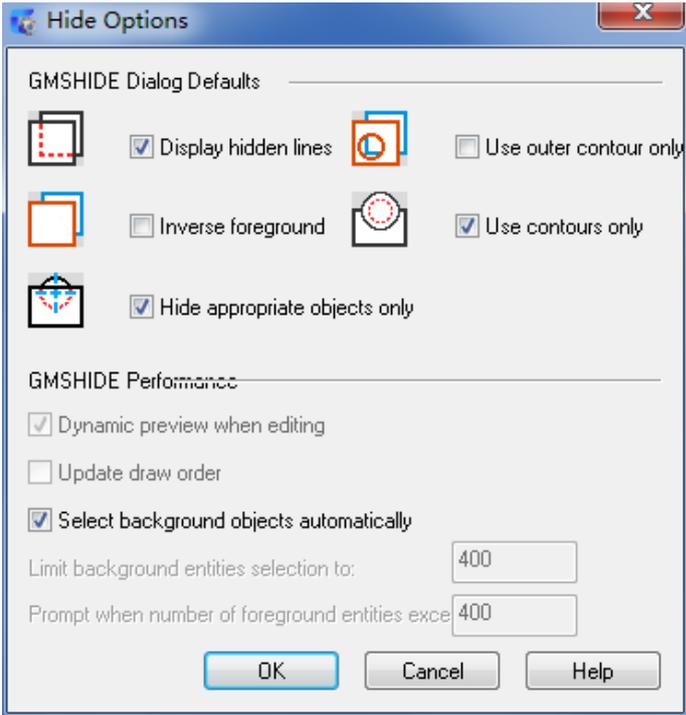
Use outer contours only

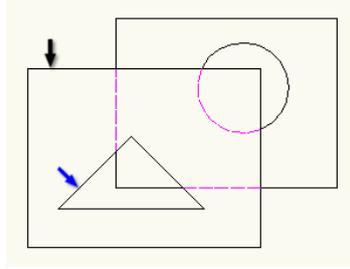
Uses only the outer contours of objects for foreground calculations. If the outer contour contains openings such as holes and slots, the program ignores them and background objects that should be visible through these openings remain hidden.

- See an illustration of the effects of the Use outer contour only check box



In this example, the foreground consists of the rectangle (black arrow) and a triangular opening (blue arrow) objects. Because Use outer contour only is selected, background objects are not visible through the opening. The program shows them using hidden lines.





Inverse foreground

Swaps the roles of foreground and background objects. The program shows objects overlapped by the foreground and hides objects not overlapped by the foreground boundary.

Swaps the roles of foreground and background objects. The program shows objects overlapped by the foreground and hides objects not overlapped by the foreground boundary.

Use contours only

Filters out all inappropriate objects from the foreground. By default, the program considers only closed contours drawn on the mechanical layers AM_0, AM_1, and AM_2 (Contour Layers) and closed contours drawn on layers created with the AutoCAD LAYER command appropriate for the foreground.

Hide appropriate objects only

Specifies that the GMSHIDE command must ignore objects that are not valid background objects (hatches and construction lines, for example).

Select background objects automatically

Specifies that the GMSHIDE command automatically selects background entities, when creating a new hide situation.

Hide Situation Dialog Box

Tree View

Graphically describes the hide situation. The name of the hide situation is the root node. Levels appear as child nodes of the root, listed in z-order. Elemental geometry such as lines and arcs selected for a level are shown as a single entity named free objects.

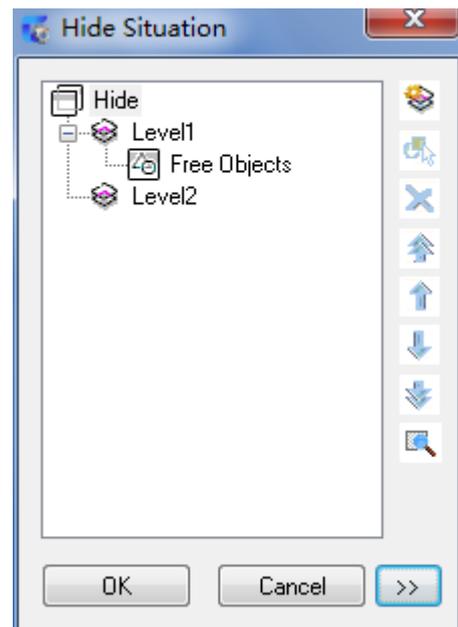
Vertical Toolbar

Contains buttons that operate on the item selected in the tree view. The availability of each button depends on its relevance to the selection.

Buttons



Creates a level and adds it below the current selection in the tree.





Temporarily hides the dialog box so you can pick objects for the currently selected level.

- To select a block instance:

Click a geometry that belongs to the block instance. The tooltip displays the name of the block.

- To select a free object :

Continue clicking the free object until you see its name in the tooltip.

- To remove an object from the selection set:

Click the object until you see its name in the tooltip window.



Removes the item selected in the tree from the hide situation.



Moves the selected level to the top.



Moves the selected level up one level.



Moves the selected level down one level.



Moves the selected level to the bottom.



Temporarily hides the dialog box to show the drawing. Press ESC or ENTER to return to the dialog box.

Hide Situation Dialog Box - Hide Settings

Hide Situation Settings

Name

Provides a space for you to name the hide situation. Hide situation names do not need to be unique.

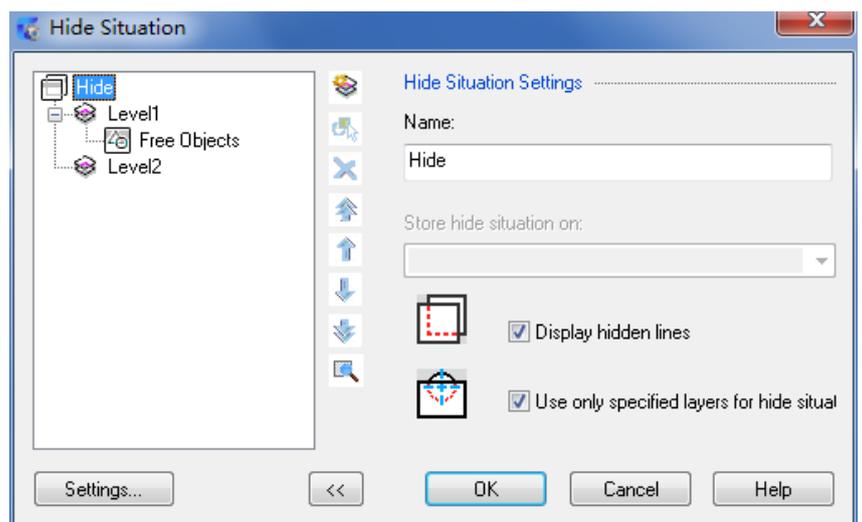
Display hidden lines

Sets the hide style of hidden lines to dashed lines (enabled by default). Clear this check box to make hidden lines invisible.

Use only specified layers for hide situations

Excludes inappropriate objects (hatches and centerlines, for example) from the hide situation.

Settings



Opens the Hide Options dialog box and enables users to set defaults for GMSHIDE.

Hide Situation Dialog box - Level Settings

Basic tab

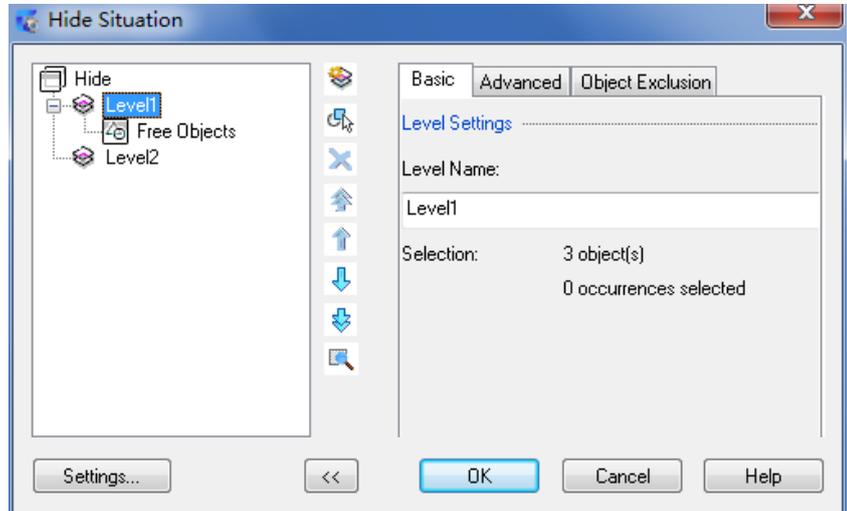
Level Settings

Level Name

Provides a space for you to specify a name for the level. The program shows this name on the tree view.

Selection

Shows the number of entities have been selected for that level. Occurrences refer to the number of mechanical structure objects that have been selected for that level.



Advanced tab

Level Settings

Use outer contour only

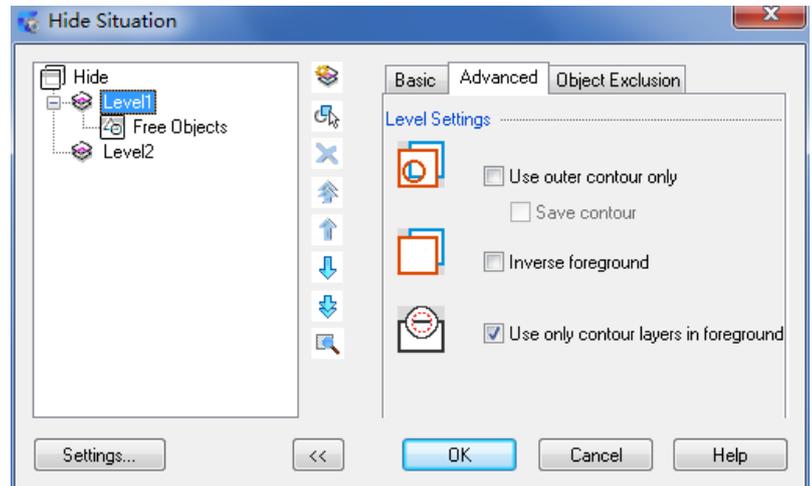
Uses only the outer contour for foreground calculations. When selected, if there are openings such as holes and slots inside the contour, background objects are not visible through them. A common use for this option is hiding background objects when the foreground object contains blind holes.

Inverse foreground

Swaps the role of the foreground and background. The program hides objects that are not overlapped by the foreground boundary and shows objects that are overlapped by the foreground.

Use only contour layers in foreground

Applies a layer filter to the foreground selection set, filtering out all layers except the contour layers and non-mechanical layers. By default, AM_0, AM_1, and AM_2 are contour layers and you can define up to four contour layers. By default, the program draws all objects that are valid foreground objects on these layers. If you clear this check box, all objects that belong to the foreground selection set appear as foreground objects, regardless of



whether they are valid foreground objects or not.

Object Exclusion tab

Objects used as Foreground

Select objects for exclusion

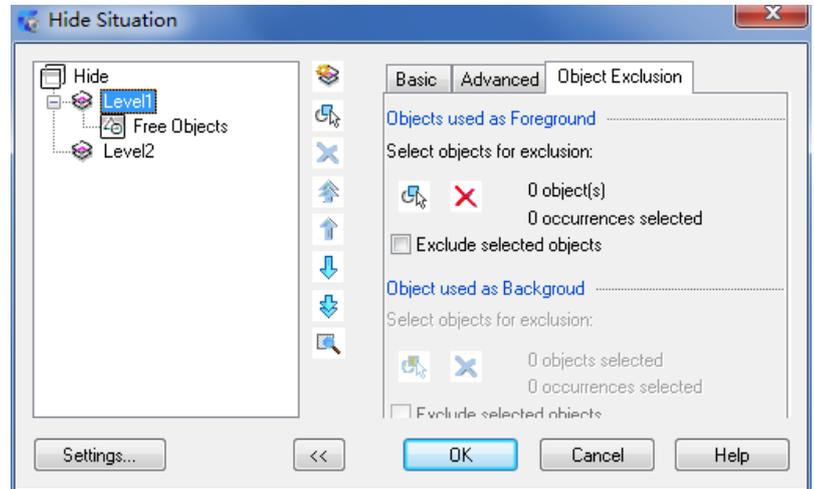


Temporarily hides the dialog box so you can select or remove objects from the foreground exclusion set.

- To select a block instance:
Click a geometry that belongs to the block instance. The tooltip displays the name of the block.
- To add a free object to the exclusion set:

Continue clicking the free object until you see its name in the tooltip.

- To remove an object from the selection set:
Click the object until you see its name in the tooltip.



Clears current selection for the exclusion set.

Exclude selected objects

Specifies that the objects for the foreground exclusion set are not calculated as part of the foreground.

Objects used as Background

Select objects for exclusion



Temporarily hides the dialog box so you can select or remove objects from the background exclusion set.

- To select a block instance:
Click a geometry that belongs to the block instance. The tooltip shows the name of the block.
- To add a free object to the exclusion set:
Continue clicking the free object until you see its name in the tooltip.
- To remove an object from the selection set:
Click the object until you see its name in the tooltip.



Clears the current selection for the exclusion set.

Exclude selected objects

Specifies that the objects selected for the background exclusion set are not hidden as part of the background.

Hide Situation Dialog box - Object Settings

Exclude from Foreground

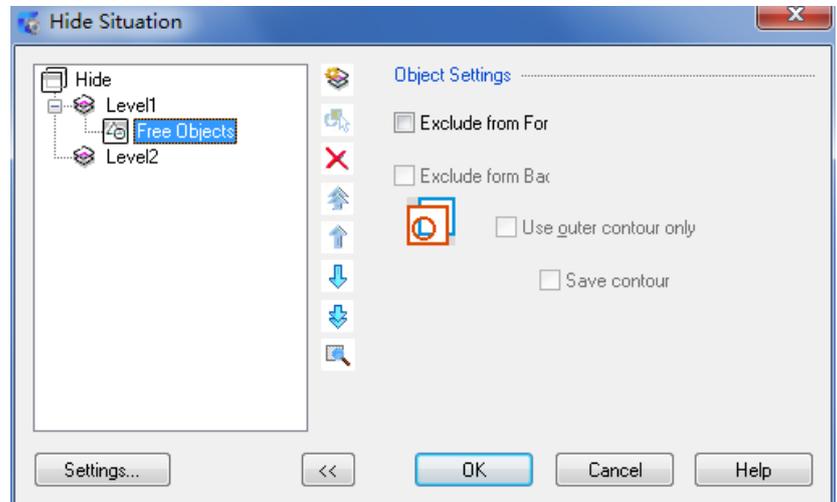
Prevents the selected entity from acting as a foreground to all the levels below its level.

Exclude from Background

Prevents the selected entity from acting as a background to all levels above its level.

Use outer contour only

Sets only the outer contour of the selected entity to be displayed as a foreground object. If you clear this check box, all contours are displayed as foreground objects.



Detail View

Ribbon: Home tab ► Detail panel ► Detail View

Keyboard command: GMDetail

Create a scaled circular, rectangular, or free defined detail view of selected geometry.

To Create a Detail View in a Layout

1. Click Home tab ► Detail panel ► Detail View.
2. To create a detail view with a circular detail boundary:
 - a) Click the point to use as the center of the area to be detailed.
 - b) Click in the drawing area to indicate the radius of the circular boundary.

3. To create a detail view with a rectangular detail boundary:

In the command line, enter R.

In the drawing area, click one corner of the area to be detailed.

Click the opposite corner.

4. To create a detail view using an existing closed contour as the detail boundary:

In the command line, enter O.

Click the object to use as the detail border.

5. In the Detail dialog box, specify the enlargement scale for the area to magnify.
6. In the Create detail view in: section, select the Layout option.

7. In the adjacent list, select the layout to create the detail view in.
8. Click OK.
9. Place the detail view in the drawing area.

List of Prompts

Center of Circle: Specifies the center of the circular area you want to create a detail view from.

- **Radius** - Specifies the radius of the circular area you want to create a detail view from.
- **Diameter** - Specifies the diameter of the circular area you want to create a detail view from.

Rectangle: Specifies a rectangular area to create a detail view from.

- **First point** - Specifies a corner point of the rectangular area you want to create a detail view from.
- **Second point** - Specifies the opposite corner point of the rectangular area you want to create a detail view from.

Object: Specifies an area encapsulated by a closed contour to create a detail view from.

Place the detail view: Specifies a location for the detail view in model space.

- **Specify next point** - Specifies the next vertex of the connection line. Click on the detail view to complete the command. This option displays only when the current detail view style specifies that the detail boundary must connect to the detail view border by a connection line.
- **No connection line** - Places the detail view without a connection line.

Target position: Specifies a location for the detail view in a layout.

Current position: Places the detail view at the center of the layout.

Detail Dialog Box

Detail scale

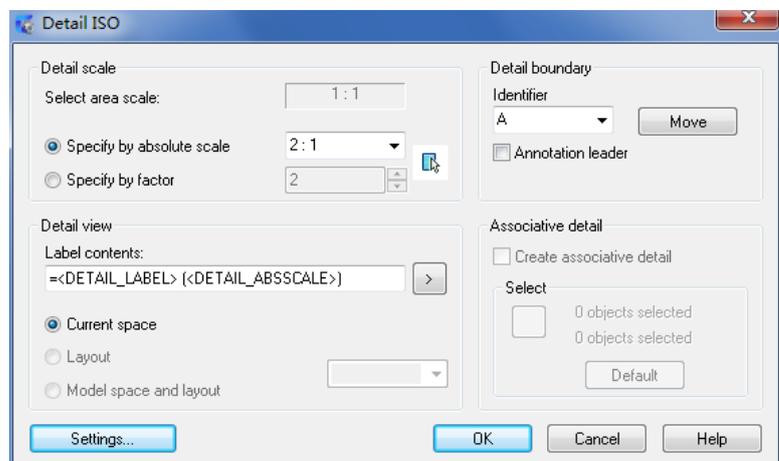
Specifies the scale for the detail view.

Selected area scale: Displays the scale of the area under the detail boundary. If the detail boundary is not in a scale area or title block, the selected area scale is the same as the model scale.

Specify by absolute scale: Sets the enlargement scale for the detail by an absolute scale. Select a preset value, or enter any desired scale.

Specify by factor

Sets the enlargement scale factor for the detail by a factor. Select a preset value, or enter any desired scale.





Picks the scale automatically from a scale area or annotation you select.

Detail boundary

Specifies properties of the border that determines the content of the detail.

Identifier: Sets descriptive text to identify the detail. You can select a valid identifier from the drop-down list or type in a value, directly. Use “-” if you want to create a detail view without a label.

Note: The system does not validate values you directly type into this box.

Annotation leader: Connects the detail label to the detail boundary with a leader line.

Move: Moves the detail border.

Detail view

Specifies additional properties of the detail.

Label contents: Specifies the textual content for the detail view label. You typically use the following fields within detail view labels.

Section Line

Ribbon: Home tab ► Detail panel ► Section Line

Keyboard command: GMSECTIONLINE

Draws a section line and inserts the corresponding section view label in the drawing area.

To Create a Section Line

1. Click Home tab ► Detail panel ► Section Line.
 2. Select a starting point for the section line .
 3. Select the next point or points for the section line and press ENTER.
 4. Enter a letter to reference the section line at the starting point.
 5. Press ENTER.
 6. Move the cursor to select the side of the cutting plane.
 7. Select an insertion point for the text of the section view.
-

List of Prompts

Select point: Defines the start point of the section line.

Next section line point: Draws a section line segment.

Half: Adds a line segment to the end of the last section and jumps to the Name option. This added line segment is perpendicular and equal in length to the first segment of the section line. You typically use this option to create a section line to show the cutting plane for half sections.

Arc: Adds an arc segment to the section line.

Center point of arc: Defines an arc segment using the start point, center, and end point.

Second point: Defines an arc segment using 3 points that the arc passes through.

Name: Specifies the cutting-plane symbols to name the section line.

- **First section symbol** - Specifies the cutting-plane symbol for the first point of the section line.
- **Next section symbol** - Specifies the cutting-plane symbol for the next vertex of the section line. GstarCAD Mechanical displays this prompt only if the Use continuous naming check box in the Section View Settings dialog box is selected.
- **Side of section** - Specifies the direction of the section arrows.
- **Origin of section view** - Specify where to insert the section view label.

Center: Creates a section line for an aligned section view of a cylindrical object.

Center point: Specifies the center of the cylindrical object you are creating the section line for.

First section symbol: Specifies the cutting-plane symbol for the first point of the section line.

Next section symbol: Specifies cutting-plane symbol for the next vertex of the section line. GstarCAD Mechanical displays this prompt only if the Use continuous naming check box in the Section View Settings dialog box is selected.

Side of section: Specifies the direction of the section arrows.

Origin of section view: Specify where to insert the section view label.

Visibility: Controls the appearance of different components of the section line.

Arrow: Toggles the visibility of arrowheads on the section line. The prompt that GstarCAD Mechanical displays after you select this option tells you what the new visibility state is.

Line: Specifies how plane lines (the section line segment between the end portion and bends) should be drawn.

Between planes: Draws plane lines using the line type and line weight assigned to the Section/Viewing Plane Line object (PHANTOM2 by default) in the Objects Property Settings dialog box.

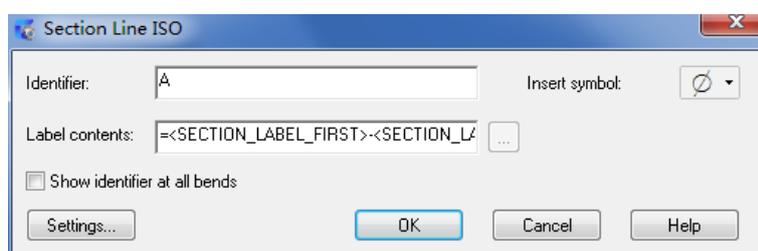
Continuous: Draws plane lines using the same line type and line weight as the ends and bends.

None: Does not draw any plane lines.

Name: Toggles the visibility of labels on the section line and section view. The prompt that GstarCAD Mechanical displays after you select this option tells you what the new visibility state is.

Plane names: Toggles the visibility of cutting-plane symbols on the bends (vertices) of the section line. The prompt that GstarCAD Mechanical displays after you select this option tells you what the new visibility state is.

Section Line Dialog box



Identifier

Specifies an alphabetic character to name the section line. If the Use continuous naming option is selected in the current section view style, this box shows the first character of the set of consecutive alphabetic characters used to name the section line.

Insert symbol

Displays a palette to enable you to insert a special character at the current cursor position. The preview in the drawing area shows the special character, while the dialog box shows the corresponding control key sequence.

Label contents

Sets the textual content for the section view label.

Show identifier at all bends

Displays the identifier at all vertices of the section line. If this check box is cleared, the identifier is displayed only at the ends of the section line.

Settings

Displays the Modify Section View Styles dialog box, which enables you to configure the defaults for the GMSECTIONLINE command.

Chapter 5 Dimensioning

Summary

With the streamlined tools in GstarCAD Mechanical 2017, you can create dimensions using abbreviated dialog boxes that conveniently control and expand only the relevant variables for manufacturing. With automatic dimensioning, you can create multiple dimensions with minimal input, resulting in instant groups of ordinate, parallel, or symmetric items that are appropriately spaced. Smart dimensioning tools force overlapping dimensions to automatically space themselves appropriately while integrating tolerance and fit list information into the design. Dimension input can even drive and change design geometry to fit certain sizes.

Create Dimensions

Power Dimension

Ribbon: Annotate tab ► Dimensions panel ► Power Dimension

Menu: Annotate ► Power Dimension

Keyboard command: GMPOWERDIM

Power Dimensions are dimensions created by GstarCAD Mechanical's dimensioning commands. We designed these dimensioning tools specifically for manufacturing, making them relevant and convenient for use within this domain.

The GMPOWERDIM command automatically recognizes objects and defaults to the appropriate dimension type. You can use the same command to place linear, angular, radial, diameter, baseline, and chain dimensions.

The supported dimension types range from vertical, horizontal, aligned, and rotated linear dimensions, to angular dimensions, to radius, diameter, jogged radius, and arc length dimensions, to baseline and chain dimensions. If required, you can change the dimension type using List of Prompts.

Commonly Steps

To Create Horizontal, Vertical, or Aligned Dimensions

1. Click Annotate tab ► Dimensions panel ► Power Dimension.
2. Select the location for dimension point or select the target for Dimension.
3. Specify the location of dimension line and complete Dimension.

To add tolerances and fits to dimensions

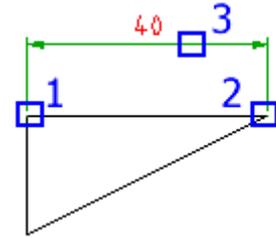
1. When specifying the dimension line location, enter ':C' in the command line.
 2. The 'Enhanced Dimension' dialog box is displayed, click 'Add Tolerances' or 'Add Fits' button.
 3. Click to preview the style of 'Tolerances' or 'Fits'.
 4. The 'Select Tolerance/Fit Type' dialog box is displayed, where user can select the required type and click OK to confirm.
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List of Prompts

Specify first extension line origin or Angle/Baseline/Continuous/Selection/eXit<Selection>:

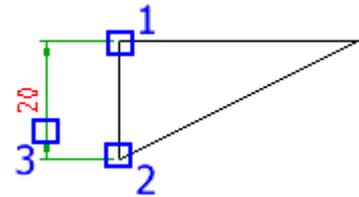
- **Linear:** Creates a linear dimension with a horizontal, vertical, or rotated dimension line.
 - ✓ **Horizontal:** set the style of dimension to horizontal linear dimension.

1. Specify the first point of linear dimension(1).
2. Specify the second point of linear dimension(2).
3. When dimension line is moved to the required location, Click to determine the location (3). The dimension line can only move along the vertical direction.



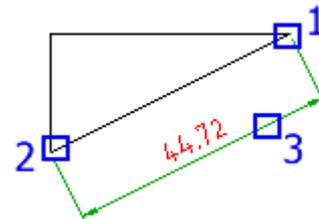
- ✓ **Vertical:** set the style of dimension to vertical linear dimension.

1. Specify the first point of linear dimension (1).
2. Specify the second point of linear dimension (2).
3. When the dimension line is moved to the required location, Click to determine the location (3). The dimension line can only move along the horizontal direction.



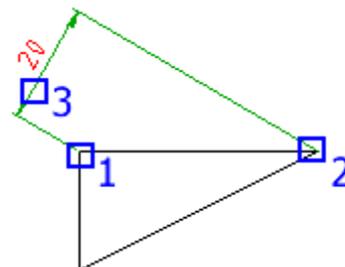
- ✓ **Aligned:** set the style of dimension to aligned linear dimension.

1. Specify the first point of linear dimension (1).
2. Specify the second point of linear dimension (2).
3. When dimension line is moved to the required location, Click to determine the location (3). The dimension line can only move along the direction vertical with the line where two points are connected.



- ✓ **Rotated:** set the style of dimension to rotated linear dimension.

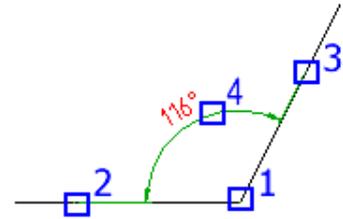
1. Specify the first point of linear dimension (1).
2. Specify the second point of linear dimension (2).
3. Specify the angle.
4. When dimension line is moved to the required location, Click to determine the location (3). The dimension line can only move along the direction vertical with the specified angle.



- **Angle:** Create an angular dimension to represent the angle formed by three points or two lines.

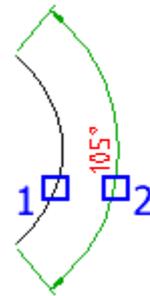
- ✓ **Specify vertex:** use three points for angular Dimension

1. Select the vertex for angular Dimension: specify the vertex of angle (1)
2. Select anchor point of the first edge for angular Dimension: specify point (2)
3. Select anchor point of the second edge for angular Dimension: specify point (3)
4. When dimension line is moved to the required location, Click to determine the location (4)



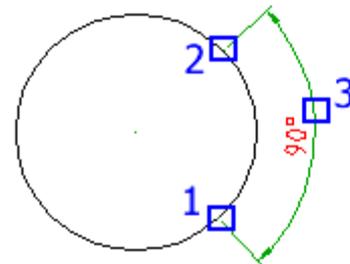
- ✓ **Select arc:** For angular Dimension of an arc, perform Dimension with the two end points and center point of the arc.

1. Select the target arc(1)
2. When dimension line is moved to the required location, Click to determine the location (2)



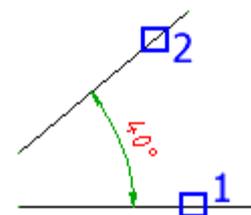
- ✓ **Select circle:** Perform angular Dimension with two points on the circle and the center of circle .

1. Select the anchor point of the first edge for angular Dimension: specify point(1)
2. Select the anchor point of the second edge for angular Dimension: specify point(2)
3. When dimension line is moved to the required location, Click to determine the location (3)



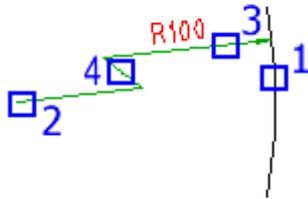
- ✓ **Select line:** select two lines as the target for angular Dimension.

1. Please select the first line: select line (1).
2. Please select the second line: select line (2).
3. When dimension line is moved to the required location, Click to determine the location.



- **Radial:** Sets the dimension type to radius dimensions.
 - ✓ **Radius:** Sets the dimension type to radius dimensions.
 1. Select arc or circle - Specifies the arc or circle to create dimensions for.
 2. Dimension line location - Specifies where to place the dimension line.
 - ✓ **Diameter:** Sets the dimension type to diameter dimensions.
 1. Select arc or circle - Specifies the arc or circle to create dimensions for.
 2. Dimension line location - Specifies where to place the dimension line.
 - ✓ **Jogged radius:** Sets the dimension type to jogged radius dimensions.

1. Select a circle or arc from the drawing.
2. Specify the center location of jogged radius.
3. Specify dimension line location.



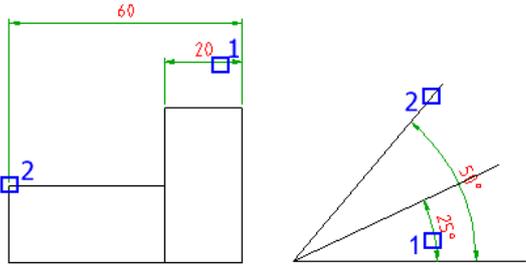
- ✓ **Arc length:** Sets the dimension type to arc length dimensions.

Select the circle or arc for Dimension, and select a c circle to proceed to the next step
 Specify dimension line location for arc length Dimension, or select an option to proceed to the next step

- **Baseline:**

Select the baseline for linear Dimension or angular Dimension

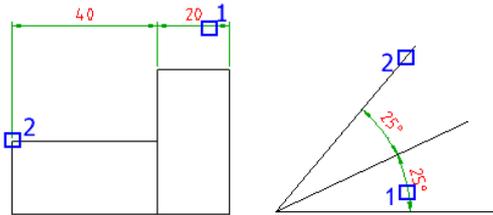
1. Select linear Dimension or angular Dimension (1)
2. Specify the second extension line origin or select (S): specify point(2), automatic baseline Dimension is completed



- **Continuous:**

Select the second extension line for linear Dimension or angular Dimension.

1. Select linear Dimension or angular Dimension (1)
2. Specify the second extension line origin or select (S): specify point(2), automatic Dimension is completed



Power Dimensioning Dialog box

General tab

Dimension Symbol

Displays a graphical representation of the current dimension text. Click the  button to the right of this box to select dimension text and special characters from predefined templates.

Representation

 Displays dimension values in both inches and millimeters. The alternate units appear between square brackets.

 Underlines the dimension value to indicate that the dimension is not to scale.

 Draws a box around the dimension value to indicate that it is a theoretically exact dimension.

 Draws a box around the dimension value to indicate that it is an inspection dimension.

 Draws parenthesis around the dimension value to indicate that it is a reference dimension.

Text

 Specifies that the dimension value must appear as part of the dimension text.

 Displays a palette of special characters to add to dimension text.

Value

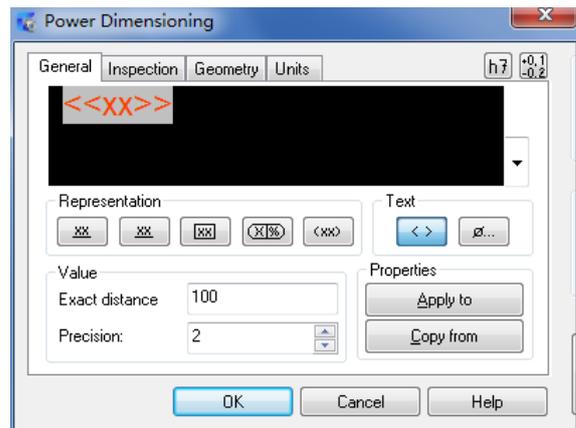
Exact Distance: Displays the actual value for the current dimension.

Precision: Specifies the number of decimals for the dimension value.

Properties

Apply to: Applies the properties of the current dimension to other dimensions. The dialog box hides enabling you to select the dimensions to apply the properties to. Thereafter the Select Properties to Copy dialog box appears enabling you to specify which properties.

Copy from: Imports the properties from another existing dimension. The dialog box hides enabling you to select the dimensions to copy the properties from. Thereafter the Select Properties to Copy dialog box appears enabling you to specify which properties.



Fit

Symbol: Specifies the fit symbol. Click the  button to display the Fits dialog box, enabling you to pick the symbol and the deviation from a list. These options are visible only if you expand the dialog box using the  buttons.

Hole: Specifies the symbol to use for the hole tolerance, in a stacked fit representation. Click the  button to display the Fits dialog box, enabling you to pick the symbol and the deviation from a list.

This option is visible only for stacked fit representations.

Shaft: Specifies the symbol to use for the shaft tolerance, in a stacked fit representation. Click the  button to display the Fits dialog box, enabling you to pick the symbol and the deviation from a list.

This option is visible only for stacked fit representations.

 Expands the dialog box so you can specify settings for the tolerance.

Deviation: Specifies the maximum and minimum tolerance value for the dimension.

Upper: Specifies the difference between the nominal dimension value and the maximum limit. This option is visible only if you expand the dialog box using the  button.

Lower: Specifies the difference between the nominal dimension value and the minimum limit. This option is visible only if you expand the dialog box using the  button.

Precision: Specifies the number of decimals for the tolerance value.

Primary: Specifies the number of decimals for the primary unit. This option is visible only if you expand the dialog box using the  button.

Alternate: Specifies the number of decimals for the alternate unit. This option is visible only if you expand the dialog box using the  button. It is available only if you select Alternate units.

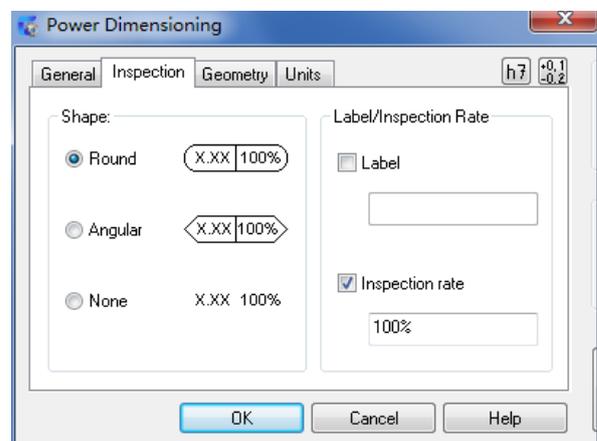
Inspection tab

Shape

Controls the shape of the frame drawn around the label, dimension value, and inspection rate of the inspection dimension.

Round

Creates a frame with semi-circles on the ends. Vertical lines separate fields within the frame.



Angular: Creates a frame with lines that form a 90-degree angle on the ends. Vertical lines separate fields within the frame.

None: Draws no frame around the values. Does not separate fields with vertical lines

Label/Inspection Rate

Controls the settings for the inspection dimension. Optionally shows a label and rate as part of the inspection dimension. You can control the label and rate fields individually, so it is not necessary to display both.

Label: Toggles the display of the label field on/off in the inspection dimension.

Label Value: Enables you to enter the label used to identify individual inspection dimensions. The label appears in the leftmost section of the inspection dimension when you select the Label check box.

Inspection Rate: Toggles the display of the rate field on/off in the inspection dimension.

Inspection Rate Value: Enables you to enter the rate used to indicate the frequency with which the part is inspected. The value is expressed as a percentage, and the valid range is 0 - 100. The inspection rate appears in the rightmost section of the inspection dimension when you select the Inspection Rate check box.

Geometry tab

Text offset from dimension line

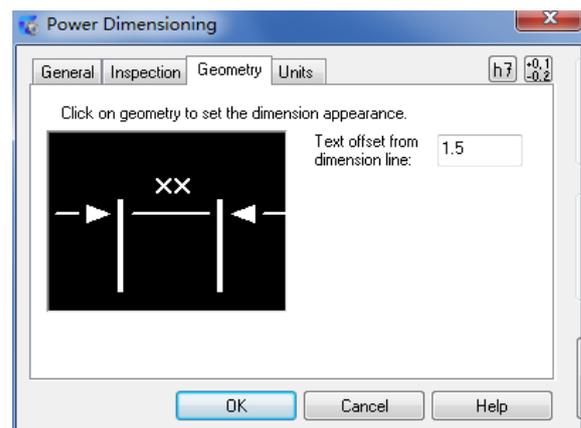
Sets the size of the gap between the text and the dimension line.

Center mark size

Sets the size of the center mark. This option is visible only if you are setting options for a radius or diameter dimension. This option is not available by default. To make this option available, click the Center mark in the graphic showing the elements of a circle.

Graphic showing extension lines and arrowheads

Controls the dimension text, arrowheads, extension lines, and force line inside options. Click the appropriate place in the graphic to customize that element. For example, click an arrowhead to display a palette of options. A tooltip shows the name of the element that can be controlled as you move the pointer over the elements of the graphic.



Units tab

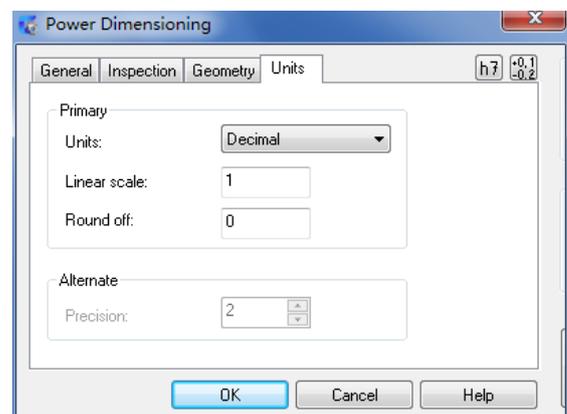
Primary

Sets options for primary units

Units: Sets a unit type for dimensioning.

Linear Scale: Sets a scale factor that multiplies the dimension value.

Round Off: Sets a value for rounding off all dimensioning distances.



Alternate

Sets options for alternate units. These options are available only if you select Alternate units in the General tab.

Precision: Sets the number of decimal places for alternate units

Properties Dialog box

Select the dimension properties

Dimension precision: Applies the precision settings of one dimension to another.

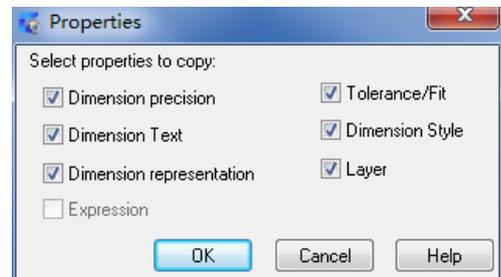
Tolerance/Fit: Applies the tolerance and fit settings of one dimension to another.

Dimension text: Applies the dimension text formatting settings found in the text editor of the Power Dimensioning dialog box.

Dimension style: Applies the dimension style of one dimension to another dimension of the same type.

Dimension representation: Applies the dimension text formatting settings found in the Representation section of the Power Dimensioning dialog box

Layer: Applies the layer settings of one dimension to another, thereby moving the dimension to the layer that the other dimension is on.



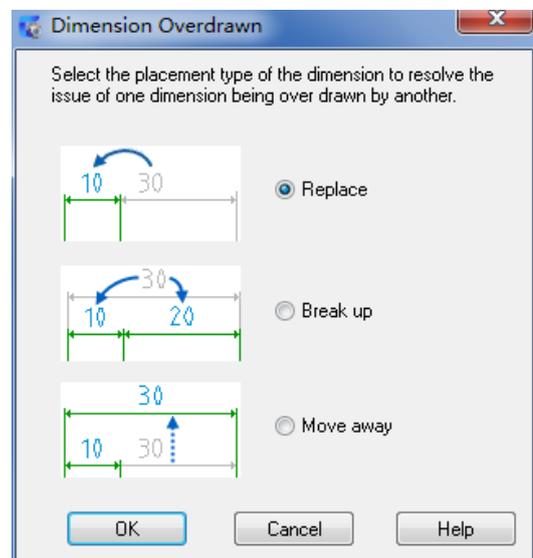
Dimension Overdrawn Dialog Box

This dialog box displays if you place a dimension in such a way that it overlaps an existing dimension. Use this dialog box to arrange the overlapping dimensions appropriately.

Replace: Deletes the existing dimension and replaces it with the one you insert.

Break up: Splits up the existing dimension into two dimensions, and arranges those dimensions into a chain dimension type.

Move away: Arranges the existing dimension and the newly inserted dimension into a baseline dimension type.

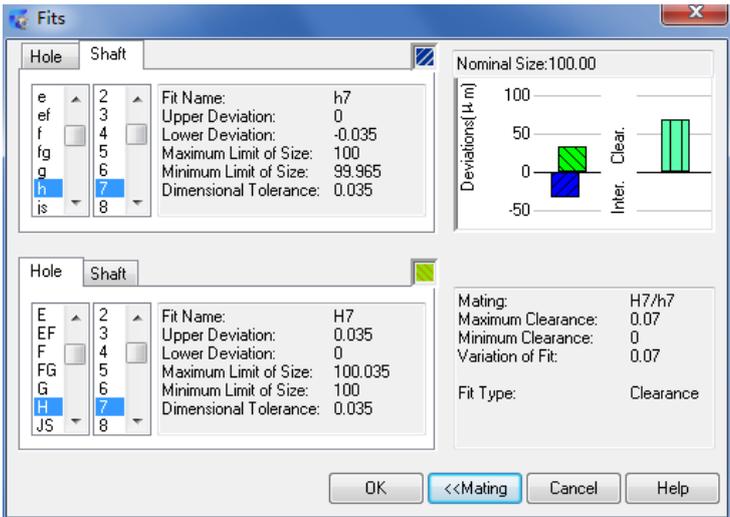


Fits Dialog Box

Use this dialog box to select a fit symbol for the dimension selected for editing. If multiple dimensions are selected for editing, the information shown (when you select a fit symbol) corresponds to the first dimension in the selection set.

Hole tab: Specifies a fit for the current dimension value of the hole. Information about the fit appears in this tab.

Shaft tab: Specifies a fit for the current dimension value of the shaft. Information about the fit appears in this tab.

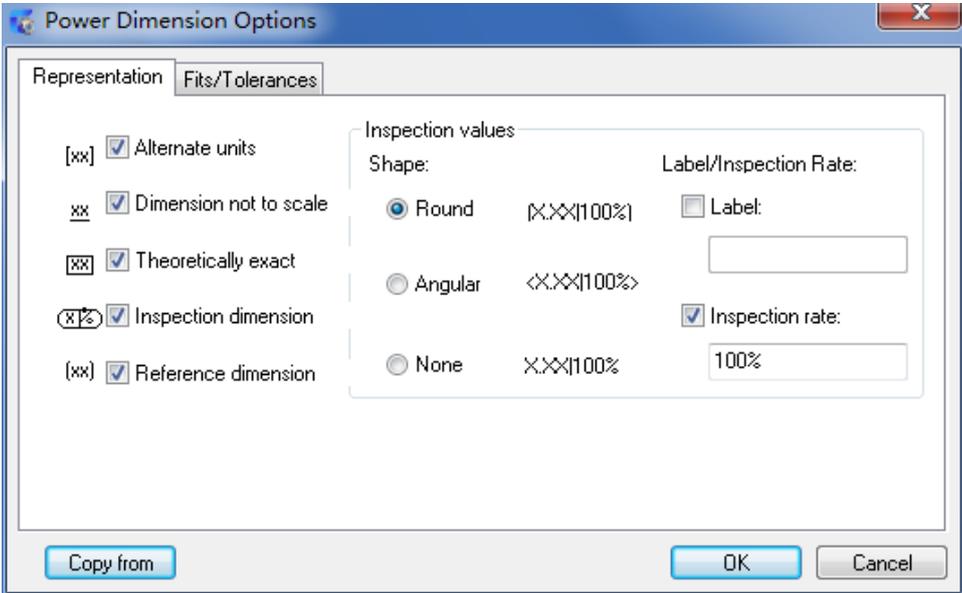


Graphical representation: Displays a graphical representation of the fit's deviation from the nominal size.

Mating: Expands the Fits dialog box to display both the Hole and Shaft tabs. A graphical combination of both fits appears. Besides the tab and shaft information section, a mating information section appears.

Power Dimension Options Dialog Box

Use this dialog box to pre-configure options for a command session.



Multiple Dimension

Ribbon: Annotate tab ► Dimensions panel ► Multiple Dimension

Menu: Annotate ► Multiple Dimension

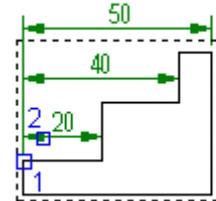
Keyboard command: GMAUTODIM

Creates multiple dimensions from a selected point to all the vertexes on the contour of selected objects.

Commonly Steps

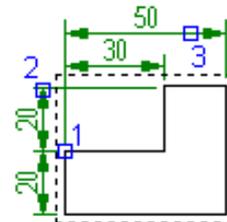
To Create 1-axis Baseline Dimensions

1. In the Automatic Dimensioning dialog box, Parallel tab, from the Type list select Baseline.
2. Verify that the Display dimension text editor for every dimension check box and Both Axes check box are cleared.
3. Click OK.
4. Select the objects to dimension, and press ENTER.
5. Specify the first extension line origin (1).
6. Place the dimensions (2).
7. Press ENTER to end dimensioning.



To Create 2-axis Baseline Dimensions

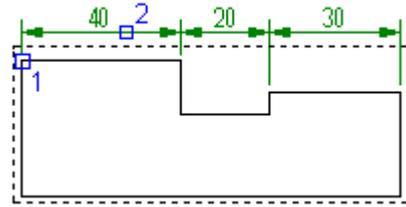
1. In the Automatic Dimensioning dialog box, Parallel tab, from the Type list select Baseline.
2. Select the Both Axes check box.
3. Verify that the Display dimension text editor for every dimension check box is cleared.
4. Select the objects to dimension, and press ENTER.
5. Specify the extension line origin (1) for the first axis.
6. Specify the extension line origin for the second axis.
7. Specify the dimension line location for the first axis (2).
8. Press ENTER to finish dimensioning the first axis.
9. Specify the dimension line location for the second axis (3).
10. Press ENTER to finish dimensioning.



To Create 1-axis Chain Dimensions

1. In the Automatic Dimensioning dialog box, Parallel tab, from the Type list select Chain.

2. Verify that the Display dimension text editor for every dimension check box and Both Axes check box are cleared.
3. Click OK.
4. Select the objects to dimension, and press ENTER.
5. Specify the first extension line origin (1).
6. Place the dimensions horizontally or vertically (2).
7. Press ENTER.



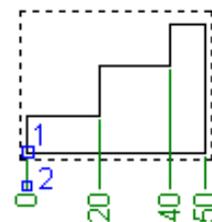
To Create 2-axis Chain Dimensions

In the Automatic Dimensioning dialog box, Parallel tab, from the Type list select Chain.

1. Select the Both Axes check box.
2. Verify that the Display dimension text editor for every dimension check box is cleared.
3. Select the objects to dimension, and press ENTER.
4. Specify the extension line origin (1) for the first axis.
5. Specify the extension line origin for the second axis.
6. Specify the dimension line location for the first axis (2).
7. Press ENTER to finish dimensioning the first axis.
8. Specify the dimension line location for the second axis (3).
9. Press ENTER to finish dimensioning.

To Create 1-axis Ordinate Dimensions

1. In the Automatic Dimensioning dialog box, Ordinate tab, verify that the Display dimension text editor for every dimension check box and Both Axes check box are cleared.
2. Click OK.
3. Select the objects to dimension, and press ENTER.
4. Specify the first extension line origin (1).
5. Place the dimensions horizontally or vertically (2).
6. Press ENTER.



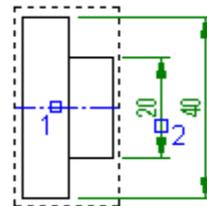
To Create 2-axis Ordinate Dimensions

1. In the Automatic Dimensioning dialog box, Ordinate tab, select the Both Axes check box.
2. Verify that the Display dimension text editor for every dimension check box is cleared.
3. Select the objects to dimension, and press ENTER.
4. Specify the extension line origin for the first axis.

5. Specify the extension line origin for the second axis.
6. Specify the dimension line location for the first axis.
7. Press ENTER to finish dimensioning the first axis.
8. Specify the dimension line location for the second axis.
9. Press ENTER to finish dimensioning.

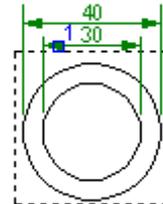
To Dimension a Shaft in Front View

1. In the Automatic Dimensioning dialog box, Shaft/Symmetric tab, from the Type list select Shaft (Front view).
2. Verify that the Display dimension text editor for every dimension check box is cleared.
3. Click OK.
4. Select the objects to dimension, and press ENTER.
5. Select the centerline (1).
6. Specify the dimension line location (2).
7. Press ENTER.



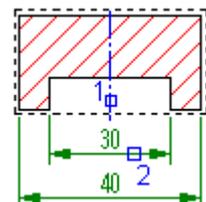
To Dimension a Shaft in the Side View

1. In the Automatic Dimensioning dialog box, Shaft/Symmetric tab, from the Type list select Shaft (Side view).
2. Verify that the Display dimension text editor for every dimension check box is cleared.
3. Click OK.
4. Select the objects to dimension, and press ENTER.
5. Specify the dimension line location (1).
6. Press ENTER.



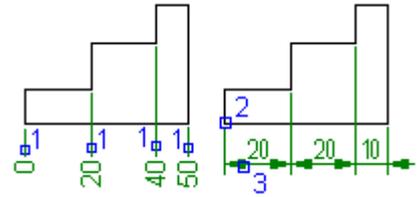
To Create a Symmetric Dimension

1. In the Automatic Dimensioning dialog box, Shaft/Symmetric tab, from the Type list select Symmetric.
2. Verify that the Display dimension text editor for every dimension check box is cleared.
3. Click OK.
4. Select the objects to dimension, and press ENTER.
5. Select the centerline (1).
6. Specify the dimension line location (2).
7. Press ENTER.



To Convert Between Chain Dimensions, Baseline Dimensions and Ordinate Dimensions

1. Click Home tab ► Dimension panel ► Power Dimension drop-down ► Multiple Dimension.
2. In the Automatic Dimensioning dialog box, select the tab corresponding to the dimension type to convert to.
3. Select the Rearrange into a New Style check box.
4. Click OK.
5. Select the dimensions to convert (1).
6. Press Enter.
7. Depending on the chosen dimension type, specify the first dimension origin point (2).
8. Specify the dimension line location (3).
9. Press Enter to finish the command.



Multiple Dimensioning Dialog Box

Parallel tab

Type: Sets the type of parallel dimension to Baseline or Chain.

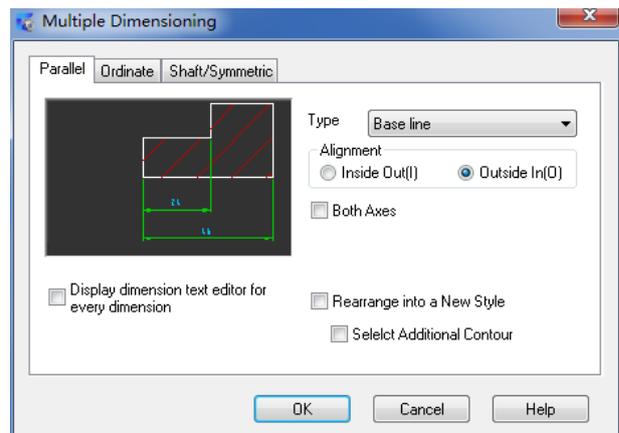
Both axes: Draws dimensions for both axes. Clear this option to draw a 1-axis dimension. A graphical representation of the dimension type selected appears in the preview section of this tab.

Display dimension text editor for every dimension

Displays the in-place dimension text editor or the Power Dimensioning dialog box (if the Ribbon is turned off or the AMPOWERDIMEDITOR system variable is set to 0) each time you place a dimension.

Rearrange into a New Style: Changes the style of existing dimensions to Parallel style, for example, from chain to baseline dimension.

Select Additional Contour: Dimensions additional contour and rearranges existing dimensions into any type of Parallel style.



Ordinate tab

Type: Sets the type of ordinate dimension to Current Standard (for example, ISO), to Equal Leader Length, or to Center Cross on Edge.

Both axes: Draws dimensions for both axes. Clear this option to draw a 1-axis dimension. A graphical representation of the dimension type selected appears in the preview section of this tab.

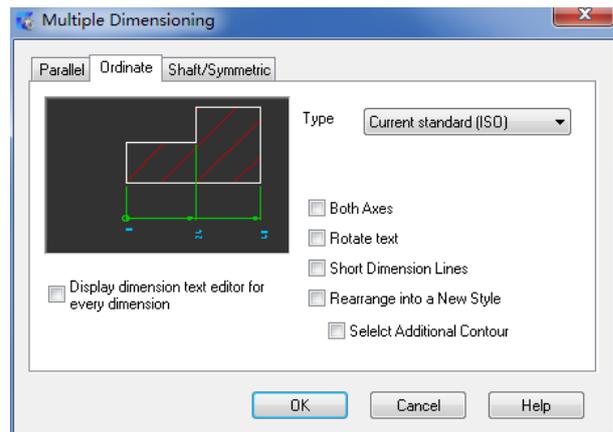
Rotate text: Rotates dimension text.

Short dimension lines: Shows only a small segment of the dimension line adjacent to the arrowhead. This option is visible only if the type of ordinate dimension is set to the current standard.

Display Power Dimensioning Dialog: Displays the Power Dimensioning dialog box each time you place a dimension.

Rearrange into a New Style: Changes the style of existing dimensions to Ordinate style (for example, from 1-axis chain dimension to 2-axes ordinate dimension).

Select Additional Contour: Dimensions additional contour and rearranges existing dimensions into any type of Ordinate style.



Shaft/Symmetric tab

Type: Sets the type of dimension to Shaft (Front View), Shaft (Side View), or Symmetric. A graphical representation of the dimension type selected appears in the preview section of this tab.

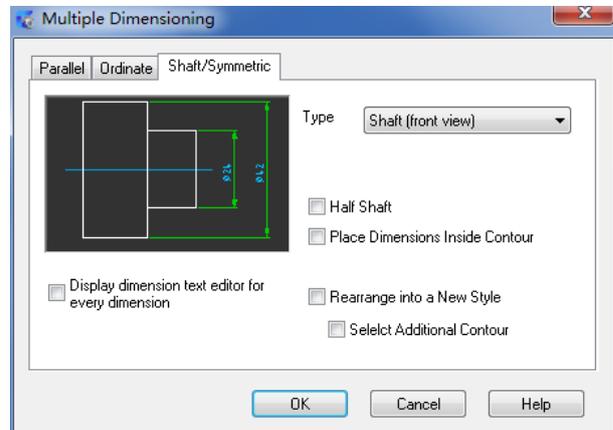
Half Shaft: Draws one extension line only for every dimension.

Place Dimension Inside Contour: Draws dimensions inside contours. Clear this option to draw dimensions outside contours.

Display Power Dimensioning Dialog: Displays the Power Dimensioning dialog box each time you place a dimension.

Rearrange into a New Style: Changes the style of existing dimensions into Shaft/Symm style.

Select Additional Contour: Dimensions additional contour and rearranges existing dimensions into any type of Shaft/Symm style.



Aligned

Ribbon: Annotate tab ► Dimensions panel ► Aligned

Menu: Annotate ► Aligned

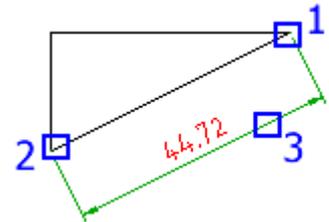
Keyboard command: GMPOWERDIM_ALI

Creates aligned linear dimensions.

To Create Aligned Dimension

Aligned: set the type of dimension to Aligned linear dimension.

1. Specifies the first point that defines the aligned linear dimension (1 in the illustration).
2. Specifies the second point that the aligned linear dimension (2 in the illustration).
3. Specifies where to place the dimension line. The dimension line moves only in the direction perpendicular to the two points (3 in the illustration).



Select objects: Draws an aligned linear dimension for the objects you select. If you select a line, the endpoints of the line are used as the origins of the extension lines. If you select a circle, the endpoints of its diameter are used as the origins of the extension line. The point used to select the circle defines the origin of the first extension line.

Horizontal

Ribbon: Annotate tab ► Dimensions panel ► Horizontal

Menu: Annotate ► Horizontal

Keyboard command: GMPOWERDIM_HOR

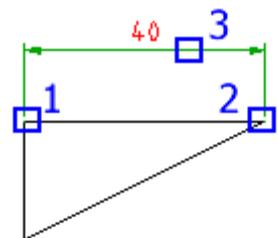
Creates horizontal linear dimensions.

To Create Horizontal Dimension

Horizontal: set the type of dimension to horizontal linear dimension.

1. Specify the first point of linear dimension (1).
2. Specify the second point of linear dimension (2).
3. When dimension line is moved to the required location, Click to determine the location (3). The dimension line can only move along the vertical direction.

Select object: Specifies an object to use as the base for the distance snap calculation.



Vertical

Ribbon: Annotate tab ► Dimensions panel ► Vertical

Menu: Annotate ► Vertical

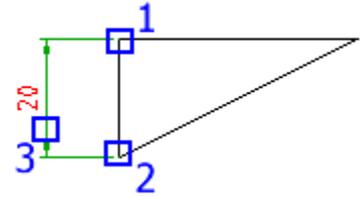
Keyboard command: GMPOWERDIM_VER

Creates vertical linear dimensions.

To Create Vertical Dimension

Vertical: Set the type of dimension to vertical linear dimension.

1. Specify the first point of linear dimension (1)
2. Specify the second point of linear dimension (2)
3. When dimension line is moved to the required location, Click to determine the location (3). The dimension line can only move along the horizontal direction.



Select object: Specifies an object to use as the base for the distance snap calculation.

Rotated

Ribbon: Annotate tab ► Dimensions panel ► Rotated

Menu: Annotate ► Rotated

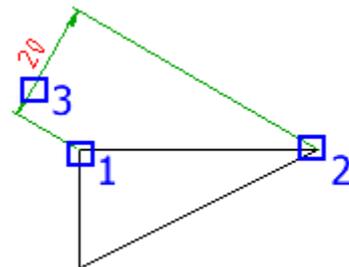
Keyboard command: GMPOWERDIM_ROT

Creates rotated linear dimensions.

To Create a Rotated Dimension

Rotated: Set the type of dimension to Rotated linear dimension.

1. Specify the first (1) and second (2) extension line origin.
2. Enter an angle for the dimension line.
3. Place the dimension line (3).



Radius

Ribbon: Annotate tab ► Dimensions panel ► Radius

Menu: Annotate ► Radius

Keyboard command: GMPOWERDIM_RAD

Creates radius dimensions for arcs and circles.

To Create a Radius Dimension

1. Specifies the arc or circle to create dimensions for.
2. Specifies where to place the dimension line.
3. Creates a linear dimension with a horizontal, vertical, or rotated dimension line.

- Depending on the direction you move the dimension, the dimension changes to a horizontal, vertical or aligned dimension.

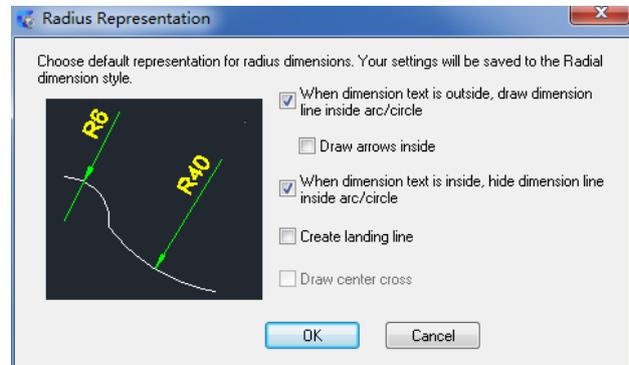
Options

Displays the Radius Dimension Options dialog box enabling you to configure the appearance of the dimension.

Radius Dimension Options Dialog Box

When dimension text is outside, draw dimension line inside arc/circle

Draws a dimension line inside the circle or arc and places the text, arrowheads, and leader outside. This option is applicable only when the radius is small and the dimension text must be outside the arc or circle. The preview updates to reflect the selection.



Draw arrows inside: Places the arrow inside the circle or arc.

When dimension text is inside, hide dimension line inside arc/circle

Suppresses the dimension line inside the circle or arc. This option is applicable only when the dimension text is inside the arc or circle. The preview updates to reflect the selection.

Create landing line: Draws dimension text horizontally when dimension text is outside the circle. This option is not available for the ANSI drafting standard.

Draw center cross: Draws a center mark if you place the dimension line outside the circle or arc. This option is available only for the ANSI drafting standard.

Diameter

Ribbon: Annotate tab ► Dimensions panel ► Diameter

Menu: Annotate ► Diameter

Keyboard command: GMPOWERDIM_DIA

Creates diameter dimensions for arcs and circles.

To Create a Diameter Dimension

- Specifies the arc or circle to create dimensions for.
- Specifies where to place the dimension line.
- Creates a linear dimension with a horizontal, vertical, or rotated dimension line.
- Depending on the direction you move the dimension, the dimension changes to a horizontal, vertical or aligned dimension.

Diameter Dimension Options Dialog Box

When dimension text is outside, draw dimension line inside arc/circle

Draws a dimension line inside the circle or arc and places the text, arrowheads, and leader outside. This option is applicable only when the diameter is small and the dimension text must be outside the arc or circle. The preview updates to reflect the selection.

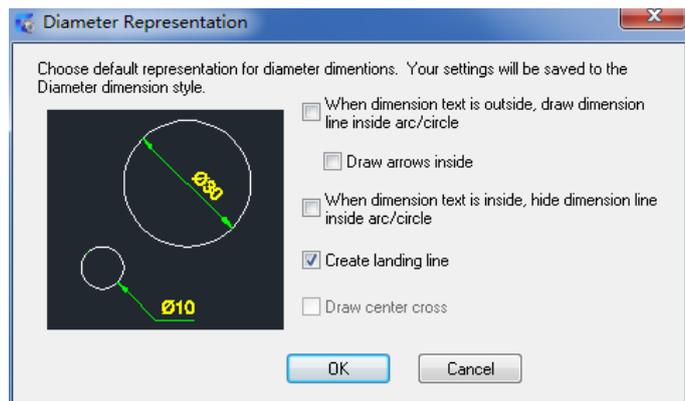
Draw arrows: Places the arrow inside the circle or arc.

When dimension text is inside, hide dimension line inside arc/circle

Suppresses the dimension line inside the circle or arc. This option is applicable only when the dimension text is inside the arc or circle. The preview updates to reflect the selection.

Create landing line: Draws dimension text horizontally when dimension text is outside the circle. This option is not available for the ANSI drafting standard.

Draw center cross: Draws a center mark if you place the dimension line outside the circle or arc. This option is available only for the ANSI drafting standard.



Jogged

Ribbon: Annotate tab ► Dimensions panel ► Jogged

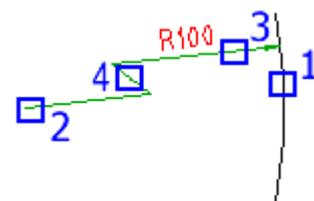
Menu: Annotate ► Jogged

Keyboard command: GMPOWERDIM_JOG

Create a radius dimension with the origin of the dimension at any location you wish and a jog at a convenient location along the dimension line.

To Create a Jogged Radius Dimension

1. Select arc or circle: Specifies the arc or circle to create dimensions for (1 in the illustration).
2. Center location override: Specifies the point to use in place of the true center point of the arc or circle (2 in the illustration).
3. Dimension line location: Indicates the angle of the dimension line (3 in the illustration). If you place the dimension such that the dimension points outside the arc, GstarCAD Mechanical automatically draws an arc extension line.
4. Jog location: Specifies where to place the jog (4 in the illustration).



Options: Displays the Power Dimension Options dialog box enabling you to pre-configure options for this command session.

Arc Length

Ribbon: Annotate tab ► Dimensions panel ► Arc Length

Menu: Annotate ► Arc Length

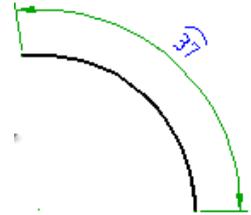
Keyboard command: GMPOWERDIM_ARC

Creates arc length dimensions for arcs and arc segments on a polyline.

To Create an arc Length Dimensio

1. **Select arc or polyline arc segment:** Specifies the arc or circle to create dimensions for.
2. **Dimension line location:** Specifies where to place the dimension line.
3. **Partial:** Creates an arc length dimension for only a section of the arc.

Options: Displays the Power Dimension Options dialog box enabling you to pre-configure options for this command session.



Angular

Ribbon: Annotate tab ► Dimensions panel ► Angular

Menu: Annotate ► Angular

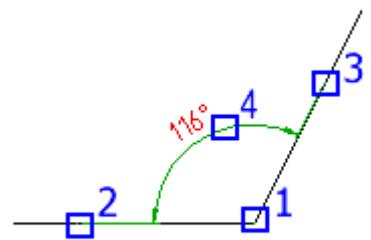
Keyboard command: GMPOWERDIM_ANG

Creates angular dimensions.

To Create Angular Dimensions

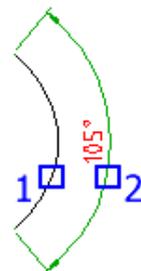
Vertex: Creates a dimension based on three points you specify.

1. **Vertex** - Specifies the point to use as the vertex of the angle (1 in the illustration).
2. **First angle endpoint** - Specifies one of the lines that form the angle (2 in the illustration).
3. **Second angle endpoint** - Specifies the other line that forms the angle (3 in the illustration).
4. **Dimension arc line location** - Specifies where to place the dimension arc line (4 in the illustration). Depending on the position of the dimension, the quadrant for the dimension changes.



Arc: Uses the endpoints of the selected arc and its center as the defining points for a three-point angular dimension. The center of the arc is the vertex.

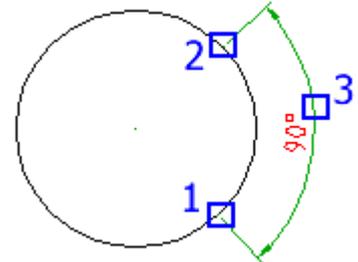
Dimension arc line location - Specifies where to place the dimension arc line (2 in the illustration). Depending on the position of the dimension, the quadrant for the dimension changes.



Quadrant - Specifies the quadrant to which the dimension is locked. When quadrant behavior is on, the dimension line extends past the extension line when the dimension text is outside of the angular dimension.

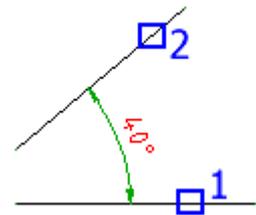
Circle: Uses two points on a circle and the center of the circle as the defining points for a three-point angular dimension.

1. The center of the circle is the vertex and the first selection point (1 in the illustration) is the origin of the first extension line.
2. Second angle endpoint - Specifies the origin of the second extension line (2 in the illustration).
3. Dimension arc line location - Specifies where to place the dimension arc line (3 in the illustration). Depending on the position of the dimension, the quadrant for the dimension changes.



Line: Creates a dimension for the angle between two lines.

1. The line you selected (1 in the illustration) is the first line that defines the angle.
2. Second line - Specifies the other line that forms the angle (2 in the illustration).
3. Dimension arc line location - Specifies where to place the dimension arc line.



Baseline: Creates an angular dimension from the baseline of a selected dimension.

1. **Base dimension:** Specifies the angular dimension to use as the base for the baseline dimension.
2. **Next extension line origin:** Specifies the next angle to dimension.

Chain: Creates a linear or angular dimension from the second extension line of a selected dimension.

1. **Base dimension:** Select the angular dimension to use as the base for the chain dimension.
2. **Next extension line origin:** Specifies the next angle to dimension.

Options: Displays the Power Dimension Options dialog box enabling you to pre-configure options for this command session.

Chamfer

Ribbon: Annotate tab ► Dimensions panel ► Chamfer

Menu: Annotate ► Chamfer

Keyboard command: GMCHAM2D_DIM

This command takes you directly to the Add Dimension option of the GMCHAM2D command. When you select the chamfer, it creates a chamfer dimension using the style specified in GMOPTIONS.

To Create a Chamfer Dimension

1. Select chamfer line: Specifies the chamfer to dimension. Press ENTER to exit the command.
2. Select first object: Selects the first of the two lines that form the chamfer.
3. Select second object: Selects the second of the two lines that form the chamfer.

To Set the Default Representation for Chamfer Dimensions

1. At the Command prompt, enter GMOPTIONS.
2. In the Standard Elements list, double-click Dimension. The Dimension Settings dialog box appears.
3. Click Chamfer. The Chamfer Representation dialog box appears.
4. From the drop-down list, select the default representation for chamfer dimensions.
5. Click OK until all dialog boxes close.

Baseline

Ribbon: Annotate tab ► Dimensions panel ► Baseline

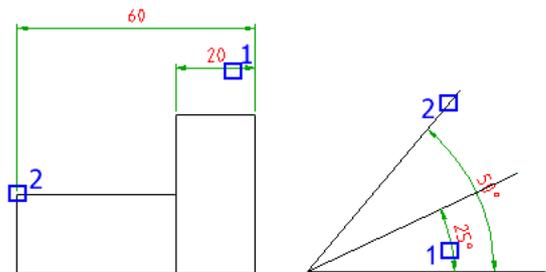
Menu: Annotate ► Baseline

Keyboard command: GMPOWERDIM_BAS

Creates a linear or angular dimension from the first extension line of a selected dimension.

To Create a Baseline Dimension

1. **Base dimension:** Specifies a linear or angular dimension to use as the base for the baseline dimension (1 in the illustration).
2. **Next extension line origin:** Specifies the next edge or angle to dimension (2 in the illustration).



Options: Displays the Power Dimension Options dialog box enabling you to pre-configure options for this command session.

Chain

Ribbon: Annotate tab ► Dimensions panel ► Chain

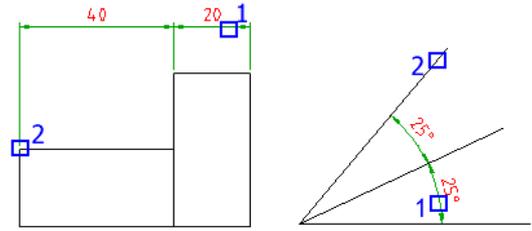
Menu: Annotate ► Chain

Keyboard command: GMPOWERDIM_CHAIN

Creates a linear, angular, or arc length dimension from the second extension line of a selected dimension.

To Create a Chain Dimension

1. **Base dimension:** Specifies a linear or angular dimension to use as the base for the chain dimension (1 in the illustration).
2. **Next extension line origin:** Specifies the next edge or angle to dimension (2 in the illustration).



Options: Displays the Power Dimension Options dialog box enabling you to pre-configure options for this command session.

Edit Dimensions

Edit Multiple

Ribbon: Annotate tab ► Dimensions panel ► Edit Multiple

Menu: Annotate ► Edit Dimensions ► Edit Multiple

Keyboard command: GMDIMMEDIT

Recognizes any object you select and invokes the most appropriate command to edit it.

To Edit Dimensions

Select object: Specifies the object to edit.

Arrange

Ribbon: Annotate tab ► Dimensions panel ► Arrange

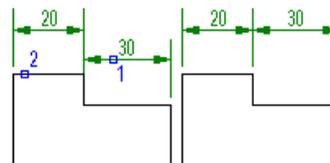
Menu: Annotate ► Edit Dimensions ► Arrange

Keyboard command: GMDIMARRANGE

Rearranges linear and ordinate dimensions, placing them at an appropriate distance from the outer contour of an object you select.

To Base Dimension Placement on a Contour

1. Select the dimensions to rearrange (1).
2. Press ENTER.
3. Select the contour on which to base these dimensions (2).
4. Press ENTER.



List of Prompts

Select dimensions and contour entities: Specifies the dimensions and the contour for the arrange operation.

Select point on contour: Specifies a point on an edge relative to the command spaces and arranges the dimensions.

Note: With GMDIMARRANGE, the dimension is not replaced; only its position changes.

Linear/ Symmetric Stretch

Ribbon: Annotate tab ► Dimensions panel ► Linear/ Symmetric Stretch

Menu: Annotate ► Edit Dimensions ► Linear/ Symmetric Stretch

Keyboard command: GMDIMSTRETCH

Resizes objects by stretching or shrinking a linear or symmetric dimension associated with the object.

To Resizes objects by stretching or shrinking a linear or symmetric dimension

Linear: Resizes objects based on the difference of value of a selected linear dimension and the value you type in.

Select dimension text: Specifies the dimension to stretch/shrink.

New dimension text: Specifies the value to stretch/shrink the dimension to.

First corner: Specifies the corner to stretch or shrink. Select the corner of the object and the origin of the extension line of the dimension using a crossing window selection box.

Select objects: Selects additional objects for stretching. Use the crossing object selection method and select one corner of the object. To remove an object from selection set, SHIFT-click the object. Press ENTER when you finish selecting objects.

Symmetric: Resizes objects based on the difference of the value of a selected dimension of symmetrical objects (shaft objects) and the value you type in.

1. **Select dimension text:** Specifies the dimension to stretch/shrink.
2. **New dimension text:** Specifies the value to stretch/shrink the dimension to.
3. **Select Centerline:** Specifies the line about which to stretch/shrink symmetrically.
4. **First corner:** Specifies the corner to stretch or shrink. Select the corner of the object and the origin of the extension line of the dimension using a crossing window selection box.
5. **Select objects:** Selects additional objects for stretching. Use the crossing object selection method and select one corner of the object. To remove an object from selection set, SHIFT-click the object. Press ENTER when you finish selecting objects.

Insert

Ribbon: Annotate tab ► Dimensions panel drop-down ► Insert

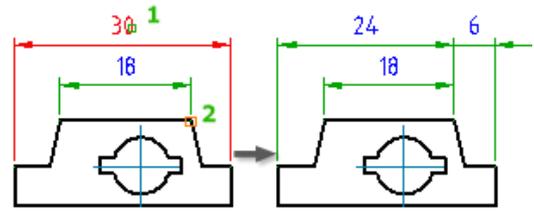
Menu: Annotate ► Edit Dimensions ► Insert

Keyboard command: GMDIMINSERT

Splits a linear or angular dimension into two individual dimensions.

To Split Dimensions

1. **Select base dimension:** Specifies the linear or angular dimension to split (1 in the illustration above).
2. **Locate extension line origin:** Specifies origin of the extension line of the new dimension (2 in the illustration above).



Join

Ribbon: Annotate tab ► Dimensions panel drop-down ► Join

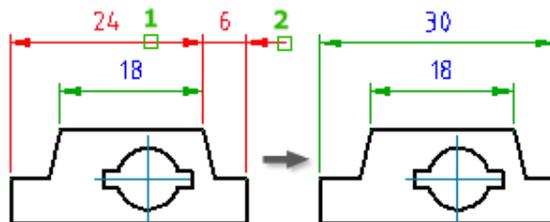
Menu: Annotate ► Edit Dimensions ► Join

Keyboard command: GMDIMJOIN

Combines two individual dimensions (of the same type) into a single dimension.

To Join Dimensions

1. **Base dimension:** Specifies a linear or angular dimension to use as the base dimension
2. **Dimensions to join:** Specifies the dimensions to combine with the base dimension.



Align

Ribbon: Annotate tab ► Dimensions panel drop-down ► Align

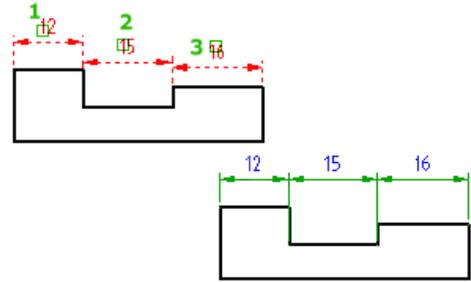
Menu: Annotate ► Edit Dimensions ► Align

Keyboard command: GMDIMALIGN

Lines up linear, ordinate or angular dimensions, with a dimension you select as the baseline.

To Align Dimensions

1. **Select base dimension:** Specifies a dimension to use as the base dimension for the align operation. You can select a linear, ordinate, or angular dimension.
2. **Select linear dimensions to align:** Specifies a set of linear dimensions to line up with the base dimension. You can only select dimensions that are parallel to the base dimension.
3. **Select angular dimensions to align:** Specifies a set of angular dimensions to line up with the base dimension. You can only select dimensions that measure angles about the same vertex as the base dimension.
4. **Select ordinate dimensions to align:** Specifies a set of ordinate dimensions to line up with the base dimension. You can only select a set of ordinate dimensions whose extension lines are parallel to the base dimension.



Check

Ribbon: Annotate tab ► Dimensions panel drop-down ► Check

Menu: Annotate ► Edit Dimensions ► Check

Keyboard command: GMCHECKDIM

Highlights or edits dimensions with dimension text overrides.

To Check Dimensions

1. **Highlight:** Highlights the overridden dimensions.
2. **Remove highlight:** Removes the highlight from the overridden dimensions.
3. **Edit:** Steps through each highlighted dimension and enables you to edit the dimension.

Chapter 6 Symbols

Summary

GstarCAD Mechanical 2017 provides 11 types of symbols to insert into a drawing. The supported symbols are:

- Datum identifier symbols
- Datum target symbols
- Edge symbols
- Feature control frame symbol
- Feature identifier symbols
- Leader note symbols
- Taper and slope symbols
- Surface texture symbols
- Welding symbol
- Marking/Stamping symbol
- Dead joint symbol

Symbols and the Drafting Standard

The drafting standard defines the behavior and characteristics of each symbol. To change the characteristics, you edit the elements of the standard. You access elements of the standard in the AM:Standards tab in the Options dialog box.

Attach behavior

While creating a symbol, you can attach it to an object. Thereafter, when you move the object, the symbol moves with it. Symbols can get detached from an object when the point of attachment is no longer valid. For example

Symbol libraries

Some symbols let you save fully configured symbols to a library. You can then directly insert them in a drawing from the ribbon, eliminating the need to specify values, or only specify the values that are from the saved symbol.

Edit Symbols

To edit symbol text of a symbol that is already inserted in a drawing, double-click the symbol. Double-clicking the leader displays the Leader tab of the symbol's settings dialog box.

Create symbols

Leader Note

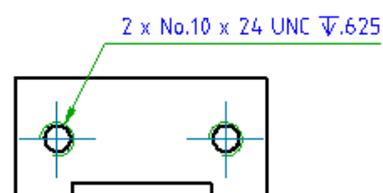
Ribbon: Annotate tab ➤ Symbol panel ➤ Leader Note

Keyboard command: GMNOTE

Creates a leader note and attaches it to an object in the drawing area

To Create a Leader Note

1. Click Annotate tab ➤ Symbol panel ➤ Leader Note.
2. Select an object to attach the leader note to.
3. If you attached the symbol to a line, in the drawing area, specify the start point for the leader.



If you attached the object to an arc, circle, ellipse or spline, skip to the next step.

4. Specify one or more points to define the vertices of the leader, and then press ENTER.
5. In the Symbol tab, specify the general appearance of the symbol.
6. Enter the note text in the multi-line text editor.

Note: In Preview display mode, select Rendered text colored to view how the leader note appears in the drawing area.

7. Click OK.

Note Dialog Box

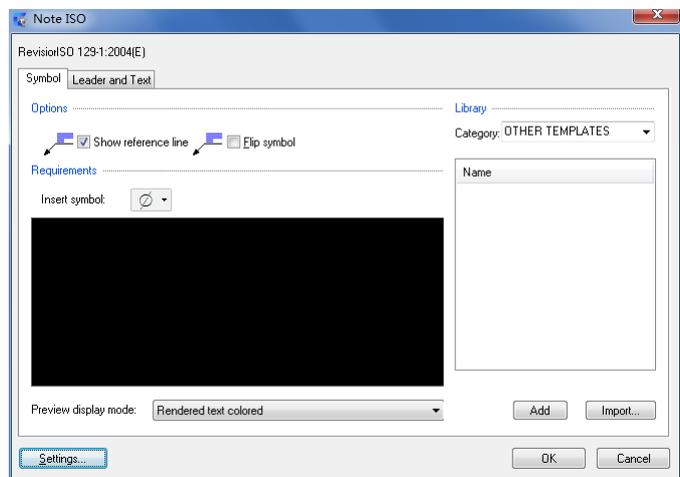
Use the Note Settings dialog box to customize the default settings of leader notes for the current drafting standard.

Symbol tab

Options

Show reference line: Adds a reference line from the leader to the note text. The icon to the left of the check box shows the result.

Flip symbol: Flips the symbol so that note text is placed to the left of the leader. The icon to the left of the check box shows the result of your selection.



Adds an additional reference line above the currently selected reference line. Note text you add thereafter is inserted on the new reference line.

This button is visible only for the GOST standard.



Deletes the current reference line. The current reference line is highlighted in the drawing.

This button is visible only for the GOST standard.



Selects the next reference line away from the leader arrowhead, and displays its settings in the dialog box.

This button is visible only for the GOST standard.



Selects the next reference line toward the leader arrowhead and displays its settings in the dialog box.

This button is visible only for the GOST standard.

Requirements

Insert symbol: Displays a palette to enable you to insert a special character at the current cursor position in the multi-line text editor.

In the rendered text mode, the editor shows the special character while in the plain text mode the editor shows the corresponding control key sequence.

Multi-line text editor: Edits note text. Use this editor to edit text, formulas, and special symbols to be displayed as note text. Use the right-click menu to access special editing tools.

Right-click menu options: Enables you to specify the formatting for the segment of text selected in the multi-line text editor. The formatting options are:

Format > Underline - Toggles underlining for the selected text

Format > Overline - Toggles overline for the selected text

Format > Align - Specifies how the selected text aligns with stacked text on the same line

Format > Stacked- Stacks two segments of text, separated by a "/" character.

Format > Text Color - Specifies a color for the selected text

Format > Text Height - Applies a scale factor for the selected text

Format > Text Font - Specifies a font for the selected text

Format > Text Width - Applies a scale factor for the width of the selected text

Format > Tracking Factor - Specifies the spacing between characters of the selected text

Dim Style >Apply Dim Style - Applies a dimension style for a numerical expression. This option is visible only when a numerical expression is selected.

Note: The remaining options in the Dim Style menu lists the available dimensions styles that you can select from.

Precision - Enables you to specify a precision for a numerical expression. This option is visible only if a numerical expression is selected. It is not available if you have applied a dimension style to the selected numerical expression.

Cut - Moves the selected text to the windows clipboard.

Copy - Copies the selected text to the windows clipboard.

Paste - Inserts the contents of the windows clipboard at the insertion point.

Delete - Erases the selected text.

Select All - Selects the entire content of the multi-line text editor.

Preview display mode

Specifies how note text appears in the multi-line text editor.

Plain Text - Displays formulas and formatting signs that define the note text.

Rendered Text Colored - Renders all formulas and formatting signs to preview the note.

Rendered Text Mono - Displays monochromatically the same text as the Rendered Text Colored mode.

Library

Category: Displays a list of object categories for which you can create note templates.

Templates: Displays the list of available templates for the selected category.

Menu options

Add: Saves the text in the multi-line text editor as a new note template. The new template is added to the symbol library.

Delete: Deletes the selected note template from the symbol library.

Rename: Makes the template name editable. GstarCAD Mechanical validates the name you provide to ensure that the names of templates for a given category are unique.

Set default: Makes the selected template the default template for the selected category. A blue tick mark appears next to default template name. When you attach a note to an object described by the selected category, the default template loads automatically.

Remove default: Removes the default status of the selected template. This menu option is available only on the default template. When no default template is selected, no note text appears in the multi-line text editor.

Add: Saves the content of the multi-line text editor as a new template. It prompts to name the new item.

Import: Imports templates from the symbol library of another drawing. You can only import symbols of the same standard and revision as the one you are editing.

Leader and Text tab

Leader

Arrowhead: Specifies the default leader arrowhead type for note symbols.

Note: If you select the arrowhead type labeled "By NameOfStandard", the arrowhead type is linked to the leader arrowhead setting of the master settings of the drafting standard. If the master settings change, the arrowhead type setting for note symbols change accordingly.

Arrow size: Edit leader segments

Add: Adds a leader, leader segment, or leader node.

Remove: Deletes a leader, or leader segment.

Edit object attachment

Attach: Attaches the symbol to an object in the drawing. When attached, if you move the object the symbol moves with it.

Detach: Detaches the symbol. The symbol becomes a free-standing object.

Reference line angle

Adjust: Hides the dialog box so that you can set the direction of the reference line. This button is not available in the GOST standard.

Color: Specifies the default leader color for note symbols.

Note: If you select the color labeled "By NameOfStandard", the leader color is linked to the leader color setting of the master settings of the drafting standard. If the master settings change, the color setting for note symbols change accordingly.

Text

Text justification: Sets the justification of text.

Note: This setting reverts to its default value if you flip the symbol.

Leader justification: Sets the position of the note text relative to the reference line.

Note: This option is disabled for the GOST standard.

Restore Defaults

Restores all settings of this dialog box to the preset default values for the drafting standard.

Surface Texture Symbols

Ribbon: Annotate tab ► Symbol panel ► Surface Texture

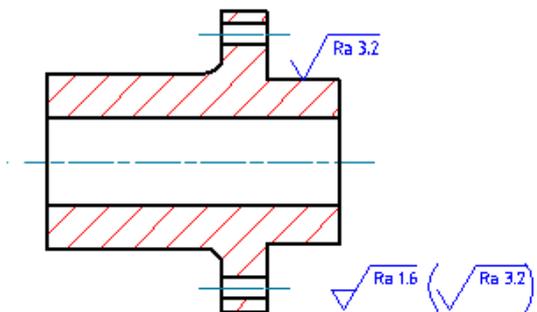
Keyboard command: GMSURFSYM

Creates a surface texture symbol and attaches it to an object in the drawing area.

Commonly Steps

To Create a Surface Texture Symbol

1. Click the Annotate tab ► Symbols panel ► Surface Texture.
2. Select an object to attach the surface texture symbol to.
3. If you attached the symbol to a line, in the drawing area, specify the start point for the leader.
4. If you attached the object to an arc, circle, ellipse, skip to the next step.
5. Specify one or more points to define the vertices of the leader, and then press ENTER.
6. In the Symbol tab, specify the surface texture requirements as required.
7. To set the symbol type, select either Basic symbol, Material removal required, or Material removal prohibited.
8. To add a tail to the surface texture symbol, select Force tail.
9. To set the symbol to be a collective indication of all other surface texture symbols in the drawing. select Majority symbol.
10. To add the all-around designation, click the All-Around button.



Note: To enter special characters such as the wave filter notation (λ), enter the corresponding Unicode character string (U+03BB in this case). Click OK.

Note: Some of these steps may not be valid for the GOST standard.

To Create a Surface Texture Symbol with a Surface Indication Leader

1. Click the Annotate tab ► Symbols panel ► Surface Texture.
2. Select an object to attach the surface texture symbol to.
3. At the command prompt, enter F.
4. Note: The surface option is available only if the revision of the standard in use permits surface indication leaders
5. Specify the start point of the surface indication leader. (1 in the illustration above).
6. Specify one or more points to define the vertices of the leader (2 in the illustration above), and then press ENTER.
7. Click OK.

To Place a Surface Texture Symbol as a Majority Symbol

1. Click the Annotate tab ► Symbols panel ► Surface Texture.
 2. Click the position you want to place the symbol and press ENTER 2 times.
 3. In the Symbol tab, specify the surface texture requirements as required.
 - To set the symbol type, select either Basic symbol, Material removal required, or Material removal prohibited.
 - To add a tail to the surface texture symbol, select Force tail.

Note: To enter special characters such as the wave filter notation (λ), enter the corresponding Unicode character string (U+03BB in this case).
 4. Select Place as majority symbol.
 5. Click OK.
-

Surface Texture Dialog Box

Use the Surface Texture dialog box to specify the surface texture requirements for a surface.

Note:

The elements in this dialog box depend on the current drafting standard and options you select. Some of the elements described in this page may not appear.

Some buttons on this dialog box are only named in tooltips. Move the pointer over the buttons to see their names.

Symbol tab

Options

Sets the options for the graphical display of the symbol. The availability of each option depends on the drafting standard and whether buttons are configured to appear or hide. The options are:

Basic symbol: Draws the basic symbol only.

Material removal required: Inserts the “machining required” designation on the symbol.

Material removal prohibited: Inserts the “machining prohibited” designation on the symbol.

Force tail: Controls the availability of a tail for a symbol without text.

Use majority symbol: Sets the symbol to be a collective indication of all other surface texture symbols in the drawing. This option is available only if you created the symbol without a leader.

Note: Click Properties to select between the simplified and full-list representations. In the simplified representation, only the basic symbol appears in parentheses to the right of the collective indication. In the full list representation, the state of any surface texture symbol present elsewhere in the drawing is fully indicated in parentheses to the right of the collective indication. Full list is not an option for the GOST standard. Simplified is the default selection.

Library

Lists all the surface texture symbols for the active drafting standard/revision in the symbol library. When you move the pointer over an item, all settings for that item appear in a tooltip. Each item in the library is associated with a context-sensitive menu.

Menu options

Load: Loads the selected symbol from the library. The current settings in the Surface Texture Symbol dialog box are overwritten.

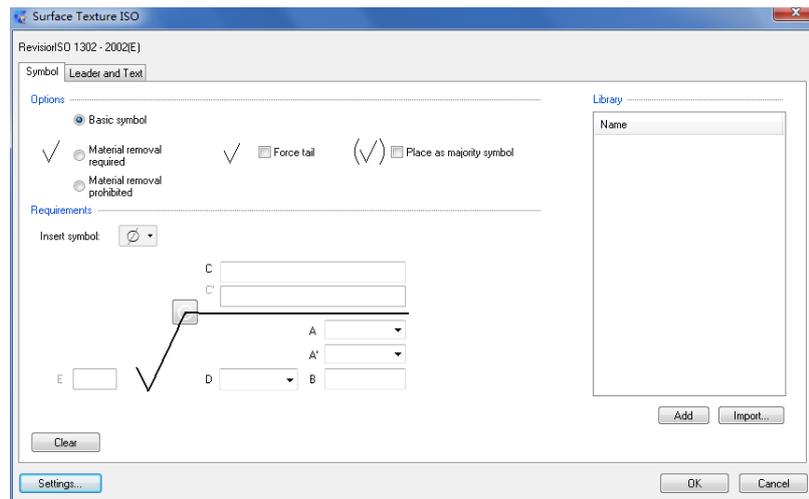
Update: Overwrites the selected symbol in the library with the current settings of the Surface Texture Symbol dialog box. The preview updates to reflect the new settings. The name of the symbol and default status do not change with this operation.

Delete: Removes the selected symbol from the library.

Rename: Makes the symbol name editable. ProdName validates the name you provide to ensure that the names of the symbols in the library are unique.

Set default: Makes the selected symbol the default symbol of the library. A blue check mark appears next to the symbol to indicate that it is the default symbol. The next time you create a surface texture symbol, the default symbol loads automatically.

Remove default: Removes the default status of the selected symbol. This menu option is available only on the default symbol. When no default symbol is selected, the surface texture symbol dialog box defaults to the settings of the surface texture symbol created or edited most recently.



Add: Saves current settings to the symbol library as a new item in the library. A prompt enables you to name the new item.

Import: Imports the surface texture symbols from the symbol library of another drawing. You can only import symbols of the same standard and revision as the surface texture symbol you are editing.

Requirements

Sets the options used to specify surface texture requirements. The boxes used for data entry appear as and when you add data. The order in which the boxes appear depends on the type of weld you select and therefore may be different to the order in which they appear in the following list. When you move the pointer over a box, the tooltip describes what information to enter to that box.

Insert symbol: Displays a palette to enable you to insert a special character at the current cursor position. The preview in the drawing area shows the special character while the dialog box shows the corresponding control key sequence.

All-around: Inserts the all-around indication to the symbol.

Machine allowance: Specifies the machining allowance for the surface. Not available in the GOST standard.

Roughness average max: Specifies the maximum roughness average for the surface.

Roughness average min: Specifies the minimum roughness average for the surface.

First requirement: Specifies the first requirement for the surface.

Second requirement: Specifies the second requirement for the surface.

Third requirement: Specifies the third requirement for the surface.

Fourth requirement: Specifies the fourth requirement for the surface.

Note: This option is available only in the GOST standard.

Texture Direction: Identifies the direction that the texture should take.

Note: This option is available only in the GOST standard.

Production method: Specifies the production requirement for the surface.

User added values that are not identical to existing values are added to the drop-down list.

Note: This option is available only in the GOST standard.

Process note: Specifies the process requirements for the surface.

Process note 1: Specifies the process requirements for the surface.

Process note 2: Specifies the process requirements for the surface.

Sampling length: Specifies the sampling length required for the surface.

Maximum spacing: Specifies the maximum spacing for the surface.

Cutoff: Specifies the cutoff value for the surface.

Reference: Specifies the reference and evaluation length for the surface.

Waviness: Specifies the waviness value for the surface.

Other roughness: Specifies the other roughness for the surface.

Other roughness max: Specifies the maximum other roughness for the surface.

Other roughness min: Specifies the minimum other roughness for the surface.

Direction of lay: Specifies the direction of lay for the surface.

Base length: Enables force tail option when a value is present. This option is available only for the 1973 GOST standard.

Clear

Clears all the data and returns the dialog box options to the default values.

Leader and Text tab

Leader

Arrowhead: Specifies the type of the leader arrowhead. When the arrowhead is set to By Standard, it acquires the arrowhead from the current drafting standard. Thereafter, if you change the arrowhead selected for the current drafting standard (in the Standard Settings dialog box. Or Surface Texture Settings dialog box.) the arrowhead for this symbol updates automatically.

Note: When the leader attached to the symbol is a surface indication leader, the name of this option changes to Surface arrowhead.

Edit leader segments

Add: Adds a leader, leader segment, or leader node.

Remove: Deletes a leader, or leader segment.

Edit object attachment

Attach: Attaches the symbol to an object in the drawing. When attached, if you move the object the symbol moves with it.

Detach: Detaches the symbol. The symbol becomes a free-standing object.

Symbol scale

Enlarges or reduces the size of the graphical symbol. The drawing area shows a dynamic preview of the symbol while you change the scale factor. During scaling, the system maintains the line weight at 10% of the symbol text height. When you enlarge or reduce the size of the symbol, the scale factor snaps only to a value for which a valid line weight is available.

Surface extension line

Controls options for extension lines to the start point of the leader. Extension lines are drawn only if you move the start point of the leader beyond the end of the line or arc it is attached to. The options in this section are available only if extension lines for this symbol exist.

Offset from object: Defines the distance between the extension line's start point and the attached object.

Sync: Loads values for the offset from the DIMEXO system variable.

Extension beyond leader: Defines the distance between the symbol's start point and the surface extension line's end point

Sync: Loads values for the extension from the DIMEXE system variable.

Text

Automatically adjust symbol position on leader: Moves the symbol to the default position on the leader.

Settings

Opens the Surface Texture Settings dialog box and enables you to edit the default settings for surface texture symbols for the current drafting standard.

Feature Control Frame

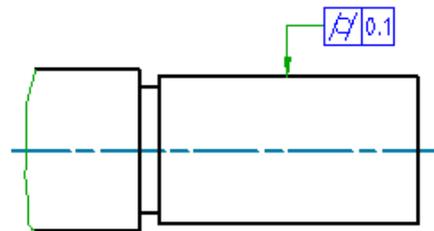
Ribbon: Annotate tab ► Symbols panel ► Feature Control Frame

Keyboard command: GMFCFRAME

Creates a feature control frame symbol and attaches it to an object in the drawing.

To Create a Feature Control Frame

1. Click Annotate tab ► Symbols panel ► Feature Control Frame .
2. Select an object to attach the feature control frame to.
3. If you attached the symbol to a line, in the drawing area, specify the start point for the leader.



If you attached the object to an arc, circle, ellipse, skip to the next step.

4. Specify one or more points to define the vertices of the leader, and then press ENTER.
5. In the Feature Control Frame dialog box, enter the data.
6. Click OK.

Note: To create the symbol without a leader, specify only the start point and press ENTER.

Feature Control Frame Dialog Box

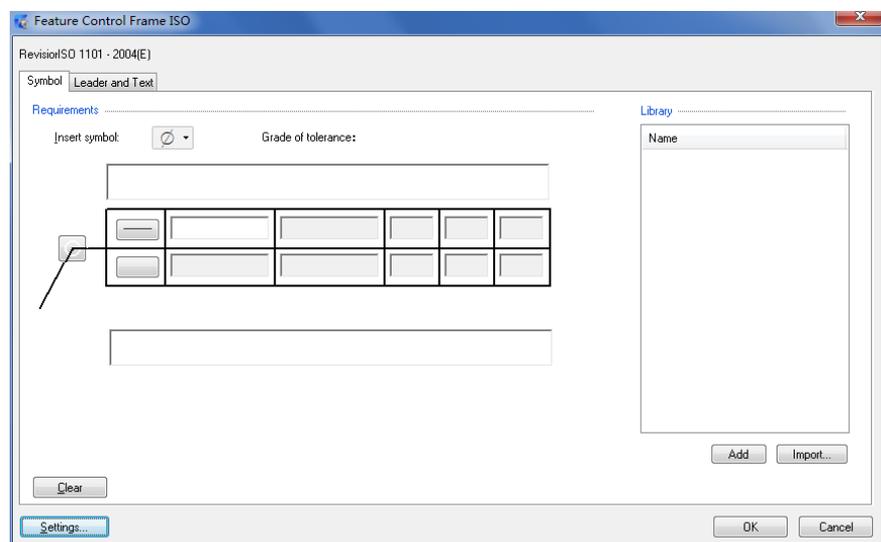
Use this dialog box to describe the allowable variation from nominal geometry.

Symbol tab

Requirements

Contains buttons and edit boxes to specify the variations. These are named only in tooltips. Move the pointer over them to identify them.

Insert symbol: Displays a palette enabling you to insert a special character at the current cursor position. The preview in the drawing area shows the special character



while the dialog box shows the corresponding control key sequence.

Top Notes: Specifies text that must display above the feature control frame. In the drawing area the notes and the feature control frame move together as a single entity.

Geometric symbol: Displays a list of symbols to enable you to specify the geometric characteristic for which you are defining the tolerance. The symbols on this list depend on the current standard.

Tolerance 1: Specifies primary tolerance data. Use the Insert symbol list to insert special characters such as the diameter symbol or the maximum material condition symbol.

Tolerance 2: Specifies secondary tolerance data. This option is available only for the ANSI standard.

Datum 1: Specifies the primary datum from which the tolerance is measured.

Datum 2: Specifies the secondary datum from which the tolerance is measured.

Datum 3: Specifies the tertiary datum from which the tolerance is measured.

Bottom Notes: Specifies text that must display below the feature control frame. In the drawing area the notes and the feature control frame move together as a single entity.

All Around: Adds the all-around indication to the symbol. This option is unavailable for the GOST standard.

Library

Lists all feature control frame symbols in the symbol library, for the current drafting standard.

Menu Options

Load: Loads the settings of the selected symbol to the Feature Control Frame dialog box. This operation overwrites the current settings of the Feature Control Frame dialog box.

Update: Overwrites the selected symbol in the library, with the current settings of the Feature Control Frame dialog box. The preview updates to reflect the new settings. The name of the symbol and default status do not change.

Delete: Removes the selected symbol from the library.

Rename: Makes the symbol name editable. AutoCAD Mechanical validates the name to ensure that the symbol names are unique.

Set default: Makes the selected symbol the default symbol of the library. A blue check mark appears next to the symbol to indicate that it is the default symbol. The next time you create a feature control frame symbol, the default symbol loads automatically.

Remove default: Removes the default status of the selected symbol. This menu option is available only on the default symbol. When there is no default symbol selected, the settings default to the settings of the feature control frame symbol you created or edited most recently.

Add: Saves current settings to the symbol library as a new item in the library. A prompt enables you to name the new item.

Import: Imports the feature control frame symbols from the symbol library of another drawing. You can only import symbols of the same standard and revision as the feature control frame symbol you are editing.

Clear

Clears all the data and returns the dialog box options to the default values.

Leader and Text tab

Leader

Arrowhead: Specifies the type of the primary leader arrowhead. When the arrowhead is set to By Standard, it acquires the arrowhead from the current drafting standard. Thereafter, if you change the arrowhead selected for the current drafting standard (in the Standard Settings dialog box or Feature Control Frame Settings dialog box) the arrowhead for this symbol updates automatically.

Secondary arrowhead: Specifies the type of the secondary leader arrowhead. When the arrowhead is set to By Standard, it acquires the arrowhead from the current drafting standard. Thereafter, if you change the arrowhead selected for the current drafting standard (in the Standard Settings dialog box or Feature Control Frame Settings dialog box) the arrowhead for this symbol updates automatically.

Surface arrowhead: Specifies the type of arrowhead to use at the end of the surface indication leader. When the arrowhead is set to By Standard, it acquires the arrowhead from the current drafting standard. Thereafter, if you change the arrowhead selected for the current drafting standard (in the Feature Control Frame Settings dialog box) , the arrowhead for this symbol updates automatically..

Note: This option is available only if a surface indication line exists for the symbol you are editing.

Edit leader/segments

Add: Adds a leader, leader segment(s), or leader node(s).

Remove: Deletes a leader, or leader segment(s).

Edit object attachment

Attach: Attaches the symbol to a geometry in the drawing.

Detach: Detaches the symbol. The symbol becomes a free-standing object.

Surface extension line

Controls options for extension lines to the start point of the leader. Extension lines are drawn only if you move the start point of the leader beyond the end of the line or arc it is attached to. The options in this section are available only if extension lines for this symbol exist.

Offset from object: Defines the distance between the extension line's start point and the attached object.

Sync: Loads values for the offset from the DIMEXO system variable.

Extension beyond leader: Defines the distance between the leader start point and the extension line end point.

Sync: Loads values for the extension from the DIMEXE system variable.

Settings

Opens the Feature Control Frame Settings dialog box, enabling you to edit the default settings for the current drafting standard.

Welding Symbol

Ribbon: Annotate tab ➤ Symbols panel ➤ Welding Symbol

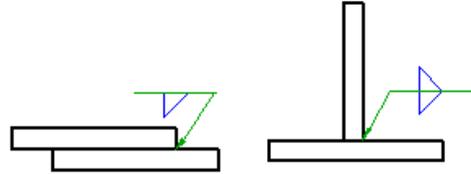
Keyboard command: GMWELDSYM

Creates a welding symbol and attaches it to the weld point.

To Create a Welding Symbol

1. Click Annotate tab Symbols panel Welding Symbol.
2. Select an object to attach the weld symbol to.
3. If you attached the symbol to a line, in the drawing area, specify the start point for the leader.

If you attached the object to an arc, circle, ellipse or spline, skip to the next step.



4. Specify one or more points to define the vertices of the leader and press ENTER.
5. In the Symbol tab, specify the general appearance of the symbol as required.
 - To align the position of the arrow side data, the identification line, or direction of the symbol tail to the left, in the Options section, select Flip symbol.
 - To stagger the alignment of intermittent fillet welds (or ANSI edge welds) on both sides, in the Options section, select Stagger.
 - To add the all-around designation, click the All-Around button.
 - To add a tail to the welding symbol, click the Add Note Tail button.
 - To add a process from a predefined list:
 - 1) Click Add Process.
 - 2) To prefix process numbers with the statement "Process ISO 4063," select the Prefix Process number with "Process ISO 4063-" check box.
 Note: This option is not available if the current drafting standard is ANSI.
 - 3) Double-click the processes to add.
 - 4) Click Close.
 - To edit processes by entering details using the keyboard:
 - 1) In the box beneath Add Process, click at the position you want to edit.
 - 2) Enter the process details.
 - To delete a process, select it and press the DELETE key.
 - To draw a box around the note, select Add note frame.
 Note: This button is not available if the current drafting standard is ANSI.
6. To specify arrow side data:
 - 1) Click the Weld Type button on the arrow side and select a weld symbol. The available options change to match the symbol you select.
 - 2) Specify parameters.
7. To specify other side data:

Click the Weld Type button on the other side and select a weld symbol. The available options change to match the symbol you select.

Specify parameters.

- Click OK.

Note: To clear all data in the Weld Symbol dialog box, click Clear.

Weld Symbol Dialog box

Use the Weld Symbol dialog box to set up the parameters of the symbol you are inserting.

Note: The visibility of elements in this dialog box depends on the current drafting standard and the options you select.

Some of the elements described in this page may not appear on the dialog box.

Some buttons on this dialog box are named only in tooltips. Move the pointer over the buttons to obtain their names.

Symbol Tab

Options: Changes the orientation of welding symbol elements.

Flip sides: Swaps symbols on the arrow side with the symbols on the other side. The icon to the left of the check box shows the result of your selection.

Flip symbol: Flips the symbol so that the weld tail is placed to the left of the arrow. The icon to the left of the check box shows the result of your selection.

Stagger: Specifies that the weld represents a staggered intermittent weld. The icon to the left of the check box shows the result of your selection. This option is available only if you select fillet welds or ANSI edge welds for both sides.

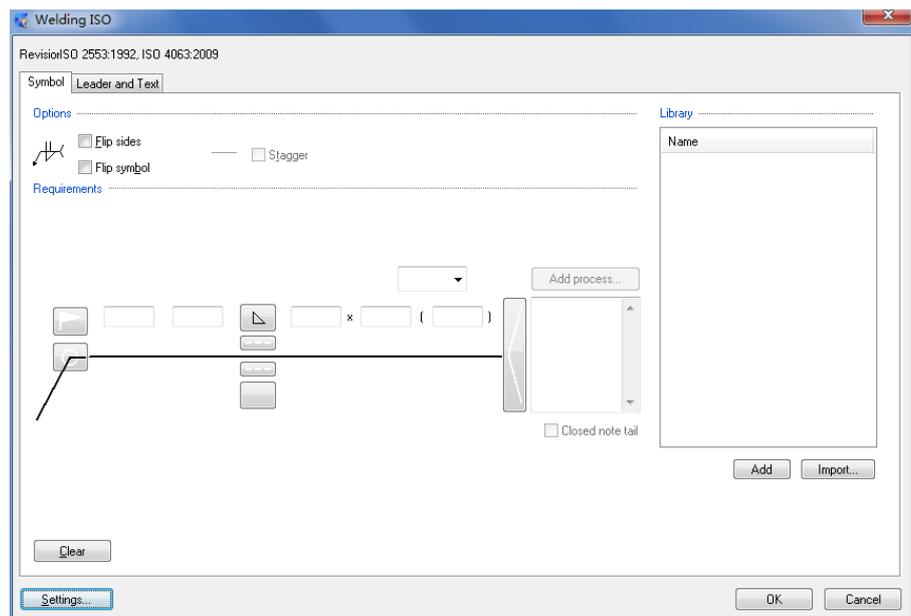
Mirror: Specifies that the weld represents a staggered intermittent weld where the shape and dimensions of the fillet on both sides are equal. The icon to the left of the check box shows the result of your selection. This option is visible only if you select JIS fillet welds for both sides.

Spacer: Adds a spacer symbol to the weld.



Adds a reference line to the welding symbol for indicating a sequence of welding operations.

This option is visible only for the ANSI standard.





Deletes the current reference line. The current reference line is highlighted in the drawing.

This option is visible only for the ANSI standard.



Selects the next reference line closer to the arrow and displays its settings in the dialog box.

This option is visible only for the ANSI standard.



Selects the reference line further from the arrow, and displays its settings in the dialog box.

This option is visible only for the ANSI standard.

Library

Lists all the welding symbols for the active drafting standard/revision in the symbol library. When you move the pointer over an item, all settings for that item appear in a tooltip. Each item in the library is associated with a context-sensitive menu.

Menu options

Load: Loads the selected symbol from the library. The current settings in the Welding Symbol dialog box are overwritten. Load operations ignore the Flip symbol check box.

Update: Overwrites the selected symbol in the library with the current settings of the Welding Symbol dialog box. The preview updates to reflect the new settings. The name of the symbol and default status do not change.

Delete: Removes the selected symbol from the library.

Rename: Makes the symbol name editable. GstarCAD Mechanical validates the name to ensure that the symbol names are unique.

Set default: Makes the selected symbol the default symbol of the library. A blue check mark appears next to the symbol to indicate that it is the default symbol. The next time you create a welding symbol, the default symbol loads automatically.

Remove default: Removes the default status of the selected symbol. This menu option is available only on the default symbol. When there is no default symbol selected, the welding symbol dialog box defaults to the settings of the welding symbol you created or edited most recently.

Add: Saves current settings to the symbol library as a new item in the library. A prompt enables you to name the new item.

Import: Imports the welding symbols from the symbol library of another drawing. You can only import symbols of the same standard and revision as the welding symbol you are editing.

Requirements

Provides a space for you to specify the welding requirements. The boxes to specify the requirements are named and described in tooltips. Only the named buttons are described below.

Add Process: Displays the Process List dialog box, enabling you to insert processes and process variants within the tail note.

Closed note tail: Draws a box around the tail note.

Clear

Clears all the data and returns the dialog box options to the default settings.

Leader and Text tab

Leader

Arrowhead: Specifies the type of the leader arrowhead. When the arrowhead is set to By Standard, it acquires the arrowhead from the current drafting standard. Thereafter, if you change the arrowhead selected for the current drafting standard (in the Standard Settings dialog box or Welding Settings dialog box) the arrowhead for this symbol updates automatically.

Edit leader segments

Add: Adds a leader, leader segment, or leader node.

Remove: Deletes a leader, or leader segment.

Edit object attachment

Attach: Attaches the symbol to an object in the drawing. When attached, if you move the object the symbol moves with it.

Detach: Detaches the symbol. The symbol becomes a free-standing object.

Text

Text Justification: Sets justification of text. Changing the symbol alignment from the General tab overwrites the text justification setting. Selecting center alignment does not change the setting.

Settings

Opens the Welding Settings dialog box and enables you to edit the default settings for welding symbols for the current drafting standard.

Datum Identifier

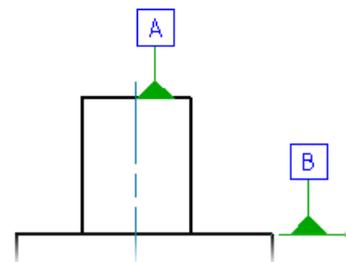
Ribbon: Annotate tab ► Symbol panel ► Datum Identifier

Keyboard command: GMDATUMID

Creates a datum identification symbol and attaches it to an object in the drawing area.

To Create a Datum Identifier Symbol

1. Click Annotate tab ► Symbol panel ► Datum Identifier.
2. In the drawing area, select an object to attach the datum identifier to.
3. If the attached object is a line:
 - Specify the start point for the leader.
4. Specify the next point (vertex) on the leader.
5. Press ENTER to end the selection.
6. In the Datum Identifier dialog box, enter the identifying letters.
7. Click OK.



Datum Identifier Dialog Box

Use this dialog box to specify datum identifier symbol details.

Symbol tab

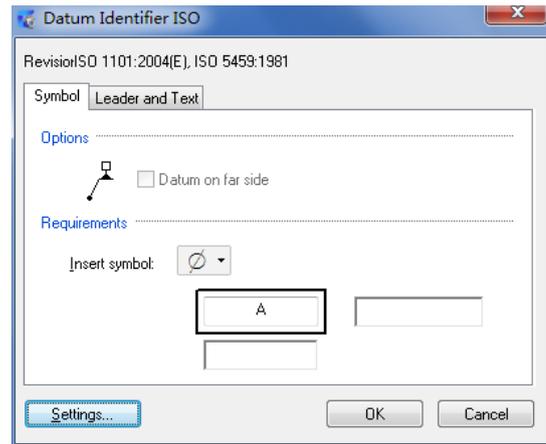
Options

Datum on far side: Changes the line type of surface indication leaders to indicate that the datum is on the far side of a component. The icon to the left of the check box changes to indicate the current state.

Note: This option is visible only in the revisions of drafting standards that permit surface indication lines.

Requirements

Contains buttons and edit boxes to specify the datum. The edit boxes are named only in tooltips. Move the pointer over them to identify them.



Insert symbol: Displays a palette enabling you to insert a special character at the current cursor position in the Datum Note or Thread Note boxes. The preview in the drawing area shows the special character while the dialog box shows the corresponding control key sequence.

Note: This option is visible only in the revisions of drafting standards that permit datum notes or thread notes.

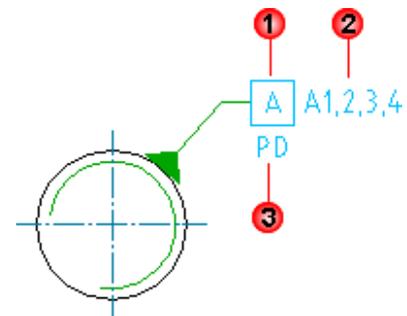
Datum identifier: Specifies a label for the datum identifier (1 in the image above). You can specify a maximum of two characters for the label.

Datum note: Specifies reference datum targets that correspond to one or more points on a surface. Typically this box contains a series of datum targets separated by commas (2 in the image above).

Note: This option is visible only in the revisions of drafting standards that permit datum notes.

Thread note: Specifies which diameter of a screw or gear to use as the datum (3 in the image above).

Note: This option is visible only in the revisions of drafting standards that permit thread notes.



Leader and Text tab

Leader

Arrowhead: Specifies the type of datum feature triangle to use at the end of the leader line. Select an arrowhead from the drop-down menu. The list of arrowheads conforms to the current drafting standard. The default arrowhead is the standard arrowhead for symbols. Set it in the Options dialog box.

Surface arrowhead: Specifies the type of arrowhead to use at the end of the surface indication leader line. When the arrowhead is set to By Standard, it acquires the arrowhead from the current drafting standard. Thereafter, if you change the arrowhead selected for the current drafting standard (in the Datum and Feature Identifier Settings dialog box), the arrowhead for this symbol updates automatically.

Note: This option is available only if a surface indication line exists for the symbol you are editing.

Edit object attachment

Attach: Attaches the symbol to a geometry in the drawing.

Detach: Detaches the symbol. The symbol becomes a free-standing object.

Surface extension line

Controls options for extension lines to the start point of the leader. Extension lines are drawn only if you move the start point of the leader beyond the end of the line or arc it is attached to. The options in this section are available only if extension lines for this symbol exist.

Offset from object: Defines the distance between the surface extension line's start point and the attached object's end point.

Sync: Loads values for the offset from the DIMEXO system variable.

Extension beyond leader: Defines the distance between the surface extension line's end point and the leader start point.

Sync: Loads values for the extension from the DIMEXE system variable.

Settings

Opens the Datum and Feature Identifier Settings dialog box and enables you to edit the default settings for the current drafting standard.

Datum Target Symbol

Ribbon: Annotate tab ► Symbol panel ► Datum Target.

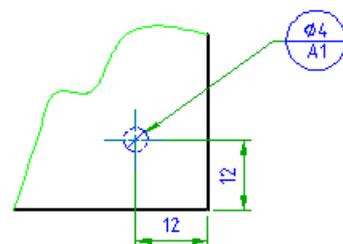
Keyboard command: GMDATUMTGT

Creates a datum target symbol and attaches it to an object in the drawing area.

Commonly Steps

To Create a Datum Target Symbol with No Termination

1. Click Annotate tab ► Symbol panel ► Datum Target .
2. In the Termination Type dialog box, select None.
3. Select an object to attach the datum target to.
4. In the drawing area, specify the start point of the leader.
5. Specify one or more points for the leader vertices, and press Enter.
6. In the Datum Target dialog box, enter the data for the datum target.
7. Click OK.



To Create a Datum Target with a Point Termination

1. Click Annotate tab ► Symbol panel ► Datum Target.
2. In the Termination Type dialog box, select Point.
3. Select an object to attach the datum target to.

4. In the drawing area, specify a point for the point termination.
5. Specify one or more points for the leader vertices, and press ENTER.
6. In the Datum Target dialog box, enter the data for the datum target.
7. Click OK.

To Create a Datum Target with a Line Termination

1. Click Annotate tab ► Symbol panel ► Datum Target.
2. In the Termination Type dialog box, select Line.
3. Select an object to attach the datum target to.
4. In the drawing area, specify a point for the start point of the line termination.
5. In the drawing area, specify a point for the endpoint of the line termination. The program draws a line to indicate that the target region lies along an edge between the two Xs. The leader start point is at the midpoint of the target line.
6. Specify one or more points for the leader vertices, and press ENTER.
7. In the Datum Target dialog box, enter the data for the datum target.
8. Click OK.

To Create a Datum Target with a Circular Termination

1. Click Annotate tab ► Symbol panel ► Datum Target.
2. In the Termination Type dialog box, select Circle.
3. Select an object to attach the datum target to.
4. In the drawing area, specify a point for the center of the circle termination.
5. Specify the radius of the circular termination by entering a value or by dragging the circle to the correct size. You can also enter d for the Diameter option, and enter a value, or select the diameter on the screen.
6. Specify one or more points for the leader vertices, and press ENTER.
7. In the Datum Target dialog box, enter the data for the datum target.
8. Click OK.

To Create a Datum Target with a Rectangular Termination

1. Click Annotate tab ► Symbol panel ► Datum Target.
2. In the Termination Type dialog box, select Rectangle.
3. Select an object to attach the datum target to.
4. In the drawing area, specify a point for the center of the rectangle termination.
5. Specify the corner or edge of the rectangle.

6. Specify one or more points for the leader vertices, and press ENTER. The leader start point is at one of the corners. As you move the pointer, the leader start point moves from one corner to the next. When you are pointing to the corner that you want, select the second point.
7. In the Datum Target dialog box, enter the data for the datum target.
8. Click OK.

To Create a Datum Target Attached to Another Datum Target

1. Click Annotate tab ► Symbol panel ► Datum Target.
2. In the Termination Type dialog box, select Attached to Another.
3. In the drawing area, select a datum target symbol to attach the new symbol to. The pick point controls the position where the program connects the datum target.
4. In the Datum Target dialog box, enter the data for the datum target.
5. Click OK.

Datum Target Dialog box

Symbol tab

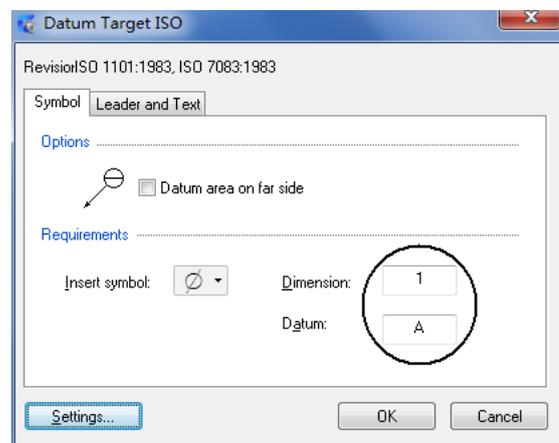
Options

Datum area on far side: Changes the leader linetype to dashed, indicating that the datum target is on the far side.

Requirements

Dimension: Specifies the size of the datum area.

Datum: Specifies a label to identify the datum. You can specify a datum reference letter and a target number of up to 3 digits.



Leader and Text tab

Leader

Arrowhead: Specifies the type of the leader arrowhead. The setting By Standard acquires the arrowhead from the current drafting standard. If you change the arrowhead selected for the current drafting standard in the Standard Settings dialog box or Datum Target Settings dialog box. The arrowhead for this symbol updates automatically.

Edit object attachment

Attach: Attaches the symbol to a geometry in the drawing.

Detach: Detaches the symbol. The symbol becomes a free-standing object.

Settings

Opens the Datum Target Settings dialog box and enables you to edit the default settings for the current drafting standard.

Feature Identifier

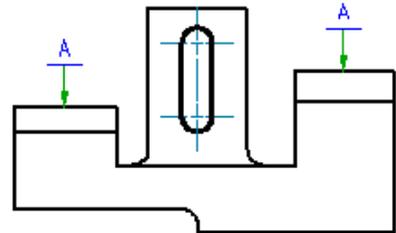
Ribbon: Annotate tab ► Symbol panel ► Feature Identifier

Keyboard command: GMFEATID

Creates a feature identification symbol and attaches it to an object in the drawing area.

To Create a Feature Identifier Symbol

1. Click Annotate tab ► Symbol panel ► Feature Identifier.
2. Select an object to attach the feature identifier to.
3. If you attached the symbol to a line, in the drawing area, specify the start point for the leader.
4. If you attached the object to an arc, circle, ellipse or spline, skip to the next step.
5. Specify one or more points to define the vertices of the leader, and then press ENTER.
6. Click OK.



Taper/Slope Symbol

Ribbon: Annotate tab ► Symbol panel ► Taper and Slope

Keyboard command: GMTAPERSYM

The cone angle and the inclination of symbols to ratio, percentage or code (such as the Morse digital) in the form of cone angle and the inclination of the symbol display. If in addition to choose taper / angle object also selects baseline, then GstarCAD Mechanical will measure inclination / rate of taper and the value is used as the default text. In addition, the baseline is placed in the drawing area, object GstarCAD Mechanical will determine the attachment is taper or inclination, automatically adjust the symbols range, and the symbol of the reference line is aligned with the baseline.

Commonly Steps

To Create a Taper/Slope Symbol, Automatically Calculating the Slope

1. Click Annotate tab ► Symbol panel ► Taper and Slope.
2. In the drawing area, select the baseline the slope must be measured from.
3. In the drawing area, select the object that the symbol must attach to.
4. In the drawing area select a point on the slope/taper.
5. GstarCAD Mechanical 2017 uses this point to calculate the slope. It also uses this point as the start point of the leader. If you want to place the symbol without a leader, the point must be the position you want to place the symbol.
6. In the drawing area, click to specify the vertices of the leader and press ENTER.
7. To place the symbol without a leader, press ENTER first and then click in the drawing area to specify the

orientation of the symbol.

8. If the calculated slope is not in the correct format, select the appropriate format from the Dimension list.

Note: The Dimension box is named only in tooltips.

9. Click OK.

To Create a Taper/Slope Symbol, Manually Specifying the Slope

1. Click Annotate tab ► Symbol panel ► Taper and Slope.
2. Press ENTER.
3. In the drawing area, select the object that the symbol must attach to. If you do not want to attach the symbol to an object, click in a vacant space in the drawing area.
4. Note: If you click in a vacant space, GstarCAD Mechanical uses that position as the start point of the leader.
5. In the drawing area, click to specify the vertices of the leader and press ENTER.
6. To place the symbol without a leader, press ENTER first and then click in the drawing area to specify the orientation of the symbol.
7. In the Dimension box, specify the slope/taper rate.

Tip: To locate the Dimension text box, move the pointer over the boxes in the dialog box and inspect the tooltips.

To insert a special character in the dimension text, place the cursor at the position you want to insert the character. From the Insert symbol list, click the character.

To Change the Type or Orientation of a Taper/Slope Symbol

1. In the drawing area, double-click the taper/slope symbol to edit.
2. In the Taper and Slope Symbol dialog box, click the Symbol Type button.
3. Tip: To locate the Symbol Type button, move the pointer over the buttons in the dialog box and inspect the tooltips.
4. Select the appropriate symbol type of the desired orientation.
5. Click OK.

To Change the Slope/Taper Rate of a Taper/Slope Symbol

1. In the drawing area, double-click the taper/slope symbol to edit.
2. In the Dimension text box, type the taper rate/slope value.

Tip: To locate the Dimension text box, move the pointer over the boxes in the dialog box and inspect the tooltips.

3. To insert a special character in the dimension text, place the cursor at the position you want to insert the character and from the Insert symbol list, and then click the character.
 4. Click OK.
-

Taper and Slope Symbol Dialog Box

Use the Taper and Slope Symbol dialog box to set up the requirements of the symbol you are inserting.

Note: Some buttons on this dialog box are named only in tooltips. Move the pointer over such buttons to obtain their names.

Revision

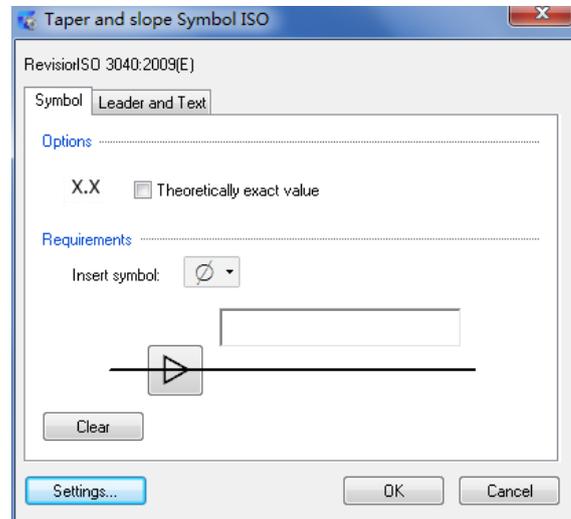
Displays the name and revision of the drafting standard the symbol is based on.

Symbol tab: Contains options to set up the graphical representation of the symbol, orientation, and textual content.

Displays the name and revision of the drafting standard the symbol is based on.

Options

Theoretically exact value: Draws a box around the dimension value to indicate that it is theoretically exact dimension. The preview to the left of the check box changes depending on your selection.



Requirements

Insert symbol: Displays a palette to enable you to insert a special character (for example, the degree symbol) at the current cursor position.

Symbol type: Displays a palette containing various taper and slope symbol representations, which enables you to change the symbol representation and its orientation.

Dimension: Provides a space to specify the slope/taper rate manually. In the drawing area, this text appears next to the symbol. The drop-down arrow displays a list that contains the calculated value for the slope/taper rate expressed in different formats.

Calculated values are available only while creating a symbol and if you selected the calculation option at the command prompt. Calculated values are not available when editing an existing symbol.

Clear

Resets all values within the Options and Requirements section to their default values.

Leader and Text Tab: Sets the leader line options of the symbol.

Leader

Arrowhead: Specifies the type of the leader arrowhead. When the arrowhead is set to By Standard, it acquires the arrowhead from the current drafting standard. Thereafter, if you change the arrowhead selected for the current drafting standard (in the Standard Settings dialog box or Taper and Slope Settings dialog box.) the arrowhead for this symbol updates automatically.

Edit object attachment

Attach: Temporarily hides the dialog box enabling you to select an object to attach the symbol to. This button is visible only if the symbol is not attached to an object.

Detach: Detaches the symbol and makes it a free-standing object. This button is visible only if the symbol is

attached to an object.

Reference line angle

Adjust: Temporarily hides the dialog box so you can change the alignment of the reference line.

Settings

Displays the Taper and Slope Symbol Settings dialog box enabling you to configure the defaults for Taper and Slope symbols.

Edge Symbol

Ribbon: Annotate tab ► Symbols panel ► Edge Symbol

Keyboard command: GMEDGESYM

Creates an edge symbol and attaches it to an object in the drawing area.

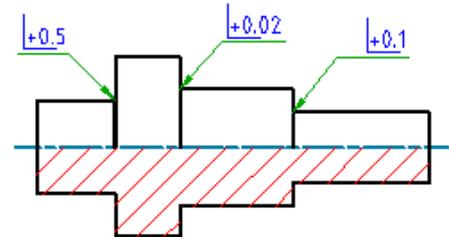
Commonly Steps

To Create an Edge Symbol

1. Click Annotate tab ► Symbols panel ► Edge Symbol.

Note: Only the DIN, GB, JIS, and ISO standards support edge symbols.

2. Select the object you want to attach the symbol to.
3. If you attached the symbol to a line, in the drawing area, specify the start point for the leader.
4. If you attached the object to an arc, circle, ellipse or spline, skip to the next step.
5. Specify the second point of the edge symbol leader and press ENTER.
6. In the Edge Symbol dialog box, specify the edge symbol requirements.



To insert an edge symbol without specific data, select Allow undefined size and click Burr or passing permitted, or Undercut required. To define the direction, select from the drop-down list.

To set the edge symbol data, enter the Edge symbol value in the upper or lower box in the parameters area or select the value from the drop-down list.

To display the standard revision next to the symbol, select Show Standard Label.

7. To add the all-round designation, click the All-round button.
8. Click OK.

To Create a Majority Symbol for Edge Symbols

1. Click Annotate tab ► Symbols panel ► Edge Symbol .

Note: Only the DIN, GB, JIS, and ISO standards support edge symbols.

2. In the drawing area, click an empty area to specify the start point for the leader.

Note: To make a symbol a majority symbol, it must not be attached to anything.

3. Specify the second point of the edge symbol leader and press ENTER.
4. In the Edge Symbol dialog box, specify the edge requirements for the edges that you do not hope to create any edge symbols for.

Select Place as majority.

5. To specify how to display the majority symbol:

Click Settings.

Select the Full List option to show the requirements of edge symbols placed elsewhere in the drawing. Select the Simplified option, to show only the basic symbol.

Click OK.

6. Click OK.

Edge Symbol Dialog box

Use the Edge Symbol dialog box to set the parameters of the symbol you are inserting.

Note

- The visibility of the elements in this dialog box depends on the current drafting standard and the options you select. Therefore some elements described in this page are not visible in the dialog box.
- Some button names are available only in the tooltips. Move the cursor over the buttons to view the names.

Revision

Displays the standard revision that controls the appearance of the symbol and determines what parameters are visible in the dialog box.

Symbol tab

Options

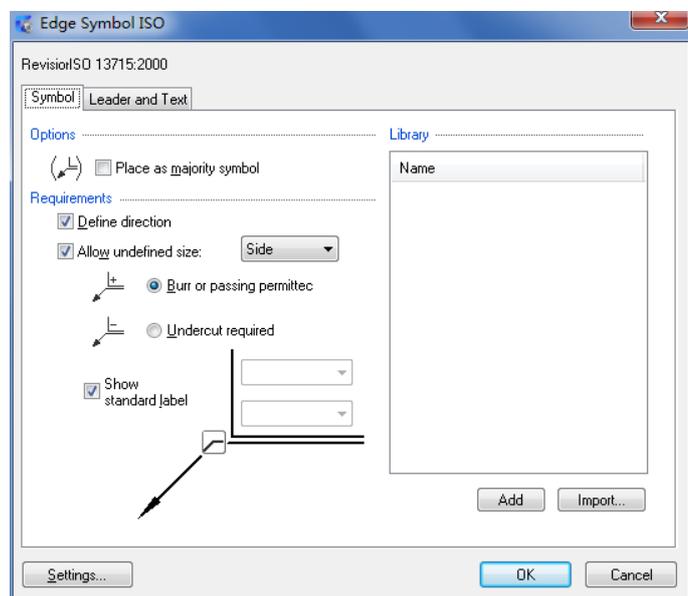
Place as majority symbol: Sets the symbol as a collective indication representing all other edge symbols in the drawing.

Requirements

Sets the options used to specify edge symbol requirements.

Define direction: Turns on the direction of the symbol data or undefined size. When selected, the value entered in the upper box appears at the top of the edge symbol or the value in the lower box appears beside the edge symbol. However, if you enter different values in both boxes, they appear in the middle of the symbol. You can also set the undefined size at top or side of the symbol.

Allow undefined size: Inserts an edge symbol without a specific size. The edge can be burred, passing, or undercut. Use only if you do not specify the exact edge number definition.



Undefined size direction: Specifies whether to insert the undefined size at the top or beside the edge symbol. It is enabled only when both Define direction and Allow undefined size options are selected.

Burr or passing permitted: Sets the edge type as burr or passing.

Undercut required: Sets the edge type as undercut.

Show Standard Label: Displays the current standard revision next to the symbol.

Edge symbol parameters: Sets edge symbol data. Select a recommended value from the drop-down list or enter a value in the box. If the upper boundary is lower than the lower boundary, a warning appears. Click Close to close the warning. The upper boundary and lower boundary are swapped.

All Around: Adds the all-around indication to the symbol. This button is available only if the symbol representation is not a majority symbol.

This button is visible only if the symbol is based on ISO 13715 - 2000. The Revision box shows what standard the symbol is based on.

Library

Lists all edge symbols in the symbol library, for the current drafting standard.

Menu Options

Load: Loads the settings of the selected symbol to the Edge Symbol dialog box. This operation overwrites the current settings of the Edge Symbol dialog box.

Update: Overwrites the selected symbol in the library, with the current settings of the Edge Symbol dialog box. The preview updates to reflect the new settings. The name of the symbol and default status do not change.

Delete: Removes the selected symbol from the library.

Rename: Makes the symbol name editable. AutoCAD Mechanical validates the name to ensure that the symbol names are unique.

Set default: Makes the selected symbol the default symbol of the library. A blue check mark appears next to the symbol to indicate that it is the default symbol. The next time you create an edge symbol, the default symbol loads automatically.

Remove default: Removes the default status of the selected symbol. This menu option is available only on the default symbol. When there is no default symbol selected, the settings default to the settings of the edge symbol you created or edited most recently.

Add: Saves current settings to the symbol library as a new item in the library. A prompt enables you to name the new item.

Import: Imports the edge symbol symbols from the symbol library of another drawing. You can only import symbols of the same standard and revision as the edge symbol you are editing.

Leader and Text tab

Leader

Arrowhead: Specifies the leader arrowhead type. When the arrowhead is set to By Standard, the arrowhead is acquired from the current drafting standard. From then on, if you change the arrowhead selected for the current

drafting standard (in the Standard Settings dialog box or Edge Settings dialog box), the arrowhead for this symbol is automatically updated.

Edit leader segments

Add: Adds a leader, leader segment, or leader node.

Remove: Deletes a leader or leader segment.

Edit object attachment

Attach: Attaches the symbol to an object in the drawing. When attached, if you move the object the symbol moves with it.

Detach: Detaches the symbol. The symbol becomes a free-standing object.

Settings

Opens the Edge Symbol Settings dialog box and enables you to edit the default settings for edge symbols for the current drafting standard.

Center Mark

Ribbon: Annotate tab ► Symbol panel drop-down ► Center Mark

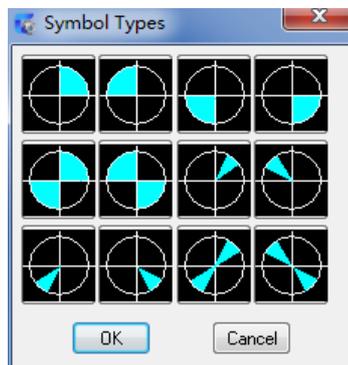
Keyboard command: GMCIRCLEMARK

Create Center Mark.

To create a Circle mark

1. Display "hole marked" dialog box, select the label form.
2. Select the target hole mark complete.

Circle mark Dialog box



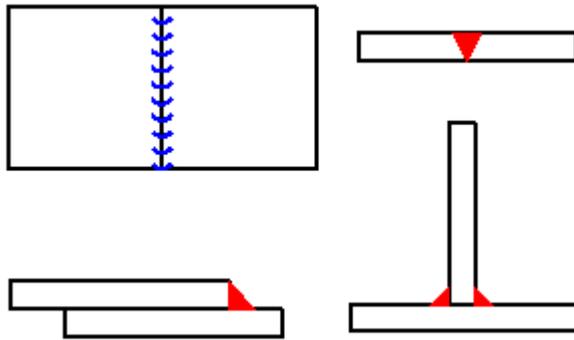
Symbol Types

Weld Representation

Ribbon: Annotate tab ► Symbol panel drop-down ► Weld Representation

Keyboard command: GMSIMPLEWELD

Draws seam and fillet welds on ellipses, circles, arcs, lines and polylines.



List of Prompts

Leg width: Specifies the width of the seam.

Select line, arc, circle, or ellipse: Specifies the taper or slope to attach the symbol to.

Start point: Specifies the starting point of the seam.

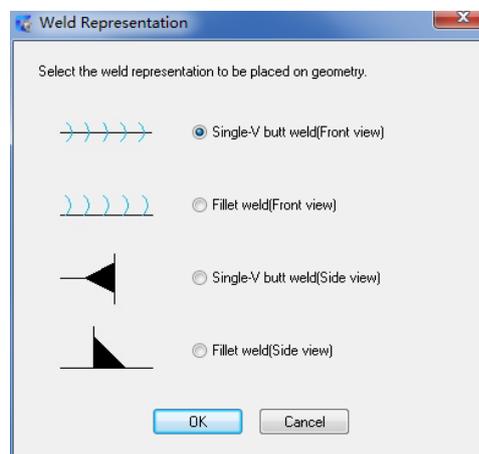
End point: Specifies the end point of the seam.

Correct side: Specifies if fillet welds are placed on the correct side of the seam. If you specify No, the program moves the welds to the other side.

Angle: Specifies the angle of the weld.

Weld Representation Dialog box

The specified welding representation style different view



Edit Symbol

Add/Remove Leader

Ribbon: Annotate tab ► Symbol panel drop-down ► Add Leader

Keyboard command: GMSYMLEADER

Adds leaders and leader segments to a symbol. Removes leaders and leader segments to a symbol.

List of Prompts

Select symbol to add leader to: Specifies the symbol to add the leader to.

Select object to attach: Specifies the object to attach the leader to. To place the leader without attaching it to anything, click the position you want to place the arrowhead of the leader. If you select an arc, circle, spline, or ellipse, you do not need to specify the leader start point. The program derives the leader start point by making the first leader segment perpendicular to the attached object. If you select a line, or do not select an object, the program prompts you to specify the leader start point.

Note: The prompts that are displayed from this point onwards depend on the type of object you select.

Start point: Specifies the position of the leader start point.

Next point: Specifies the location of the next vertex of the leader.

Attach/Detach Leader

Ribbon: Annotate tab ► Symbol panel drop-down ► Attach/Detach Leader

Keyboard command: GMSYMLEADER

Enables you to select an object to attach the symbol to. Detaches the symbol and makes it a free-standing object.

List of Prompts

Select object to attach

Select leader node or leader segment

Chapter 7 BOM Data

Summary

In GstarCAD Mechanical 2017, You can use the BOM to generate parts lists and balloons.

The BOM is a table that contains information about the parts and assemblies within a drawing. The program uses this information to generate parts lists and balloons.

When you select a drafting standard, the default settings for parts lists and balloons change to match that standard. The BOM settings change to support the change in parts list and balloon settings. You can customize these settings to suit your drafting requirements.

Configuring the BOM, parts lists, and balloons are interlinked operations. Customize the component properties, including:

- How properties are displayed in the BOM.
- The parts list layout
- Balloons
- What information the program captures from the parts and assemblies in the drawing, to be displayed in the BOM
- We recommend customizing BOM Configuration before you start drawing. If you change the BOM configuration after you create a BOM, the BOM can include inconsistent data.

Create Part References

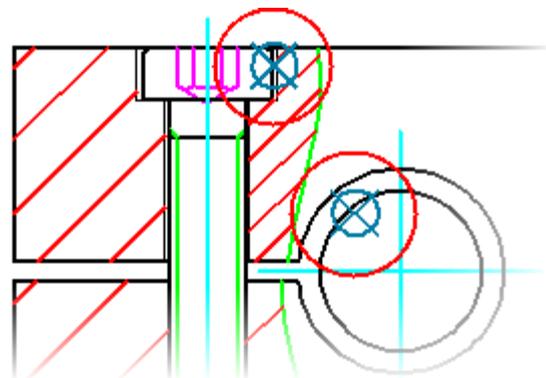
Ribbon: Annotate tab ➤ BOM panel ➤ Part Reference drop-down ➤ Create

Keyboard command: GMPARTREF

Annotate tab BOM panel Part Reference drop-down Create.

Creates a part reference and places it in the drawing area.

You use Part References (GMPARTREF command) to tag geometry as parts. The BOM counts the Part References and lists them in the BOM dialog box as parts. When you create balloons for your drawing, you can appoint balloon leader to a part reference, enabling you to correlate the geometries with parts lists quickly. Part references enable you to specify the component properties of each part. The BOM automatically picks up this information and lists the properties in the BOM dialog box as well as parts lists.



To Create a Part Reference

1. Click Annotate tab ► BOM panel ► Part Reference drop-down ► Create.
2. In the drawing area, click a geometry (or a blank space in the drawing area) to place the part reference.
3. In the Part Reference dialog box, enter to the data to store in the BOM.
4. Click OK.

List of Prompts

The following prompts are displayed.

Point: Inserts a part reference at the point you specify and displays the Part Reference dialog box. If you place the part reference on an object, the part reference attaches itself to that object.

- **Start point** - Specifies the start point of the balloon leader.
- **Next point** - Specifies the bend points that define leader segments. Press ENTER to place the balloon in the drawing area.

Block: Creates a part reference for a block reference. If the block has an attribute that has the same name as a component property, the part reference automatically imports its value into the component property.

Copy: Creates a complete copy of a part reference, including its data. The new part reference becomes a new BOM item.

Select reference - Specifies the part reference to copy.

Reference: Creates a part reference that marks an object as another instance of a part that has already been tagged with a part reference. In the BOM, the new part reference increments the quantity of the original part reference instead of adding another row.

Select reference - Specifies the part reference you want to add another instance of.

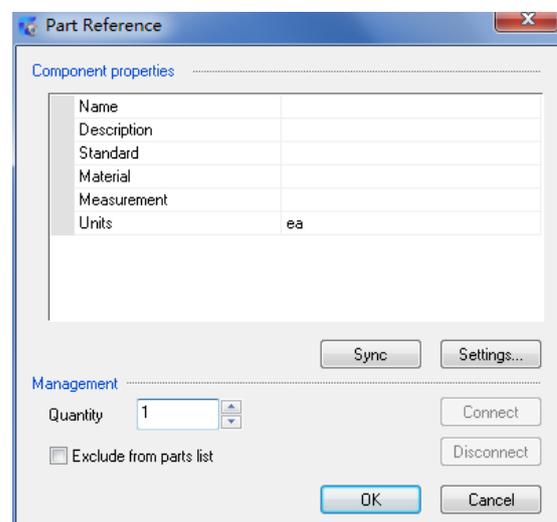
Select point - Specifies the point to place the part reference.

Part Reference Dialog Box

Component Properties

Provides the ability to enter data for component properties. Only the component properties pre-configured for display in this dialog box appear.

Note: The Name component property is set to a formula, by default. This formula enables GstarCAD Mechanical to pick up the name of objects such as blocks, automatically. However, if you specify a name manually (even when the formula does not pick up a name automatically), GstarCAD Mechanical considers it an override and highlights it in blue.



Settings: Displays the BOM Settings dialog box, where you can set what component properties are visible in this dialog box.

Management

Quantity: Provides the option to override the default quantity of 1, with any quantity, as required.

Exclude from Parts List

Excludes the component from all parts lists. The component remains in the BOM but is excluded from parts lists. You are prevented from creating balloons for this component as well.

Edit part reference

Ribbon: Annotate tab ► BOM panel ► Part Reference drop-down ► Edit

Keyboard command: GMPARTREFEDIT

Edits a part reference. You can edit the component properties of a part or edit the assembly properties of a drawing.

To Modify the Component Properties of a Part Reference

1. Click Annotate tab ► BOM panel ► Part Reference drop-down ► Edit.
 2. Click the part reference to edit. The Part Reference dialog box is displayed.
 3. In the Component Properties section, change anything.
 4. Click OK.
-

List of Prompts

Select object: Displays the Part Reference dialog box, enabling you to edit BOM-related information for the part or assembly that is identified with the part reference you select.

Balloon

Create Balloon

Ribbon: Annotate tab ► Balloon panel ► Balloons

Keyboard command: GMBALLOON

Creates and places balloons in the drawing area.

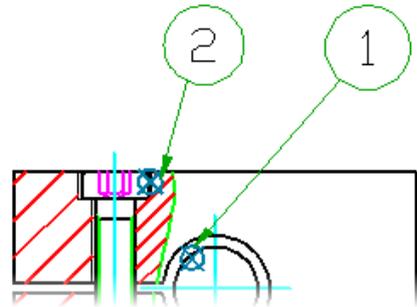
You can create a single balloon or create multiple balloons simultaneously. For multiple balloons, you have the option to align the balloons horizontally, vertically, or in any straight line. You can also distribute balloons around a rectangular area.

By default, the balloon leaders start at the center of a Part Reference or are attached to the component (part or assembly). If necessary, you can create balloons with leader start points at positions other than on the Part Reference or

component.

To Create a Balloon

1. Click Annotate tab ► Balloon panel ► Balloons.
2. In the drawing area, select a part reference or component (part or assembly).
3. In the drawing area, click to indicate the second point for the balloon leader, and continue until the balloon is in the location you want.
4. Press ENTER to place the balloon.
5. Press ENTER to end the procedure.



List of Prompts

[auTo/autoAll/set Bom/Collect/arrow Inset/Manual/One/Renumber/rEorganize/annotation View]:

Auto: Creates balloons for the selected part references. This option lets you select only the part references that have not been ballooned yet. If you select only one part reference, GstarCAD Mechanical automatically jumps to the “One” command line option.

Auto All: Creates balloons for all selected part references, including those that already have balloons attached to them. If you select only one part reference, GstarCAD Mechanical automatically jumps to the “One” command line option.

Set BOM: Specifies the BOM to set as the current BOM.

To select or create the main BOM, enter M. To select or create a Border BOM, click the corresponding border in model space. To see a list of existing BOMs, enter ?. To create or set the current database to one of the listed BOM databases, type the name of the BOM and press Enter.

Collect: Creates collected balloons for the selected part references.

Manual: Creates a part reference with a balloon. This option forcibly changes the current BOM to the border BOM that corresponds to the border that the part reference is placed in. If you want the balloon to be based on the main BOM, delete the balloon, set the current BOM to MAIN and balloon the part reference again.

Point: Inserts a part reference at the point you specify and displays the Part Reference dialog box. If you place the part reference on an object, the part reference attaches itself to that object.

Start point - Specifies the start point of the balloon leader.

Next point - Specifies the bend points that define leader segments. Press ENTER to place the balloon in the drawing area.

Block: Creates a part reference for a block reference. If the block has an attribute that has the same name as a component property, the part reference automatically imports its value into the component property.

Copy: Creates a complete copy of a part reference, including its data. The new part reference becomes a new BOM item.

Select reference - Specifies the part reference to copy.

Reference: Creates a part reference that marks an object as another instance of a part that has already been tagged with a part reference. In the BOM, the new part reference increments the quantity of the original part reference instead of adding another row.

Select reference - Specifies the part reference you want to add another instance of.

Select point - Specifies the point to place the part reference.

One: Places a single balloon in the drawing area. Unlike with the Auto and Auto all options, you must specify the start point for the balloon leader manually.

Renumber: Renumbers balloons and changes the item numbers of the selected parts in the current BOM.

Starting item number: Specifies the number for the first selected balloon.

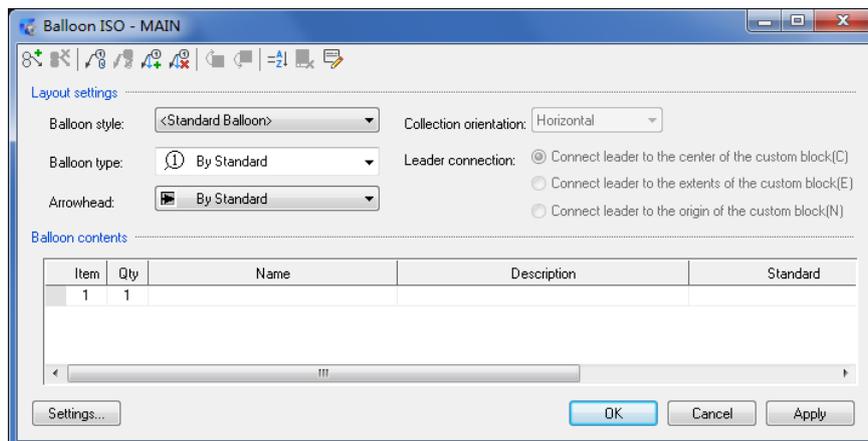
Increment: Specifies the increment value for the next selected balloons.

Balloon: Specifies the balloons to renumber, in sequence. Press ENTER to end the selection.

Reorganize: Lines up the balloons horizontally, vertically, or in a specified angle. You can also convert the balloons to standalone balloons (balloons without leaders).

Balloon Dialog Box

Use this dialog box to manage the appearance of the balloon or balloon collection you selected for editing.



Toolbar buttons



Collects other balloons on the leader of the balloon you are editing. The dialog box temporarily hides to enable you select balloons to collect. Collected balloons are added to the Balloon contents list.



Deletes the balloon selected in the Balloon contents list. This button is available only if there is more than one balloon in the Balloon contents list.



Detaches the balloon from the object it is attached to. This button is visible only if the balloon is attached to an object.



Attaches the balloon to an object in the drawing area. The dialog box temporarily hides and all objects that you can

be attached to are highlighted in the drawing area. This button is visible only when the balloon is not attached to anything.



Adds a leader or leader segment to the balloon. The dialog box temporarily hides, enabling you to select the attach point. To add a new leader, the last selected point must be on the original balloon leader. To add new leader segments, the last selected point must be the start point of the original leader.



Deletes a leader or leader segment. The dialog box temporarily hides, enabling you to select the leader to delete. If you select an internal leader segment, this operation deletes all its successors. If you remove all leader segments, the balloon converts to a stand-alone balloon.



Displays the Sort dialog box, which provides various options to sort the balloons displayed in the Balloon contents list.



Removes overrides from the current selection and restores their default values. This button is available only when overrides exist within the current selection.



Sets the values for a range of cells in a column. Changes you make are written to the corresponding BOM.

Layout settings

Balloon style: Enables you to select between standard-based balloons or a block that can be used as a custom balloon. If the block is in an external file, click the Browse option at the bottom of the list.

Balloon type: Displays a panel from which you can select a standard compliant shape for the balloon. This option is available only for standard-based balloons.

Arrowhead: Displays a palette containing standard-compliant arrows for leader termination.

Collection orientation: Defines how balloon collections line up. This option is available only if you have collected more than one balloon on the leader of the balloon being edited.

Leader Connection: Defines where on a custom balloon the leader connects to it. This option is not available if the current balloon style is a standard based balloon.

Balloon contents

Displays the properties of the ballooned item. The columns are the same as the columns in the BOM.

Settings

Displays the Balloon Settings dialog box, enabling you to change the default settings for balloons.

Organizing Balloons

To Renumber Balloons

1. Click Annotate tab ► Balloon panel ► Balloons.
2. At the Command prompt, enter R to select the Renumber option.

3. Enter the lowest item number for the balloons you plan to select.
4. Enter the increment for the subsequent balloons.
5. Select the balloons to renumber one by one. Select the balloon with the lowest item number first, followed by the balloon with the next highest number, and so on. The program renumbers the balloons and they remain highlighted.
6. Press Enter to end the renumbering.

To Collect Balloons

1. In the drawing area, double-click the balloon to collect other balloons to. The Balloon dialog box is displayed.
2. In the toolbar of the Balloon dialog box, click .
3. In the drawing area, select the balloons to collect, and press Enter to end the selection.
4. Pick the orientation of the collected balloon.
5. Click OK.

Parts List and Bills of Materials

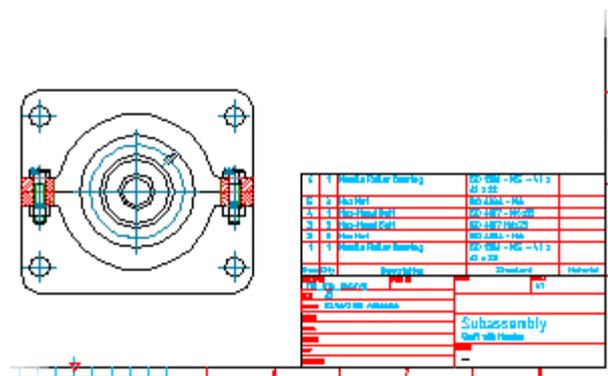
Parts List

Ribbon: Annotate tab ► Sheet panel ► Parts List.

Keyboard command: GMPARTLIST

Creates a parts list and enables you to place it in the drawing area.

To create a parts list that contains all the parts and assemblies in the drawing, the main BOM must be set as the current BOM. To create a parts list that only contains the parts and assemblies within a particular drawing border in model space, the corresponding Border BOM must be set as the current BOM.



Commonly Steps

To Create a Parts List for the Entire Drawing

1. Click Annotate tab ► Sheet panel ► Parts List.
2. At the Command prompt, enter MAIN.

3. If necessary, modify the layout of the parts list.
4. If necessary, modify how data appears in the parts list.
5. Click OK. The dialog box closes and a preview of the parts list appears in the drawing area.

To Create a Parts List for a Drawing Border

1. In the drawing area, click to indicate where to place the parts list. The program inserts the parts list into the drawing.
2. Click the drawing border that contains the parts and assemblies to include in the parts list. The Parts List dialog box is displayed.
3. If necessary, modify how data appears in the parts list (Click here for more information).
4. Click OK. The dialog box closes and a preview of the parts list appears in the drawing area.
5. In the drawing area, click to indicate where to place the parts list. The program inserts the parts list into the drawing.

List of Prompts

The following prompts are displayed.

Specify BOM

Specifies the BOM to set as the current BOM.

To select or create the main BOM, enter M. To select or create a Border BOM, click the corresponding border in model space. To see a list of existing BOMs, enter?. To create or set the current database to one of the listed BOM databases, type the name of the BOM and press Enter.

Parts List Dialog Box

Use this dialog box to edit the layout of the parts list and edit parts list data. **Toolbar buttons**

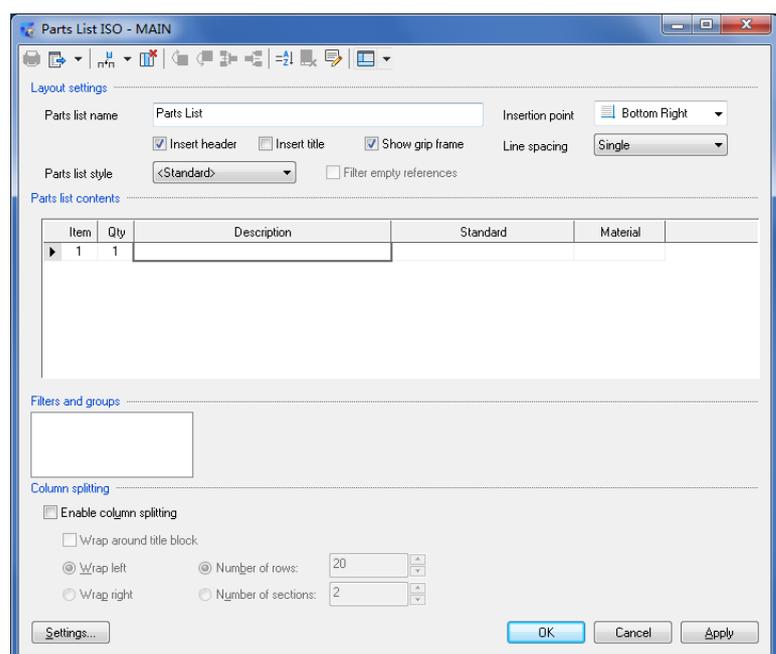
 Exports a parts list as a spreadsheet or database.

 Inserts a column to the left of the selected position.

Insert column drop-down

Insert column left-Inserts a column to the left of the current selection.

Insert column right-Inserts a column to the right of the current selection.





Deletes the selected column.



Merges two or more selected rows. Only rows that have identical properties can be merged.



Splits a row that has a quantity greater than 1 into two separate rows. When you click this button, a prompt enables you to select the part references to move to a new row. You typically use this option to split rows that you merged previously.



Displays the Sort dialog box, which provides various options to sort the items in the parts list.



Removes overrides from the current selection and restores their default values. This button is available only when overrides exist within the current selection.



Sets the values for a range of cells in a column and updates the corresponding BOM.

Show/Hide drop-down

Formula bar: Displays an area below the toolbar (the formula bar), to enable you to enter and edit formulas. A check mark next to the menu item indicates that the formula bar is displayed.

Results bar: Displays a row at the bottom of the list of parts (the results bar) to show the totals of the properties configured to be summed. A check mark next to the menu item indicates that the results bar is displayed.

Layout Settings

Parts list name: Provides a space for you to enter the name of the parts list. The name appears on the title bar of the parts list.

Insertion point: Sets the insertion point placement for the parts list and where the title of the parts list appears. If the insertion point is at the bottom, the header and title appear at the bottom of the parts list and the parts are listed from bottom to top. Similarly, if the insertion point is at the top, the header and title appear at the top of the parts list and the parts are listed from top to bottom.

Insert header: Controls whether a header row appears in the parts lists. The header row displays the column headers of the parts lists columns.

Insert title: Controls whether parts lists have title bars. The title bar shows the parts list name.

Show grip frame: Controls whether the grip frame of the parts list appears. The grip frame is a frame that shows the extent of the parts list, which, under certain conditions, is larger than the visible extent of the parts list. The grip frame acts as a visual aid for column splitting by dragging. This method of column splitting has been superseded by the options in the Column Splitting section of this dialog box. The Show grip frame option is included only for backward compatibility.

Parts list style: Switches the parts list style between a standard based parts list and custom parts list. The list shows all blocks that can be used as custom parts lists. If the blocks to use are in an external drawing, select Browse (last entry in the list).

Filter empty references: Controls whether body blocks of a custom parts list are hidden if it contains no data. This

option is available only if the parts list style is a custom parts list.

List of parts and assemblies

Displays a list of BOM items shown in the parts list. The columns correspond to the component properties pre-configured for display in parts lists.

Filters and Groups

Filters/Groups list: Displays the parts list filters. Right-click in the window to add, or remove a filter, or to group parts list rows.

Details for filter: Displays a description of the selected filter, or provides options to set up a custom filter.

Options for setting custom filter criteria: Displays two drop-down lists and a text box to set the filter criteria. The first drop-down list enables you to select the column to filter by. The second drop-down list enables you to select the filter condition, and in the third box, the information to filter on. These options are visible only when you select a custom filter in the Filters/Groups list.

Group by: Displays a series of drop-down lists enabling you to select the columns to group by.

Column splitting

Includes options to configure whether to perform column splitting is performed and, if so, how. Column splitting refers to the ability for the entire parts list to wrap as the number of data rows grows.

Enable column splitting

Turns on column splitting.

Wrap around title block-Wraps the parts list around the title block as in the following image. Specify the title block before inserting the part list.

Wrap left Wraps-the parts list to the left as in the following image.

Wrap right Wraps-the parts list to the right as in the following image.

Number of rows-Performs split after the number of rows exceeds the number you specify in the adjacent box.

Number of sections-Splits the parts list into the specified number of sections and distributes the entries evenly.

Settings: Displays the Parts List Settings dialog box, enabling you to change the default parts list settings.

Bills of Materials

Ribbon: Annotate tab ► Bom panel ► Bom.

Keyboard command: GMBOM

Creates, edits or deletes Bills of Materials (BOMs). The Bill of Materials (BOM) is a table that contains information about the parts in a drawing.

You can use the BOM to generate parts lists and balloons. A drawing file can have several BOMs. The “main” BOM contains information about all the parts and assemblies in a drawing. A parts list generated from the main BOM contains all the parts and assemblies in that drawing. You can also create a “border” BOM, which contains only the parts and assemblies within a drawing border. Parts lists generated from a border BOM do not contain parts from elsewhere in the drawing.

Commonly Steps

To Create a BOM for the Entire Drawing

1. Click Annotate tab ➤ BOM panel ➤ BOM.
2. At the Command prompt, enter M.

To Create a BOM for a Drawing Border in Model Space

1. Click Annotate tab ➤ BOM panel ➤ BOM.
2. In model space, click the drawing border for which you want to create the BOM. The program creates the BOM and shows it in the BOM dialog box.

List of Prompts

The following prompts are displayed.

Specify BOM: Specifies the BOM to create or set as the current BOM. To select or create the main BOM, enter M. To select or create a Border BOM, click the corresponding border in model space. To see a list of existing BOMs, enter? To set the current database to one of the listed BOMs, type the name of the BOM and press ENTER.

BOM Dialog Box

Use this dialog box to manage BOMs and BOM data in the drawing file.

BOM List panel



Sets the selected BOM as the current BOM and displays its contents within the BOM panel.



Exports the current BOM as a spreadsheet or database.



Deletes the BOM selected in the list of BOMs.

List of BOMs

Displays a list of BOMs that exist in the current drawing.



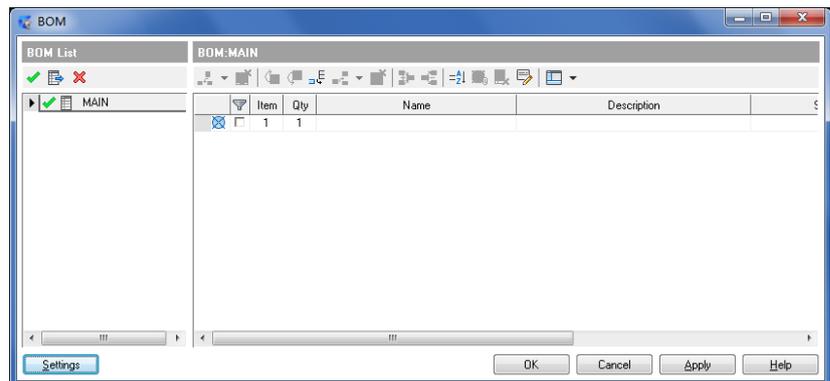
Displays the icon to indicate the current BOM.

BOM Type

Indicates the BOM type using an icon.



Border BOM





Main BOM

Right-click Menu Items

Set Current: Sets the selected BOM as the current BOM and displays its contents within the BOM panel.

Zoom to: Temporarily hides the BOM dialog box and zooms to the geometry associated with the BOM. If the BOM is a Border BOM, this menu option zooms to the corresponding border. If the BOM is an Assembly BOM, it zooms to the corresponding component.

Highlight: Temporarily hides the BOM dialog box and highlights to the geometry associated with the BOM. If the BOM is a Border BOM, this menu option highlights to the corresponding border. If the BOM is an Assembly BOM, it highlights all the views of the corresponding component.

Rename: Places the cursor within the name of the BOM, enabling you to rename it.

Delete: Deletes the selected BOM.

BOM Panel



Inserts a column to the left of the current selection in the list of BOM items.

Insert column drop-down

Insert column left-Inserts a column to the left of the current selection in the list of BOM items. **Insert column right-**Inserts a column to the right of the current selection in the list of BOM items.



Deletes the selected column.



Adds a row to the bottom of the BOM. Rows inserted in this manner are user-defined items and do not represent any geometry in the drawing area.



Inserts a row above the current selection. Rows inserted in this manner are user-defined items and do not represent any geometry in the drawing area.

Insert item drop-down

Insert item above-Inserts a row above the current selection. Rows inserted in this manner are user-defined items and do not represent any geometry in the drawing area.

Insert item below-Inserts a row below the current selection. Rows inserted in this manner are user-defined items and do not represent any geometry in the drawing area.



Deletes the selected row. If there are any rows nested below this row, they are deleted as well.



Merges two or more selected rows. Only rows that have identical properties can be merged. You typically merge rows when the system shows the same part as two separate entries in the BOM.



Splits a row that has a quantity greater than 1 into two separate rows. This operation is not applicable to mechanical

structure components. When you click this button, a prompt enables you to select the part references to move to a new row.

You typically use this option to split rows that you merged previously.



Displays the Sort dialog box, which provides various options to sort the items in the BOM.



Removes overrides from the current selection and restores their default values. This button is available only when overrides exist within the current selection.



Sets the values for a range of cells in a column.

List of BOM items

Displays a list of BOM items.

If you override the default value of a cell, by specifying a value manually, GstarCAD Mechanical highlights that cell in blue. When you move the mouse over an overridden cell, a tooltip shows the default value of the cell.

You can override the QTY component property and any other component property that uses a formula to display their values. You can override any component property of a top-level xref item. BOMs attached to Part References are also considered xref items.

You can remove overrides and restore the default value of a cell by right-clicking it and selecting Remove overrides.



Part Reference



Block Part Reference (created using the Block option of the GMPARTREF command)

Right-click menu items

Insert Item

- Insert item above-Inserts a row above the current selection. Rows inserted in this manner are user-defined items and do not represent any geometry in the drawing area.
- Insert item below-Inserts a row below the current selection. Rows inserted in this manner are user-defined items and do not represent any geometry in the drawing area.

Delete item

Deletes the selected row. If there are any rows nested below this row, they are deleted as well.

Merge items

Merges two or more selected rows. Only rows that have identical properties can be merged.

Rows are typically merged when the system shows the same part as two separate entries in the BOM

Split item

Splits a row that has a quantity greater than 1 into two separate rows. This operation is not applicable to mechanical structure components. When you click this button, a prompt enables you to select the part references to move to a new row.

Typically, this option is used to split rows that were merged previously.

Insert column

- **Left**-Inserts a column to the left of the current selection.
- **Right**-Inserts a column to the right of the current selection.

Delete

Deletes the selected column.

Set values

Sets the values for a range of cells in a column.

Settings

Displays the BOM settings dialog box, enabling you to configure component properties, among others.

Chapter 8 Standard Parts Library

Summary

GstarCAD Mechanical 2017 standard parts library, it'll be free from heavy and complicated work for designers, make drawing more quickly, more accurate, more standard.

Development system of Serialized Parts Design

Ribbon: Library tab ► Library panel ► All Parts

Keyboard command: GM_SPART_OUT

According to requirements of different professions in manufacturing industry, GstarCAD Mechanical supplies serials of national standard parts libraries and they are open to manage, maintain and customize. Users could easily create, retrieve, modify and expand the company's own serialization standard graphic library.

To Draw a Standard Part

1. Click Library tab ► Library panel ► All Parts
2. Shows "Development system of Serialized Parts Design" dialog box.
3. Select the parts type in the directory tree of parts.
4. Select parts parameters values in the parameters lists.
5. Select view options and export options content.
6. Click "Export" button.
7. Specify position and angle of standard part in the drawing, standard part drawing is complete.

To Edit a Standard Parts

1. Double click the drawn standard part to edit
2. Shows "Development system of Serialized Parts Design " dialog box, and locate the standard in the parts directory tree.
3. Select the "corresponding options" or "other parameters", and click OK.

Note: after standard editing, parts position and direction in the drawings will not change.

Development system of Serialized Parts Design Dialog Box

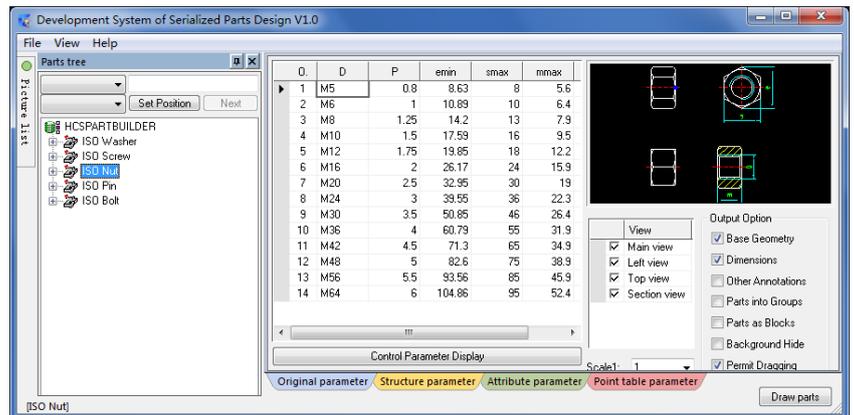
Parts tree

Parts directory: Choose all the parts or some parts, parts category of parts library.

Query input box: Input box for querying current parts directory content.

Set Position: Set the position of current query content in the parts directory.

Next: Locate the next position of current query content in the parts directory.



Parameter list

Original parameter: Select original parameter value of series parts.

Structure parameter: Choose parts parameter data list a part contains options parameter data, including the original parameters, label content drawing ratio, etc.

Property parameter: Parameter data list a part contains property data content, including dimension information, code, name, etc.

Point table parameter: Select parameter data list a parts after calculated coordinate values, coordinates formula content.

Draw parts: Draw selected part in the drawing.

Output options

Basic entity: In the drawing draw parts as entity.

Dimensions: In the drawing draw parts dimension.

Other annotation: In the drawing draw parts other annotation.

Parts as group: In the drawing draw parts as a group.

Parts as block: In the drawing draw parts as a block.

Background Hide: It creates hide situation automatically between parts.

Permit to drag: In the engineering graphics rendering parts if you can point in any zone other than drawing, otherwise only permitted in basis points as the insertion point rendering.

Scale: In the engineering drawings rendering parts and components cannot automatically adapt to the proportion, and dimension content of parts change proportionally.

Chapter 9 Commonly Used Tool

Summary

GstarCAD Mechanical 2017 provides a set of common tools, including text, table, and computing, it covers all aspects of daily drawing. The commonly used tool will help designers to complete the drawing quickly, reduce the burden of the designer.

Text annotation

Ribbon: Annotate tab ► Text panel ► Multiline Text drop-down ► Text annotation.

Keyboard command: GMTEXT

The Text Annotation feature allows user to use some special text processing methods such as text rotation, width factor setting and text obliquity setting, which provide a useful complement to the text processing methods in GSTARCAD.

To Create Text Annotation

1. Click Annotate tab ► Text panel ► Multiline Text drop-down ► Text annotation.
2. The 'Text Annotation' dialog box is displayed, where data can be entered
3. Select the location of dimension

List of Prompts

Please select start point for text

Select the location for Text Annotation in the drawing

Text Annotation dialog box

Text Contents: Enter the required content of Text Annotation

Text Height: The list of Text Annotation height setting available for selection, user may as we manually input a value.

Rotation Angle: Setting of Text Annotation rotation angle

Width factor: Setting of Text Annotation width factor

Rotation Angle: Setting of Text Annotation obliquity

Horizontal Alignment: List of Text Annotation horizontal alignment

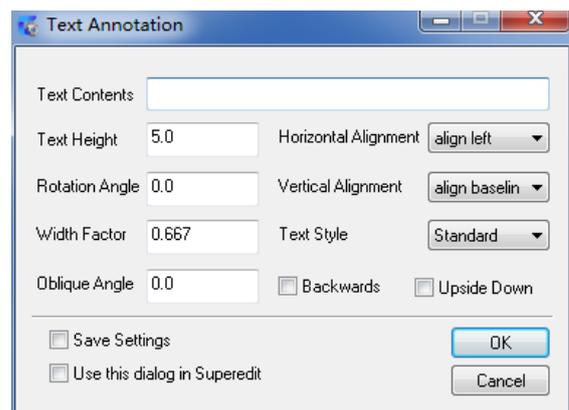
Vertical Alignment: List of Text Annotation vertical alignment

Text Style: List of Text Annotation styles

Backwards: Back-to-Front setting of Text Annotation

Upside Down: Upside down setting for multi-line Text Annotation

Save settings: Save various Text Annotation settings, which will be maintained for subsequent Text Annotation.



Technical requirements

Menu: Text ► Technical requirements

Keyboard command: GMTECHREQUEST

Technical requirement is indispensable for any drawing, and similar type of parts normally involve similar technical requirements. This allows users to edit or modify existing technical requirement, and make text rotation, width factor setting as well as text obliquity setting without changing font type. Users may also extract technical requirement from common technical requirement document, or save common technical requirements into a technical requirement document.

To creat technical requirement

1. Click Text menu ► Technical requirements
2. The 'Technical requirements' dialog box is displayed, where data can be entered, click 'Generate' to complete adding technical requirement
3. Specify the drawing area for technical requirement

List of Prompts

Specify first point of technical requirement block: Select the upper left corner of technical requirement area in a drawing

Specify second point of technical requirement block: Select the lower right corner of technical requirement area in a drawing

Technical requirements dialog box

Title: The title of technical requirement

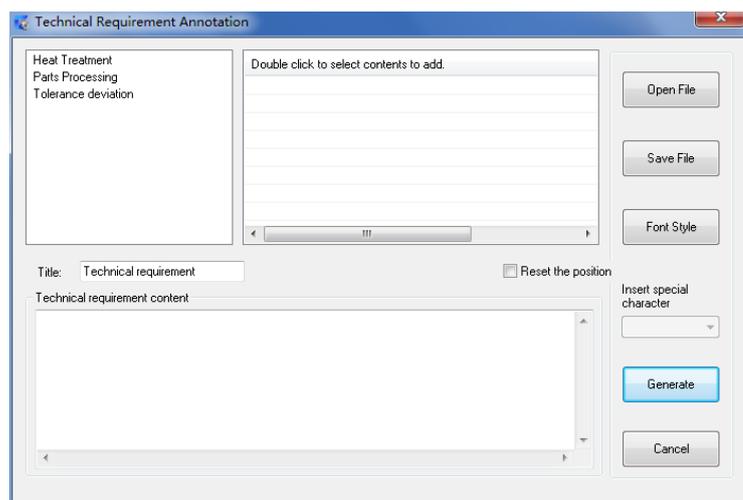
Reset the position: User may re-specify the location for technical requirement after the location is selected

Technical library: Repeatedly invoke the content of technical requirement from the word library

Open File: Read content from an existing technical requirement document

Save file: Save user-defined technical requirement into a technical requirement document

Font Style: Perform font style for technical requirement.



Form tool

Hole Chart

Ribbon: Annotate tab ► Table panel ► Hole Chart.

Keyboard command: GMHOLECHAR

A Hole Chart is a table that lists the position and dimensions of selected holes (any circle or standard hole) in a drawing. The GMHOLECHART command creates hole charts, when you select the holes and the origin from where the position is defined. When you create a hole chart, a hole label is placed next to each hole in order to uniquely identify each hole in the hole chart. If holes are deleted, the hole chart updates automatically. However, if you draw new holes, the hole chart does not update automatically. You must explicitly add the hole to the hole chart. The format of hole labels and hole charts is controlled by the drafting standard. Configure the drafting standard settings for hole charts before you create hole charts.

Hole Tables:

The ANSI standard defines a table referred to as a hole table, in addition to a hole chart. Hole tables assigns a label for each unique size of the holes selected for the hole chart. It lists the label, the size and the number of holes of that size in the hole chart. When you create a hole chart, you are prompted to place the hole table in the drawing area as well.

Nested Hole Charts:

You can also create "nested hole charts". A nested hole chart is a consolidated table of holes that belong to more than one origin.

Hole Labels"

A hole label uniquely identifies a hole in a hole chart. The hole label format comes from the drawing standard.

Commonly Steps

To Create a Hole Chart

1. Click Annotate tab ► Sheet panel ► Hole Chart.
2. Specify a origin to use for the hole chart.
3. Specify the angle by which the Hole Chart coordinates are to be deviated from the UCS, and then press ENTER.
4. Specify a name for the origin, and then press ENTER.
5. In the drawing area, select the holes to include in the hole chart, and then press ENTER.
6. Place the hole chart in the drawing area.

Note: If the hole chart is too long, you can split the hole chart into multiple hole charts.

To Create a Hole Chart with a Hole Table

Hole Tables can be created only if the current drafting standard is ANSI

1. Click Annotate tab ► Sheet panel ► Hole Chart.
2. Specify a origin use for the hole chart.
3. Specify the angle by which the Hole Chart coordinates are to be deviated from the UCS, and then press ENTER.

4. Specify a name for the origin, and then press ENTER.
5. In the drawing area, select the holes to include in the hole chart, and then press ENTER.
6. Specify an insertion point to place the hole chart.
7. Specify an insertion point to place the hole table.

To Create a Nested Hole Chart

A nested hole chart is a consolidated table that describes the coordinate positions and specifications of holes that belong multiple origins.

1. Click Annotate tab ► Sheet panel ► Hole Chart.
2. Specify a origin to use for the nested hole chart.
3. Specify the angle by which the Hole Chart coordinates are to be deviated from the UCS, and then press ENTER.
4. Specify a name for the point origin, and then press ENTER.
5. At the "Select holes or [Block/POint]:" prompt, select the origins of the hole charts to include in the nested hole chart, and then press ENTER.
6. Specify an insertion point to place the hole chart. The program creates the nested hole chart and erases the individual hole charts.

To set Precision for Numeric Value in a Hole Chart or Table

1. In the drawing area, double-click a hole chart or a hole table.
2. In the Hole Chart dialog box, click .
3. In the Hole Chart Columns dialog box, select the row to set precision for by clicking the blank cell at the beginning of the row.
4. Click OK in all dialog boxes.

To Sort a Hole Chart or Table

1. In the drawing area, double-click a hole chart or a hole table.
2. In the Hole Chart dialog box, choose the Sort Data button.
3. In the Sort dialog box, select the settings and click OK.
4. In the Hole Chart dialog box, click OK.

To Toggle the Visibility of a Hole Chart or Hole Table Column

1. In the drawing area, double-click the hole chart or hole table to edit.
2. In the Hole Chart dialog box, click the Column Properties button.
3. In the Hole Chart Columns dialog box, select the column to make visible in the hole chart.
4. Under Visible in Hole Chart, select Cartesian to make the selected column visible in a Cartesian hole chart.

5. Select Polar to make the selected column visible in a Polar hole chart.
6. Click OK in all dialog boxes.

Hole Chart Dialog Box

Use this dialog box to edit hole charts.

Toolbar buttons

Specifies whether the dialog box displays hole chart or hole table details. Select “List of Coordinates” for hole chart and “Hole Table” for hole table.



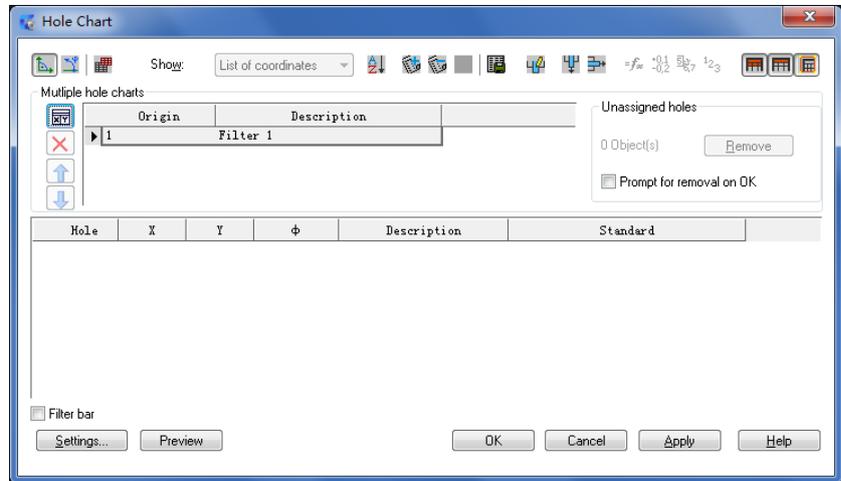
Sets coordinates to the Cartesian type.



Sets coordinates to the polar type.



Creates a hole table. The ANSI standard enables this option by default.



Show

Specifies whether the dialog box displays hole chart or hole table details. Select “List of Coordinates” for hole chart and “Hole Table” for hole table.



Sorts columns in a selected hole chart or hole table.



Closes this dialog box temporarily, enabling you to select a hole from the drawing and add it to the current selection set.



Closes this dialog box temporarily, enabling you to select a hole from the drawing and remove it from the current selection set.



Inserts hole chart into the drawing.



Exports data to MS-Excel, to a text editor, or to other supported formats.



Displays the Hole Chart Columns dialog box.



Removes a selected column.



Deletes one or more selected rows.



Displays the column headings in the drawing.



Displays the column title in the drawing.



Controls the display of the grip frame.

Multiple Hole Charts

Contains options to group the selected holes by a filter criteria, and display each group in a separate hole chart. The hole charts thus created share the same origin.



Adds a hole chart to the bottom of the list. A prompt enables you to place the hole chart in the drawing area and the hole chart populates automatically with the holes that not picked up by the other hole charts in the list.

Note: If you use the Add Holes button on nested hole charts, they convert to normal hole charts. Individual origins are erased and the holes are relabeled relative to the origin of the nested hole chart.



Deletes the selected hole chart. The holes that belonged to that hole chart become available in the hole charts below it in the list, provided they satisfy the filter conditions. Holes that do not satisfy a filter condition become unassigned holes.



Moves the selected hole chart one level up. The grouping of holes in the selected chart and the charts below it change. Filters are applied from top to bottom and only the holes that do not satisfy the filter conditions for the hole charts above are available for a given hole chart.



Moves the selected hole chart one level down. The grouping of holes in the selected chart and the charts below it changes. Filters are applied from top to bottom and only the holes that do not satisfy the filter conditions for the hole charts above are available for a given hole chart.

Hole Chart list: Displays the hole charts in the current selection set. Click in any description cell to provide a meaningful name for the hole chart.

Unassigned Holes: Contains options to handle the holes that do not satisfy any of the filter criteria. Unassigned holes are labeled “?” in the drawing area.

Remove button: Removes unassigned holes from the selection set.

Prompt for removal on OK

Prompts for confirmation to remove unassigned holes automatically when you click the OK button.

Main window

Displays coordinate dimensions and diameter sizes of selected holes. To add a tolerance or fit to a dimension, select a cell in this table and right-click, and then click Fits/Tolerances.

Filters

Contains options to set filters for the hole chart selected in the multiple hole chart list. Note: Filters do not apply to nested hole charts.

Filter list: Displays the types of filters set for the hole chart. Additionally, contains a right-click menu to add/remove filters.

Right-click menu items:

Add custom filter - Adds a filter, for which you can select from a set of predefined filter conditions that are available from a drop-down list.

Add hole selection filter - Adds a filter that enables you to select holes from the drawing area manually. You can add only one hole selection filter per hole chart.

Delete - Deletes the filter selected in the filters list.

Details for filter: Contains options enabling you to define the filter criteria for the filter selected in the Filters list.

Drop-down list: Contains predefined filter conditions. This list is visible only if the selected filter type in the filters list is a custom filter.

Edit box: Specifies the operand for the filter condition selected in the drop-down list. This edit box is visible only if the selected filter type in the filters list is a custom filter.

Select: Temporarily hides the Hole Chart dialog box enabling you to select holes from the drawing area. To select a hole, in the drawing area, click the hole. Press ENTER or ESC to return to the Hole Chart dialog box. This button is visible only if the selected filter type in the filters list is a hole selection filter.

Settings button

Displays the Hole Chart Settings dialog box.

Preview

Temporarily closes the dialog box to show the drawing area. Highlights the holes that belong to the hole chart selected in the multiple hole chart list. Press ENTER to return to the Hole Chart dialog box.

Super Card

Menu: Aids Tools ► Super Card

Keyboard command: Super Card

To meet the small and medium-sized enterprise to carry on the design summary card processing, processing technology card definition and filling and process card management needs, provide a powerful, very practical super card function.

The user can easily draw out all kinds of cards, generate some kind of summary cards and process cards.

The biggest feature of Super Card is you can edit the generated card , and you can edit several properties block of card at the same time.

To Creat Super Card

1. Click Aids Tools menu ► Super Card
2. In the "Select card" dialog, select the desired card;
3. In the drawing specified card's insertion point position;
4. In the current active card interface, fill in each tab page of card with data;
5. Click the "draw" button, the card will be drawn automatically at the designated position in accordance with settings.

Card Edit

Menu: Aids Tools ► Card Edit

Keyboard command: GMSUPERCARDEDIT

For the convenience of users to edit single data or whole card, provide card edit function, at the same time users can use "V" command to edit any property block of card separately.

To Edit Card

1. Click aids tools menu ► Card Edit
2. Select the target that needs to edit in the drawing.
3. In the current active card interface to fill in or modify each tab page's data.
4. Click the "draw" or "refresh" button, the revised data will be shown in the card.

Computational tools

Calculate Area

Summary

Menu: Tools ► Inquiry ► Calculate Area

Keyboard command: GMAREA

The calculation of materials area and volume in design is a necessary segment. Quick calculation of the area and volume of raw materials can contribute to effective control of material cost in design.

To Calculate Area

1. Click tools menu ► Inquiry ► Calculate Area
2. Select the area to be calculated
3. Press the 'Enter' key to complete the area computation of currently selected graphics and bring up the dialog box of 'Calculate Area, Perimeter, and Quality'

4. If the selected area is complicate, control the selected area with 'Add' or 'Subtract' in the dialog box where range, area, perimeter, and quality are displayed on a real-time basis

List of Prompts

Pick a point inside the outline (Press ENTER to Exit) :

Select the position of area to calculate in the engineering drawing.

Dialog Box of 'Calculate Area, Perimeter, and Quality'

Area: Calculated values of area of the selected area in the drawings

Perimeter: Calculated values of perimeter of the selected area in the drawings

Density: Preset material density value

Material: Preset material list

Area: Enter the cross-sectional area value of the selected area

Result: Enter the calculated value of the material corresponding to the cross-sectional area value of the selected area

Height: Enter the height value of the selected area

Result: Enter the calculated value of the material corresponding to the height value of the selected area

Calculation of Area, Perimeter and Mass

Area(mm2):	100	Density(kg/dm3):	7.85
Perimeter(mm):	40	Material:	Carbon steel

Mass Calculating

Input cross section area	Input thickness of flat
Area(mm2): 100	Height(mm): 100
Result(Kg): 0.0314	Result(Kg): 0.0785

Add Subtract OK

Chapter 10 Configure

Summary

GstarCAD Mechanical 2017 supplies comfortable and accurate drafting standards in line with standards in different countries. Support for standards as ANSI/BSI/CSN/DIN/GB/GOST/ISO/JIS. Help engineers communicate their design intent and improve production and processing based on standards-based drafting environments.

Select a Drafting Standard

Summary

GstarCAD Mechanical groups all settings governed by drafting standards and saves them to a profile referred to as a standard. GstarCAD Mechanical includes eight such profiles (ANSI, BSI, CSN, DIN, GB, ISO JIS, and GOST), each configured to match a drafting standard. These settings govern how GstarCAD Mechanical's built-in tools create geometry.

Default Drafting Standard

Because GstarCAD Mechanical 2017 is standards-driven, assigning a drafting standard to a drawing is critical. When you create a drawing, GstarCAD Mechanical automatically assigns a drafting standard to the drawing.

If you open an GstarCAD drawing or create a drawing using an GstarCAD template (.dwt), GstarCAD Mechanical assigns the most appropriate standard for the drawing (see also the following discussion "making standard overrides and custom standards available for other drawings"). You can change the drafting standard easily.

Settings Controlled by the Drafting Standard

The drafting standard controls the following elements of the standard:

- Mechanical object behavior.
- The properties of these layers.
- Text heights and colors.
- Dimension styles.
- Hole chart settings and formats.
- Centerline format.
- Section line format.
- Note text and leader formats.
- Symbology formats.
- Bill of Materials (BOM), parts list and balloon formats.

Commonly Steps

To Configure Elements of the Drafting Standard

1. The "IM:Standards Summary" dialog box contains a list of features that are controlled by the drafting standard, which are referred to as elements of the standard. Examples of such elements are drawing sheet formats, dimension styles, symbol formats, and so on. You can override the default settings of an element as follows:
2. At the Command prompt, enter GMOPTIONS.
3. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click the entry you want to change. The corresponding settings dialog box appears.
4. Make the necessary overrides and click OK until all dialog boxes close.

Note: Overrides are applied only to the objects created under the current drafting standard. For example, if you created a note while the standard was set to ISO, and then switched the standard to DIN, and you then override the text color settings for notes, only the notes created after you switched to DIN are affected. The note you created under ISO does not change color.

To Change the Current Drafting Standard

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, from the Standards list, select a new standard.
3. Then choose OK.

Note: None of the drawing objects that exist at the time the standard is switched are affected. Only the new objects you draw follow the standard you switched to.

To Configure Master Settings for Text Height

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click the root node (the standard name). The Standard Settings dialog box appears.
3. To change the text height, in the text section, in the Base height box, select a text height.

To Configure Master Settings for Text Color

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click the root node (the standard name). The Standard Settings dialog box appears.
3. To change the text color, in the text section, in the Base color box, select a text color.

To Create a Custom Text Style

1. At the Command prompt, enter STYLE.
2. Click New. The New Text Style dialog box is displayed.
3. In the Style Name box, type the name of the style to create and click OK. You return to the Text Style dialog box.
4. In the Font Name box, select the font to use.
5. In the Height box, type 0 (zero).

Note: GstarCAD Mechanical automatically manipulates text heights. To preserve this behavior, set text heights of custom text styles to 0.

6. Click Apply.
7. Click Close.

To Link Text and Leader Master Settings to Dimension Styles

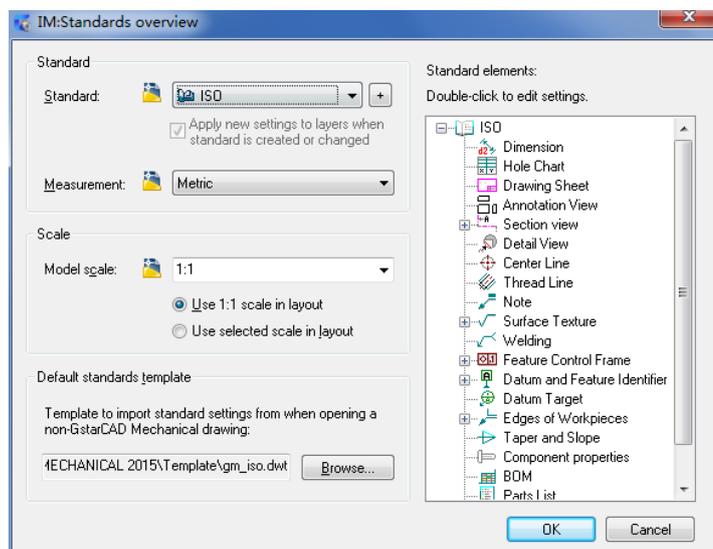
1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click the root node (the standard name). The Standard Settings dialog box appears.
3. Select the Link to dimension style check box.
4. To link to the Leader sub style only, select the Link to leader dimension style only check box.
5. Note: If you clear the Link to leader dimension style only check box, the leader and text settings link to the base dimension style. If you modify text or leader settings, all dimension sub styles update unless the sub style overrides the setting.
6. In the Style list, select the dimension style to link to.
7. Click OK until all dialog boxes close.

IM:Standards Summary dialog box

Use this dialog box to set preferences that are typically governed by a drafting standard.

Standard

Standard: Specifies the drafting standard to be used in the drawing. If you switch the standard, settings for all elements of the standard change accordingly. However, none of the existing drawing objects are affected. Only the new objects you draw follow the new standard. To create a custom standard (user-defined standard), type a name for the standard in this box and press Enter.



Measurement: Sets the units of measurement to be used in the current drawing.

Model Scale

Sets the scale to be used for text and symbols for the drawing in model space.

Use 1:1 Scale in Layout

Specifies that a 1:1 scale is used in layout, irrespective of the scale selected for model space.

Use Selected Scale in Layout

Specifies that the model space scale is used in layout as well.

Default standards template

Specifies the template the system picks the standard settings from, in case you open an GstarCAD Drawing (as opposed to an GstarCAD Mechanical drawing) or if you choose to create a drawing without using a template.

Note: Only standard dependent settings come from the default template.

If there is a unit mismatch between the default template and the GstarCAD drawing you are opening, the default template setting is ignored and the system behaves as though no default template is specified.

If you are creating a drawing without a template and the units you selected do not match the default template units, the default template setting is ignored and the system behaves as though no default template is specified.

If no default template is specified, the system automatically selects the most appropriate standard based on the type of units in the drawing you are opening.

Standard Elements

Displays a list of elements of the standard in the form of a tree.

The root of the tree shows the current drafting standard. Double-click the root to display a dialog box containing the master settings for the standard. Double-click an element to display its properties dialog box. From this dialog box, you can override the default settings as well as the settings inherited from the master settings.

Overrides are applied only to the objects created under the current drafting standard. For example, if you created a note while the standard was set to ISO, and then switched the standard to DIN, and you then override the text color settings for notes, only the notes created after you switched to DIN are affected. The note you created under ISO does not change color.

Tip: To find out what standard an object was created under, display the properties of that object.

For some items such as Feature Control Frames, more than one revision of a standard is available. Right click the revision you want to make active and select Set Current. If Set Current is not available, select Insert Standard.

Right-click menu items

Remove: Makes the selected item unavailable for implementation. For example, if you remove notes, and later on if you invoke the GMNOTE command, a message appears to inform you that the feature is unavailable for the selected standard.

Settings: Displays a dialog box containing the properties for the selected item. This dialog box is used to override the default settings for this item.

Restore Defaults: Removes all overrides for the selected item and resets its properties to the default settings.

Insert standard: Inserts an item that was removed previously. If the item contained overrides before removal, they

are not available when you reinsert the item.

Standard Settings Dialog Box

Sets up master settings for the current drafting standard. The elements of the standard that have their settings configured as `ByNameOfCurrentDraftingStandard`, automatically acquire their settings from this dialog box.

Dimension link

Link to dimension style: Links the text and leader settings to a dimension style, so that dimension styles are controlled by the drafting standard. Any change you make on this dialog box is applied to the linked dimension style. Any change you make to the linked dimension style is applied back to the text and leader settings.

When this option is selected, you can specify non-standard text heights.

Link to leader dimension style only: Restricts linking of dimension styles to the Leader dimension sub-style. Hence changes to text and leader settings are applied only to the Leader sub-style.

If this option is not selected, you link to the base dimension style. Changes to the base dimension style are automatically applied to all sub-styles, unless that setting is overridden in the sub-style. This means that if you modify the text and leader settings in this dialog box, the change is applied across all sub-styles, unless the settings are explicitly overridden in the sub-style.

Style: Displays the name of the dimension style to which leader and text settings are linked.

Text

Style: Sets the text style for use with all annotations. This setting is controlled by the drafting standard. Hence this setting can be changed only if the current drafting standard is a custom drafting standard or if the text and leader settings are linked to a dimension style.

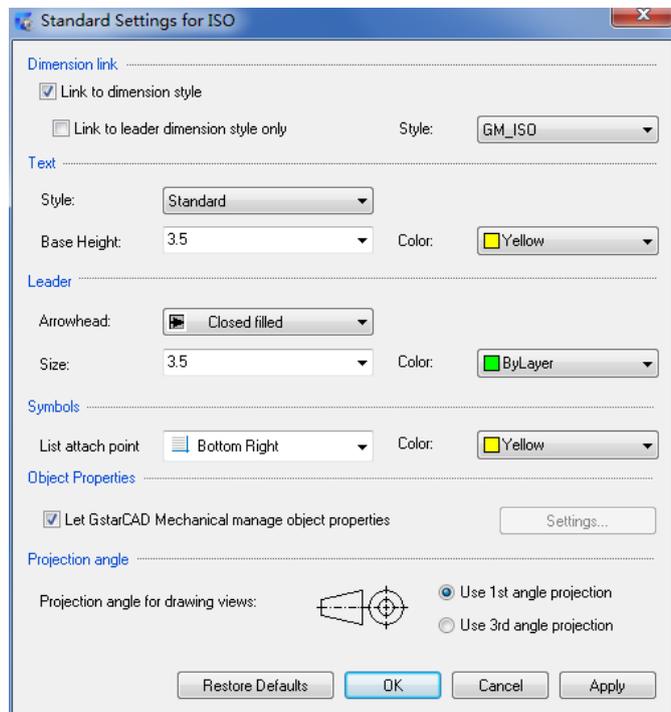
Base height: Sets the text height to use as the base height for annotation text. If the Link to dimension style check box is selected, or if the current drafting standard is a custom drafting standard, you can specify non-standard text heights. If not, you can only select the standard text heights that are available on the list.

By default, all symbols use the base text height for symbol text. Annotations such as annotation view labels, section line labels, detail view labels and balloons, use a text height that is one size larger than the base height.

Color: Sets the color for the text category selected for base text. If you click Select Color (at the bottom of the Color list), the Select Color dialog box appears. You can also enter a color name or number.

Leader

Arrowhead: Sets the default leader arrowhead type for all annotations.



Arrow size: Sets the default leader arrowhead size for all annotations.

Color: Displays and sets the leader color for all annotations. If you click Select Color (at the bottom of the Color list), the Select Color dialog box appears. You can also enter a color name or number. If the Link to dimension style check box is selected, this setting affects the dimension line and extension line colors.

Symbols

List attach point: Sets the default attachment point for lists such as parts lists and hole charts, which are controlled by the drafting standard.

Color: Sets the color for all symbol geometry. If you click Select Color (at the bottom of the Color list), the Select Color dialog box appears. You can also enter a color name or number.

Restore Defaults

Returns all values to the default values for the current standard.

Configure Dimension Style

When you select a drafting standard for a drawing, a set of dimension styles that comply with the standard are automatically added to the drawing.

The dimension styles associated with a drafting standard consists of a base dimension style, which defines the basic settings for dimensions for that standard, and substyles, which explicitly define styles for the different dimension types. In most cases, substyles are identical to the base style. In some cases, when it is required to enforce a unique characteristic on a dimension type, the default settings for that dimension style have at least one setting that is different from the base dimension style.

Commonly Steps

To Configure Distance Snap Settings

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click the entry you want to change. The dimension settings dialog box appears.
3. To enable distance snaps, select the Use distance snap for placement check box.
4. In the Snap value box, type the distance.
5. Click OK until all dialog boxes close.

To Set the Default Representation for Radius Dimensions

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Dimension. The dimension settings dialog box appears.
3. Click Radius. The Radius Representation dialog box appears.

4. Select the check boxes corresponding to the behaviors to enable. The preview updates.
5. Click OK until all dialog boxes close.

To Set the Default Representation for Diameter Dimensions

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Dimension. The dimension settings dialog box appears.
3. Click Diameter. The Diameter Representation dialog box appears.
4. Select the check boxes corresponding to the behaviors to enable. The preview updates.
5. Click OK until all dialog boxes close.

To Set the Default Representation for Chamfer Dimensions

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Dimension. The dimension settings dialog box appears.
3. Click Chamfer. The Chamfer Representation dialog box appears.
4. From the drop-down list, select the default representation for chamfer dimensions.
5. Click OK until all dialog boxes close.

To Add a New Dimension Text Template

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Dimension. The dimension settings dialog box appears.
3. Click Predefined Text. The Predefined Dimension Text dialog box appears.
4. From the Predefined Text list, select the dimension text template to add.
5. Click add.
6. Click OK until all dialog boxes close.

To Delete a Dimension Text Template

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Dimension. The dimension settings dialog box appears.
3. Click Predefined Text. The Predefined Dimension Text dialog box appears.
4. From the Predefined Text list, select the dimension text template to delete.

5. Click Remove.
6. Click OK until all dialog boxes close.

Dimension Settings Dialog Box

Use this dialog box to set preferences for Power Dimension commands for the current standard.

Dimension style

Base dimension style: Sets the dimension style to be used as the base dimension style. All the dimension types are based on the dimension style or a variation (a sub-style).

Edit: Displays the Dimension Styles dialog box, enabling you to create, modify, and assign dimension styles.

Force power dimensions to use this dimension style: Prevents the dimension style from being overridden with the DIMSTYLE command.

Default representations

Chamfer: Displays the Chamfer Representation dialog box, enabling you to select the default chamfer dimension representation from a predefined list of representations.

Radius: Displays the Radius Representation dialog box, enabling you to select the default locations for dimension line, dimension text, and arrowhead for radius.

Diameter: Displays the Diameter Representation dialog box, enabling you to select the default locations for dimension line, dimension text, and arrowhead for diameter.

Dimension Text

Ignore GstarCAD scale factor for linear dimension measurements: Makes Power Dimensioning ignore the DIMLFAC system variable and other settings that control the linear scale.

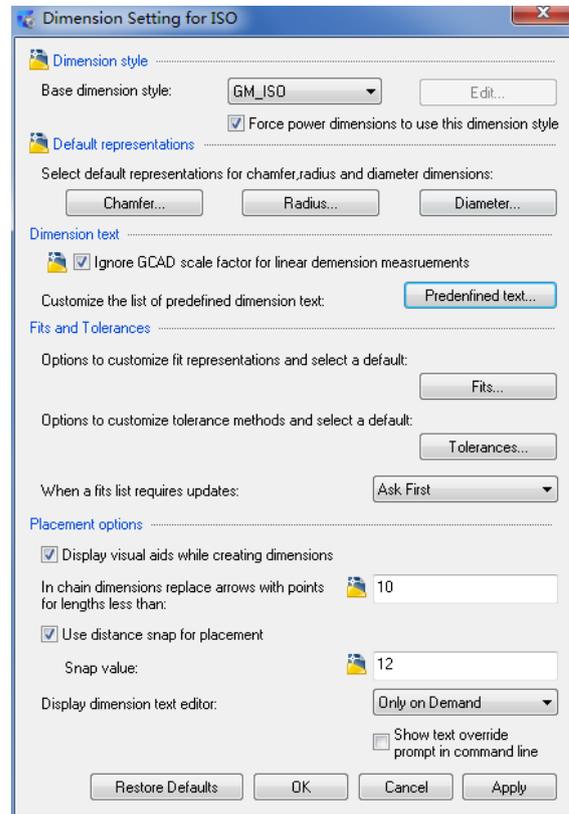
GstarCAD Mechanical uses several scaling mechanisms such as scale areas, detail views, and annotation views that are not available in GstarCAD. Power dimensioning recognizes the scales and automatically compensates for them. If you choose to honor GstarCAD scale factors and use different scales in different places in a drawing, GstarCAD Mechanical's scaling mechanisms may not reflect the correct dimension value.

Predefined Text: Displays the Predefined Text dialog box, which enables you to create predefined dimension text templates for Power Dimensioning commands at draw time.

Fits and Tolerances

Fits: Displays the Fits Representation dialog box, which enables you to set the default fit representation.

Tolerances: Displays the Tolerance Representation dialog box, which enables you to set the default tolerance method.



When a fits list requires updates: Configures how fits lists rebuild. You can choose to update fits lists schematically, have the system prompt you, or update fits lists on demand by selecting the manual option.

Placement Options

Display visual aids when creating dimensions: Displays an icon on the crosshairs when a dimension is created, indicating the type of dimension being created.

In chain dimensions, replace arrows with points for lengths less than: Defines the distance at which arrowheads are replaced with points in chain dimensions.

Use distance snaps for placement: Turns on distance snaps, which cause dimensions to snap into place as you move the dimension line in the drawing area.

Snaps value: Sets the distance from the snap position to the object being dimensioned.

Display dimension text editor: Sets how often the dimension text editor appears during a command session.

Show text override prompt: Enables the display of the text override prompt enabling you to modify the dimension text on the command line during the dimensioning process. By default, this check box is cleared.

Restore defaults

Resets all settings to the factory settings for the current standard. If the current standard is a custom standard, the settings default to the factory settings of the standard it is derived from.

Configure the Layout of Hole Charts

AutoCAD Mechanical enables you to pre-configure the format of hole charts, saving it with the drafting standard. At draw time, hole charts based on that standard default to that format automatically. You can:

- Set the Text height and color for heading and data rows separately.
- Set the gap between the heading and data rows, as well as the gap between adjacent data rows.
- Determine the hole chart title.
- Select which columns to show in a hole chart.
- Add custom columns to a hole chart.

Note: If you change the settings in the middle of a drawing session, existing hole charts will not reflect the change. However, any new hole chart you create will reflect the changes in the settings.

In the drawing area, hole labels tag and identify the holes in the hole chart. The hole label is generated dynamically with a formula. The default formula for hole labels ensures that holes are relabeled whenever holes are added, deleted, or moved. In some cases, this behavior is not desirable. In the DIN, ISO, GB, and CSN standards, you can prevent holes from being relabeled by activating the User Item Order, a sort option under Sorting Styles.

In addition to customizing the formula for hole labels, you can:

- Set text height and color.
- Add a leader to the hole label
- Select how holes are marked

Commonly Steps

To Configure Text Height and Color for Hole Charts

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Hole Chart. The Hole Chart settings dialog box appears.
3. To change the text height of hole chart column labels, in the List of coordinates section, in the Heading text height list, select the text height.

Note: To acquire the base height setting from the master text settings, select the text height marked “By NameOfCurrentDraftingStandard” in the respective lists.
4. To change the text height of hole chart data rows, in the List of coordinates section, in the Data text height list, select the text height.
5. To change the text height of the origin labels, in the Point and Origin Cartesian section, in the Text height list, select the text height.
6. To change the text color of hole chart column labels, in the List of coordinates section, in the Heading text color list, select the color.
7. To change the text color of hole chart data rows, in the List of coordinates section, in the Data text color list, select the color.
8. To change the text color of the origin labels, in the Point and Origin Cartesian section, in the Text color list, select the color.
9. Click OK until all dialog boxes close.

To Configure the Gaps Between Rows in a Hole Chart

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Hole Chart. The Hole Chart settings dialog box appears.
3. In the List of coordinates section, in the Heading gap box, type or select the gap between the heading row and data rows.
4. In the Row gap box, type or select the gap between adjacent data rows.
5. Click OK until all dialog boxes close.

To Configure Which Columns are Visible in a Hole Chart

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Hole Chart. The Hole Chart settings dialog box appears.
3. Click . The Hole Chart Columns dialog box appears.
4. To make a hole chart column visible in all types of hole charts, select the check box in column 1.

5. To make a column visible in Cartesian Hole charts only, in the (1) Visible in hole Charts section, select the Cartesian check box. and clear the Polar check box.
6. To make a column visible in Polar Hole charts only, in the (1) Visible in hole Charts section, clear the Cartesian check box and select the Polar check box.
7. Click OK until all dialog boxes close.

Hole Chart Settings Dialog Box

Use this dialog box to specify hole chart settings for the current drafting standard.

Toolbar



Sets the default coordinates to the cartesian type. Once set, all new hole charts you create will have their coordinate type set to cartesian coordinates. This button is available only if you arrived at this dialog box via the IM:Standards tab.



Sets the default coordinates to the polar type. Once set, all new hole charts you create will have their coordinate type set to polar coordinates. This button is available only if you arrived at this dialog box via the IM:Standards tab.



Enables creation of a hole table by default.

Consequently, whenever you place a hole chart in the drawing area, the GMHOLECHART command prompts you to place a hole table as well. This button is available only if you arrived at this dialog box via the IM:Standards tab. You typically use this option with ANSI hole charts.

Show

Specifies whether this dialog box displays hole chart or hole table details. Select "List of Coordinates" for hole chart details and "Hole Table" for hole table details. This option is available only if you arrived at this dialog box via the IM:Standards tab.



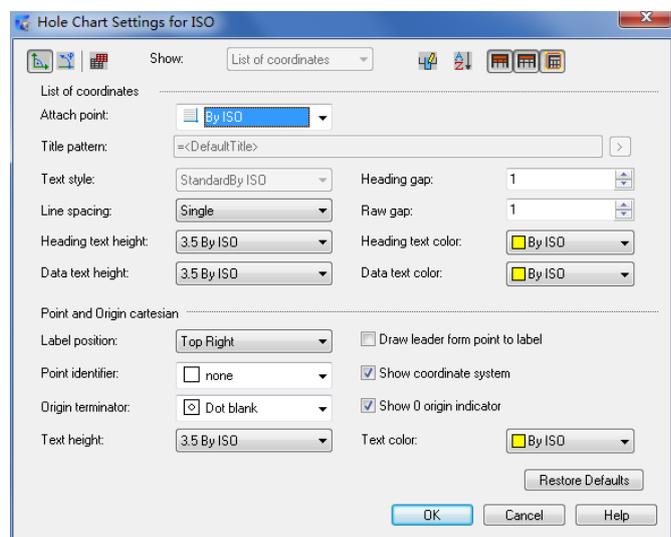
Displays the Hole Chart Columns dialog box. Use this dialog box to set the column defaults for hole charts. This button is available only if you arrived at this dialog box via the IM:Standards tab.



Displays the Sort dialog box to set the default sort order. This button is available only if you arrived at this dialog box via the IM:Standards tab.



Specifies that hole charts must display column headings by default. This button is available only if you arrived at this dialog box via the IM:Standards tab.





Specifies that hole charts must display a table title by default. Once set, all new hole charts you create will contain titles. This button is available only if you arrived at this dialog box via the IM:Standards tab.



Specifies that hole charts should display a grip frame by default. Once set, all new hole charts you create will contain grip frames. This button is available only if you arrived at this dialog box via the AM:Standards tab.

List of Coordinates/Hole Table

Attach Point: Sets the placement for the grip.

Title pattern: Sets the table title.

Text style: Specifies the text style to use with hole chart text.

Heading Gap: Sets the gap between heading and the first row in the table.

Row Gap: Sets the spacing between the rows in the table.

Line Spacing: Sets the height of the rows.

Heading text height: Sets the height of the text in the heading row.

Note: If you select the text height labelled "By NameOfStandard", the text height is linked to the base height setting in the master settings of the drafting standard. If the master setting changes, the heading text height changes accordingly.

Heading text color: Defines the text color to be used for the heading row. If the text height changes, the color automatically changes to the default color corresponding to that text height. Text object colors are set from the Object **Property Settings dialog box**.

Note: If you select the color labelled "By NameOfStandard", the heading text color is linked to the text color setting in the master settings of the drafting standard. If the master setting changes, the text color setting for heading text change accordingly.

Data text height: Defines the text height to be used for data rows.

Note: If you select the text height labelled "By NameOfStandard", the text height is linked to the base height setting in the master settings of the drafting standard. If the master setting changes, the data text height changes accordingly.

Data text color: Defines the text color to be used for data rows. If the text height changes, the color automatically changes to the default color corresponding to that text height. Text object colors are set from the Object Property Settings dialog box.

Note: If you select the color labelled "By NameOfStandard", the data row text color is linked to the text color setting in the master settings of the drafting standard. If the master setting changes, the data row text color changes accordingly.

Points and Origin Cartesian

Label position: Specifies how labels are positioned in the drawing.

Point identifier: Specifies the format of points in the drawing.

Origin terminator: Specifies the format of the point origin in the drawing.

Text height: Sets the text height to use for hole labels and origin labels. If the text height changes, the color automatically changes to the default color corresponding to that text height. Text object colors are set from the

Object Property Settings dialog box.

Notefy you select the text height labelled "By NameOfStandard", the text height is linked to the base height setting in the master settings of the drafting standard. If the master setting changes, the label text heights change accordingly.

Text color: Defines the text color to be used for hole labels and origin label. If the text height changes, the color automatically changes to the default color corresponding to that text height. Text object colors are set from the Object Property Settings dialog box.

Note:If you select the color labelled "By NameOfStandard", the label text color is linked to the text color setting in the master settings of the drafting standard. If the master setting changes, the label text color changes accordingly.

Draw leader from point to label: Draws a leader at the point origin.

Show coordinate system: Displays coordinate system in the drawing.

Show 0 origin Indicator: Displays 0 at the point origin in the drawing.

Restore Defaults

Resets all values to default settings.

Configure Drawing Sheets

When you select a drafting standard for a drawing, the drawing sheet formats that correspond to the standard become available for selection in the GMTITLE command. Drawing sheet formats consist of two individually configurable areas: The drawing border and The title block. Each of these items exists as separate .dwg files, which you can customize. Usually, you do not modify the original files supplied with the product, but rather create a copy and modify that. Configure your system to use the modified files with the GMTITLE command. Once you add the required drawing borders and title blocks, you may want to remove all other selections from the GMTITLE command.

Note: If you change the settings in the middle of a drawing session, they may not take effect if you already have a drawing border in the drawing area. To make the latest settings take effect, delete existing drawing borders and purge all blocks and then invoke the GMTITLE command.

Commonly Steps

To Add a Title Block to the List of Available Title Blocks

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Drawing Sheet. The Drawing Sheet Properties dialog box is displayed.
3. In the Default Title section, click the Additional button. The Select File dialog box is displayed.

Note: The program stores title blocks in theTitle subfolder of the folder containing title border files (See related links).

4. Select the file containing the desired title block and click Open.
5. Click OK until all dialog boxes close.

Note: The program saves the title block details to the current drafting standard. When you execute the

GMTITLE command after changing the title block details, the selected title block is available for selection.

To Remove a Title Block from the List of Available Title Blocks

1. The drafting standard keeps track of the names of the files selected for use as title blocks. Renaming a file removes the title block from the selection set, and therefore the GMTITLE command. To remove unnecessary selections from the GMTITLE command:
2. At the Command prompt, enter GMOPTIONS.
3. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Drawing Sheet. The Drawing Sheet Properties dialog box is displayed.
4. In the Default Title Block section, click the Additional button. The Select File dialog box is displayed.
5. Right-click the file to remove from the selection set and rename it.
6. Click Cancel until you return to the IM:Standards tab of the Options dialog box.
7. In the Standard Elements list, double-click Drawing Sheet again. The Drawing Sheet Properties dialog box is displayed.
8. In the Default Title section, click the Additional button. The Select File dialog box is displayed.
9. Right-click the file you renamed and restore it to its original name.
10. Click Cancel until you return to the IM:Standards tab of the Options dialog box.
11. Click OK.

Note: The program saves the title block details to the current drafting standard. When you execute the GMTITLE command after changing the title block details, the renamed title block is no longer available for selection.

To Add a Drawing Border to the Paper Formats List

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Drawing Sheet. The Drawing Sheet Properties dialog box is displayed.
3. In the Default Paper Format section, click the Additional button. The Select File dialog box is displayed.

Note: The program stores drawing borders in the Format subfolder of the folder containing title border files (See related links).

4. Select the file containing the desired drawing border and click Open.
5. Click OK until all dialog boxes close.

Note: The program saves the drawing border details to the current drafting standard. When you execute the GMTITLE command after changing the drawing border details, the selected drawing border is available for selection.

To Remove a Drawing Border from Paper Formats List

The drafting standard keeps track of the names of the files selected for use as title blocks. Renaming a file removes the drawing border from the paper formats list, and therefore the dialog box opened by the GMTITLE command. To remove unnecessary drawing border formats:

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Drawing Sheet. The Drawing Sheet Properties dialog box is displayed.
3. In the Default Paper Format section, click the Additional button. The Select File dialog box is displayed.
4. Right-click the file to remove from the selection set and rename it.
5. Click Cancel until you return to the IM:Standards tab of the Options dialog box.
6. In the Standard Elements list, double-click Drawing Sheet again. The Drawing Sheet Properties dialog box is displayed.
7. In the Default Paper Format section, click the Additional button. The Select File dialog box is displayed.
8. Right-click the file you renamed and restore it to its original name.
9. Click Cancel until you return to the IM:Standards tab of the Options dialog box.
10. Click OK.

Note: The program saves the drawing border details to the current drafting standard. When you execute the GMTITLE command after changing the drawing border details, the renamed drawing border is no longer available for selection.

Drawing Sheet Settings Dialog Box

Use this dialog box to specify entries for the title block, paper format, and scale factor of drawings for the current standard.

Revision

Displays most recent revision of the standard this symbol complies with.

Default Paper Format

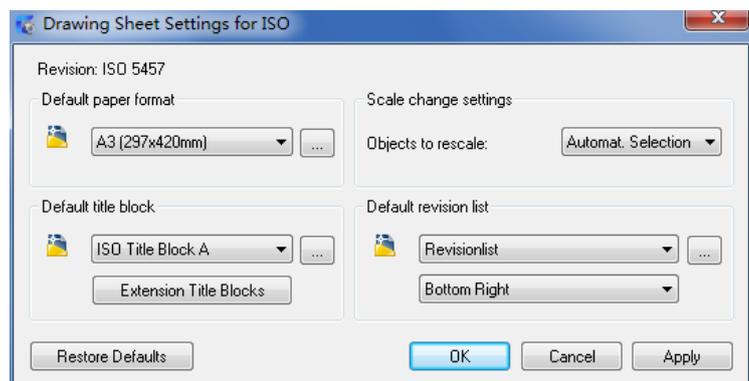
Drop-down list: Sets the default drawing border for the current drawing.



Displays the Select File dialog box, in which you can add a drawing file (containing a drawing border) to the drop-down list and set it as the default drawing border.

Default Title Block

Drop-down list: Sets the default title block for the current drawing.





Displays the Select File dialog box, in which you can add a drawing file (containing a title block) to the drop-down list and set it as the default drawing border.

Extension Title Blocks: Displays a dialog box so that you can place the part number diagonally opposite the title block.

Note: This option is available only if the current drafting standard is set to GB.

Default Revision List

Note Revision lists are not available in the GOST standard.

First Drop-down list: Selects a default revision list block for the current drawing.

Displays the Select File dialog box, in which you can add a drawing file (containing a revision block) to the drop-down list and set it as the default revision list.

Second Drop-down list: Sets a base point for the revision block.

Scale Change Settings

Objects to Rescale: Specifies whether the drawing objects are to be selected automatically or manually.

Restore Defaults: Resets all settings to their default values.

Configure Section View

Sets properties for section view styles. The sample image on each tab displays the effects of each option. The New and Modify Section View Style dialog boxes have 3 tabs (Identifier and Arrows tab, Cutting Plane tab, View Label tab).

Identifier and Arrows Tab: Sets the properties for the identifiers and direction arrows used by section lines.

Identifier

Controls the format of identifiers on section lines.

Text Style: Sets the text style to use for identifiers.

Note: If you select the text style labeled "By NameOfCurrentDraftingStandard", identifiers are linked to the text style setting of the master settings of the drafting standard.

Text color: Sets the text style to use for identifiers.

Text height: Sets the height of identifiers.

Note: If you select the text height labeled "By NameOfCurrentDraftingStandard", identifier text height is based on the base height setting in the master settings of the drafting standard. When this option is selected, the identifier height is always one size larger than the base height setting.

Exclude characters: Sets the characters that must not be used as identifiers.

Show identifier at all bends: When selected, displays section line identifiers at all bends.

Use continuous labeling: When selected, names end and bends with consecutive letters of the alphabet.

Direction arrows

Controls the format of the direction arrows on the section line.

Show direction arrows: When selected, displays direction arrows at the end points of section lines.

Start symbol: Sets the arrowhead type to use at the start of the section line.

Note: If you select the arrowhead type labeled “By NameOfCurrentDraftingStandard”, the arrowhead type is linked to the leader arrowhead setting of the master settings of the drafting standard.

End symbol: Sets the arrowhead type to use at the end of the section line.

Note If you select the arrowhead type labeled “By NameOfCurrentDraftingStandard”, the arrowhead type is linked to the leader arrowhead setting of the master settings of the drafting standard.

Symbol color: Sets the color to use for direction arrows. Both direction arrows uses the same color. If you click Select Color (at the bottom of the Color list), the Select Color dialog box is displayed. You can also enter a color name or number.

Note: If you select the color labeled “By NameOfCurrentDraftingStandard”, the arrowhead color is linked to the leader color setting of the master settings of the drafting standard.

Symbol size: Sets the size of the arrowheads. You can select a typical size from the list, or enter any numeric value for the size. The program validates the value you enter to prevent you from specifying invalid values.

Note: If you select the size labeled “By NameOfCurrentDraftingStandard”, the arrowhead size is linked to the arrow setting of the master settings of the drafting standard.

Extension length: Sets the length of the extension lines used by the direction arrows.

Arrangement

Controls the placement of section line identifiers and direction arrows, in relation to the start and end point of the section line.

Note: The arrangement is also controlled by the projection angle (first or third).

Identifier position: Sets the position and alignment of the section line identifier.

Identifier offset: Sets the distance to offset the section line identifier from the position specified by the Identifier position setting.

Arrow direction: Specifies whether the direction arrows should point towards the cutting plane or away from the cutting plane.

Restore Defaults

Returns all values to the default values for the drafting standard and revision the style is based on.

Cutting Plane Tab

End and bend lines

Controls the properties of the end and bend line segments of section lines.

Show end and bend lines: When selected, displays the end and bend line segments of section lines.

Line color: Sets the color for the end and bend segments. All end and bend line segments use the same color. If

you click Select Color (at the bottom of the Color list), the Select Color dialog box is displayed. You can also enter a color name or number.

Linetype: Sets the linetype for the end and bend segments. All end and bend line segments use the same linetype.

Lineweight: Sets the lineweight of the end and bend segments. All end and bend line segments use the same lineweight.

End line length: Sets the length of the end segments.

End line overshoot: Sets the length of the extension beyond the direction arrows.

Bend line length: Sets the length of the bend line segment on either side of the bend vertices.

Cutting Plane lines

Controls the properties for plane line segments of section lines.

Show cutting plane lines: When selected, displays line segments to indicate the entire cutting plane.

Line color: Sets the color of cutting plane line segments. All plane line segments use the same color. If you click Select Color (at the bottom of the Color list), the Select Color dialog box is displayed. You can also enter a color name or number.

Linetype: Sets the linetype of cutting plane line segments. All plane line segments use the same linetype. If you click Other, the Select Linetype dialog box is displayed.

Lineweight: Sets the lineweight of cutting plane lines. All cutting plane line segments use the same lineweight.

View Label Tab

Label

Show view label: When selected, displays view labels on section views.

Text style: Sets the text style to use for the section view label.

Note: If you select the text style labeled "By NameOfStandard", section view labels are linked to the text style setting of the master settings of the drafting standard.

Text style Button: Displays the Text Style dialog box, where you can create or modify text styles.

Text color: Sets the color to use for the section view label. If you click Select Color (at the bottom of the Color list), the Select Color dialog box is displayed. You can also enter a color name or number.

Note: If you select the color labeled "By NameOfCurrentDraftingStandard", section view label color is linked to the color corresponding to the text height of the section view label.

Text height: Sets the height of the section view label.

Note: If you select the text height labeled "By NameOfCurrentDraftingStandard", the section view label size is based on the base height setting in the master settings of the drafting standard. When this option is selected, the section view label height is always one size larger than the base height setting.

Label position: Sets the placement of the section view label, in relation to the section view.

Distance from view: Sets the gap between the edge of the section view and the section view label.

Configure Detail View

GstarCAD Mechanical 2017 supplies intelligent detail view design tools, the drawing will be automatically updated as design changing. It could magnify the partial region by box selection in the drawing and easily create linked partial views with different scales.

To Configure Text Height and Color for Detail Views

By default, detail view labels are one size larger than the base height specified by the master settings for the drafting standard. You can override these settings, if required.

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, expand the Detail View node.
3. Double-click the current detail view style. The Modify Detail View dialog box appears.
4. To change the detail view label text attributes: specify the text color and text height.

Note: To derive the text height from the base text height specified by the master settings of the drafting standard, select the text height marked "By NameOfCurrentDraftingStandard".

5. To change the detail view identifier text attributes: specify the text color and text height.

Note: To derive the text height from the base text height specified by the master settings of the drafting standard, select the text height marked "By NameOfCurrentDraftingStandard".

6. Click OK until all dialog boxes close

Modify Detail View Style Dialog Box

Label

Label contents: Specifies the default textual content for detail view labels.

Label position: Specifies the placement of the detail view label, in relation to the detail view.

Text

Text style: Specifies the text style to use for the detail view label.

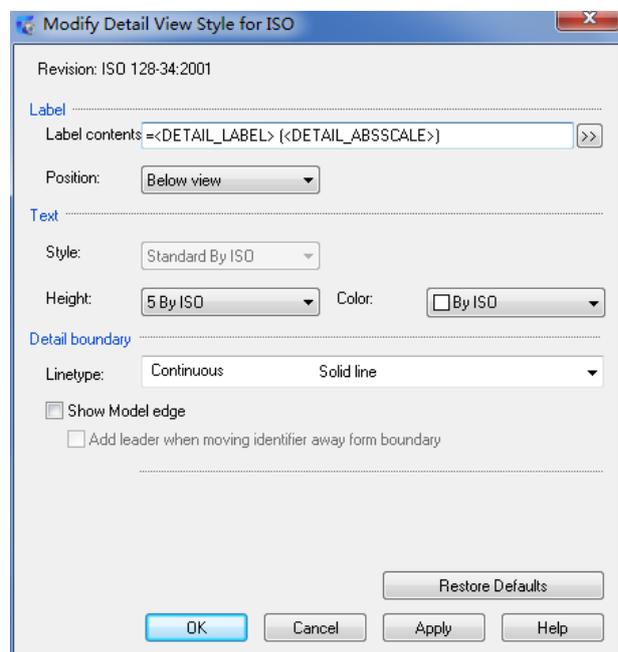
Text height: Specifies the text height of the detail view label.

Text color: Specifies the color to use for the detail view label.

Detail Boundary

Linetype: Sets the linetype for the detail boundary line. If

Show Mode edge: Sets the cut lines on the models in detail views.



Show Connection line: Displays a connection line between the detail border and the detail view boundary.

Restore Defaults

Returns all values to the default values for the drafting standard and revision the style is based on.

Configure Leader Notes

When you select the drafting standard (and the revision), the default settings for notes change to match that drafting standard. You can customize these settings to suit your drafting requirements. Because text and leader lines are the most common objects used for annotations, GstarCAD Mechanical 2017 provides a single configuration point (the Standard Settings dialog box) enabling you to specify their default properties. These settings are the master settings. In the Note Settings dialog box, the text and leader settings default to "By NameOfCurrentDraftingStandard." If you modify the master settings, note symbols automatically acquire the new settings. If you do not want note symbols to follow the master settings, you can override the text height and select a specific value.

Commonly Steps

To Configure the Leader Settings of Leader Notes

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Note. The Note Settings dialog box appears.
3. Click the Leader and Text tab.
4. To change the Arrowhead shape, in the Leader section, in the Arrowhead list, select the arrowhead.
5. To change the arrow size, in the Leader section, in the Arrow size list, select the size.
6. To change the leader color, in the Leader section, in the Color list, select the color.

Note: To acquire the arrowhead shape, arrow size, and leader color from master settings, select the list item marked "By NameOfCurrentDraftingStandard" in the respective lists.

7. Click OK until all dialog boxes close.

To Configure Text Height and Color for Leader Notes

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Note. The Note Settings dialog box appears.
3. Click the Leader and Text tab.
4. To change the text height, in the Text section, in the Height list, select the text height.

Note: To acquire the base height setting from the master text settings, select the text height marked "By NameOfCurrentDraftingStandard" in the respective lists.

5. To change the text color, in the Text section, in the Color list, select the color.
6. Click OK until all dialog boxes close.

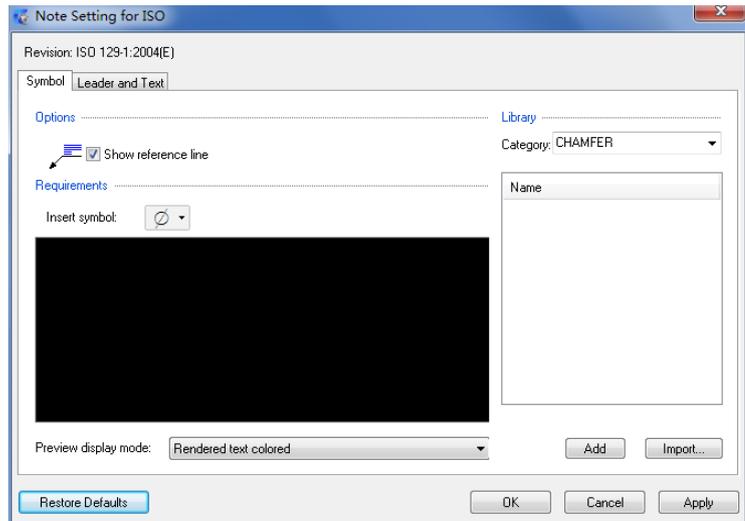
Note Settings Dialog Box

Use the Note Settings dialog box to customize the default settings of leader notes for the current drafting standard.

Symbol tab

Options

Show reference line: Adds a reference line from the leader to the note text. The icon to the left of the check box shows the result.



Requirements

Insert symbol: Displays a palette to enable you to insert a special character at the current cursor position in the multi-line text editor. In the rendered text mode, the editor shows the special character while in the plain text mode the editor shows the corresponding control key sequence.

Right-click menu options: Enables you to specify the formatting for the segment of text selected in the multi-line text editor. The formatting options are:

Format > Underline - Toggles underlining for the selected text

Format > Overline - Toggles overlining for the selected text

Format > Align - Specifies how the selected text aligns with stacked text on the same line

Format > Stacked- Stacks two segments of text, separated by a “/” character.

Format > Text Color - Specifies a color for the selected text

Format > Text Height - Applies a scale factor for the selected text

Format > Text Font - Specifies a font for the selected text

Format > Text Width - Applies a scale factor for the width of the selected text

Format > Tracking Factor - Specifies the spacing between characters of the selected text

Dim Style >Apply Dim Style - Applies a dimension style for a numerical expression. This option is visible only when a numerical expression is selected.

Note:The remaining options in the Dim Style menu lists the available dimensions styles that you can select from.

Precision - Enables you to specify a precision for a numerical expression. This option is visible only if a numerical expression is selected. It is not available if you have applied a dimension style to the selected numerical expression.

Cut - Moves the selected text to the windows clipboard.

Copy - Copies the selected text to the windows clipboard.

Paste - Inserts the contents of the windows clipboard at the insertion point.

Delete - Erases the selected text.

Select All - Selects the entire content of the multi-line text editor.

Preview display mode: Specifies how note text appears in the multi-line text editor.

Plain Text - Displays formulas and formatting signs that define the note text.

Rendered Text Colored - Renders all formulas and formatting signs to preview the note.

Rendered Text Mono - Displays monochromatically the same text as the Rendered Text Colored mode.

Library

Category: Displays a list of object categories for which you can create note templates.

Templates: Displays the list of available templates for the selected category.

Menu options

Add: Saves the text in the multi-line text editor as a new note template. The new template is added to the symbol library.

Delete: Deletes the selected note template from the symbol library.

Rename: Makes the template name editable. AutoCAD Mechanical validates the name you provide to ensure that the names of templates for a given category are unique.

Set default: Makes the selected template the default template for the selected category. A blue tick mark appears next to the default template name. When you attach a note to an object described by the selected category, the default template loads automatically.

Remove default: Removes the default status of the selected template. This menu option is available only on the default template. When no default template is selected, no note text appears in the multi-line text editor.

Add: Saves the content of the multi-line text editor as a new template. Prompts to name the new item.

Import: Imports templates from the symbol library of another drawing. You can only import symbols of the same standard and revision as the one you are editing.

Leader and Text tab

Leader

Arrowhead: Specifies the default leader arrowhead type for note symbols.

Note: If you select the arrowhead type labelled "By NameOfStandard", the arrowhead type is linked to the leader arrowhead setting of the master settings of the drafting standard. If the master settings change, the arrowhead type setting for note symbols change accordingly.

Arrow size: Specifies the default leader arrowhead size for note symbols.

Note: If you select the size labelled "By NameOfStandard", the arrow size is linked to the arrow setting of the master settings of the drafting standard. If the master settings change, the arrow size value for note symbols change accordingly.

Color: Specifies the default leader color for note symbols.

Note: If you select the color labelled "By NameOfStandard", the leader color is linked to the leader color setting of the master settings of the drafting standard. If the master settings change, the color setting for note symbols change accordingly.

Text

Height: Specifies the default text height for the symbol text of note symbols. If the current standard is a "custom standard", you can specify a non-standard text height. If not, AutoCAD Mechanical restricts you to selecting a value from the list.

Note: If you select the text height labelled "By NameOfStandard", the text height is linked to the base height setting in the master settings of the drafting standard. If the master settings change, the text height setting for note symbols change accordingly.

Color: Specifies the default color for the symbol text of note symbols. If the text height changes, the color automatically changes to the default color corresponding to that text height. Text object colors are set from the Object Property Settings dialog box.

Note: If you select the color labelled "By NameOfStandard", the text color is linked to the base text color setting in the master settings of the drafting standard. If the master settings change, the text color setting for note symbols change accordingly.

Leader justification: Sets the position of the note text in relation to the leader end point.

Note: This option is disabled for the GOST standard. However, it is available for custom standards based on the GOST standard.

Restore Defaults

Restores all settings of this dialog box to the preset default values for the drafting standard.

Configure Surface Texture Symbols

When you select the drafting standard (and the correct revision), the default settings for surface texture symbols change to match that drafting standard. You can customize these settings to suit your drafting requirements. Because text and leader lines are the most common objects used for annotations, GstarCAD Mechanical provides a single configuration point (the Standard Settings dialog box) enabling you to specify their default properties. These settings are the master settings.

In the Surface Texture Settings dialog box, the text and leader settings default to "By NameOfCurrentDraftingStandard." If you modify the master settings, surface texture symbols automatically acquire the new settings. If you do not want surface texture symbols to follow the master settings, you can override the text height and select a specific value.

You can also configure the options for symbol type, the appearance of the tail, and the appearance of the all-around sign. When you insert a surface texture symbol into a drawing, these settings are the initial settings. You can change them. We recommend that you set these options to match the most frequently used settings to minimize the amount of customizing required when you insert a symbol.

Commonly Steps

To Configure the Default Symbol Type for Surface Texture Symbols

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click the Surface Texture symbol.

The Surface Texture Settings dialog box appears.

3. In the Symbol type section, select the options to set as default.
4. Click OK until all dialog boxes close.

To Configure Leader Settings for Surface Texture Symbols

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click the Surface Texture symbol. The Surface Texture Settings dialog box appears.
3. To change the Arrowhead shape, in the Leader section, in the Arrowhead list, select the arrowhead.
4. To change the arrow size, in the Leader section, in the Arrow size list, select the size.
5. To change the leader color, in the Leader section, in the Color list, select the color.

Note: To acquire the arrowhead shape, arrow size, and leader color from master settings, select the list item marked “By NameOfCurrentDraftingStandard” in the respective lists.

6. Click OK until all dialog boxes close.

To Configure Text Height and Color for Taper and Slope Symbols

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click the Surface Texture symbol. The Surface Texture Settings dialog box appears.
3. To change the text height, in the Text section, in the Height list, select the text height.

Note: To acquire the base height setting from the master text settings, select the text height marked “By NameOfCurrentDraftingStandard” in the respective lists.

4. To change the text color, in the Text section, in the Color list, select the color.
5. Click OK until all dialog boxes close.

Note: If the current drafting standard is a custom standard, you can click inside the Height box and type any text height.

Surface Texture Settings Dialog Box

Use the Surface Texture Settings dialog box to specify the default settings of surface texture symbols, for the current drafting standard.

Revision

Displays most recent revision of the standard this symbol complies with.

Symbol type

Make circle in the veer small: Sets the size of the Machining Prohibited designation. This option is available only in custom standards.

Force Tail: Controls the default state of the Force Tail button in the Surface Texture dialog box. This button controls the availability of a tail for symbols without text.

Allow all Around: Controls the visibility of the All Around button in the Surface Texture dialog box. This option is not visible in the ANSI standard.

Add Multiple tail: Sets symbols to have an additional tail when multiple lines of text is available above the tail. This option is not visible in the ANSI standard.

Note: This option is not available for the GOST standard.

Preset lists

Contains a set of buttons that let you customize the drop-down list that are displayed in the Surface Texture dialog box.

Edit list: Displays a dialog box that lets you customize the list at the position represented by the button location.

Majority Symbol

Sets options for the majority symbol, which indicates the state common to all surfaces, using a single collective symbol. This option is not visible of the drafting standard is ANSI.

Simplified: Sets the majority symbol to its simplified representation. Accordingly, only the basic symbol appears in parenthesis at the right of the collective indication.

Full list: Sets the majority symbol to its full representation. Accordingly, if surface texture symbols are present elsewhere in the drawing, their state is fully indicated in parenthesis at the right of the collective indication.

Note: This option is not available for the GOST standard.

Leader

Arrowhead: Specifies the default leader arrowhead type for surface texture symbols.

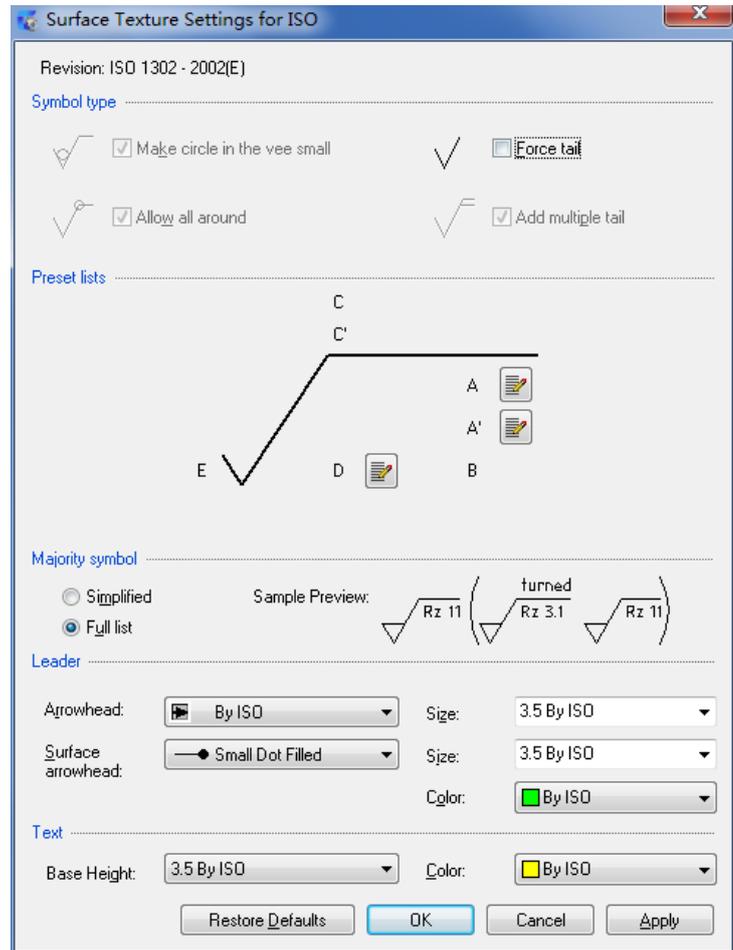
Note If you select the arrowhead type labeled "By NameOfStandard", the arrowhead type is linked to the leader arrowhead setting of the master settings of the drafting standard. If the master settings change, the arrowhead type setting for surface texture symbols change accordingly.

Size: Specifies the default leader arrowhead size for surface texture symbols.

Note: If you select the size labeled "By NameOfStandard", the arrow size is linked to the arrow setting of the master settings of the drafting standard. If the master settings change, the arrow size value for surface texture symbols change accordingly.

Surface arrowhead: Specifies the arrowhead to use for surface indication leaders.

Note: This option is visible only for the standards that permit surface indication leaders.



Color: Specifies the default leader color for surface texture symbols.

Note: If you select the color labeled "By NameOfStandard", the leader color is linked to the leader color setting of the master settings of the drafting standard. If the master settings change, the color setting for surface texture symbols change accordingly.

Text

Height: Specifies the default text height for the symbol text of surface texture symbols.

Note: If you select the text height labeled "By NameOfStandard", the text height is linked to the base height setting in the master settings of the drafting standard. If the master settings change, the text height setting for surface texture symbols change accordingly.

Color: Specifies the default color for the symbol text of surface texture symbols.

Note If you select the color labeled "By NameOfStandard", the text color is linked to the base text color setting in the master settings of the drafting standard. If the master settings change, the text color setting for surface texture symbols change accordingly.

Restore Defaults

Returns all values to the default values for the drafting standard.

Configure Welding Symbols

When you select the drafting standard (and the revision), the default settings for welding symbols change to match that drafting standard. If necessary, you can customize these settings to suit your drafting requirements

Because text and leader lines are the most common objects used for annotations, GstarCAD Mechanical provides a single configuration point (the Standard Settings dialog box) enabling you specify their default properties. These settings are the master settings. In the Welding Settings dialog box, the text and leader settings default to "By NameOfCurrentDraftingStandard." If you modify the master settings, welding symbols automatically acquire the new settings. If you do not want welding symbols to follow the master settings, you can override the text height and select a specific value. You can also specify which supplementary weld symbols are available for selection when you insert a welding symbol into a drawing. You can also configure the gap between the identification line and reference line.

Commonly Steps

To Configure Leader Settings for Welding Symbols

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Welding. The Welding Settings dialog box appears.
3. To change the Arrowhead shape, in the Leader section, in the Arrowhead list, select the arrowhead.
4. To change the arrow size, in the Leader section, in the Arrow size list, select the size.
5. To change the leader color, in the Leader section, in the Color list, select the color.

Note: To acquire the arrowhead shape, arrow size, and leader color from master settings, select the list

item marked "By NameOfCurrentDraftingStandard" in the respective lists.

6. Click OK until all dialog boxes close.

To Configure Text Height and Color for Welding Symbols

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Welding. The Welding Settings dialog box appears.

3. To change the text height, in the Text section, in the Height list, select the text height.

Note: To acquire the base height setting from the master text settings, select the text height marked "By NameOfCurrentDraftingStandard" in the respective lists.

4. To change the text color, in the Text section, in the Color list, select the color.
5. Click OK until all dialog boxes close.

To Configure Which Supplementary Weld Symbols are Available for Insertion

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Welding. The Welding Settings dialog box appears.
3. In the Show symbols for list, select a category.
4. In the list of supplementary weld symbols, select the check boxes adjacent to the symbols to make available and clear the others.
5. Repeat steps 4 and 5 for all required categories.
6. Click OK until all dialog boxes close.

Welding Settings Dialog Box

Revision

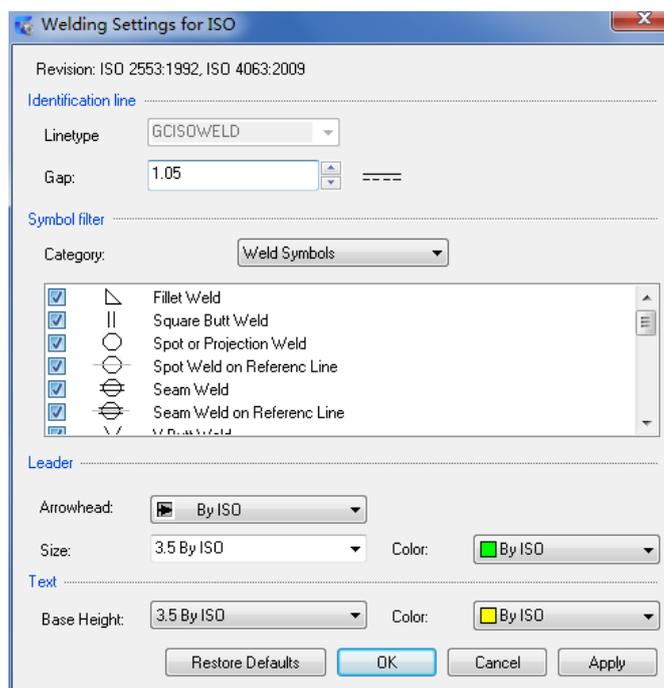
Displays most recent revision of the standard this symbol complies with.

Identification Line

Defines the appearance of identification lines. This section is not visible if the active drafting standard is ANSI, GOST, or JIS.

Linetype: Defines the default linetype to use for identification lines. This option is available only for custom standards.

Gap: Sets the distance between the identification line (if added) and the reference



line.

Symbol Filter

Category: Specifies what category of symbols to display in the Symbols list.

Symbols list: Displays a list of supplementary weld symbols supported by the current drafting standard. Select the check box in front of a weld symbol to make it available in the Weld Symbol dialog box.

Leader

Arrowhead: Specifies the default arrowhead type for welding symbols.

Note: If you select the arrowhead type labeled "By NameOfStandard", the arrowhead type is linked to the leader arrowhead setting of the master settings of the drafting standard. If the master setting change, the arrowhead type setting for welding symbols change accordingly.

Arrow size: Specifies the default leader arrowhead size for welding symbols.

Note: If you select the size labeled "By NameOfStandard", the arrow size is linked to the arrow setting of the master settings of the drafting standard. If the master setting change, the arrow size value for welding symbols change accordingly.

Color: Specifies the default leader color for welding symbols.

Note: If you select the color labeled "By NameOfStandard", the arrowhead color is linked to the leader color setting of the master settings of the drafting standard. If the master setting change, the color setting for welding symbols change accordingly.

Text

Height: Specifies the default text height for the symbol text of welding symbols.

Note: If you select the text height labeled "By NameOfStandard", the text height is linked to the base height setting in the master settings of the drafting standard. If the master setting change, the text height setting for welding symbols change accordingly.

Color: Specifies the default color for the symbol text of welding symbols. If the text height changes, the color automatically changes to default color corresponding to that text height.

Note: If you select the color labeled "By NameOfStandard", the text color is linked to the text color setting in the master settings of the drafting standard. If the master setting change, the text color setting for welding symbols change accordingly.

Restore Defaults: Returns all values to the default values for the drafting standard.

Configure Feature Control Frames

When you select the drafting standard (and the revision), the default settings for feature control frames change to match that drafting standard. If necessary, you can customize these settings to suit your drafting requirements

Because text and leader lines are the most common objects used for annotations, GstarCAD Mechanical 2017 provides a single configuration point (the Standard Settings dialog box) enabling you to specify their default properties. These settings are the master settings. In the Feature Control Frame Settings dialog box, the text and leader settings default to

“By NameOfCurrentDraftingStandard.” If you modify the master settings, feature control frame symbols automatically acquire the new settings. If you do not want feature control frame symbols to follow the master settings, you can override the text height and select a specific value. You can also specify what geometric dimensioning and tolerance (GD&T) symbols are available for selection when you insert a feature control frame into a drawing.

Commonly Steps

To Configure Text Height and Color for Feature Control Frame Symbols

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Feature Control Frame. The Feature Control Frame Settings dialog box appears.
3. To change the text height, in the Text section, in the Height list, select the text height.

Note: To acquire the base height setting from the master text settings, select the text height marked “By NameOfCurrentDraftingStandard” in the respective lists.
4. To change the text color, in the Text section, in the Color list, select the color.
5. Click OK until all dialog boxes close.

To Configure Leader Settings for Feature Control Frame Symbols

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Feature Control Frame. The Feature Control Frame Settings dialog box appears.
3. To change the Arrowhead shape, in the Leader section, in the Arrowhead list, select the arrowhead.
4. To change the arrow size, in the Leader section, in the Arrow size list, select the size.
5. To change the leader color, in the Leader section, in the Color list, select the color.

Note: To acquire the arrowhead shape, arrow size, and leader color from master settings, select the list item marked “By NameOfCurrentDraftingStandard” in the respective lists.
6. Click OK until all dialog boxes close.

To Configure Which GD&T Symbols are Available for Insertion

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Feature Control Frame. The Feature Control Frame Settings dialog box appears.
3. In the standard elements list, double-click the Feature Control Frame symbol.
4. In the Show symbols for list, select a category.
5. In the list of Geometric Dimensioning and Tolerancing symbols, select the check boxes adjacent to the symbols to make available and clear the others.
6. Repeat steps 4 and 5 for all required categories.

- Click OK until all dialog boxes close.

Feature Control Frame Settings Dialog Box

Use the Feature Control Frame Settings dialog box to configure the default settings of Feature Control Frames for the current standard.

Revision

Displays most recent revision of the standard this symbol complies with.

Merge Styles

Provides options to merge cells by removing duplication of tolerance types, tolerance zones, and datum references.

Merge symbol: Merges symbol cells, if the cells contain the same symbol.

Merge tolerance: Merges vertically adjacent tolerance cells, if the tolerance values are identical.

Merge datums: Merges vertically adjacent datum cells, if the tolerances are identical.

Adjust cells to vertically align: Pads cell contents with spaces so that cells of similar zones are of the same width.

Symbol filter

Provides options to select which geometric dimensioning and tolerance symbols will be available for insertion in Feature Control Frames.

Category: Selects a symbols category to display in the list of symbols.

Symbols list: Displays a list of for the selected category. Select the check box in front of a symbol to make it available for insertion via the GMFCFRAME command.

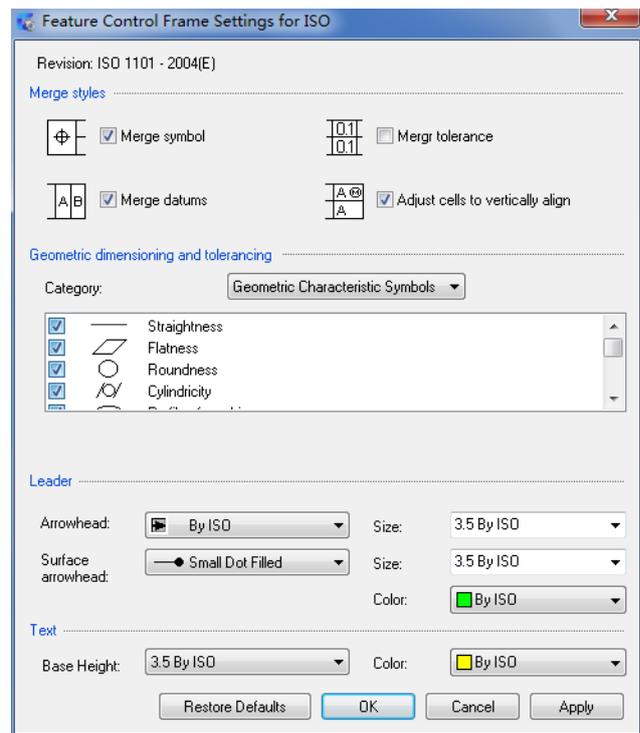
Leader

Arrowhead: Specifies the default leader arrowhead type for primary and secondary leaders of feature control frame symbols.

Note: If you select the arrowhead type labeled "By NameOfStandard", the arrowhead type is linked to the leader arrowhead setting of the master settings of the drafting standard. If the master setting change, the arrowhead type setting for feature control frame symbols change accordingly.

Size: Specifies the default leader arrowhead size for feature control frame symbols.

Note: If you select the size labeled "By NameOfStandard", the arrow size is linked to the arrow setting of the master settings of the drafting standard. If the master setting change, the arrow size value for feature control frame symbols change accordingly.



Surface arrowhead: Specifies the arrowhead to use for surface indication leaders.

Note: This option is visible only for the standards that permit surface indication leaders.

Color: Specifies the default leader color for feature control frame symbols.

Note: If you select the color labeled "By NameOfStandard", the arrowhead color is linked to the leader color setting of the master settings of the drafting standard. If the master setting change, the color setting for feature control frame symbols change accordingly.

Text

Height: Specifies the default text height for the symbol text of feature control frame symbols.

Note: If you select the text height labeled "By NameOfStandard", the text height is linked to the base height setting in the master settings of the drafting standard. If the master setting change, the text height setting for feature control frame symbols change accordingly.

Color: Specifies the default color for the symbol text of feature control frame symbols. If the text height changes, the color automatically changes to the default color corresponding to that text height.

Note: If you select the color labeled "By NameOfStandard", the text color is linked to the base text color setting in the master settings of the drafting standard. If the master setting change, the text color setting for feature control frame symbols change accordingly.

Restore Defaults

Returns all values to the default values for the drafting standard.

Configure Datum Identifier Symbols

When you select the drafting standard, the default settings for datum identifier symbols and feature identifier symbols change to match that drafting standard. If necessary, you can customize these settings to suit your drafting requirements. You cannot change settings of datum identifier symbols independently of feature identifier symbols. Both symbols derive their settings from the same set of configuration parameters. The only configurable options available for these symbols are the text and leader settings.

Because text and leader lines are the most common objects used for annotations, GstarCAD Mechanical provides a single configuration point (the Standard Settings dialog box) enabling you to specify their default properties. These settings are the master settings. In the Datum And Feature identifier Settings dialog box, the text and leader settings default to "By NameOfCurrentDraftingStandard." If you modify the master settings, datum and feature identifier symbols automatically acquire the new settings. However, if you do not want datum and feature identifier symbols to follow the master settings, you can override the text height and select a specific value.

Commonly Steps

To Configure Text Height and Color for Datum and Feature Identifier Symbols

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Datum and Feature Identifier. The Datum and Feature Identifier Settings dialog box appears.

- To change the text height, in the Text section, in the Height list, select the text height.

Note: To acquire the base height setting from the master text settings, select the text height marked "By NameOfCurrentDraftingStandard" in the respective lists.

- To change the text color, in the Text section, in the Color list, select the color.
- Click OK until all dialog boxes close.

To Configure Leader Settings for Datum and Feature Identifier Symbols

- At the Command prompt, enter GMOPTIONS.
- In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Datum and Feature Identifier. The Datum and Feature Identifier Settings dialog box appears. To change the Arrowhead shape, in the Leader section, in the Arrowhead list, select the arrowhead.
- To change the arrow size, in the Leader section, in the Arrow size list, select the size.
- To change the leader color, in the Leader section, in the Color list, select the color.

Note: To acquire the arrowhead shape, arrow size, and leader color from master settings, select the list item marked "By NameOfCurrentDraftingStandard" in the respective lists.

- Click OK until all dialog boxes close.

Datum and Feature Identifier Settings Dialog Box

Use this dialog box to specify the defaults settings of datum and feature identifier symbols, for the current drafting standard.

Revision

Displays most recent revision of the standard this symbol complies with.

Leader

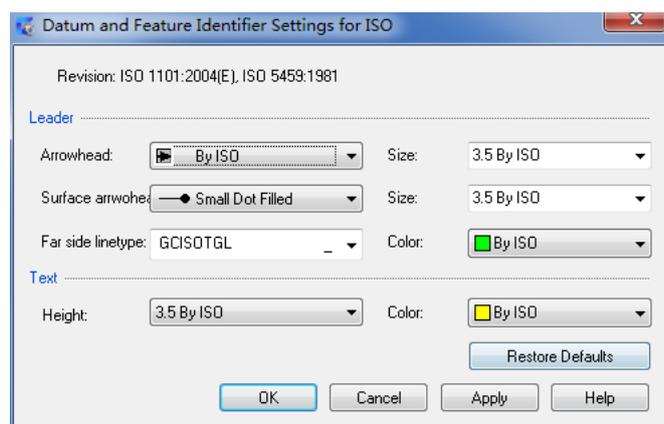
Arrowhead: Specifies the default leader arrowhead type for datum and feature identifier symbols.

Note: If you select the arrowhead type labeled "By NameOfStandard", the arrowhead type is linked to the leader arrowhead setting of the master settings of the drafting standard. If the master setting change, the arrowhead type setting for datum and feature identifier symbols change accordingly.

Size: Specifies the default leader arrowhead size for datum and feature identifier symbols.

Note: If you select the size labeled "By NameOfStandard", the arrow size is linked to the arrow setting of the master settings of the drafting standard. If the master setting change, the arrow size value for datum and feature identifier symbols change accordingly.

Surface arrowhead: Specifies the arrowhead to use for surface indication leaders.



Note: This option is visible only for the standards that permit surface indication leaders.

Size: Specifies the size of the surface indication leader arrowhead.

Note: This option is visible only for the standards that permit surface indication leaders.

Far side linetype: Specifies the linetype to use for surface indication lines that point to hidden surfaces.

Note This option is visible only for the standards that permit surface indication leaders.

Color: Specifies the default leader color for datum and feature identifier symbols.

Note: If you select the color labeled "By NameOfStandard", the arrowhead color is linked to the leader color setting of the master settings of the drafting standard. If the master setting change, the color setting for datum and feature identifier symbols change accordingly.

Text

Height: Specifies the default text height for the symbol text of datum and feature identifier symbols.

Note: If you select the text height labeled "By NameOfStandard", the text height is linked to the base height setting in the master settings of the drafting standard. If the master setting change, the text height setting for datum and feature identifier symbols change accordingly.

Color: Specifies the default color for the symbol text of datum and feature identifier symbols. If the text height changes, the color automatically changes to default color corresponding to that text height.

Note: If you select the color labeled "By NameOfStandard", the text color is linked to the text color setting in the master settings of the drafting standard. If the master setting change, the text color setting for datum and feature identifier symbols change accordingly.

Restore Defaults: Returns all values to the default values for the drafting standard.

Configure Datum Target Symbols

When you select the drafting standard (and the correct revision), the default settings for datum target symbols change to match that drafting standard. If necessary, you can customize these settings to suit your drafting requirements. Because text and leader lines are the most common objects used for annotations, GstarCAD Mechanical 2017 provides a single configuration point (the Standard Settings dialog box) enabling you to specify their default properties. These settings are the master settings. In the Datum Target Settings dialog box, the text and leader settings default to "By NameOfCurrentDraftingStandard." If you modify the master settings, datum target symbols automatically acquire the new settings. If you do not want datum target symbols to follow the master settings, you can override the text height and select a specific value. For datum target symbols with circular and rectangular terminations, you can specify the hatch angle, hatch spacing, and boundary line type. For datum target symbols with point terminations, you can specify the size of the point of termination and specify whether or not the symbol includes an arrowhead.

Commonly Steps

To Configure Text Height and Color for Datum Target Symbols

1. At the Command prompt, enter GMOPTIONS.

2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Datum Target. The Datum Target Settings dialog box appears.
3. To change the text height, in the Text section, in the Height list, select the text height.

Note: To acquire the base height setting from the master text settings, select the text height marked “By NameOfCurrentDraftingStandard” in the respective lists.
4. To change the text color, in the Text section, in the Color list, select the color.
5. Click OK until all dialog boxes close.

To Configure Leader Settings for Datum Target Symbols

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Datum Target. The Datum Target Settings dialog box appears.
3. To change the Arrowhead shape, in the Leader section, in the Arrowhead list, select the arrowhead.
4. To change the arrow size, in the Leader section, in the Arrow size list, select the size.
5. To change the leader color, in the Leader section, in the Color list, select the color.

Note: To acquire the arrowhead shape, arrow size, and leader color from master settings, select the list item marked “By NameOfCurrentDraftingStandard” in the respective lists.
6. Click OK until all dialog boxes close.

To Configure Circular and Rectangular Terminations

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Datum Target. The Datum Target Settings dialog box appears.
3. In the Datum Target Settings dialog box, in the Area hatch distance box, select the hatch spacing for the termination.
4. In the Hatch angle box, specify the hatch angle.
5. In the Boundary linetype list, select the boundary line type for the termination.
6. Click OK until all dialog boxes close.

To Configure Point Terminations

1. At the Command prompt, enter GMOPTIONS.
 2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click Datum Target. The Datum Target Settings dialog box appears.
 3. To draw an arrowhead at the leader start point, In the Datum Target Settings dialog box, select the Draw arrowhead for point termination check box.
 4. Click OK until all dialog boxes close.
-

Datum Target Settings Dialog Box

Use the Datum Target Settings dialog box to configure the default settings of datum target symbols for the current standard.

Revision

Displays most recent revision of the standard this symbol complies with.

Style

Target point size: Sets the size of the target point for datum targets with point terminations. The value you set refers to half the width of the "X" the target point.

Area hatch distance: Sets the distance between hatch lines for datum targets with rectangular and circular terminations.

Hatch angle: Sets the hatch angle for datum targets with rectangular and circular terminations.

Boundary linetype: Specifies the linetype to use for the datum target boundary for datum targets with rectangular and circular terminations.

Leader

Hidden linetype: Specifies the linetype to use for the leader of datum target symbols that indicate that the datum target is on the far side.

Draw arrowhead for point termination: Specifies whether arrowheads are drawn for datum targets with point terminations. This setting has no effect on datum target symbols that already exist in a drawing. It only affects new symbols you create.

Arrowhead: Specifies the default leader arrowhead type for datum target symbols.

Note: If you select the arrowhead type labeled "By NameOfStandard", the arrowhead type is linked to the leader arrowhead setting of the master settings of the drafting standard. If the master settings change, the arrowhead type setting for feature control frame symbols change accordingly.

Arrow size: Specifies the default leader arrowhead size for feature control frame symbols.

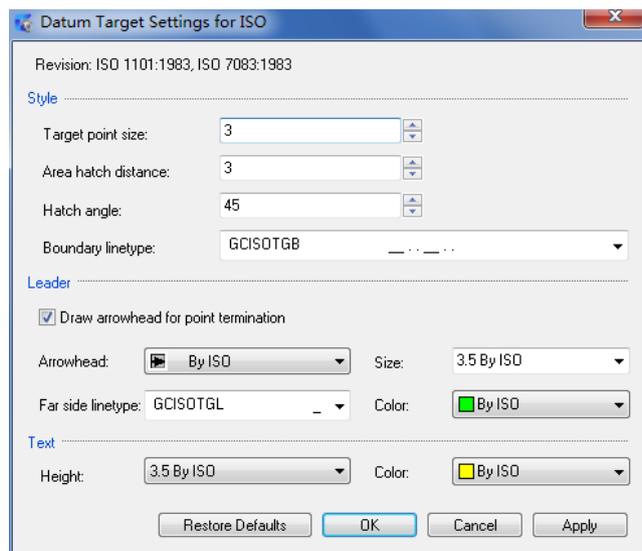
Note: If you select the size labeled "By NameOfStandard", the arrow size is linked to the arrow setting of the master settings of the drafting standard. If the master settings change, the arrow size value for datum target symbols change accordingly.

Color: Specifies the default leader color for datum target symbols.

Note: If you select the color labeled "By NameOfStandard", the arrowhead color is linked to the leader color setting of the master settings of the drafting standard. If the master settings change, the color setting for datum target symbols change accordingly.

Text

Height: Specifies the default text height for the symbol text of datum target symbols.



Note: If you select the text height labeled "By NameOfStandard", the text height is linked to the base height setting in the master settings of the drafting standard. If the master settings changes, the text height setting for datum target symbols change accordingly.

Color: Specifies the default color for the symbol text of datum target symbols. If the text height changes, the color automatically changes to the default color corresponding to that text height.

Note: If you select the color labeled "By NameOfStandard", the text color is linked to the base text color setting in the master settings of the drafting standard. If the master settings change, the text color setting for datum target symbols change accordingly.

Restore Defaults

Returns all values to the default values for the drafting standard.

Configure Edge Symbols

When you select the drafting standard (and the revision), the default settings for edge symbols change to match that drafting standard. If necessary, you can customize these settings to suit your drafting requirements.

Because text and leader lines are the most common objects used for annotations, GstarCAD Mechanical 2017 provides a single configuration point (the Standard Settings dialog box) enabling you to specify their default properties. These settings are the master settings. In the Edge Settings dialog box, the text and leader settings default to "By NameOfCurrentDraftingStandard." If you modify the master settings, edge symbols automatically acquire the new settings. However, if you do not want welding symbols to follow the master settings, you can override the text height and select a specific value.

Commonly Steps

To Configure Majority Symbol Representation of Edge Symbols

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Edges of Work pieces.
3. In the Majority symbol section, select the default representation.
4. Click OK until all dialog boxes close.

Edge Settings Dialog Box

Use the Edge Settings dialog box to specify the defaults settings of edge symbols, for the current drafting standard.

Revision

Displays the current standard.

Majority Symbol

Sets options for the majority symbol, which indicates the state common to all edges, using a single collective symbol.

Simplified: Sets the majority symbol to its simplified representation. Accordingly, only the basic symbol appears in parenthesis, at the right of the collective indication.

Full list: Sets the majority symbol to its full representation. Accordingly, if edge symbols are present elsewhere in the drawing, their state is fully indicated in parenthesis at the right of the collective indication.

Leader

Arrowhead: Specifies the default leader arrowhead type for edge symbols.

Note: If you select the arrowhead type labeled "By NameOfStandard", the arrowhead type is linked to the leader arrowhead setting of the master settings of the drafting standard. If the master setting change, the arrowhead type setting for feature control frame symbols change accordingly.

Arrow size: Specifies the default leader arrowhead size for edge symbols.

Note: If you select the size labeled "By NameOfStandard", the arrow size is linked to the arrow setting of the master settings of the drafting standard. If the master setting change, the arrow size value for edge symbols change accordingly.

Color: Specifies the default leader color for edge symbols.

Note: If you select the color labeled "By NameOfStandard", the arrowhead color is linked to the leader color setting of the master settings of the drafting standard. If the master setting change, the color setting for edge symbols change accordingly.

Text

Height: Specifies the default text height for the symbol text of edge symbols.

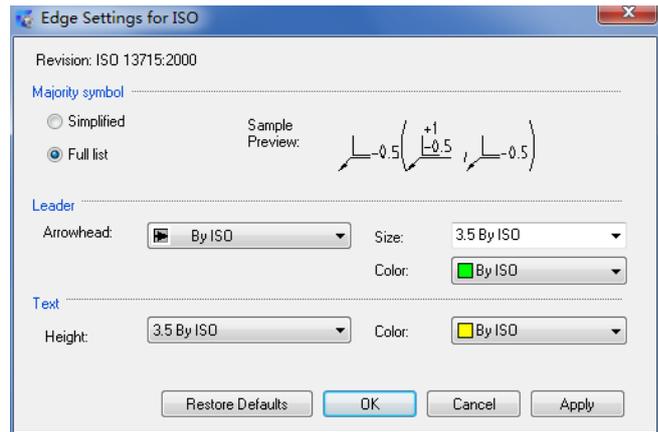
Note: If you select the text height labeled "By NameOfStandard", the text height is linked to the base height setting in the master settings of the drafting standard. If the master setting change, the text height setting for edge symbols change accordingly.

Color: Specifies the default color for the symbol text of edge symbols. If the text height changes, the color automatically changes to the default color corresponding to that text height.

Note: If you select the color labeled "By NameOfStandard", the text color is linked to the base text color setting in the master settings of the drafting standard. If the master setting change, the text color setting for edge symbols change accordingly.

Restore Defaults

Returns all values to the default values for the drafting standard.



Configure Taper and Slope Symbols

When you select the drafting standard (and the correct revision), the default settings for Taper and Slope Symbols

change to match that drafting standard. If necessary, you can customize these settings to suit your drafting requirements.

Taper and Slope Settings Dialog Box

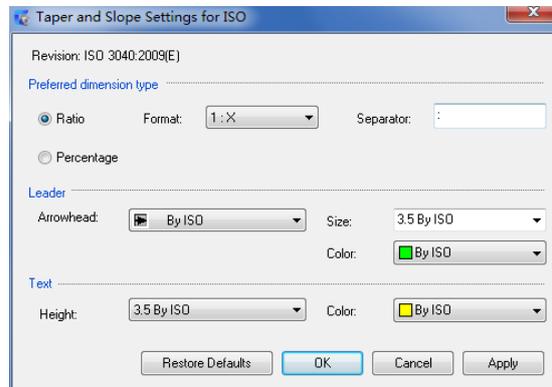
Use the Taper and Slope Settings dialog box to configure the default settings of taper and slope symbols for the active standard.

Revision

Displays the name and revision of the drafting standard on which the symbol is based.

Preferred dimension type

Defines whether the calculated slope/taper rates are expressed as a percentage or a ratio, by default.



Format: Defines the format in which ratios are expressed.

Separator: Defines the ratio separator to use in the ratio format. You can use alphanumeric characters, including spaces. If you do not specify a separator, the GMTAPERSYM command uses the colon character (:) as the separator.

Leader

Arrowhead: Specifies the default leader arrowhead type for taper and slope symbols.

Note: If you select the arrowhead type labeled "By NameOfStandard", the arrowhead type is linked to the leader arrowhead setting of the master settings of the drafting standard. If the master setting change, the arrowhead type setting for taper and slope symbols change accordingly.

Arrow size: Specifies the default leader arrowhead size for taper and slope symbols.

Note: If you select the size labeled "By NameOfStandard", the arrow size is linked to the arrow setting of the master settings of the drafting standard. If the master setting change, the arrow size value for taper and slope symbols change accordingly.

Color: Specifies the default leader color for taper and slope symbols.

Note: If you select the color labeled "By NameOfStandard", the arrowhead color is linked to the leader color setting of the master settings of the drafting standard. If the master setting change, color setting for taper and slope symbols change accordingly.

Text

Height: Specifies the text height for symbol text. When the height is set to ByNameOfCurrentDraftingStandard, it acquires the text height from the master settings of the current drafting standard. If master setting changes, text height of all taper/slope symbols based on the current drafting standard change automatically.

Color: Specifies the default color for the symbol text of taper and slope symbols. If text height changes, the color automatically changes to default color corresponding to that text height.

Note: If you select the color labeled "By NameOfStandard", the text color is linked to the text color setting in the master settings of the drafting standard. If the master setting change, the text color setting for taper and slope symbols change accordingly.

Restore Defaults: Returns all values to the default values for the drafting standard.

Custom BOM Properties

The entry point for all configuration operations for BOMs, balloons, and parts lists is the BOM Settings dialog box. This dialog box contains a list of available component properties. You can drag-and-drop properties to make them visible within parts lists, balloons, and the Component Properties sections of the Part Reference (for non-mechanical structure parts/assemblies) dialog boxes. These properties appear in the BOM automatically. If necessary, you can add properties to the BOM directly. You can create user-defined component properties. We recommend that you use the predefined component properties USER1 to USER8 and customize them instead. For numeric data, you can set the precision and specify whether the property is summed in BOMs and Parts Lists. You can customize the caption of a property, which appears as the column heading in BOMs and parts lists. You can also customize component properties to:

- Contain formulas so that the properties are populated with context-sensitive information automatically as you add objects to a drawing. If necessary, you can lock the property, ensuring that the formula is not overridden.
- Populate from a customizable drop-down list.

We recommend configuring component properties before you start drawing. If you add component properties after you create a BOM, the BOM may contain inconsistent data.

Commonly Steps

To Configure which Component Properties are Visible

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click BOM. The BOM Settings dialog box is displayed.
3. Select the tab corresponding to the entity you want to make the property visible in:
 - To make the property visible in Parts Lists, click the Parts List tab.
 - To make the property visible in Part References or the Component Properties dialog box, click the Component Properties/Part Reference tab.
 - To make the property visible in the BOM only, click the BOM tab.

Note:When you add a component property to the Parts List or the Component Properties/Part Reference tab, the program adds it automatically to the BOM tab as well.
4. In the Available component properties list, click the row heading of the component property to make it visible, and then click Add to. The program adds the component property to the bottom of the list in the tab.

Note:The Add to button is unavailable if the selected component property already appears in the list.
5. Use the Move up or Move down buttons to position the property.
6. Click OK.

To Create a User-defined Component Property

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click BOM. The BOM Settings dialog box is displayed.
3. Click New. The program positions the insertion point in the first column of the last row in the Available component properties list.
4. Add data to the new row, column by column. The program adds the new property to the Available component properties list as well as the BOM tab.
5. Use the Move up or Move down buttons to position the property.
6. Click OK.

To Set the Data Type and Format of a Component Property

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click BOM. The BOM Settings dialog box is displayed.
3. In the Available component properties list, in the row corresponding to the component property you want to change, select the data type from the drop-down list in the Data Type column.
4. In the Precision column, click and select the precision from the drop-down list.
5. In the Leading Zeros column, click and select the number of leading zeros from the drop-down list.

Note:Steps 5 and 6 apply to numeric data only.

6. Click OK.

To Set a Component Property to be Automatically Summed

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click BOM. The BOM Settings dialog box is displayed.
3. In the Available component properties list, in the row corresponding to the component property to be totaled, select the check box in the Sum column.
4. Click OK.

BOM Settings Dialog Box

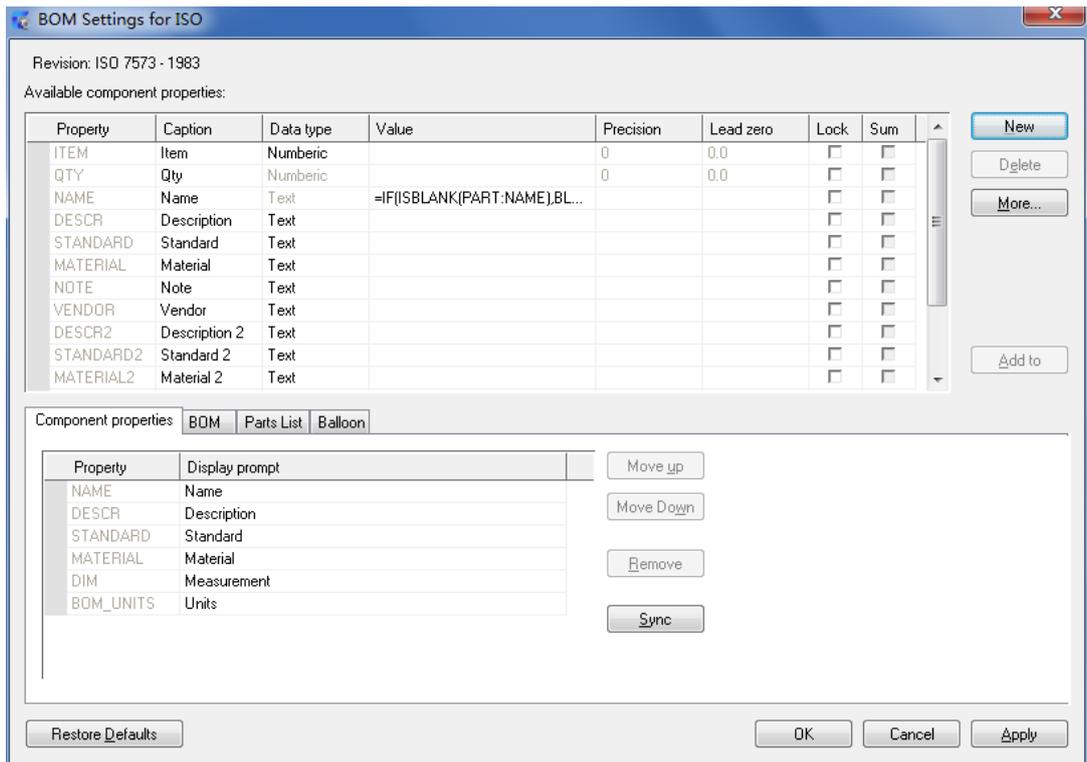
Use this dialog box to configure component properties and set up defaults for parts lists, balloons and data capture for the BOM.

Revision

Displays the name and the revision of the standard that controls the default settings of parts lists.

Available component properties

Contains a list of component properties that can be used as columns for BOMs and



parts lists. To make a property visible in the BOM, Parts list or in the Component Property section of the Part Reference dialog box and Properties dialog box, drag the property from this list to the list of properties in the corresponding tab.

Property: Defines the name of the component property. This name uniquely identifies a component property. This is the name by which a BOM column is referred to in formulas.

Caption: Defines the column header to be displayed for this property, when it is used as a column in a BOM table or parts list.

Data type: Specifies the data type for the property.

The text data type allows all alphanumeric characters while the numeric data type allows only numeric characters 0 -9. If you need to store a value such as 4 1/2", you must set the data type to Text.

Value: Specifies the default value for the property. .

Precision: Specifies the number of decimals to be used round off numerical data for that property. Precision is defined in the form of 0.000 where the number of zeros (0) after the decimal point indicates the number of decimals to use.

Lead zero: Specifies the number of leading zeros (0) to be used when displaying numerical data for that property. Leading zeros are defined in the form of 00.0 where the number of zeros (0) before the decimal point indicates the minimum number of digits that must exist before the decimal point.

Lock: Prevents the component property data from being edited in the BOM, Parts List, Part Reference and Properties dialog boxes.

Sum: Controls if the component property should be summed and displayed in the result bar of the BOM and Parts List dialog boxes.

New: Displays the last row of the Available component properties list, to enable you to add a new row and thereby create a user-defined component property.

Tippet is better to use the pre-defined component properties User1 to User8 (available via the More button) as these are better supported by the system. For example, when creating custom parts lists, to be able to properly fit text into columns, you must use property ids. The property ids for user defined columns are not known before hand and hence cannot be supported. However, the property ids for the pre-defined columns are known.

Delete: Removes the row selected in the Available component properties list. If the property is a user-defined property, the property is deleted. If not the property is added to the list of additional pre-defined component properties. The additional component properties are shown in the more component properties dialog box, which appears when you click the More button. This Delete button is available only if the entire row is selected.

More: Opens a list of additional pre-defined component properties to add to the Available component properties list.

Add to: Adds the row selected in the Available component property list, to the list of properties in the currently selected tab below the Available component properties list. This button is not available if the entire row is not selected or if the current tab is the Balloon tab.

Component properties tab

Component properties list: Displays the component properties that are visible in the Part Reference dialog box and Properties dialog box.

Property: Displays the name of the component property.

Display prompt: Defines the prompt to be displayed for that property in the Part References dialog box and Properties dialog box. For example, you can set the caption for the Name property to Enter the name of the component.

Move Up: Moves the selected row or range of rows in the Component properties list, one position up. This button is available only if entire rows are selected.

Move Down: Moves the selected row or range of rows in the Component properties list, one position down. This button is available only if entire rows are selected.

Remove: Removes the selected row or range of rows from the Component properties list. This button is available only if entire rows are selected.

Sync: Synchronizes the component properties list with the component properties list in the BOM tab. Consequently, all component properties visible in the BOM will be made visible in the Part Reference dialog box and the Properties dialog box.

Note: Sync does not add QTY and ITEM.

BOM tab

Component properties list: Displays the component properties that are visible in the BOM dialog box.

Property: Displays the name of the component property.

Caption alignment: Defines the how the column headings are aligned in the BOM dialog box.

Value alignment: Defines the how the data rows are aligned in the BOM dialog box.

Move Up: Moves the selected row or range of rows in the Component properties list, one position up. This button is available only if entire rows are selected.

Move Down: Moves the selected row or range of rows in the Component properties list, one position down. This button is available only if entire rows are selected.

Remove: Removes the selected row or range of rows from the Component properties list. This button is available only if entire rows are selected.

Parts list tab

Note: The Parts list tab is not visible if you are working in the GOST standard.

Component properties list: Displays the component properties that are visible in Parts Lists, by default.

Property: Displays the name of the component property.

Caption alignment: Defines the how the column heading are aligned in parts lists.

Value alignment: Defines the how the data rows are aligned in parts lists.

Width: Defines the column width for the property in the parts list.

Move Up: Moves the selected row or range of rows in the Component properties list, one position up. This button is available only if entire rows are selected.

Move Down: Moves the selected row or range of rows in the Component properties list, one position down. This button is available only if entire rows are selected.

Remove: Removes the selected row or range of rows from the Component properties list. This button is available only if entire rows are selected.

Sync

Synchronizes the component properties list with the component properties list in the BOM tab. Consequently, all component properties visible in the BOM will be made visible in parts lists.

Default style

Standard: Switches the parts list style to a standard based parts list.



Displays a dialog box, which enables you to customize settings applicable to standard based parts lists.

Custom



Switches the parts list style to a custom parts list .The list shows all blocks that can be used as custom parts lists. If the blocks to use are in an external drawing, select Browse (last entry in the list).

Balloon tab

Category: Displays different categories of references or functions that you can use in the expression for balloon text. This option is not available for custom balloons (because the expression to use for balloon text is defined within the block used for the custom balloon).

Select: Displays a list of references that belong to the selected category or a list of functions, as the case may be.

Add: Inserts the reference selected in the Select list to the Expression box. The reference is inserted at the insertion point within the Expression box. If there is no explicit insertion point, the reference is appended to the end of any existing expression.

Expression: Defines the formula to be used for balloon text.

Default style

Standard: Switches the appearance of the balloon to that defined by the drafting standard.

Displays a dialog box, which enables you to customize settings applicable to standard based balloon.

Custom: Switches the appearance of the balloon to a custom style defined via a block. If the block to use is in an external drawing, select Browse (last entry in the list).

Restore Defaults

Resets all settings to the default settings for current drafting standard.

Configure the BOM

Configuring the BOM is the foundation for configuring parts list and balloons. Configure the following options for the BOM:

- Configure which component properties are visible in the BOM.

When you add a component property to a parts list or to a part reference, the property is added to the BOM automatically. You can add component properties that parts lists do not usually use, directly to the BOM. For example, you can add component properties for downstream consumption by product lifecycle management tools.

When you remove a component property from the BOM, the property disappears from parts lists or part references as well.

- Change the order in which component properties appear in the BOM.

The order in which component properties appear in the BOM Settings dialog box is the order in which they appear (as BOM columns) in the BOM. Use the Move Up and Move Down buttons in the BOM tab of the BOM Settings dialog box to change the order.

Commonly Steps

To Add a Component Property to the BOM

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click BOM. The BOM Settings dialog box is displayed.
3. In the Available component properties list, click the row heading of the component property to add to the BOM. The program selects the entire row.
4. Click Add to. The program adds the component property to the bottom of the list in the BOM tab.
Note: The Add to button is unavailable if the selected component property already appears in the list.
5. Use the Move Up or Move Down buttons to position the property.
6. Click OK.

To Remove a Component Property from the BOM

1. At the Command prompt, enter GMOPTIONS.

2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click BOM. The BOM Settings dialog box is displayed.
3. In the BOM tab, click the row heading of the component property to delete. The program selects the entire row.
4. Click Remove.
5. Click OK.

To Change the Order Component Properties Appear in the BOM

1. At the Command prompt, enter GMOPTIONS.
2. In “IM:Standards Summary” dialog box, in the Standard Elements list, double-click BOM. The BOM Settings dialog box is displayed.
3. In the BOM tab, move the rows around to obtain the desired order.
4. To move a row:
 - Click the row heading of the row to move. The program selects the entire row.
 - Click Move Up or Move Down, as appropriate.
5. Click OK.

Set Up Standard Based Parts Lists

When you select the drafting standard, the default settings for parts lists change to match that drafting standard. If necessary, you can customize these settings to suit your drafting requirements. Configuring a parts list involves:

Specifying which columns appear in parts lists.

- In parts lists, each column corresponds to a component property, so each column behaves in accordance to the settings of the component property. For example, the caption, data type, and precision set in the component properties are the same for the corresponding parts list column.
- Note: When you add a component property to a parts list, it appears in the BOM automatically.

Formatting parts list columns.

- After you add columns to a parts list, you can set the width of each column. You can also specify the alignment of the column header and the data rows.
- The order in which component properties appear in the Parts List tab of the BOM Settings dialog box is the order in which the columns appear in parts lists. Use the Move Up and Move Down buttons to change the order in which columns appear.

Setting up the layout of parts lists.

- You can set the visibility of the parts list title and column heading. You can specify whether they appear at the top or at the bottom of the parts list. You can set up the row and column margins for the heading row and data

rows separately. You can select whether a grip frame appears, enabling grip editing in the drawing area.

Configuring parts list text options.

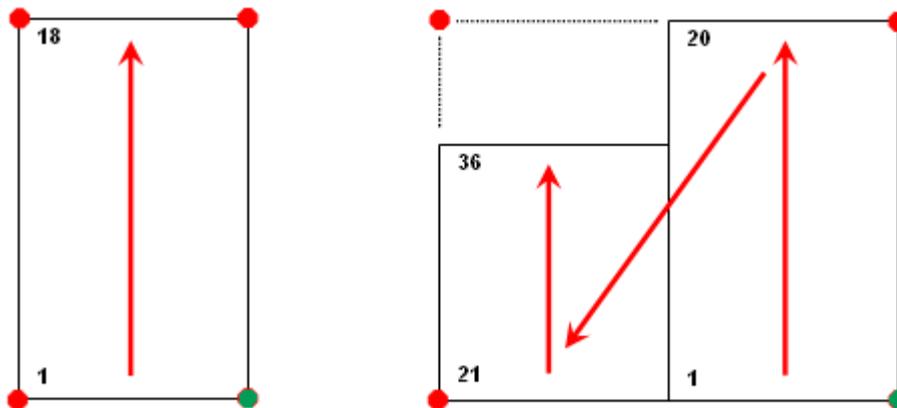
- Because text and leader lines are the most common objects used for annotations, GstarCAD Mechanical provides a single configuration point (the Standard Settings dialog box) enabling you to specify their default properties. These settings are the master settings.
- When the text size for the parts list is "By NameOfCurrentDraftingStandard," it acquires the base height master setting automatically. If you modify the master settings, the text size for parts lists automatically updates. If you do not want the text size for parts lists to follow the master settings, you can override the setting to a specific value.
- You can also configure how text appears within a parts list column. You can fit the text within the column, wrap it to the next line, or let it flow to the next column.

Specifying the location of the heading row.

- The system enables you to configure where the insertion point for parts lists appears. If the insertion point is at the top left or top right, the heading row is at the top of the parts list. If the insertion point is at the bottom left or bottom right, the heading row is at the bottom.

Enabling column splitting.

- Column splitting is the ability of parts list columns to wrap when the number of data rows exceeds a configurable threshold value. The following image shows one parts list with 18 data rows and another with 36. Column splitting occurs when the number of rows exceeds the threshold value of 20. The insertion point appears as a green dot.



- Instead of specifying the threshold value, you can define the number of sections to which the columns must split. In this case the data rows split evenly among the number of sections you define.
- We recommend that you configure parts lists before you start drawing. Parts list follow the options you set. If you change the part list configuration settings after you create part lists, only text height settings apply to them. None of the other settings apply to existing parts lists.

Commonly Steps

To Specify Which Columns to Display in Parts Lists

1. At the Command prompt, enter GMOPTIONS.

2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Parts List. The BOM Settings dialog box is displayed.
3. To add a column to parts lists:

In the Available component properties list, click the row heading of the component property to add as a Parts List column. The program selects the entire row.

Click Add to. The program adds the component property to the bottom of the list in the Parts List tab.

Note: The Add to button is unavailable if the selected component property already appears in the list.
4. Use the Move up or Move down buttons to position the property.
5. To remove a parts list column:

In the Parts List tab, click the row heading of the component property to remove.

Click Remove.
6. Click OK until all dialog boxes close.

To Change the Column Order in Parts Lists

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Parts List. The BOM Settings dialog box is displayed.
3. In the Parts List tab, Use the Move up or Move down buttons to position the property.
4. Click OK until all dialog boxes close.

To Format Parts List Columns

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Parts List. The BOM Settings dialog box is displayed.
3. To set the column width, enter the width in the Parts List tab, in the Width column.
4. To set the heading row caption alignment, click in the Caption alignment column and select the alignment from the drop-down list.
5. To set the data row alignment, click in the Value alignment column and select the alignment from the drop-down list.
6. Click OK until all dialog boxes close.

To Set Up the Table Layout for a Parts List

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Parts List. The BOM Settings dialog box is displayed.

3. In the Parts List tab, click .
4. To insert the column headings and title at the top of the table, in the Attach Point list, select Top Left or Top Right. To insert the heading and the title at the bottom of the table, select Bottom Right or Bottom Left.
5. To insert headings for each part list column, in the Layout section, select the Insert heading check box. You can override this setting while inserting a parts list.
6. To insert a title line for the parts list, in the Layout section, select the Insert Title check box. You can override this setting while inserting a parts list.
7. To include a grip frame to display the extents of the parts list, in the Layout section select the Show Grip Frame check box.
8. To set the spacing between the heading row and the body rows, in the Internal margins section, set the Heading Top margin or the Bottom margin.
9. To set the spacing between the body rows, in the Internal margins section, set the Data Top margin and/or the Bottom margin as required.
10. Click OK until all dialog boxes close.
11. Note: Changes to parts list table layout have no impact on parts lists that exist in a drawing.

To Configure Text Settings for Parts Lists

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Parts List. The BOM Settings dialog box is displayed.
3. In the Parts List tab, click .
4. To change the text height of part list column captions, in the Text section, below the Header row, from the Height list, select the text height.

Note: To acquire the base height setting from the master text settings, select the text height marked "ByNameOfCurrentDraftingStandard".
5. To change the text height of the data rows, in the Text section, below the Data row, from the Height list, select the text height.
6. To change the text color of part list column captions, in the Text section, below the Header row, from the Color list, select the color.
7. To change the text color of the data rows, in the Text section, below the Data row, from the Color list, select the color.
8. To specify how text works with columns:
9. To resize text to fit the column width, in the Text section, in the appropriate Control list, select Fit.
10. To wrap text into multiple lines, in the Text section, in the appropriate Control list, select Wrap.
11. Click OK until all dialog boxes close.

To Specify the Location of the Heading Row

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Parts List. The BOM Settings dialog box is displayed.
3. In the Parts List tab, click .
4. To insert the column headings and title at the top of the table, in the Attach Point list, select Top Left or Top Right.
5. To insert the heading and the title at the bottom of the table, select Bottom Right or Bottom Left.
6. Click OK until all dialog boxes close.
7. Note:Changes to heading row location have no impact on parts lists that exist in a drawing.

To Enable Column Splitting on Parts Lists

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Parts List. The BOM Settings dialog box is displayed.
3. In the Parts List tab, click .
4. Select the Enable column split check box.
5. To split the column based on the number of data rows, select the Number of rows option and enter the row number to split.
6. To split the column by distributing the data rows evenly among multiple sections, select the Number of sections option and enter the number of sections to split to.
7. Click OK until all dialog boxes close.
8. Note:Changes to heading row location have no impact on parts lists that exist in a drawing.

To Set a Custom Parts List as the Default Parts List Style

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Parts List. The BOM Settings dialog box is displayed.
3. In the Parts List tab, in the Default style section, select Custom. The drop-down list next to Custom becomes available.
4. In the drop-down list, select the name of the custom parts list scheme. To use a custom parts list scheme from an external file, select Browse and open the external file.
5. Click OK until all dialog boxes close.

Parts List Settings Dialog Box

Use this dialog box to customize the default settings for parts lists.

Revision

Displays the name and the revision of the standard that controls the default settings of parts lists.

Insertion

Insertion point: Specifies the location of the base point for parts lists. This also controls where the title of the parts list appears. If the insertion point is at the bottom, the headings and title are displayed at the bottom of the parts list and the parts are listed from bottom to top. Similarly, if the insertion point is at the top, the headings and title are displayed at the top of the parts list and the parts are listed from top to bottom.

Note: If you select the option labeled "By NameOfStandard", the insertion point is linked to the List Attach Point setting of the master settings of the current drafting standard. If the master settings change, the insertion point automatically follows the change.

Text

Controls the default text settings for parts list heading and body rows.

Header

Style: Defines the text style to be used for the heading row.

Control: Defines how text fits within a column, if the length of the text exceeds the column width. The fit option compresses text to fit within one line. The wrap option wraps text to the next line. The None option lets text flow to the next column.

Height: Defines the text height to be used for the heading row.

Note: If you select the text height labeled "By NameOfStandard", the text height is linked to the base height setting in the master settings of the drafting standard. If the master settings change, the header text size changes accordingly.

Color: Defines the text color to be used for the heading row.

Note: If you select the color labeled "By NameOfStandard", the text color is linked to the base text color setting in the master settings of the drafting standard. If the master settings change, the text color of the header row changes accordingly.

Data

Style: Defines the text style to be used for the data rows.

Control: Defines how text fits within a column, if the length of the text exceeds the column width. The fit option compresses text to fit within one line. The wrap option wraps text to the next line. The None option lets text flow to the next column.

Height: Defines the text height to be used for data rows.

Note: If you select the text height labeled "By NameOfStandard", the text height is linked to the base height setting in the master settings of the drafting standard. If the master settings change, the data row text size changes accordingly.

Color: Defines the text color to be used for the data rows.

Note: If you select the color labeled "By NameOfStandard", the text color is linked to the base text color setting in the master settings of the drafting standard. If the master settings change, the text color of the data row changes accordingly.

Spacing: Defines the gap between two data rows.

Internal Margins

Header

Sets the amount of white space for the heading row.

Top: Sets the distance from the top of the row to the top of the text.

Bottom: Sets the distance from the bottom of the row to the bottom of the text.

Left: Sets the distance from the left edge of a column to the left edge of left aligned text.

Right: Sets the distance from the right edge of a column to the right edge of right aligned text.

Data

Sets the amount of white space for the data rows.

Top: Sets the distance from the top of the row to the top of the text.

Bottom: Sets the distance from the bottom of the row to the bottom of the text.

Left: Sets the distance from the left edge of a column to the left edge of left aligned text.

Right: Sets the distance from the right edge of a column to the right edge of right aligned text.

Layout

Insert header: Controls whether a header row appears in the parts lists. The header row displays the column headings of the parts lists columns.

Insert title: Controls whether parts lists have a title bar. The title bar displays the name of the parts list.

Show grip frame: Controls whether the grip frame of the parts list appears. The grip frame is a frame that shows the actual extent of the parts list, which, under certain conditions, may be larger than the visible extent of the parts list.

Bold style: Sets the line style on title, header and summation rows to bold.

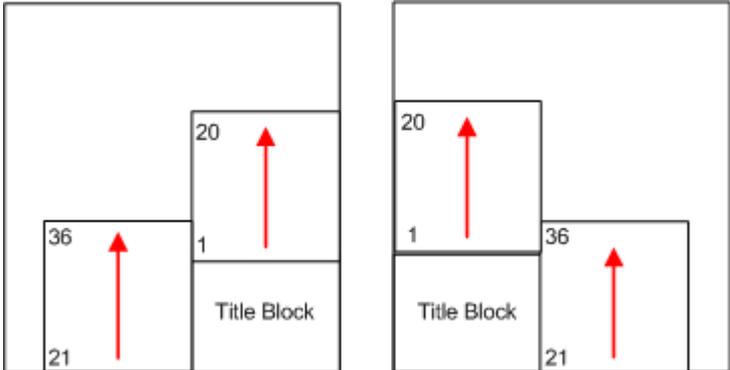
Filter empty references: Controls if body blocks (of a custom parts list) should be hidden, if it contains no data.

Column splitting

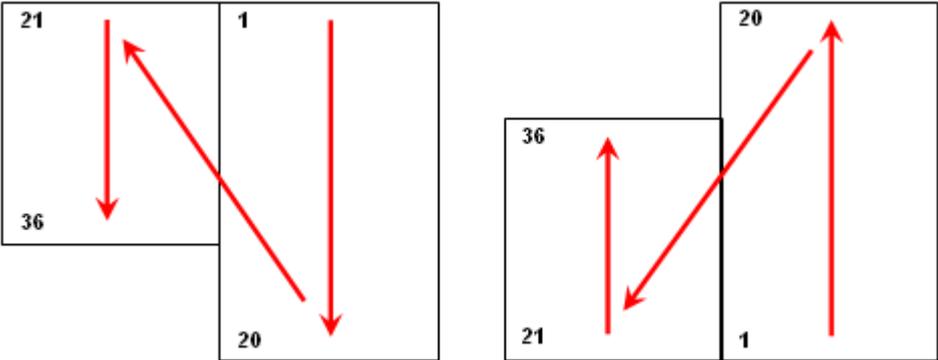
Contains options to configure if column splitting is to be performed and if so, how. Column splitting refers to the ability for the entire parts list to wrap as the number of data rows grow.

Enable column splitting: Turns on column splitting.

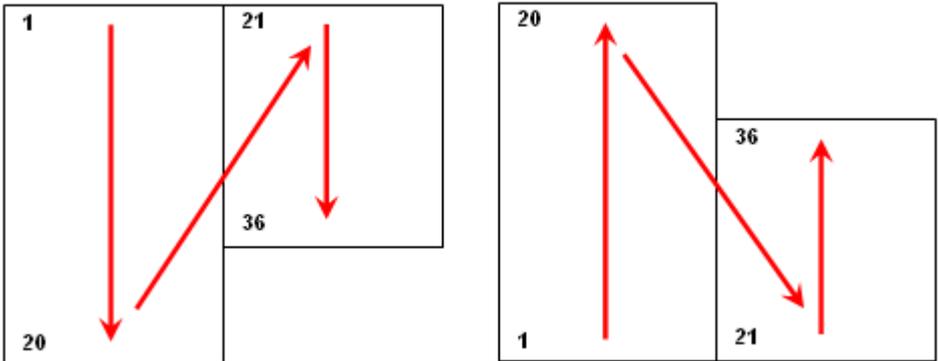
Wrap round title block: Wraps the parts list around the title block as in the following image. You must specify the title block before inserting the part list.



Wrap left: Wraps the parts list to the left as shown below.



Wrap right: Wraps the parts list to the right as shown below.



Number of rows: Performs split after the number of rows exceeds the number you specify in the adjacent box.

Number of sections: Splits the parts list onto the specified number of sections and distributes the entries evenly.

Restore Defaults

Resets all settings to the default settings for the current drafting standard.

Set Up Standard Based Balloon

When you select the drafting standard, the default settings for balloons change to match that drafting standard. If necessary, you can customize these settings to suit your drafting requirements. The following configuration operations are available:

- Select the default balloon type.

For any selected drafting standard, the system supports several balloon types that comply with it. It selects one automatically as the default balloon type. You can change the default balloon type.

- Set up the balloon size.

GstarCAD Mechanical enables you to select a size factor for balloons. It calculates the balloon size by multiplying the balloon text height by the size factor.

- Configure balloon text and leaders.

Because text and leader lines are the most common objects used for annotations, AutoCAD Mechanical provides a single configuration point (the Standard Settings dialog box) enabling you to specify their default properties. These settings are the master settings.

In the Balloon Settings dialog box, the text and leader settings default to "By NameOfCurrentDraftingStandard." If you modify the master settings, balloons update automatically. If you do not want balloons to follow the master settings, select a specific setting.

- Configure what to display in balloons.

Although it is customary to display the item number in balloons, you can display the value of any component property in a balloon.

- Configure how the program spaces balloons.

Select the vertical and horizontal distance between balloons. The system uses this setting when it aligns balloons.

During a drawing session, you can override most settings made at configuration time. If you change a balloon setting during a drawing session, the changes apply to all balloons except ones for which you override that setting (the program retains overrides).

Commonly Steps

To Select the Default Balloon Type

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Balloon. The BOM Settings dialog box is displayed.
3. In the Default style section, click . The Balloon Settings dialog box appears.

4. In the Balloon list, select the balloon type.

Note: If the appropriate balloon type is not visible, use the scroll bar to bring it into view.

5. Click OK until all dialog boxes close.

To Set Up the Balloon Size

1. At the Command prompt, enter GMOPTIONS.

2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Balloon. The BOM Settings dialog box is displayed.

3. In the Default style section, click . The Balloon Settings dialog box appears.

4. In the Balloon size factor box, specify how much larger than the balloon text the balloon must be.

Note: The balloon size is determined by multiplying the balloon text height with the size factor.

5. Click OK until all dialog boxes close.

To Configure Leaders for Balloons

1. At the Command prompt, enter GMOPTIONS.

2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Balloon. The BOM Settings dialog box is displayed.

3. In the Default style section, click . The Balloon Settings dialog box appears.

4. In the Arrowhead list, select the arrowhead.

5. In the Color list, select the color.

Note: To acquire the arrowhead shape and leader color from master settings, select the list item marked "ByNameOfCurrentDraftingStandard".

6. Click OK until all dialog boxes close.

To Configure Text Height and Color for Balloons

1. At the Command prompt, enter GMOPTIONS.

2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Balloon. The BOM Settings dialog box is displayed.

3. In the Default style section, click . The Balloon Settings dialog box appears.

4. To change the text height, in the Text Section, in the Height list, select the text height.

Note: To derive the text height from the master text settings automatically, select the text height marked "ByNameOfCurrentDraftingStandard".

5. To change the text color, in the Text Section, in the Color list, select the color.

6. Click OK until all dialog boxes close.

To Configure how Balloons are Distributed

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Balloon. The BOM Settings dialog box is displayed.
3. In the Default style section, click . The Balloon Settings dialog box appears.
4. In the Horizontal spacing box, specify the horizontal distance to maintain when balloons are aligned automatically.
5. In the Vertical spacing box, specify the vertical distance to maintain when balloons are aligned automatically.
6. Click OK until all dialog boxes close.

To Set a Custom Balloon as the Default Balloon Style

1. At the Command prompt, enter GMOPTIONS.
2. In "IM:Standards Summary" dialog box, in the Standard Elements list, double-click Balloon. The BOM Settings dialog box is displayed.
3. In the Balloon tab, in the Default style section, select Custom. The drop-down list next to Custom becomes available.
4. If the block to use as the custom balloon is in the current drawing, select the name of the block in the drop-down list.
5. If the block is in another file:
6. In the drop-down list, select Browse.
7. Open the file that contains the blocks. The names of the blocks appear in the drop-down list.
8. In the drop-down list, select the name of the block.
9. In the Default style section, click . The Balloon Settings dialog box is displayed.
10. In the Leader connection section, select the position of the leader connection.
11. Click OK until all dialog boxes close.

Balloon Settings Dialog Box

Use this dialog box to configure the default settings for balloons.

Revision

Displays the name and the revision of the standard that controls the default settings of balloons.

Balloon

Provides options for setting the appearance for standard based balloons.

Balloon type: Displays a palette of predefined standard based balloons. The current selection is boxed. Click a balloon to set it as the default.

Balloon size factor: Specifies how many times larger than the balloon text a balloon is.

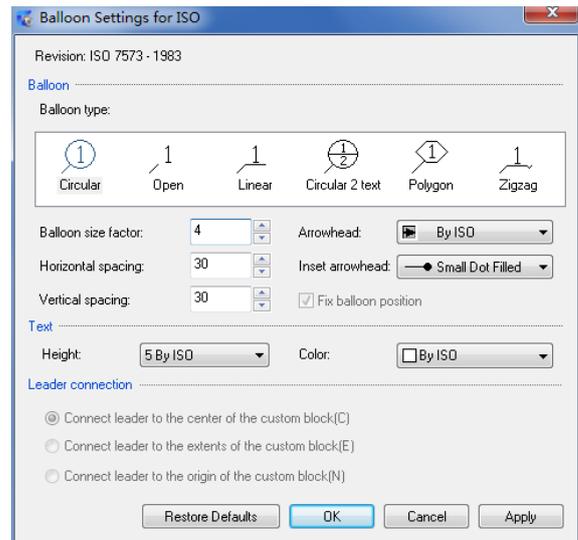
Horizontal spacing: Sets the horizontal distance (between the centers of two adjacent balloons) that must be maintained when balloons are automatically aligned.

Vertical spacing: Sets the vertical distance (between the centers of two adjacent balloons) that must be maintained when balloons are automatically aligned.

Arrowhead: Sets the arrowhead to be used for the leader termination. A palette containing standard compliant arrows appears to assist you select the desired arrowhead.

Inset Arrowhead: Sets the arrowhead to be used for leaders that do not terminate on object lines or the part reference icon (⊗). A palette containing standard compliant arrows appears to assist you select the desired arrowhead.

Fix balloon position: Locks balloons in the place you place them such that they won't move when the objects they are attached to are moved.



Text

Provides options to set the text height and color for standard based balloons.

Height: Sets the default text height for balloon text.

Note: If you select the text height labeled "By NameOfStandard", the balloon text height is based on the base height setting in the master settings of the drafting standard. When this option is selected, the balloon text height is always one size larger than the base height setting. If the master settings change, the text height setting for balloons change accordingly

Color: Sets the default color for balloon text.

Note If you select the color labeled "By NameOfStandard", balloon text color is linked to the color corresponding to the "By NameOfStandard" text height.

Leader Connection

Defines where on a custom balloon the leader connects to it. This option is not available if the current balloon style is a standard based balloon.

Restore Defaults

Resets all settings to the default settings for the current drafting standard.

Create a Customized Drawing Border

GstarCAD Mechanical 2017 contains a set of predefined boards in line with standards in different countries. It could quickly create a drawing with standard board. Double-clicking the board could modify its size. Users could customize the board according to their own enterprise standard.

To Create a Customized Drawing Border

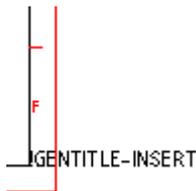
1. At the Command prompt, enter OPEN.
2. Select the file containing the drawing border closest to the one you need.

Note: The program stores drawing borders in the Format subfolder of the folder containing title border files.

3. Save the file under another name.
4. Change the border (change color, add objects, and so on).

Note: If necessary, you can add attributes. When you insert the border (through the GMTITLE command), the program prompts you to specify the attributes.

5. Save and close the DWG file.
6. In the file to add the new border to, add this file to the list of available drawing borders.



Note: GMTITLE uses keywords to define the insertion point for the title block and for the available vacant drawing surface. To calculate these points, GMTITLE searches in the inserted border block for the text! GENTITLE-INSERT or the attribute INSERT that defines the insertion point of the title block. Therefore, when you create a drawing border, it is necessary to enter! GENTITLE-INSERT into the border with the MTEXT command.

Customize Title Block

GstarCAD Mechanical 2017 contains a set of predefined titles in line with standards in different countries. It could quickly create a drawing with standard title block. Double-clicking the board could change the title block style. Users could customize the tile according to their own enterprise standard.

To Customize Title Block

1. At the Command prompt, enter OPEN.
2. Select the file containing the title block closest to the one you need.

Note: The program stores title blocks in the Title subfolder of the folder containing title border files (D:\Gstarsoft\GstarCAD Mechanical 2017\Dwg\Title).

3. Save the file with another name in the same folder.

4. From the View menu, select Zoom> Zoom All. The title block appears. Static text (title block captions such as the text "File Name and variable text (title block entries such as the file name) are shown as tags.

Illustration of an ISO title block

<small>Igenmsg"gentitis"60}{22.7}</small> GEN-TITLE-DWG{13.6}	<small>Igenmsg"gentitis"68}{15.7}</small> GEN-TITLE-FSCM{8.7}	<small>Igenmsg"gentitis"64}{19.2}</small> GEN-TITLE-SHEET{11.8}	<small>Igenmsg"gentitis"62}{18.9}</small> GEN-TITLE-SCA{12.7}
<small>Igenmsg"gentitis"43}{14.5}</small> GEN-TITLE-SIZ{22.6}	<small>Igenmsg"gentitis"41}{14.5}</small> GEN-TITLE-DAT{7.1}		
<small>Igenmsg"gentitis"43}{14.5}</small> GEN-TITLE-NAME{14.9}	<small>Igenmsg"gentitis"41}{14.5}</small> GEN-TITLE-DACT{21}		
<small>Igenmsg"gentitis"43}{14.5}</small> GEN-TITLE-CHKM{14.9}	<small>Igenmsg"gentitis"41}{14.5}</small> GEN-TITLE-DES1{12.3}		
<small>Igenmsg"gentitis"46}{15.5}</small> GEN-TITLE-APPM{14.9}	<small>Igenmsg"gentitis"41}{14.5}</small> GEN-TITLE-DES2{24.5}		
<small>Igenmsg"gentitis"47}{15.5}</small> GEN-TITLE-ISSM{14.9}	<small>Igenmsg"gentitis"41}{14.5}</small> GEN-TITLE-ISSD{7.1}		
<small>Igenmsg"gentitis"90}{15.5}</small> GEN-TITLE-REV{22.6}	<small>Igenmsg"gentitis"61}{40.4}</small> GEN-TITLE-NR{12.3}		
<small>Igenmsg"gentitis"69}{15.5}</small> GEN-TITLE-CTR{19.4}			

5. To edit a static text, double-click the static text and type the new caption.

Note: Although default title block captions are implemented as tags (for example, Igenmsg "gentitis60{22.7}), when you insert your own captions, they do not need to be inserted as tags. Type the text to display.

Illustration of a custom title block

<small>FILE NAME</small> GEN-TITLE-DWG{13.6}	<small>SHEET</small> GEN-TITLE-SHEET{11.8}	<small>SCALE</small> GEN-TITLE-SCA{12.7}
<small>SIZE</small> GEN-TITLE-SIZ{22.6}	<small>Igenmsg"gentitis"41}{14.5}</small> GEN-TITLE-DACT{21}	
<small>DRAWN</small> GEN-TITLE-NAME{14.9}	<small>Igenmsg"gentitis"41}{14.5}</small> GEN-TITLE-DAT{7.1}	
<small>CHECK</small> GEN-TITLE-CHKM{14.9}	<small>Igenmsg"gentitis"41}{14.5}</small> GEN-TITLE-DES1{12.3}	
<small>Igenmsg"gentitis"43}{14.5}</small> GEN-TITLE-CHKD{7.1}	<small>Igenmsg"gentitis"41}{14.5}</small> GEN-TITLE-DES2{24.5}	
<small>ISSUED</small> GEN-TITLE-ISSM{14.9}	<small>DWG NO</small> GEN-TITLE-NR{12.3}	
<small>Igenmsg"gentitis"47}{15.5}</small> GEN-TITLE-ISSD{7.1}		

6. To edit a variable text:
 - 1) Double-click the variable text. The Edit attributes dialog box is displayed.
 - 2) In the Tag box, type the name of the title block attribute, ending it with a pair of curly brackets.
 - 3) Between the curly brackets, type the width to height ratio of the variable text. For example, if the text height is 5 units, and the available width is 100 units, enter 20 between the curly brackets.
 - 4) In the prompt box, type the caption for this variable text.

Note: Although default title block captions are implemented as tags (for example, Igenmsg "gentitis60{22.7}), custom captions do not need to be inserted as tags. Type the text to display.
 - 5) In the default box, type the default value for the variable text.
 - 6) Click OK
7. Use the MOVE and ERASE commands to reorganize entries as required.
8. Save and close the file.
9. In the file you want to add the title block to, add this file to the list of available title blocks.

Set Up Custom Parts Lists

Customize the shape of the parts lists by creating a custom parts list using blocks.

Custom parts lists require at least two blocks: one block for the parts list heading row, and another for the body row. If necessary, you can have several blocks for the body row. The heading block and the body blocks must have the same name, with a digit at the end to differentiate them.

The heading block usually contains a frame with a heading text or attribute inside. The name of the heading block ends with the digit 0. For example MYPLIST0 is a valid name for the heading block.

The body blocks become a template for one parts list row. The name of the first body block must end with the digit 1, the second body block must end with the digit 2, and so on. For example, MYPLIST1 and MYPLIST2 are valid body block names for the heading block MYPLIST0.

Body blocks usually contain the frame for a row and attributes that indicate which component property they show. Set the tag name of the attribute to the name of the property to show. For example, if the Body block shows the component property Material, name the attribute tag MATERIAL.

One limitation of using the component property name as the attribute tag is that there is no mechanism to fit text within the space reserved for it. For example, if you use DESCRIPTION as the tag name and reserve a column width of 35 for it, if the description of an item in the parts lists exceeds 35, it will overflow to the next column. To work around this issue, use the internal representation.

The internal representation takes the form #GENST{#}, where #GENST is the Component Property ID and {#} is a number to indicate text fit. For example, the Component Property ID for DESCRIPTION is 5GENST (See the topic Component Property Id Reference for full list). If the available space for the description is 35 and the text height is 3.5, divide the available space (35) by the text height (3.5), enter the resulting value (10) between curly brackets, and append it to the Component Property ID (5GENST{10}, in this case). If the description of an item exceeds 35 in width, the text compresses to fit into the available space.

You can create the heading block and the body blocks in the current drawing or in an external drawing. If you use an external drawing, the drawing must have the same name as the blocks. For example, the file containing the parts list blocks MYPLIST0, MYPLIST1, and MYPLIST2 must be saved as myplist.dwg.

After you create the blocks for the custom parts list, you can configure the system to use the custom parts list as the default parts list style. You can also set up whether the heading row appears at the top of the parts list or at the bottom and set up column splitting.

Additional, the system enables you to keep standard style parts lists as well as custom style parts lists in the same drawing. You can select the parts list style when you create the parts list.

To Create a Block Scheme for Custom Parts Lists

1. Create a block for the heading row. To create the block:

In the drawing area, create the frame and heading text for the heading row.

Use the BLOCK command to convert the geometry and text into a block.

Note: The block name must end with the digit 0.

2. Create blocks for the parts list data rows. To create the blocks:
 - 1) In the drawing area, create the geometry for the parts list row pattern.
 - 2) Use the ATTDEF command to add attributes to display row data.

Note:

Place the attributes at the position you intend to display the component.

The tags of the attributes must be the component property name or the component property ID (See the topic Component Property Id Reference for a full list of component).

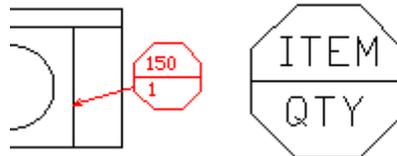
3. Use the BLOCK command to convert the geometry and attributes into blocks.

Note:

- The body blocks must have the same name as the heading block, but must end with a digit other than 0.
 - The name of the first body block must end with the digit 1, the second with the digit 2, and so on.
4. In the drawing area, move the blocks to obtain the desired appearance of a data row.

Set Up Custom Balloons

GstarCAD Mechanical 2017 gives you the ability to set up balloons of custom shape using blocks and attributes.



To create a custom shaped balloon, you must create the custom shape and then, in the place where you want the balloon text to appear, you must insert an attribute.

To create a custom shaped balloon, you first create the balloon geometry (the custom shape) and then, in the position you want the balloon text to appear, you must place an attribute. The tag name of the attribute must be the same as the name of the component property you want to show on the balloon. You can insert more than 1 attribute, if required. Finally you must create a block and add the custom shape balloon geometry and the attributes to it.

You can create blocks for custom balloons in the current drawing or in an external drawing. You can set the custom balloon as the default balloon style.

You can have standard-based balloons and custom balloons in the same drawing. In the BOM Settings dialog box, set the default balloon style to the style you intend to use for the majority of balloons. After you create balloons, you can edit them and change the balloon style to a standard-based balloon or custom balloon.

To Create Blocks for Custom Balloons

1. Create a geometry for the custom balloon.

2. Click Home tab ► Block panel drop-down ► Define Attributes.
3. Place the attributes at the position you want the balloon text displayed. The tags of the attributes must be the same as the component property you want the balloon to display.
4. Use the BLOCK command to convert the geometry and attributes to a block.

Appendix: GstarCAD Mechanical 2017 Command Table

Name	Command	Description
About GstarCAD Mechanical	GMABOUT	Display information about the product.
Angle Line	GMANGLELINE	Draws an angle line in the drawing area.
Calculate Area	GMAREA	Calculate the closed area in the drawing.
Automatic Construction Lines	GMAUTOCLINES	Creates vertical and horizontal construction lines on all endpoints of selected objects.
Multiple Dimensioning	GMAUTODIM	Creates multiple dimensions from a selected point to all the vertexes on the contour of selected objects.
Balloons	GMBALLOON	Creates and places balloons in the drawing area.
Collect Balloons	GMBALLOON C	Creates a balloon collection on a single leader.
Renumber Balloons	GMBALLOON R	Renumbers balloons that already exist in the drawing area.
Angle Bisector Line	GMBISECTOR	Draws a line that bisects an angle
BOM Database	GMBOM	Creates edits or delete Bills of Materials (BOMs).
Break	GMBREAKATPT	Breaks the selected object at one point.
Break Symbol	GMBREAKSYMBOL	Creates a break symbol with line in the drawing.
Center Mark	GMCEN	Creates the center mark or the center lines of circles and arcs.
Centerline Cross with Angles	GMCENCRANGLE	Creates concentric pitch circles and places holes on it at specified angles.
Centerline Cross in Corner	GMCENRCORNER	Places a hole at a specified distance from 2 contour lines.
Centerline Cross On Full Circle	GMCENCRFULLCIRCLE	Creates concentric pitch circles and places a specified number of holes on them, distributed uniformly.
Centerline Cross with Hole	GMCENCRHOLE	Places a hole at a specified point.
Centerline Cross in Holes	GMCENCRINHOLE	Places a centerline cross on a circle.
Centerline Cross	GMCENCROSS	Places a centerline cross at a specified point.
Centerline Cross on Plate	GMCENCRPLATE	Places holes at the corners of a closed contour that represents a plate.
Centerline In-between	GMCENINBET	Draws a centerline in between two lines to mark them as symmetrical.
Center Line	GMCENTERLINE	Draws a centerline at a specified place in the drawing area.
Center Line	GMCENTLINE	Draws a centerline at a specified place in the drawing area.
Chamfer	GMCHAM2D	Connects two non-parallel objects by extending or trimming them to intersect or to join with a beveled line.
Chamfer	GMCHAM2D_DIM	Creates dimensions for chamfers .
Check Dimension	GMCHECKDIM	Highlights or edits dimensions with dimension text overrides.
Center Mark	GMCIRCLEMARK	Creates the center mark or the center lines of circles and arcs.
Layers, Construction Lines Lock/Unlock	GMCLINEL	Locks or unlocks the construction line layer.

Common Tangent Line	GMCOMMONTANGENT	Draws 2 line that are tangential to 2 circles, arc, ellipse
Circle	GMCONST_CIRCLE	Draws a construction line circle.
Circle Tangent to Line	GMCONSTC2	Draws a construction line circle that uses a specified line as a tangent.
Concentric Circle	GMCONSTCC	Draws a construction line circle that is concentric to a specified circle.
Create Circle to end of Shaft	GMCONSTCCREA	Draws a construction line circle to represent a top view of a shaft or hole.
Rectangle on Circle	GMCONSTCIRCLI	Draws a construction line rectangle around a circle.
Cross	GMCONSTCRS	Draws a construction line cross.
2 points Or Angle	GMCONSTHB	Draws a construction line when you specify two points or a point and an angle.
Bi-sect	GMCONSTHM	Draws a construction line that bisects an angle.
Horizontal	GMCONSTHOR	Draws a horizontal construction line.
Relative Angle from Line	GMCONSTHW	Draws a construction line through a point, by specifying an angle relative to an apparent line that goes through that point.
Circle Tangent to 2-Lines	GMCONSTKR	Draws a construction line that is tangential to two specified circles.
Draw Construction Lines	GMCONSTLINES	Draws construction lines.
Perpendicular to Line	GMCONSTLOT	Draws a construction line that is perpendicular to a specified line.
Perpendicular to 2-Points	GMCONSTLOT2	Draws a construction line through a specified point that is a perpendicular to the direction specified by another point or angle.
Parallel with full Distance	GMCONSTPAR	Draws a construction line parallel to an existing line at a specified distance.
Parallel with half Distance	GMCONSTPAR2	Draws a construction line parallel to an existing line and bisects the distance between the selected line and a specified point or a specified distance.
Switch Ray / Xline	GMCONSTSWI	Switches between construction lines that extend to infinity in both directions or lines that extend in one direction only
Tangent	GMCONSTTAN	Constrains two curves to maintain a point of tangency to each other or their extensions
2-circle Tangent	GMCONSTTC	Draws 2 construction lines that are tangential to 2 specified circles.
Vertical	GMCONSTVER	Draws a vertical construction line.
Lines at Point	GMCONSTXLINE	Draws a construction line through a point, which extends to infinity in both directions.
Rays at Point	GMCONSTXRAY	Draws a construction line starting from a point and extends to infinity in one direction.
Z Direction	GMCONSTZ	Draws a construction line along the z axis.
Contour Inside	GMCONTIN	Creates a closed polyline that traces the inner contour of an enclosed area.
Continuous Copy	GMCONTINUOUSCOPY	It's stronger than copy of CAD, and it can remember the distance and direction.
Contour Outside	GMCONTOUT	Creates a closed polyline that traces the outer contour of an enclosed area.
Contour Trace	GMCONTRACE	Creates a closed contour that traces the contour of an area by letting you

		specify the boundaries segment by segment.
Statistical Block	GMCOUNTBLOCK	The function can statistic the block to be used and the result prints to the command line.
Super Card	GMCREATECARD	Creates a card.
Datum Identifier	GMDATUMID	Creates a datum identification symbol and attaches it to an object in the drawing area.
Datum Target	GMDATUMTGT	Creates a datum target symbol and attaches it to an object in the drawing area.
Detail	GMDetail	Create a scaled circular, rectangular, or free defined detail view of selected geometry.
Align Dimensions	GMDIMALIGN	Lines up linear, ordinate or angular dimensions, with a dimension you select as the baseline.
Arrange Dimension	GMDIMARRANGE	Rearranges linear and ordinate dimensions, placing them at an appropriate distance from the outer contour of an object you select.
Insert Dimension	GMDIMINSERT	Splits a linear or angular dimension into two individual dimensions.
Join Dimensions	GMDIMJOIN	Combines two individual dimensions (of the same type) into a single dimension.
Multi Edit	GMDIMMEDIT	Edits multiple dimensions simultaneously.
Linear/Symmetric Stretch	GMDIMSTRETCH	Resizes objects by stretching or shrinking a linear or symmetric dimension.
Dynamic Extensions	GMDYNGMICEXTEND	Extends an object to meet another object.
Edge	GMEDGESYM	Creates an edge symbol and attaches it to an object in the drawing area.
Erase all Construction Lines	GMERASEALLCL	Erases all construction lines.
Erase Construction Lines	GMERASECL	Erases selected construction lines.
Merge Overlapped	GMERASEREPETITION	Cleans up overlapping geometry by removing duplicated and/or unneeded objects.
Export mechanical drawings	GMEXPORT	Export mechanical drawings
Feature Control Frame	GMFCFRAME	Creates a feature control frame symbol and attaches it to an object in the drawing.
Feature Identifier	GMFEATID	Creates a feature identification symbol and attaches it to an object in the drawing area.
Fillet	GMFILLET2D	Rounds and fillets the edges of objects with an arc of a specified radius.
Add Rect Around	GMFRAMETEXT	Add rect around
Hatch 135 deg. 11 mm/0.4 inch	GMHATCH_135_11	Fills an enclosed area with a 135-degree, 11 mm/0.4" hatch.
Hatch 135 deg. 2.7 mm/0.12 inch	GMHATCH_135_2	Fills an enclosed area with a 135-degree, 2.7 mm/0.12" hatch.
Hatch 135 deg. 4.7 mm/0.19 inch	GMHATCH_135_4	Fills an enclosed area with a 135-degree, 4.7 mm/0.19" hatch.
Hatch 45 deg. 13	GMHATCH_45_13	Fills an enclosed area with a 45-degree, 13 mm/0.5" hatch.

mm/0.5 inch		
Hatch 45 deg. 2.5 mm/0.1 inch	GMHATCH_45_2	Fills an enclosed area with a 45-degree, 2.5 mm/0.1" hatch.
Hatch 45 deg. 5 mm/0.22 inch	GMHATCH_45_5	Fills an enclosed area with a 45-degree, 5 mm/0.22" hatch.
Double Hatch 45/135 deg. 2.3 mm / 0.09 inch	GMHATCH_DBL	Fills an enclosed area with a 45 and 135 degree, 2.3 mm/0.09 cross hatch.
Hole Chart	GMHOLECHART	Creates coordinate dimensions for holes in a work piece, dimensions the size of those holes, and generates a hole chart for that work piece.
Smart Line	GMINTELLIGENTLINE	Creates straight line segments by the smart way
Intersect	GMINTERSECT	Trims line or arc intersecting, and deletes superfluous sections.
Join Entities	GMJOIN	Connects parallel lines, polylines, arcs, and circles.
To 0 Layer	GMLAYER0	Moves objects from one layer to 0.
Thick Line Layer	GMLAYER1	Moves objects from one layer to Thick Line.
Thin Line Layer	GMLAYER2	Moves objects from one layer to Thin Line.
Centerline Layer	GMLAYER3	Moves objects from one layer to Centerline.
Dashed Layer	GMLAYER4	Moves objects from one layer to Dashed.
Hatch Layer	GMLAYER5	Moves objects from one layer to Hatch.
Text Layer	GMLAYER6	Moves objects from one layer to Text .
Dimensions Layer	GMLAYER7	Moves objects from one layer to Dimensions .
Symbols Layer	GMLAYER8	Moves objects from one layer to Symbols.
Divide Line Layer	GMLAYER9	Moves objects from one layer to Divide Line.
Definition Card	GMMAKECARDMODEL	Defines cards
Definition Form	GMMAKETABLEMODEL	Defines forms
One-quarter Mirror	GMMIRROR4	Creates three mirror image copy of objects.
Leader Note	GMNOTE	Creates a leader note and attaches it to an object in the drawing area.
Standards Configuration	GMOPTIONS	Sets a drafting standard
Parallel/Perpendicular	GMPARALLELLINE	Draws a line parallel to existing line at a specified distance
Part list	GMPARTLIST	Creates a parts list and enables you to place it in the drawing area.
Part Reference	GMPARTREF	Creates a part reference and places it in the drawing area.
Part Reference Edit	GMPARTREFEDIT	Edits a part reference.
Perp Bisector Line	GMPEPBISECTOR	Draws a line Vertical and split to existing line
Pipe Line	GMPIPELINE	Draws a pipe line in the drawing area
Power Dimensioning	GMPOWERDIM	Creates multiple types of dimensions within a single command session and provides the ability to specify tolerances or fits as appropriate.
Aligned	GMPOWERDIM_ALI	Creates aligned linear dimensions.
Angular	GMPOWERDIM_ANG	Creates angular dimensions.
Arc Length	GMPOWERDIM_ARC	Creates radius dimensions for arcs and circles.
Baseline	GMPOWERDIM_BAS	Creates a linear or angular dimension from the first extension line of a selected dimension.
Chain	GMPOWERDIM_CHAIN	Creates a linear, angular, or arc length dimension from the second extension line of a selected dimension.
Diameter	GMPOWERDIM_DIA	Creates diameter dimensions for arcs and circles.

Horizontal	GMPOWERDIM_HOR	Creates horizontal linear dimensions.
Jogged	GMPOWERDIM_JOG	Create a radius dimension with the origin of the dimension at any location you wish and a jog at a convenient location along the dimension line.
Radius	GMPOWERDIM_RAD	Creates a radius dimension for a circle or an arc.
Rotated	GMPOWERDIM_ROT	Creates rotated linear dimensions.
Vertical	GMPOWERDIM_VER	Creates vertical linear dimensions.
Super Edit	GMPOWEREDIT	Edits annotation, serial number and other entities.
Power Recall	GMPOWERRECALL	Recognizes the object you select and invokes the command that it was created with.
Projection OFF/ON	GMPROJO	Creates projection lines to assist you create orthographic views.
Radiation	GMRADIATION	Draws a radiation line in the drawing area
Rectangle	GMRECTANG	Draws a rectangle using a closed polyline.
Rename Block	GMRENAMEBLOCK	Renames the block.
Rescale	GMRESCALE	Enlarges or shrinks annotations, without impacting drawing geometry.
Revision On/Off	GMREV	Turns revisions on or off.
Add one Revision Line	GMREVLIN	Adds a revision line to a revisions list.
Update Revision Line	GMREVUPDATE	Updates the revision block.
Scale xy	GMSCALEXY	Sets scale of X and Y axes independently of each other .
Scale Area	GMSCAREA	Creates a scale area in model space.
Scale Monitor	GMSCMONITOR	Displays the scale of a scale area or viewport.
Section Line	GMSECTIONLINE	Draws a section line and inserts the corresponding section view label in the drawing area.
Truncation line	GMSECTIONSYPHOL	Creates truncation line.
Define Layer Set	GMSETLAYER	Sets layer of the enterprise.
Create Hide Situation	GMSHIDE	Draws hidden lines to represent hidden edges, when you specify what objects.
Edit Hide Situation	GMSHIDEEDIT	Edits a hide situation created with the GMSHIDE command.
Weld Representation	GMSIMPLEWELD	Draws seam and fillet welds on ellipses, circles, arcs, lines and polylines.
Card Editor	GMSUPERCARDEDIT	Edits the card.
Surface Texture	GMSURFSYM	Creates a surface texture symbol and attaches it to an object in the drawing area.
Add Leader	GMSYMLEADER ADD	Add a leader line to an existing multileader object.
Attach Object	GMSYMLEADER ATTACH	Attaches the symbol to an object in the drawing.
Detach Object	GMSYMLEADER DETACH	Detaches the symbol.
Remove Leader	GMSYMLEADER REMOVE	Removes a leader line from an existing multileader object.
Symmetrical Line	GMSYMLINE	Draws symmetrical polylines.
Tangent Line	GMTANGENTLINE	Snaps to a point on the phantom line that passes tangentially through an arc endpoint.
Taper and Slope	GMTAPERSYM	Creates a taper or slope symbol.
Technical Requirements	GMTECHREQUEST	Creates a technical requirement.

Text Annotation	GMTEXT	Creates text annotations
Drawing Title/Borders	GMTITLE	Inserts a drawing border with a title block.
Trace Contour	GMTRCONT	Traces contours on construction lines or construction line circles.
Text Underline	GMUNDERLINETEXT	Adds a line under the text.
Words cancel theory frame space	GMUNFRAMETEXT	Words cancel theory frame space
User-Defined Hatch	GMUSERHATCH	Fills an enclosed area with a user-defined hatch pattern.
Viewport	GMVPORT	Creates a viewport in layout mode or edits an existing viewport.
Wavyline	GMWAVILNESSLINE	Draws a Wave line in the drawing area.
Welding	GMWELDSYM	Creates a welding symbol and attaches it to the weld point.
Zigzag Line	GMZIGZAGLINE	Draws a zigzag line.
License Management	register	License Management
Construction Lines On/Off	(command "-LAYER" "OFF" "AM_CL" "")(princ)	Freezes or thaws construction line layers.
Construction Lines On/Off	(command "-LAYER" "ON" "AM_CL" "")(princ)	Freezes or thaws construction line layers.
Fasteners(ISO)	(command "GM_SPART_OUTN" "ISO Fasteners.ini")	Inserts fasteners from the Serial Part library, such as screw, bolt.
Parametric Library	GM_PARA_OUT	Inserts parts from the Parametric Library
Serial Part	GM_SPART_OUT	Inserts parts from the Serial Part library.