Alphabetical Listing of CorelCAD Commands and Functions

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.AddCmdPoint (CAD)

.AddCmdPoint .X=*double*, .Y=*double*, .Z=*double*

This command adds a command point to an existing object. The **.AddCmdPoint** command must be in a contiguous block of one or more **.AddCmdPoint** commands. The first **.AddCmdPoint** command in the block must be preceded by the **.StartAddCmdPoint** command, and the last must be followed by the **.EndAddCmdPoint** command.

Syntax	Description
.X	Specifies the X-coordinate for the command point in the document's default unit of measurement.
.Y	Specifies the Y-coordinate for the command point in the document's default unit of measurement.
.Z	Specifies the Z-coordinate for the command point in the document's default unit of measurement.

Note

- You can use the **.AddCmdPoint** command for an existing object with start and end points.
- If you are adding more than one command point, you must set the number of points in the **.StartAddCmdPoint** command.

Example

```
REM units in inches
.StartAddCmdPoint 3
.AddCmdPoint 20.0, 0.0, 0.0
.AddCmdPoint 25.0, 10.0, 0.0
.AddCmdPoint 40.0, 10.0, 0.0
.EndAddCmdPoint
```

{button ,AL(`CAD_AddCmdPoint_Menu;cad_point;;;;',0,"Defaultoverview",)} **Related Topics**

.AddLayerToLayerGroup (CAD)

.AddLayerToLayerGroup .LayerName=*string*, GroupName=*string*

This command adds a layer to a layer group.

Syntax	Description
.LayerName	Specifies the name of the existing layer.
.GroupName	Specifies the name of the layer group.

Note

- The layer name specified for **.LayerName** must not be part of the **.GroupName**.

Example
```
.AddLayerToLayerGroup "layer 2" "mygroup"
```

The above example adds the layer called "layer 2" to the layer group called "mygroup".

{button ,AL(`CAD_AddLayerToLayerGroup_Menu;cad_toolsmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.ApplyMaterial (CAD)

.ApplyMaterial .Category=*string*, .Material=*string*

This command applies materials to the selected object(s).

Syntax	Description
.Category	Specifies the material category as follows: Glass Jewel Metallic Mineral Miscellaneous Others Plastics Woods
.Material	Specifies the material. Click on the category to display a list of materials. Glass Jewel Metallic Mineral Miscellaneous Others Plastics Woods

Note

- This command corresponds to the Material Roll-Up command on the Tools menu. Click Tools, Material Roll-Up.
- You can only apply materials to solids and surfaces.

Example
```
REM units in inches
.Box -1, -33.2181, 14.6993, 0.0, 36.3049, -41.6133, 80.6979
.ApplyMaterial "Metallic", "Bronze"
```

The above example creates a box and then applys the Bronze material from the Metallic category.

{button ,AL(`CAD_ApplyMaterial_Menu;cad_toolsmenu;;;;',0,"Defaultoverview",)} Related Topics

.ApplyName (CAD)

.ApplyName .Name=*string*

This command applies a name to the selected object(s).

Syntax	Description
.Name	Specifies the name of the selected object.

Note
- Multiple objects can have the same name.
- An object must be selected before using this command.

Example
```
REM units in inches
.Box -1, -33.2181, 14.6993, 0.0, 36.3049, -41.6133, 80.6979
.SelectPointAt -25.633, -41.2995, 80.6979, 0
.ApplyName "Canada"
```

{button ,AL(`CAD_ApplyName_Menu;cad_name;;;;',0,"Defaultoverview",)} **Related Topics**

.Arc3Points (CAD)

.Arc3Points .SaveAsFlag=*Integer*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2 =*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command draws an arc through three points.

Syntax	Description
.SaveAsFlag	Specifies whether the arc is saved as a wire, surface center, or surface end points. Set to 0 to save the arc as a wire; set to 1 to save the arc as a surface center and set to 2 to save the arc as surface end points.
.x1	Specifies the X-coordinate for the first (starting point) of the arc in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first and (starting point) of the arc in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first and (starting point) of the arc in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point of the arc in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point of the arc in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point of the arc in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the third (end point) of the arc in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the third (end point) of the arc in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the third (end point) of the arc in the document's default unit of measurement.

Note
- This command corresponds to the Arc through 3 points command on the Arc flyout of the Draw menu. Click Draw, Arc, Arc through 3 points.

Example
```
REM units in inches
.Arc3Points 0, -13, 6.0, -9.0, 5.0, 0.0, -12.0, 2.0, 0.0
```

{button ,AL(`CAD_Arc3Points_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.ArcAngle (CAD)

.ArcAngle .SaveAsFlag=*Integer*, .angle=*double*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command draws an arc according to the specified angle and two points.

Syntax	Description
.SaveAsFlag	Specifies whether the arc is saved as a wire, surface center, or surface end points. Set to 0 to save the arc as a wire; set to 1 to save the arc as a surface center and set to 2 to save the arc as surface end points.
.angle	Specifies the angle of the arc in degrees.
.x1	Specifies the X-coordinate for the arc's center point in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the arc's center point in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the arc's center point in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the arc's starting point in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the arc's starting point in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the arc's starting point in the document's default unit of measurement.

Note

▪ This command corresponds to the Arc Center, Start, Angle command on the Arc flyout of the Draw menu. Click Draw, Arc, Arc Center, Start, Angle .

Example
```
REM units in inches
.ArcAngle 0, 60, -30.0, -10.0, 0.0, 0.0, -10.0, 0.0
```

{button ,AL(`CAD_ArcAngle_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.ArcCSE (CAD)

.ArcCSE .SaveAsFlag=*integer***, .x1=***double***, .y1=***double***, .z1=***double***, .x2=***double***, .y2=***double***, .z2=***double***, .x3=***double***, .y3=***double***, .z3=***double***

This command draws an arc determined by three points.

Syntax	Description
.SaveAsFlag	Specifies whether the arc is saved as a wire, surface center, or a surface end points. Set to 0 to save the arc as a wire; set to 1 to save the arc as a surface center and set to 2 to save the arc as surface end points.
.x1	Specifies the X-coordinate for the arc's center point in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the arc's center point in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the arc's center point in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the arc's start point in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the arc's start point in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the arc's start point in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the arc's end point in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the arc's end point in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the arc's end point in the document's default unit of measurement.

Note
- This command corresponds to the Arc Center, Start, End command on the Arc flyout of the Draw menu. Click Draw, Arc, Arc Center, Start, End.

Example
```
REM units in inches
.ArcCSE 2, -6.0, 1.5, 0.0, -4.0, 4.5, 0.0, -3.0, 3.5, 0.0
```

{button ,AL(`CAD_ArcCSE_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.ArcEllipse (CAD)

.ArcEllipse .SaveAsFlag=*Integer,* .x1=*double,* .y1=*double,* .z1=*double,* .x2=*double,* .y2=*double,* .z2=*double,* .x3=*double,* .y3=*double,* .z3=*double,* .x4=*double,* .y4=*double,* .z4=*double,* .x5=*double,* .y5=*double,* .z5=*double*

This command creates an ellipse arc.

Syntax	Description
.SaveAsFlag	Specifies whether the arc is saved as a wire, surface center, or surface end points. Set to the following: 0 to save the arc as a wire 1 to save the arc as a surface center 2 to save the arc as surface end points.
.x1	Specifies the X-coordinate of the ellipse arc's center point in the document's default units of measurement.
.y1	Specifies the Y-coordinate of the ellipse arc's center point in the document's default units of measurement.
.z1	Specifies the Z-coordinate of the ellipse arc's center point in the document's default units of measurement.
.x2	Specifies the X-coordinate for the first point used to determine the ellipse arc's radius in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the first point used to determine the ellipse arc's radius in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the first point used to determine the ellipse arc's radius in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the second point used to determine the ellipse arc's radius in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the second point used to determine the ellipse arc's radius in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the second point used to determine the ellipse arc's radius in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the start point of the ellipse arc in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the start point of the ellipse arc in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the start point of the ellipse arc in the document's default unit of measurement.
.x5	Specifies the X-coordinate for the end point of the ellipse arc in the document's default unit of measurement.
.y5	Specifies the Y-coordinate for the end point of the ellipse arc in the document's default unit of measurement.
.z5	Specifies the Z-coordinate for the end point of the ellipse arc in the document's default unit of measurement.

Note
- This command corresponds to the Elliptical Arc command on the Arc flyout of the Draw menu. Click Draw, Arc, Elliptical Arc.

Example
```
REM units in inches
.ArcEllipse 0, -10.0, 10.0, 0.0, 0.0, 10.0, 0.0, -10.0, 15.0, 0.0, 0.0, 10.0, 0.0,
-10.0, 20.0, 0.0
```

{button ,AL(`CAD_ArcEllipse_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.ArcRSE (CAD)

.ArcRSE .SaveAsFlag=*Integer*, .radius=*double*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2 =*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command draws an arc according to the specified radius, start point, and end point.

Syntax	Description
.SaveAsFlag	Specifies whether the arc is saved as a wire, surface center, or surface end points. Set to the following: 0 to save the arc as a wire 1 to save the arc as a surface center 2 to save the arc as surface end points.
.radius	Specifies the radius of the arc.
.x1	Specifies the X-coordinate of the arc's start point in the document's default unit of measurement.
.y1	Specifies the Y-coordinate of the arc's start point in the document's default unit of measurement.
.z1	Specifies the Z-coordinate of the arc's start point in the document's default unit of measurement.
.x2	Specifies the X-coordinate of the arc's end point in the document's default unit of measurement.
.y2	Specifies the Y-coordinate of the arc's end point in the document's default unit of measurement.
.z2	Specifies the Z-coordinate of the arc's end point in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the point which determines which side of the start and end points the arc is to be formed in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the point which determines which side of the start and end points the arc is to be formed in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the point which determines which side of the start and end points the arc is to be formed in the document's default unit of measurement.

Note
- This command corresponds to the Arc Radius, End Points on the Arc flyout of the Draw menu. Click Draw, Arc, Arc Radius, End Points.

Example
```
REM units in inches
.ArcRSE 1, 5,-9.0, 2.0, 0.0, -4.5, 3.5, 0.0, -3.0, -8.0, 0.0
```

{button ,AL(`CAD_ArcRSE_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.ArrowLine (CAD)

.ArrowLine .ArrowSize=*double*, .ArrowHead=*integer*

This command creates a line through preset points with an arrow on the last point.

Syntax	Description
.ArrowSize	Specifies the size of the arrowhead.
.ArrowHead	Sets the type of arrowhead to one of four types (1, 2, 3, 4).

Note

- You must precede this command with either **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint** or **.SetPointXYZ**.
- This command corresponds to the Arrow Line command on the Line flyout of the Draw menu. Click Draw Line, Arrow Line.

Example
```
REM units in inches
.SetPointXYZ -12.0, 4.5, 0.0
.SetPointXYZ -2.0, 1.0, 0.0
.SetPointXYZ -0.5, 0.5, 0.0
.ArrowLine 10, 4

REM units in inches
.StartAddCmdPoint 4
.AddCmdPoint -85.3, 52.3, 0.0
.AddCmdPoint -37.4, 45.5, 0.0
.AddCmdPoint -27.0, 1.0, 0.0
.AddCmdPoint 6.3, 6.8, 0.0
.EndAddCmdPoint
.ArrowLine 10, 4
```

The first example creates an arrow using the **.SetPointXYZ** command. The second example uses **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint**.

{button ,AL(`CAD_Arrow_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.BezierSegment (CAD)

.BezierSegment .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*, .x4=*double*, .y4=*double*, .z4=*double*

This command creates a Bezier line segment.

Syntax	Description
.x1	Specifies the X-coordinate for the Bezier line segment's starting point in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the Bezier line segment's starting point in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the bezier line segment's starting point in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the Bezier line segment's second point in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the Bezier line segment's second point in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the Bezier line segment's second point in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the Bezier line segment's third point in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the Bezier line segment's third point in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the Bezier line segment's third point in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the Bezier line segment's end point in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the Bezier line segment's end point in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the Bezier line segment's end point in the document's default unit of measurement.

Note

- This command corresponds to the Bezier Segments command on the Curve flyout of the Draw menu. Click Draw, Curve, Bezier Segments.

Example
```
REM units in inches
.BezierSegment -15.0, 6.0, -7.5, 2.0, 0.0, -7.5, -8.0, 0.0, 0.0, -10.0, 0.0
```

{button ,AL(`CAD_BezierSegemnt_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.BezierCurve (CAD)

.BezierCurve

This command creates a Bezier curve object. You must precede this command with the **.SetPointXYZ** and/or the **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint** commands to create the points for the Bezier curve.

Example
```
REM units in inches
.StartAddCmdPoint 4
.AddCmdPoint 0.0, -10.0, 0.0
.AddCmdPoint 5.0, 0.0, 0.0
.AddCmdPoint 20.0, 0.0, 0.0
.AddCmdPoint 20.0, 0.0, 0.0
.EndAddCmdPoint
.SetPointXYZ -10.0, -10.0, 0.0
.SetPointXYZ -10.0, 0.0, 0.0
.BezierCurve
```

{button ,AL(`CAD_BezierCurve_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.Box (CAD)

.Box .SaveAsSolid=*Boolean***, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*
, *x3=double, y3=double, z3=double, x4=double, y4=double, z4=double*

This command creates a box.

Syntax	Description
.SaveAsSolid	Specifies whether the box is saved as a solid or a surface. Set to TRUE (-1) to save the box as a solid; set to FALSE (0) to save the box as a surface. The default setting is TRUE.
.x1	Specifies the X-coordinate for the starting point of the box in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the starting point of the box in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the starting point of the box in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point (opposite corner for a 2 point box, width for a three point box) in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point (opposite corner for a 2 point box, width for a three point box) in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point (opposite corner for a 2 point box, width for a three point box) in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the third point (length of a 3 point box) of the box in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the third point (length of a 3 point box) in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the third point (length of a 3 point box) in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the fourth point (depth of a 3 point box) of the box in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the fourth point (depth of a 3 point box) in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the fourth point (depth of a 3 point box) in the document's default unit of measurement.

Note
- The third and fourth points are optional. If only 2 points are specified, the box is a 2 point box. If all four points are specified, the box is a 3 point box.
- .SaveAsSolid is optional. If not specified, it is set to TRUE.

Example
```
REM units in inches
.Box -1, -14.5, 4.0, -1.5, -13.5, 3.0, -2.5
.Box -1, -15.0, -5.5, -10.0, 7.5, -5.5, -11.0, 4.0, -5.5, -11.0, 4.0, -10.0
```

The first example creates a 2 point box, and the second example creates a 3 point box.

{button ,AL(`CAD_Box_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.Chamfer (CAD)

.Chamfer .firstDistance=*double*, .secondDistance=*double*, .x1=*double*, .y1=*double*, .z1=*double*, .x2 =*double*, .y2=*double*, .z2=*double*

This command chamfers 2D line corners.

Syntax	Description
.firstDistance	Specifies the first distance in inches.
.secondDistance	Specifies the second distance in inches.
.x1	Specifies the X-coordinate for the point located on the first of two adjacent lines in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point located on the first of two adjacent lines in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point located on the first of two adjacent lines in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point located on the second of two adjacent lines in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point located on the second of two adjacent lines in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point located on the second of two adjacent lines in the document's default unit of measurement.

Note
- You must have drawn a 2D object before using this command.
- This command corresponds to the Chamfer command on the Fillet flyout of the Transform menu. Click Transform, Fillet, Chamfer.

Example
```
REM units in inches
.Rectangle -1, -59.688, 49.6767, -29.5, -52.157, 45.463, -29.5
.Chamfer 1, 1, -54.9577, 45.3767, -29.5, -52.2194, 46.8948, -29.5
```

The example above creates a rectangle and then chamfers a corner.

{button ,AL(`CAD_Chamfer_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} __Related Topics__

.ChamferEdge (CAD)

.ChamferEdge .firstDistance=*double*, .secondDistance=*double*, x1=*double*, x1=*double*, z1=*double*, x2=*double*, y2=*double*, z3=*double*

This command chamfers the edge of the 3D object.

Syntax	Description
.firstDistance	Specifies the first distance in inches.
.secondDistance	Specifies the second distance in inches.
.x1	Specifies the X-coordinate for the point on the edge to be chamfered in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point on the edge to be chamfered in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point on the edge to be chamfered in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point which determines the face of the object to be associated with the first distance in the document's default unit of measurement. The second point is used only if the first and second distances are **not** equal.
.y2	Specifies the Y-coordinate for the point which determines the face of the object to be associated with the first distance in the document's default unit of measurement. The second point is used only if the first and second distances are **not** equal.
.z2	Specifies the Z-coordinate for the point which determines the face of the object to be associated with the first distance in the document's default unit of measurement. The second point is used only if the first and second distances are **not** equal.

Note
- This command corresponds to the Chamfer Edge command on the Fillet flyout of the Transform menu. Click Transform, Fillet, Chamfer Edge.

Example
```
REM units in inches
.Box -1, -34.0, 61.0, -58.5, -29.0, 57.5, -62.0
.ChamferEdge 3, 3, -32.5, 65.0, -62.0
.Box -1, -34.0, 61.0, -58.5, -29.0, 57.5, -62.0
.ChamferEdge 3, 2, -32.5, 61.5, -62.0, -32.5, 61.5, -62.0
```

The first example has the first and second distances equal and only one point. In the second example, the first and second distances are not equal and the command requires two points.

{button ,AL(`CAD_ChamferEdge_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.ChangeColor (CAD)

.ChangeColor .Red=*integer*, .Green=*integer*, .Blue=*integer*

This command changes the color of a selected object(s).

Syntax	Description
.Red	Specifies the command's red setting (RGB color model). Valid values range from 0 to 255.
.Green	Specifies the command's green setting (RGB color model). Valid values range from 0 to 255.
.Blue	Specifies the command's blue setting (RGB color model). Valid values range from 0 to 255.

Note

- You must have selected an object before using this command.

Example

```
.ChangeColor 255, 0, 0
```

This example sets the color to red.

{button ,AL(`CAD_ChangeColor_Menu;;;;;',0,"Defaultoverview",)} **Related Topics**

.Circle3Points (CAD)

.Circle3Points .SaveAsSurface=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command creates a circle based on the coordinates of three points.

Syntax	Description
.SaveAsSurface	Specifies whether the circle is saved as a surface or a wire. Set to TRUE (-1) to save the circle as surface; set to FALSE (0) to save as a wire.
.x1	Specifies the X-coordinate for the first point of the circle in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point of the circle in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point of the circle in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point of the circle in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point of the circle in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point of the circle in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the third point of the circle in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the third point of the circle in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the third point of the circle in the document's default unit of measurement.

Note

- This command corresponds to the 3 Points command on the Circle flyout of the Draw menu. Click Draw, Circle, 3 Points.

Example
```
REM units in inches
.Circle3Points -1, -18.0, 26.0, -23.0, -14.5, 24.0, 23.0, -15.0, 23.5, -23.0
```

{button ,AL(`CAD_Circle3Points_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.CircleDiameter (CAD)

.CircleDiameter .SaveAsSurface=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command creates a circle based on diameter.

Syntax	Description
.SaveAsSurface	Specifies whether the circle is saved as a surface or a wire. Set to TRUE (-1) to save the circle as surface; set to FALSE (0) to save as a wire.
.x1	Specifies the X-coordinate for the first point of the circle's diameter in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point of the circle's diameter in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point of the circle's diameter in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point of the circle's diameter in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point of the circle's diameter in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point of the circle's diameter in the document's default unit of measurement.

Example
```
REM units in inches
.CircleDiameter TRUE, -18.0, 26.0, -23.0, -14.5, 24.0, 23.0
```

{button ,AL(`CAD_CircleDiameter_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.CircleRadius (CAD)

.CircleRadius .SaveAsSurface=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command creates a circle based on the center point and the radius.

Syntax	Description
.SaveAsSurface	Specifies whether the circle is saved as a surface or a wire. Set to TRUE (-1) to save the circle as surface; set to FALSE (0) to save as a wire.
.x1	Specifies the X-coordinate of the circle's center in the document's default unit of measurement.
.y1	Specifies the Y-coordinate of the circle's center in the document's default unit of measurement.
.z1	Specifies the Z-coordinate of the circle's center in the document's default unit of measurement.
.x2	Specifies the X-coordinate of the circle's radius in the document's default unit of measurement.
.y2	Specifies the Y-coordinate of the circle's radius in the document's default unit of measurement.
.z2	Specifies the Z-coordinate of the circle's radius in the document's default unit of measurement.

Note
- This command corresponds to the Center and Radius command on the Circle flyout of the Draw menu. Click Draw, Circle, Center and Radius.

Example
```
REM units in inches
.CircleRadius -1, -18.0, 22.5, -23.0, -14.0, 21.5, -23.0
```

{button ,AL(`CAD_CircleRadius_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.CircularArray (CAD)

.CircularArray .Copies=*long*, .angle=*double*, .RotateObjects=*Boolean*, .x1=*double*, .y1=*double*, .z1= *double*, .x2=*double*, .y2=*double*, .z2=*double*, x3=*double*, y3=*double*, z3=*double*

This command duplicates an object(s) along the path of a circle.

Syntax	Description
.Copies	Specifies the number of copies.
.angle	Specifies the angle of the circle (in degrees) the objects will cover when duplicated. For example, if 180 degrees is specified, the copies of the object(s) are placed on the circle clockwise from 0 to 180 degrees creating a semicircle. If -180 degrees is specified, the copies of the object(s) are placed on the circle counterclockwise from 0 to -180 degrees. This will create a semicircle on the bottom.
.RotateObjects	Specifies whether the object(s) is rotated. Set to TRUE (-1) to rotate the object; set to FALSE (0) to not rotate the object. If **.RotateObjects** is set to TRUE, the number of points required is one. The second point is optional. If **.RotateObjects** is set to FALSE, the number of points required is two and the third point is optional.
.x1	If **.RotateObjects**=TRUE, specifies the X-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
	If **.RotateObjects**=FALSE, specifies the X-coordinate for the base point in the document's default unit of measurement.
.y1	If **.RotateObjects**=TRUE, specifies the Y-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
	If **.RotateObjects**=FALSE, specifies the Y-coordinate for the base point in the document's default unit of measurement.
.z1	If **.RotateObjects**=TRUE, specifies the Z-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
	If **.RotateObjects**=FALSE, specifies the Z-coordinate for the base point in the document's default unit of measurement.
.x2	If **.RotateObjects**=TRUE, specifies the X-coordinate for the point that defines the direction of the axis in the document's default unit of measurement. If you do not specify the second point, it is assumed to be perpendicular to the current view. For example, if you are in Top View, the second point is perpendicular to the z axis.
	If **.RotateObjects**=FALSE, specifies the X-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
.y2	If **.RotateObjects**=TRUE, specifies the Y-coordinate for the point that defines the direction of the axis in the document's default unit of measurement. If you do not specify the second point, it is assumed to be perpendicular to the current view. For example, if you are in Top View, the second point is perpendicular to the z axis.
	If **.RotateObjects**=FALSE, specifies the Y-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
.z2	If **.RotateObjects**=TRUE, specifies the Z-coordinate for the point that defines the direction of the axis in the document's default unit of measurement. If you do not specify the second point, it is assumed to be perpendicular to the current view. For example, if you are in Top View, the second point is perpendicular to the z axis.
	If **.RotateObjects**=FALSE, specifies the Z-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the point that defines the direction of the axis in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the point that defines the direction of the axis in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the point that defines the direction of the axis in the document's default unit of measurement.

Note

- You must have drawn an object before using this command.
- If **.RotateObjects**=FALSE and you do not select a base point (**.x1**, **.y1**, **z1**) on the body of the object, the object is rotated relative to the base point.
- This command corresponds to the Circular Array command on the Array flyout of the Edit menu. Click Edit, Array, Circular Array.

Example
```
REM units in inches
.Box -1, -36.0, 64.0, -64.0, -34.5, 63.0, -65.5
.CircularArray 20, 360, -1, -30.0, 65.0, -65.5, -32.0, 74.0, -65.5
.Box -1, -36.0, 64.0, -64.0, -34.5, 63.0, -65.5
.CircularArray 20, 360, 0, -36.0, 63.0, -65.5, -29.5, 64.0, -65.5, -31.5, 72.5, -65.5
```

These two examples illustrate the difference in the parameters when **.RotateObjects** is set to TRUE (first example) and when **.RotateObjects** is set to FALSE (second example). In the first example, there are 2 points. In the second example, there are 3 points.

{button ,AL(`CAD_CircularArray_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} Related Topics

.CircularSweep (CAD)

.CircularSweep .Copies=*integer*, .angle=*double*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .SweepType=*integer*

This command sweeps a 2D object around an axis in a circle creating a 3D object

Syntax	Description
.Copies	Specifies the number of copies of the object.
.angle	Specifies the angle of rotation in degrees.
.x1	Specifies the X-coordinate for the first point defining the axis in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point defining the axis in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point defining the axis in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point defining the axis in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point defining the axis in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point defining the axis in the document's default unit of measurement.
.SweepType	Specifies the sweep type. Set to 0 for Continuous, set to 1 for Discrete.

Note
- You must select an object before using this command.
- The object selected must be a 2D object.
- This command corresponds to the Circular Sweep command on the Extrude flyout of the Draw menu. Click Draw, Extrude, Circular Sweep.

Example
```
REM units in inches
.Rectangle -1, -33.0, 68.0, -67.5, -30.0, 66.5, -67.5, -31.0, 65.0, -67.5
.CircularSweep 20, 360, -27.5, 68.0, -67.5, -27.5, 72.0, -67.5
```

The above example creates a rectangle and then sweeps 20 copies of the rectangle in a circle 360 degrees around the axis specified by the two points.

{button ,AL(`CAD_CircularSweep_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.CombineLines (CAD)

.CombineLines .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command combines two lines into one line.

Syntax	Description
.x1	Specifies the X-coordinate for the point that selects the first line in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point that selects the first line in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point that selects the first line in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point that selects the second line in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point that selects the second line in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point that selects the second line in the document's default unit of measurement.

Note

- The endpoints of the two lines must be at the same location.
- This command corresponds to the Combine command on the Join Lines flyout of the Transform menu. Click Transform, Join Lines, Combine.

Example

```
REM units in inches
.SetPointXYZ -105.169, 33.0232, 0.0
.SetPointXYZ -47.8601, -16.9682, 0.0
.LineSegment
.SetPointXYZ -47.8601, -16.9682, 0.0
.SetPointXYZ -39.5749, 53.667, 0.0
.LineSegment
.CombineLines -69.3919, 2.25158, 0.0, -44.2062, 13.0848, 0.0
```

The above example creates two lines with end points at the same location and then combines the lines.

{button ,AL(`CAD_CombineLines_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.Cone (CAD)

.Cone .SaveAsSolid=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command creates a cone based on the coordinates of the three points.

Syntax	Description
.SaveAsSolid	Specifies whether the cone is saved as a solid or a surface. Set to TRUE (-1) to save the cone as a solid; set to FALSE (0) to save the cone as a surface.
.x1	Specifies the X-coordinate for the center of the cone's base in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the cone's base in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the cone's base in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the edge of the cone in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the edge of the cone in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the edge of the cone in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the height of the cone in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the height of the cone in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the height of the cone in the document's default unit of measurement.

Note
- This command corresponds to the Cone, Center and Radius command on the Solids flyout of the Draw menu. Click Draw, Solids, Cone, Center and Radius.

Example
```
REM units in inches
.Cone -1, -30.5, 68.5, -67.5, -28.5, 69.0, -67.5, -31.5, 78.0, -67.5
```

{button ,AL(`CAD_Cone_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.ConeDiameter (CAD)

.ConeDiameter .SaveAsSolid=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command creates a cone based on the diameter set by two points and the height set by the third point.

Syntax	Description
.SaveAsSolid	Specifies whether the cone is saved as a solid or a surface. Set to TRUE (-1) to save the cone as a solid; set to FALSE (0) to save the cone as a surface.
.x1	Specifies the X-coordinate for the first point that determines the cone's diameter in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that determines the cone's diameter in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that determines the cone's diameter in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that determines the cone's diameter in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that determines the cone's diameter in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that determines the cone's diameter in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the height of the cone in the document's default unit of measurement.
.y3	Specifies the Z-coordinate for the height of the cone in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the height of the cone in the document's default unit of measurement.

Example
```
REM units in inches
.ConeDiameter 0, -29.0, 71.5, -67.5, 26.0, 72.5, -67.5, -28.0, 78.0, -67.5
```

{button ,AL(`CAD_ConeDiameter_Menu;cad_solid;;;;',0,"Defaultoverview",)} Related Topics

.CountObject (CAD)

ReturnValue& = .CountObject .Name=*string*

This command counts all of the objects with the specified name and returns the number of objects.

Syntax	Description
ReturnValue&	Specifies the numeric variable that is passed the return value corresponding to the number of objects. This variable must be declared using a <u>DIM</u> statement or implicitly declared using a <u>type-declaration suffix</u>.
.Name	Specifies the name of the objects to count.

Note
- You must have named objects before using this command.

Example
```
REM units in inches
.Torus -1, -9.0, -0.5, 0.0, -7.0, -2.0, 0.0, -8.0, -1.0, 0.0, -7.5, -1.5, 0.0
.SelectPointAt -9.0, -0.5, 0.0
.ApplyName "Torusname"
CountTs& = .CountObject "Torusname"
```

This example creates a torus, selects a point on the torus and applies a name. The torus is then scaled; and a the scaled version is placed in a different location. A pyramid is drawn and finally all objects with name "Torusname" are selected.

The variable **CountTs** is passed the number of objects named "Torusname".

{button ,AL(`CAD_CountObject_Menu;cad_name;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.Curve (CAD)

.Curve

This command creates a curve object.

Note
- This command must be preceded by the **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint** or **.SetPointXYZ**.
- This command corresponds to the Curve command on the Curve flyout of the Draw menu. Click Draw, Curve, Curve.

Example
```
REM units in inches
.StartAddCmdPoint 4
.AddCmdPoint 0.0, -10.0, 0.0
.AddCmdPoint 5.0, 0.0, 0.0
.AddCmdPoint 20.0, 0.0, 0.0
.AddCmdPoint 20.0, 0.0, 0.0
.EndAddCmdPoint
.SetPointXYZ -10.0, -10.0, 0.0
.SetPointXYZ -10.0, 0.0, 0.0
.Curve
```

{button ,AL(`CAD_Curve_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.Cylinder (CAD)

.Cylinder .SaveAsSolid=*Boolean***, .x1=***double***, .y1=***double***, .z1=***double***, .x2=***double***, .y2=***double***, .z2=***double***, .x3=***double***, .y3=***double***, .z3=***double***

This command creates a cylinder.

Syntax	Description
.SaveAsSolid	Specifies whether the cylinder is saved as a solid or a surface. Set to TRUE (-1) to save the cylinder as a solid; set to FALSE (0) to save the cylinder as a surface.
.x1	Specifies the X-coordinate for the center of the cylinder in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the cylinder in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the cylinder in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the radius of the cylinder in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the radius of the cylinder in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the radius of the cylinder in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the plane (height) of the cylinder in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the plane (height) of the cylinder in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the plane (height) of the cylinder in the document's default unit of measurement.

Note
- This command corresponds to the Cylinder, Center and Radius command on the Solids flyout of the Draw menu. Click Draw, Solids, Cylinder, Center and Radius.

Example
```
REM units in inches
.Cylinder -1, -7.0, 1.5, 0.0, 3.5, 1.5, 0.0, -3.0, -6.0, 0.0
```

{button ,AL(`CAD_Cylinder_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.CylinderDiameter (CAD)

.CylinderDiameter .SaveAsSolid=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command creates a cylinder.

Syntax	Description
.SaveAsSolid	Specifies whether the cylinder is saved as a solid or a surface. Set to TRUE (-1) to save the cylinder as a solid; set to FALSE (0) to save the cylinder as a surface.
.x1	Specifies the X-coordinate for the first point that determines the diameter of the cylinder in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that determines the diameter of the cylinder in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that determines the diameter of the cylinder in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that determines the diameter of the cylinder in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that determines the diameter of the cylinder in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that determines the diameter of the cylinder in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the plane (height) of the cylinder in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the plane (height) of the cylinder in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the plane (height) of the cylinder in the document's default unit of measurement.

Example
```
REM units in inches
.CylinderDiameter -1, -10.0, 5.0, 0.0, -7.5, 4.5, 0.0, -8.5, -1.0, 0.0
```

{button ,AL(`CAD_CylinderDiameter_Menu;cad_solid;;;;',0,"Defaultoverview",)} **Related Topics**

.DeleteSelection (CAD)

.DeleteSelection

This command deletes the selected objects.

Note
- You must have selected objects before using this command.
- If you want to cut an object and place it on the clipboard, use **.EditCut**.

Example
```
REM units in inches
.Box -1, -14.5, 4.0, -1.5, -13.5, 3.0, -2.5
.Cylinder -1, -7.0, 1.5, 0.0, 3.5, 1.5, 0.0, -3.0, -6.0, 0.0
.SelectWithinRegion -16.0, 4.0, 0.0, 4.0, -9.5, 0.0
.DeleteSelection
```

If you create a script identical to the example, you will not be able to see the result. The objects will have been created, selected and deleted. To see how the command works, you should create two scripts. The first one containing the **.Box** and **.Cylinder** commands and the second one containing the **.SelectWithinRegion** and **.DeleteSelection** commands.

_____

{button ,AL(`CAD_DeleteSelection_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.DimensionAngle (CAD)

.DimensionAngle . .Dynamic=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*, .x4=*double*, .y4=*double*, .z4=*double*

This command draws an angular dimension line determined by the position of four points

Syntax	Description
.Dynamic	Specifies whether the dimension line is dynamic or static. Set to TRUE (-1) to make the dimension line dynamic, set to FALSE (0) to make the dimension line static.
.x1	Specifies the X-coordinate for the first point that, with points 2 and 3, determines the angle value in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that, with points 2 and 3, determines the angle value in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that, with points 2 and 3, determines the angle value in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that, with points 1 and 3, determines the angle value in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that, with points 1 and 3, determines the angle value in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that, with points 1 and 3, determines the angle value in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the third point that, with points 1 and 2, determines the angle value in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the third point that, with points 1 and 2, determines the angle value in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the third point that, with points 1 and 2, determines the angle value in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the point that determines the dimension line's location in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the point that determines the dimension line's location in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the point that determines the dimension line's location in the document's default unit of measurement.

Note
- This command corresponds to the Angle command on the Dimension menu. Click Dimension, Angle.

Example
```
REM units in inches
.DimensionAngle -1, -8.5, 5.0, 0.0, -4.0, 3.0, 0.0, -7.5, 1.0, 0.0, -7.5, 2.5, 0.0
```

{button ,AL(`CAD_DimensionAngle_Menu;cad_dimenmenu;;;;',0,"Defaultoverview",)} Related Topics

.DimensionBaseline (CAD)

.DimensionBaseline .Dynamic=*Boolean*, .Direction=*integer*

This command draws baseline dimension lines with spans of up to 200.

Syntax	Description
.Dynamic	Specifies whether the dimension line is dynamic or static. Set to TRUE (-1) to make the dimension line dynamic; set to FALSE (0) to make the dimension line static.
.Direction	Specifies the direction of the baseline dimension. 0 x direction 1 y direction 2 z direction

Note

▪ Command points have to be set in advance using **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint**. The last point specifies the location of the text.

▪ This command corresponds to the Continuous Baseline command on the Dimension menu. Click Dimension, Continuous Baseline.

Example
```
REM units in inches
.StartAddCmdPoint 3
.AddCmdPoint -11.0, 5.0, 0.0
.AddCmdPoint -6.0, 2.5, 0.0
.AddCmdPoint -6.0, -1.0, 0.0
.EndAddCmdPoint
.DimensionBaseline -1, 0
```

The above example sets three command points and then creates a baseline dimension line. The text is displayed at the last point.

{button ,AL(`CAD_DimensionBaseline_Menu;cad_dimenmenu;;;;',0,"Defaultoverview",)} Related Topics

.DimensionContinuous (CAD)

.DimensionContinuous .Dynamic=*Boolean*, .Direction=*integer*

This command draws continuous dimension lines.

Syntax	Description
.Dynamic	Specifies whether the dimension line is dynamic or static. Set to TRUE (-1) to make the dimension line dynamic; set to FALSE (0) to make the dimension line static.
.Direction	Specifies the direction of the continuous dimension. 0 x direction 1 y direction 2 z direction

Note
- Command points have to be set in advance using **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint**. The last point specifies the location of the text.
- This command corresponds to the Continuous Linear command on the Dimension menu. Click Dimension, Continuous Linear.

Example
```
REM units in inches
.StartAddCmdPoint 3
.AddCmdPoint -9.5, 5.5, 0.0
.AddCmdPoint -5.0, 3.5, 0.0
.AddCmdPoint -5.0, -2.0, 0.0
.EndAddCmdPoint
.DimensionContinuous -1,0
```

The above example sets three command points and then creates a Continuous dimension line. The text is displayed at the last point.

{button ,AL(`CAD_DimensionContinuous_Menu;cad_dimenmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.DimensionDiameter (CAD)

.DimensionDiameter .Dynamic=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command draws a dimension line along the diameter of a circle.

Syntax	Description
.Dynamic	Specifies whether the dimension line is dynamic or static. Set to TRUE (-1) to make the dimension line dynamic; set to FALSE (0) to make the dimension line static.
.x1	Specifies the X-coordinate for the point lying on the circle edge in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point lying on the circle edge in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point lying on the circle edge in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point that determines the location of the text in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point that determines the location of the text in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point that determines the location of the text in the document's default unit of measurement.

Note
- You must have drawn a circle before using this command.
- This command corresponds to the Diameter command on the Dimension menu. Click Dimension, Diameter

Example
```
REM units in inches
.CircleDiameter 0, -13.0, 12.0, -8.0, -9.0, 7.5, -12.5
.DimensionDiameter -1, -9.0, 7.5, -12.5, -8.0, 7.0, -12.5
```

This command creates a dimension line along the diameter of the circle. You do not have to use **.CircleDiameter** command to create the circle. Any of the circle commands or CorelCAD tools can be used.

{button ,AL(`CAD_DimensionDiameter_Menu;cad_dimenmenu;;;;',0,"Defaultoverview",)} Related Topics

.DimensionLinear (CAD)

.DimensionLinear .Dynamic=*Boolean*, .Direction=*integer*, .x1=*double*, .y1=*double*, .z1=*doueble*, .x2 =*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command draws linear dimension lines.

Syntax	Description
.Dynamic	Specifies whether the dimension line is dynamic or static. Set to TRUE (-1) to make the dimension line dynamic; set to FALSE (0) to make the dimension line static.
.Direction	Specifies the direction of the linear dimension. 0 x direction 1 y direction 2 z direction 3 freehand 4 ortho
.x1	Specifies the X-coordinate for the first point that determines the length of the dimension line in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that determines the length of the dimension line in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that determines the length of the dimension line in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that determines the length of the dimension line in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that determines the length of the dimension line in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that determines the length of the dimension line in the document's default unit of measurement.
.x3	Specifies the X-coordinate that determines the location of the dimension line in the document's default unit of measurement.
.y3	Specifies the Y-coordinate that determines the location of the dimension line in the document's default unit of measurement.
.z3	Specifies the Z-coordinate that determines the location of the dimension line in the document's default unit of measurement.

Note
- This command corresponds to the Free, X Linear, Y Linear, Z Linear and Ortho commands on the Dimension menu. The different settings for the **.Direction** parameter equate to the commands on the menu.

Example
```
REM units in inches
.Box -1, -14.0, 15.5, -12.5, -10.0, 13.0, -16.5
.DimensionLinear -1, 3, -14.0, 15.5, -12.5, -14.0, 15.5, -14.5, -17.0, 15.0, -14.5
```

This example creates a linear dimension line. The **.Direction** parameter is set to Free (3) and corresponds to the Free command on the Dimension menu.

```
.Box -1, -14.0, 15.5, -12.5, -10.0, 13.0, -16.5
.DimensionLinear -1, 0, -12.0, 15.5, -12.5, -12.5, -9.5, 18.0, -12.5
```

The **.Direction** parameter is set to X (0) and corresponds to the X Linear command on the Dimension menu.

{button ,AL(`CAD_DimensionLinear_Menu;cad_dimenmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.DimensionRadius (CAD)

.DimensionRadius .Dynamic=*Boolean*, **.x1=***double*, **.y1=***double*, **.z1=***double*, **.x2=***double*, **.y2=***double* **, .z2=***double*

This command draws a dimension line along the radius of a circle.

Syntax	Description
.Dynamic	Specifies whether the dimension line is dynamic or static. Set to TRUE (-1) to make the dimension line dynamic; set to FALSE (0) to make the dimension line static.
.x1	Specifies the X-coordinate for the point lying on the circle edge in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point lying on the circle edge in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point lying on the circle edge in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point that determines the location of the text in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point that determines the location of the text in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point that determines the location of the text in the document's default unit of measurement.

Note
- This command corresponds to the Radius command on the Dimension menu. Click Dimension, Radius.
- You must have drawn a circle before using this command.

Example
```
REM units in inches
.Circle3Points 0, -16.0, 14.0, -12.5, -12.0, 13.0, -12.5, -12.0, 13.5, -12.5
.DimensionRadius -1, -16.0, 14.0, -12.5, -15.5, 12.0, -12.5
```

This command creates a dimension line along the radius of the circle. You do not have to use **.Circle3Points** command to create the circle. Any of the circle commands or CorelCAD tools can be used.

{button ,AL(`CAD_DimensionRadius_Menu;cad_dimenmenu;;;;',0,"Defaultoverview",)} Related Topics

.Duplicate (CAD)

.Duplicate

This command duplicates selected objects.

Note

- You must have selected an object(s) before running this command.
- The duplicate is offset from the original at the distance specified in the Units & Angles dialog box. Click Tools, Layout, Units and Angle.
- This command corresponds to the Duplicate command on the Edit menu. Click Edit, Duplicate.

Example
```
REM units in inches
.Box -1, -14.5, 4.0, -1.5, -13.5, 3.0, -2.5
.Cylinder -1, -7.0, 1.5, 0.0, 3.5, 1.5, 0.0, -3.0, -6.0, 0.0
.SelectWithinRegion -16.0, 4.0, 0.0, 4.0, -9.5, 0.0
.Duplicate
```

The above example creates a box and a cylinder, selects them and then duplicates them.

{button ,AL(`CAD_Duplicate_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} Related Topics

.EditCopy (CAD)

.EditCopy

This command copies the selected object(s) to the Clipboard.

Note
- An object must be selected before using this command.
- This command corresponds to the Copy command on the Edit menu. Click Edit, Copy.

Example
```
REM units in inches
.Box -1, -14.5, 4.0, -1.5, -13.5, 3.0, -2.5
.Cylinder -1, -7.0, 1.5, 0.0, 3.5, 1.5, 0.0, -3.0, -6.0, 0.0
.SelectWithinRegion -16.0, 4.0, 0.0, 4.0, -9.5, 0.0
.EditCopy
```

The above example creates a box and a cylinder, selects them, and copies them to the clipboard.

{button ,AL(`CAD_EditCopy_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.EditCut (CAD)

.EditCut

This command cuts the selected object(s) from the document and copies it to the Clipboard.

Note
- An object must be selected before using this command.
- If you want to cut an object without placing it on the clipboard, use **.DeleteSelection**.
- This command corresponds to the Cut command on the Edit menu. Click Edit, Cut.

Example
```
REM units in inches
.Box -1, -14.5, 4.0, -1.5, -13.5, 3.0, -2.5
.Cylinder -1, -7.0, 1.5, 0.0, 3.5, 1.5, 0.0, -3.0, -6.0, 0.0
.SelectWithinRegion -16.0, 4.0, 0.0, 4.0, -9.5, 0.0
.EditCut
```

The above example creates a box and a cylinder, selects them and then cuts them. The cut objects are placed on the clipboard.

{button ,AL(`CAD_EditCut_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} Related Topics

.EditPaste (CAD)

.EditPaste .X=*double*, .Y=*double*, .Z=*double*

This command pastes the object(s) from the Clipboard to a location determined by the coordinates of one point.

Syntax	Description
.X	Specifies the X-coordinate for the location point in the document's default unit of measurement.
.Y	Specifies the Y-coordinate for the location point in the document's default unit of measurement.
.Z	Specifies the Z-coordinate for the location point in the document's default unit of measurement.

Note

- You must have used the **.EditCopy** command to copy object(s) to the Clipboard before using **.EditPaste**.
- This command corresponds to the Paste command on the Edit menu. Click Paste, Copy.

Example
```
REM units in inches
.Box -1, -14.5, 4.0, -1.5, -13.5, 3.0, -2.5
.Cylinder -1, -7.0, 1.5, 0.0, 3.5, 1.5, 0.0, -3.0, -6.0, 0.0
.SelectWithinRegion -16.0, 4.0, 0.0, 4.0, -9.5, 0.0
.EditCut
.EditPaste 2.0, -0.5, 0.0
```

The above example creates a box and a cylinder, selects them and then cuts them. The cut objects are placed on the clipboard. Then the objects are pasted at the location specified by the coordinates.

{button ,AL(`CAD_EditPaste_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} Related Topics

.Ellipse (CAD)

.Ellipse .SaveAsSurface=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2 =*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command creates an ellipse based on the coordinates of three points.

Syntax	Description
.SaveAsSurface	Specifies whether the ellipse is saved as a surface or a wire. Set to TRUE (-1) to save as a surface; set to FALSE (0) to save as a wire.
.x1	Specifies the X-coordinate for the center of the ellipse in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the ellipse in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the ellipse in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the first radius in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the first radius in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the first radius in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the second radius in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the second radius in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the second radius in the document's default unit of measurement.

Note

- This command corresponds to the Ellipse command on the Draw menu. Click Draw, Ellipse.

Example

```
REM units in inches
.Ellipse 0, -8.5, 3.0, 0.0, -2.5, 1.5, 0.0, -5.5, 1.5, 0.0
```

{button ,AL(`CAD_Ellipse_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.EllipticalCylinder (CAD)

.EllipticalCylinder .SaveAsSurface=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*, x4=*double*, y4=*double*, z4=*double*

This command creates an elliptical cylinder.

Syntax	Description
.SaveAsSolid	Specifies whether the elliptical cylinder is saved as a solid or a surface. Set to TRUE (-1) to save as a solid; set to FALSE (0) to save as a surface.
.x1	Specifies the X-coordinate for the center of the base in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the base in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the base in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the first radius in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the first radius in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the first radius in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the second radius in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the second radius in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the second radius in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the height in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the height in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the height in the document's default unit of measurement.

Example
```
REM units in inches
.EllipticalCylinder -1, -9.0, 4.0, -1.5, -5.5, 3.5, -1.5, -5.5, 3.5, -2.5, -5.5, 8.0-2.5
```

{button ,AL(`CAD_EllipticalCylinder_Menu;cad_solid;;;;',0,"Defaultoverview",)} **Related Topics**

.EllipticalCone (CAD)

.EllipticalCone .SaveAsSurface=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*, x4=*double*, y4=*double*, z4=*double*

This command creates an elliptical cone.

Syntax	Description
.SaveAsSolid	Specifies whether the elliptical cylinder is saved as a solid or a surface. Set to TRUE (-1) to save as a solid; set to FALSE (0) to save as a surface.
.x1	Specifies the X-coordinate for the center of the base in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the base in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the base in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the first radius in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the first radius in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the first radius in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the second radius in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the second radius in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the second radius in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the height in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the height in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the height in the document's default unit of measurement.

Example
```
REM units in inches
.EllipticalCone -1, -4.5, -17.0, 17.0, 0.5, -15.5, 17.0, 1.0, -17.0, 17.0, 1.0, -17.0, 24.5
```

{button ,AL(`CAD_EllipticalCone_Menu;cad_solid;;;;',0,"Defaultoverview",)} Related Topics

.EllipticalFrustum (CAD)

.EllipticalFrustum .SaveAsSurface=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*, x4=*double*, y4=*double*, z4=*double*, x5=*double*, y5=*double*, z5=*double*

This command creates an elliptical frustum.

Syntax	Description
.SaveAsSolid	Specifies whether the elliptical frustum is saved as a solid or a surface. Set to TRUE (-1) to save as a solid; set to FALSE (0) to save as a surface.
.x1	Specifies the X-coordinate for the center of the base in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the base in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the base in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the first radius in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the first radius in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the first radius in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the second radius in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the second radius in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the second radius in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the height in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the height in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the height in the document's default unit of measurement.
.x5	Specifies the X-coordinate for the scale of top in the document's default unit of measurement.
.y5	Specifies the Y-coordinate for the scale of the top in the document's default unit of measurement.
.z5	Specifies the Z-coordinate for the scale of the top in the document's default unit of measurement.

Example
```
REM units in inches
.EllipticalFrustum -1, -13.5, 11.5, -12.5, -11.0, 11.5, -12.5, -13.5, 15.5, -12.5,
-12.0, 15.0, -12.5, -13.0, 17.5, -12.5
```

{button ,AL(`CAD_EllipticalFrustum_Menu;cad_solid;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.EndAddCmdPoint (CAD)

.EndAddCmdPoint

This command specifies the end point for a group of set points.

Note

- This command must be preceded by the **.StartAddCmdPoint**, and **.AddCmdPoint** commands.

Example
```
REM units in inches
.StartAddCmdPoint 3
.AddCmdPoint -10.0, 0.0, 0.0
.AddCmdPoint -5.0, 10.0, 0.0
.AddCmdPoint 10.0, 10.0, 0.0
.EndAddCmdPoint
```

{button ,AL(`CAD_EndAddCmdPoint_Menu;cad_point;;;;',0,"Defaultoverview",)} Related Topics

.Extend (CAD)

.Extend .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command extends a line to the bounding line (the line which you want the extended line to meet).

Syntax	Description
.x1	Specifies the X-coordinate for the point on the bounding line in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point on the bounding line in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point on the bounding line in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point on the line to be extended in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point on the line to be extended in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point on the line to be extended in the document's default unit of measurement.

Note

- You must have already drawn a line and a bounding line before using the **.Extend** command.
- This command corresponds to the Extend command on the Transform menu. Click Transform, Extend.

Example
```
REM units in inches
.SetPointXYZ -105.143, 3.18187, 0.0
.SetPointXYZ -56.5978, 18.5399, 0.0
.LineSegment
.SetPointXYZ -54.5165, 77.0873, 0.0
.SetPointXYZ -18.3887, -28.562, 0.0
.LineSegment
.Extend -35.8766, 22.1553, 0.0,-72.2746, 14.1379, 0.0
```

The above example creates two line segments. The second line segment is the bounding line. Points are selected on the bounding line and the other line. Then the other line is extended to the bounding line.

{button ,AL(`CAD_Extend_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.Extrude (CAD)

.Extrude .ProfileOrientation=*short*, .TaperAngle=*double*

This command extrudes an existing object to a specified scale.

Syntax	Description
.ProfileOrientation	Specifies type of extrusion. Set to 0 for Normal Extrude; and set to 1 for Rigid Extrude
.TaperAngle	Specifies the amount (in degrees) by which the object becomes narrower at the top.

Note

- You must have selected an object before using this command.
- Points used to extrude the object must be set using the **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint** commands.
- This command corresponds to the Extrude command on the Extrude flyout of the Draw menu. Click Draw, Extrude, Extrude.

Example
```
REM units in inches
.Rectangle 1, -95.6901, 10.3969, 14.0229, -55.371, -21.6615, 14.0229
.StartAddCmdPoint 2
.AddCmdPoint -95.6901, -21.6615, 14.0229
.AddCmdPoint -95.4886, -21.5704, 58.4729
.EndAddCmdPoint
.Extrude 1, 45
```

The above example creates a rectangle, sets the command points, and then extrudes the object.

{button ,AL(`CAD_Extrude_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.ExtrudeNormal (CAD)

.ExtrudeNormal .Height=*double*, **.Scale=*double*, x1=*double*, y1=*double*, z2=*double*, x2=*double*, y2=*double*, z2=*double***

This command extrudes an existing object.

Syntax	Description
.Height	Specifies the height of the extruded object in inches. This parameter is optional.
.TaperAngle	Specifies the amount (in degrees) by which the object becomes narrower atthe top.
.x1	If .Height is set, specifies the X-coordinate for the base point of the extruded object in the document's default unit of measurement.
	If .Height is not set, specifies the X-coordinate for the first of two points that determine the height of the extruded object in the document's default unit of measurement.
.y1	If .Height is set, specifies the Y-coordinate for the base point of the extruded object in the document's default unit of measurement.
	If .Height is not set, specifies the Y-coordinate for the first of two points that determine the height of the extruded object in the document's default unit of measurement.
.z1	If .Height is set, specifies the Z-coordinate for the base point of the extruded object in the document's default unit of measurement.
	If .Height is not set, specifies the Z-coordinate for the first of two points that determine the height of the extruded object in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second of two points that determine the height of the extruded object in the document's default unit of measurement. This parameter is required only if .Height is not set.
.y2	Specifies the Y-coordinate for the second of two points that determine the height of the extruded object in the document's default unit of measurement. This parameter is required only if .Height is not set.
.z2	Specifies the Z-coordinate for the second of two points that determine the height of the extruded object in the document's default unit of measurement. This parameter is required only if .Height is not set.

Note

- You must have already created and selected an 2D object before using this command.
- This command corresponds to the Extrude Normal command on the Extrude flyout of the Draw menu. Click Draw, Extrude, Extrude Normal.

Example
```
REM units in inches
.Rectangle 0, -210.1, 362.0, -331.2, -171.8, 330.7, -331.2
.ExtrudeNormal 5, 2, -161.0, 375.5, 331.2
```

The above example creates a rectangle and then extrudes it.

{button ,AL(`CAD_ExtrudeNormal_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.FileClose (CAD)

.FileClose

This command closes the current document.

- **Note**

    If this command is not preceded by the **.FileSave** command, diagram changes will be lost.

Example

```
.FileClose
```

The above example closes the active CorelCAD document.

{button ,AL(`CAD_FileClose_Menu;cad_filemenu;;;;',0,"Defaultoverview",)} Related Topics

.FileCreateTemplate (CAD)

.FileCreateTemplate .FileName=*string*, .WithObjects=*Boolean*

This command creates a template from an open document.

Syntax	Description
.FileName	Specifies the name and path of the template.
.WithObjects	Specifies whether objects from the document are to be included in the new template. Set to TRUE (-1) to include the objects, set to FALSE (0) to discard the objects. The default setting is FALSE (0).

Note
- If the filename used already exists, this command does not overwrite it.
- You must have a document open before using this command.

Example
```
.FileCreateTemplate "C:\CORELCAD\CAD\TEMPLATES\MYTEMP.CCT", -1
```

The above example creates a new CorelCAD template**.**

{button ,AL(`CAD_FileCreate_Menu;cad_filemenu;;;;',0,"Defaultoverview",)} Related Topics

.FileExit (CAD)

.FileExit

This command ends the CorelCAD session.

Note
- If this command is not preceded by the **.FileSave** command, diagram changes will be lost.

Example
`.FileExit`

The above example ends the CorelCAD session.

{button ,AL(`CAD_File_Menu;cad_filemenu;;;;',0,"Defaultoverview",)} Related Topics

.FileExport (CAD)

.FileExport .FileName=*string*, .FilterID=*long*, .Width=*long*, .Height=*long*, .XResolution=*long*, .YResolution=*long*, .ImageType=*long*

This command saves the current document in a format that other programs can read.

Syntax	Description
.FileName	Specifies the new exported filename and path with an appropriate extension. If you do not specify a pathname, the file is saved to the last folder you used to save a file. To export to an AutoCAD (.DWG) or (.DXF), Stereolithography (.STL), or an ACIS (.SAT), specify the file name and the extension.
.FilterID	Specifies the type of file filter. If this parameter is omitted, the file is exported to AutoCAD (.DWG) or (.DXF), Stereolithography (.STL), or an ACIS (.SAT). 769 Windows Bitmap (BMP) 770 Paintbrush (PCX) 771 Targa Bitmap (TGA) 772 TIFF Bitmap (TIF) 773 CompuServe Bitmap (GIF) 774 JPEG Bitmaps (JPG) 776 Scitex CT Bitmap (SCT) 777 Wavelet Compressed Bitmap (WVL) 790 MACPaint Bitmap (MAC) 1792 Corel PHOTO-PAINT Image (CPT)
.Width	Specifies the width of the image in pixels. If this parameter is omitted, the file is exported to AutoCAD (.DWG) or (.DXF), Stereolithography (.STL), or an ACIS (.SAT).
.Height	Specifies the height of the image in pixels. If this parameter is omitted, the file is exported to AutoCAD (.DWG) or (.DXF), Stereolithography (.STL), or an ACIS (.SAT).
.XResolution	Specifies the horizontal resolution of the image in dots per inch (dpi). If this parameter is omitted, the file is exported to AutoCAD (.DWG) or (.DXF), Stereolithography (.STL), or an ACIS (.SAT).
.YResolution	Specifies the vertical resolution of the image in dots per inch (dpi). If this parameter is omitted, the file is exported to AutoCAD (.DWG) or (.DXF), Stereolithography (.STL), or an ACIS (.SAT).
.ImageType	Specifies the image type. If this parameter is omitted, the file is exported to AutoCAD (.DWG) or (.DXF), Stereolithography (.STL), or an ACIS (.SAT). 1 Monochrome bitmap 3 8-bit paletted color bitmap 4 24-bit RGB color bitmap 6 32-bit CMYK bitmap 10 4-bit, 16 colors (standard VGA palette)

Example

```
.FileExport "C:\CORELCAD\TEMP1.BMP", 769, 320, 400, 72, 72, 4
```

The above example exports a CorelCAD file to a Windows bitmap named "TEMP1.BMP".

```
.FileExport "C:\CORELCAD\BEARINGS.DWF"
```

The above example exports a CorelCAD file to a AutoCAD (.DWF) file named BEARINGS.DWG.

{button ,AL(`CAD_FileExport_Menu;cad_filemenu;;;;',0,"Defaultoverview",)} Related Topics

.FileImport (CAD)

.FileImport .FileName=*string*

This command brings files from other programs into CorelCAD.

Syntax	Description
.FileName	Specifies the name of the file to import.

Example
```
.FileNew
.FileImport "C:\COREL\CAD\TEMP1.WMF"
```

The above example imports a Windows Metafile named "TEMP1.WMF" into a new CorelCAD file.

{button ,AL(`CAD_FileImport_Menu;cad_filemenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.FileNew (CAD)

.FileNew

This command creates a new CorelCAD document.

Note

- If you execute a CorelCAD script while CorelCAD is closed, the first command after the **WITHOBJECT**CorelCAD command must be a .**FileNew** or .**FileOpen** command.

Example
```
.FileNew
```

This example creates a new CorelCAD document. It does not close drawings that are currently open.

{button ,AL(`CAD_FileNew_Menu;cad_filemenu;;;;',0,"Defaultoverview",)} Related Topics

.FileNewFromTemplate (CAD)

.FileNewFromTemplate .FileName=*string*, .WithObjects=*Boolean*

This command creates a new document from a template.

Syntax	Description
.FileName	Specifies the name and path of the template.
.WithObjects	Specifies whether objects from the template are to be included in the new document. Set to TRUE (-1) to include the objects, set to FALSE (0) to discard the objects. The default setting is FALSE (0).

Note
- This command corresponds to the From Template command from the New flyout on the File menu. Click File, New, From Template.

Example

```
.FileNewFromTemplate "C:\CORELCAD\CAD\TEMPLATES\TEMPLATE.CCT", -1
```

The above example creates a new CorelCAD diagram using the template called **TEMPLATE.CCT.**

{button ,AL(`CAD_FileNewFromTemplate_Menu;cad_filemenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.FileOpen (CAD)

.FileOpen .FileName=*string*

This command loads a previously saved document into CorelCAD. You can also use this command to open CorelCAD templates.

Syntax	Description
.FileName	Specifies the name of the document to open. If you do not specify a pathname, the last folder you saved a file to is used.

Note
- If you execute a CorelCAD script while CorelCAD is closed, the first command after the **WITHOBJECT** CAD command must be a **.FileNew** or .**FileOpen** command.
- A diagram opened with the .**FileOpen** command becomes the active diagram.

Example
```
.FileOpen "C:\CORELCAD\MYFILES\TEST1.CCD"
```

The above example opens a CorelCAD diagram named **TEST1.CCD.**

```
.FileOpen "C:\CORELCAD\MYFILES\TEMPLATE1.CCT"
```

The above example opens a CorelCAD template named **TEMPLATE1.CCT.**

{button ,AL(`CAD_FileOpen_Menu;cad_filemenu;;;;',0,"Defaultoverview",)} Related Topics

.FileSave (CAD)

.FileSave .FileName=*string*

This command saves the current document to a specified file.

Syntax	Description
.FileName	Specifies the name and path of the file. If you do not specify a pathname, the file is saved to the folder last used to save a file.

Note

The filename must have the extension CCD.

Example

`.FileSave "C:\Corel\CAD\Samples\mycaddoc.ccd"`

The above example saves the current document to the samples folder.

{button ,AL(`CAD_FileSave_Menu;cad_filemenu;;;;',0,"Defaultoverview",)} Related Topics

.Fillet (CAD)

.Fillet .radius=*double*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command fillets 2D line corners.

Syntax	Description
.radius	Specifies the corner radius in inches.
.x1	Specifies the X-coordinate for the point located on the first of two adjacent lines in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point located on the first of two adjacent lines in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point located on the first of two adjacent lines in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point located on the second of two adjacent lines in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point located on the second of two adjacent lines in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point located on the second of two adjacent lines in the document's default unit of measurement.

Note
- This command corresponds to the Fillet command on the Fillet flyout of the Transform menu. Click Transform, Fillet, Fillet.

Example
```
REM units in inches
.Rectangle 0, -10.0, 5.5, 0.0, -4.0, 0.0, 0.0
.Fillet 3, -6.5, 5.5, 0.0, -4.0, 3.5, 0.0
```

The above example creates a rectangle and then fillets a corner of the rectangle.

{button ,AL(`CAD_Fillet_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} <u>**Related Topics**</u>

.FilletEdgeMitered (CAD)

.FilletEdgeMitered .radius=*double*, .X=*double*, .Y=*double*, .Z=*double*, .Continuous=*Boolean*

This command fillets the edge of the 3D object with mitered corners.

Syntax	Description
.radius	Specifies the radius of the corner in inches.
.X	Specifies the X-coordinate for the point on the edge of the object in the document's default unit of measurement.
.Y	Specifies the Y-coordinate for the point on the edge of the object in the document's default unit of measurement.
.Z	Specifies the Z-coordinate for the point on the edge of the object in the document's default unit of measurement.
.Continous	Specifies whether the fillet being applied is continuous. Set to TRUE (-1) to apply a continuous fillet, set to FALSE (0) to apply a non-continuous fillet. The default is FALSE. This parameter is optional.

Note

▪ This command corresponds to the Fillet Edge command on the Fillet flyout of the Transform menu. Click Transform, Fillet, Fillet Edge.
▪ This command fillets only one edge.
▪ The **.Continuous** parameter is identical to holding the shift key down when picking your edges in the Fillet Edge dialog box.

Example
```
REM units in inches
.Box -1, -12.5, 6.0, -4.0, -7.5, 3.0, -9.0
.FilletEdgeMitered 3, -11.0, 10.5, -9.0, 0
.FilletEdgeMitered 3, -9.0, 9.0, -9.0, 0.0
```

This example fillets two edges of a box with mitered corners.

{button ,AL(`CAD_FilletEdgeMitered_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.FilletEdgeRound (CAD)

.FilletEdgeRound .radius=*double*, .SetbackVal=*double*, .X=*double*, .Y=*double*, .Z=*double*, .Continuous
=*Boolean*

This command fillets the edge of the 3D object with round corners.

Syntax	Description
.radius	Specifies the radius of the corner in inches.
.SetbackVal	Specifies the distance the original fillet edge is set back.
.X	Specifies the X-coordinate for the point on the edge of the object in the document's default unit of measurement.
.Y	Specifies the Y-coordinate for the point on the edge of the object in the document's default unit of measurement.
.Z	Specifies the Z-coordinate for the point on the edge of the object in the document's default unit of measurement.
.Continuous	Specifies whether the fillet being applied is continuous. Set to TRUE (-1) to apply a continuous fillet, set to FALSE (0) to apply a non-continuous fillet. The default is FALSE. This parameter is optional.

Note
- This command corresponds to the Fillet Edge command on the Fillet flyout of the Transform menu. Click Transform, Fillet, Fillet Edge.
- This command fillets only one edge.
- The **.Continuous** parameter is identical to holding the shift key down when picking your edges in the Fillet Edge dialog box.

Example
```
REM units in inches
.Box -1, -18.0, 27.0, -23.0, -12.5, 23.5, -28.0
.FilletEdgeRound 4, 1, -14.0, 30.0, -29.0, 0
```

This example creates a box and then fillets a corner of the box with round edges.

{button ,AL(`CAD_FilletEdgeMitered_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.Frustum (CAD)

.Frustum .SaveAsSolid=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*, .x4=*double*, .y4=*double*, .z4=*double*

This command creates a frustum.

Syntax	Description
.SaveAsSolid	Specifies whether the frustum is saved as a solid or a surface. Set to TRUE (-1) to save the frustum as a solid; set to FALSE (0) to save the frustum as a surface.
.x1	Specifies the X-coordinate for the center of the frustum's base in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the frustum's base in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the frustum's base in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the radius of the frustum's base in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the radius of the frustum's base in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the radius of the frustum's base in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the height of the frustum's second plane in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the height of the frustum's second plane in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the height of the frustum's second plane in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the radius of the frustum's second plane in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the radius of the frustum's second plane in the document's default unit of measurement.
.z4	Specifies the X-coordinate for the radius of the frustum's second plane in the document's default unit of measurement.

Note

- This command corresponds the Frustum, Center and Radius command on the Solids flyout of the Draw menu. Click Draw, Solids, Frustum, Center and Radius.

Example
```
REM units in inches
.Frustum -1, -10.5, 0.0, 0.0, -8.5, 0.0, 0.0, -10.0, 7.5, 0.0, -10.0, 8.5, 0.0
```

{button ,AL(`CAD_Frustum_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.FrustumDiameter (CAD)

.FrustumDiameter .SaveAsSurface=*Boolean*, .NumberEdges=*integer*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*, .x4=*double*, .y4=*double*, z4=*double*

This command creates a circular two point frustum.

Syntax	Description
.SaveAsSurface	Specifies whether the frustum is saved as a solid or a surface. Set to TRUE (-1) to save the frustum as solid; set to FALSE (0) to save as a surface.
.NumberEdges	Specifies the number of sides.
.x1	Specifies the X-coordinate for the center of the base in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the base in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the base in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point on the vertex in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point on the vertex in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point on the vertex in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the height in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the height in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the height in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the scale of the top in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the scale of the top in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the scale of the top in the document's default unit of measurement.

Example
```
REM units in inches
.FrustumDiameter -1, -17.0, 8.0, -12.5, -11.5, 9.0, -12.5, -13.5, 14.0, -12.5, -14.0, 15.0, -12.5
```

{button ,AL(`CAD_FrustumDiameter_Menu;cad_solid;;;;',0,"Defaultoverview",)} Related Topics

.GetName (CAD)

ReturnValue$ = .GetName ( )

This function returns the name of the selected object.

Syntax	Description
ReturnValue$	Specifies the string variable that is passed the return value corresponding to the name of the selected object. This variable must be declared using a <u>DIM</u> statement or implicitly declared using <u>type-declaration suffix</u>.

Note

- If more than one object is selected and they all have the same name, the name of the selected objects is returned. If more than one object is selected and they do not have the same name, nothing is returned.
- The object(s) must be selected before using this command.

Example
```
REM units in inches
.Frustum -1, -14.0, 2.0, 0.0, -12.0, 1.0, 0.0, -12.0, 4.5, 0.0, -11.5, 4.0, 0.0
.Sphere -1, -10.0, 0.5, 0.0, -8.5, -0.5, 0.0
.SelectWithinRegion -19.0, 7.5, 0.0, 1.0, 5.5, 0.0
.ApplyName "Canada"
ObjName$ = .GetName()
```

This example creates a frustum and a sphere. The object name "Canada" is applied to the two objects and then returns the name of the objects.

{button ,AL(`CAD_GetName_Menu;cad_name;;;;',0,"Defaultoverview",)} **<u>Related Topics</u>**

.GetTotalNumberObjects (CAD)

ReturnValue& = .GetTotalNumberObjects ( )

This function returns the number of objects in the active document.

Syntax	Description
ReturnValue&	Specifies the numeric variable that is passed the return value corresponding to the number of objects. This variable must be declared using a <u>DIM</u> statement or implicitly declared using a <u>type-declaration suffix</u>.

Example

```
NumberObjs& = .GetTotalNumberObjects ()
```

This example counts all of the objects in the active document and returns the number of objects.

{button ,AL(`CAD_GetTotalNumberObjects_Menu;cad_name;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.Group (CAD)

.Group

This command groups selected objects.

Note
- The objects must be selected before using this command.
- This command corresponds to the Group command on the Transform menu. Click Transform, Group.

Example
```
REM units in inches
.Frustum -1, -14.0, 2.0, 0.0, -12.0, 1.0, 0.0, -12.0, 4.5, 0.0, -11.5, 4.0, 0.0
.Sphere -1, -10.0, 0.5, 0.0, -8.5, -0.5, 0.0
.SetPointXYZ -8.5, 8.5, 0.0
.SetPointXYZ -4.5, -4.0, 0.0
.LineSegment
.SelectWithinRegion -19.0, 7.5, 0.0, 1.0, 5.5, 0.0
.Group
```

This command creates a frustum, sphere and a line, then selects all of the objects and groups them.

_____

{button ,AL(`CAD_Group_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.Hemisphere (CAD)

.Hemisphere .SaveAsSolid=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, . z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command creates a hemisphere.

Syntax	Description
.SaveAsSolid	Specifies whether the hemisphere is saved as a solid or a surface. Set to TRUE (-1) to save the hemisphere as a solid; set to FALSE (0) to save the hemisphere as a surface.
.x1	Specifies the X-coordinate for the center of the hemisphere in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the hemisphere in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the hemisphere in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the radius of the hemisphere in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the radius of the hemisphere in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the radius of the hemisphere in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the plane of the hemisphere in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the plane of the hemisphere in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the plane of the hemisphere in the document's default unit of measurement.

Note
- This command corresponds to the Hemisphere, Center and Radius command on the Solids flyout of the Draw menu. Click Draw, Solids, Hemisphere, Center and Radius.

Example
```
REM units in inches
.Hemisphere -1, -10.0, 2.0, 0.0, -5.0, 3.0, 0.0, -6.0, 4.5, 0.0
```

{button ,AL(`CAD_Hemisphere_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.HemisphereDiameter (CAD)

.HemisphereDiameter .SaveAsSolid=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2 =*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command creates a hemisphere.

Syntax	Description
.SaveAsSolid	Specifies whether the hemisphere is saved as a solid or a surface. Set to TRUE (-1) to save the hemisphere as a solid; set to FALSE (0) to save the hemisphere as a surface.
.x1	Specifies the X-coordinate for the first point that determines the diameter of the hemisphere in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that determines the diameter of the hemisphere in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that determines the diameter of the hemisphere in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that determines the diameter of the hemisphere in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that determines the diameter of the hemisphere in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that determines the diameter of the hemisphere in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the plane of the hemisphere in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the plane of the hemisphere in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the plane of the hemisphere in the document's default unit of measurement.

Example
```
REM units in inches
.HemisphereDiameter -1, -15.0, 13.0, -12.5, -6.0, 15.5, -12.5, -7.5, 20.0, -12.5
```

{button ,AL(`CAD_HemisphereDiameter_Menu;cad_solid;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.HideEntireView (CAD)

.HideEntireView .ShowText=*Boolean*, .ShowDimension=*Boolean*, .IsSihouette=*Boolean*, .IsAllViews= *Boolean*

This command hides the hidden lines for the entire view.

Syntax	Description
.ShowText	Specifies whether the text is displayed. If set to TRUE (-1), the text is displayed; if set to FALSE (0) the text is not displayed.
.ShowDimension	Specifies whether dimensions are displayed. If set to TRUE (-1), the dimensions are shown; if set to FALSE (0), the dimensions are not displayed.
.IsSihouette	Specifies whether the silhouette is displayed. If set to TRUE (-1), the silhouette is displayed; if set to FALSE (0), the silhouette is not displayed.
.IsAllViews	Specifies whether the hidden lines in all views are displayed. If set to TRUE (-1), hidden lines are not displayed in all views; if set to FALSE (0), only the hidden lines in the active view are not displayed.

Note

- This command corresponds to the Hide command on the View menu. Click View, Hide.

Example
```
REM units in inches
.HideEntireView 0, -1, 0, 0
```

This example hides the hidden lines from only the active view. The text and silhouette are not displayed, but dimension lines are.

{button ,AL(`CAD_HideEntireView_Menu;cad_viewmenu;;;;',0,"Defaultoverview",)} Related Topics

.HideSectionView (CAD)

.HideSectionView .ShowText=*Boolean*, .ShowDimensions=*Boolean*, .IsSilhouette=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command hides the hidden lines in the view specified by the coordinates of two points.

Syntax	Description
.ShowText	Specifies whether the text is displayed. If set to TRUE (-1), the text is displayed; if set to FALSE (0) the text is not displayed.
.ShowDimension	Specifies whether dimensions are displayed. If set to TRUE (-1), the dimensions are shown; if set to FALSE (0), the dimensions are not displayed.
.IsSihouette	Specifies whether the silhouette is displayed. If set to TRUE (-1), the silhouette is displayed; if set to FALSE (0), the silhouette is not displayed.
.x1	Specifies the X-coordinate for the first point (starting corner) of the region in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point (starting corner) of the region in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point (starting corner) of the region in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point (opposite corner) of the region in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point (opposite corner) of the region in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point (opposite corner) of the region in the document's default unit of measurement.

Note
- This command corresponds to the Hide command on the View menu. Click View, Hide.

Example
```
REM units in inches
.HideSectionView 0, 0, -1, -21.0, 21.0, -12.5, -1.5, 8.0, -12.5
```

This example hides the hidden lines in the region specified by the coordinates of the two points. Text and dimension lines are not displayed, but the silhouette is.

{button ,AL(`CAD_HideSectionView_Menu;cad_viewmenu;;;;',0,"Defaultoverview",)} Related Topics

.HideSelected (CAD)

.HideSelected .ShowText=*Boolean*, .ShowDimension=*Boolean*, .IsSilhouette=*Boolean*

This command hides the hidden lines in the selected object.

Syntax	Description
.ShowText	Specifies whether the text is displayed. If set to TRUE (-1), the text is displayed; if set to FALSE (0) the text is not displayed.
.ShowDimension	Specifies whether dimensions are displayed. If set to TRUE (-1), the dimensions are shown; if set to FALSE (0), the dimensions are not displayed.
.IsSihouette	Specifies whether the silhouette is displayed. If set to TRUE (-1), the silhouette is displayed; if set to FALSE (0), the silhouette is not displayed.

Note
- The object must be selected before running this command.

Example
```
REM units in inches
.HemisphereDiameter -1, -15.0, 13.0, -12.5, -6.0, 15.5, -12.5, -7.5, 20.0, -12.5,
.HideSelected 0, 0, -1
```

This example hides the hidden lines of the selected object. Text and dimension lines are not displayed, but the silhouette is displayed.

{button ,AL(`CAD_HideSelected_Menu;cad_viewmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.IntersectingLine (CAD)

.IntersectingLine .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command draws an intersecting line between two surfaces. The two surfaces are determined by the coordinates of two points.

Syntax	Description
.x1	Specifies the X-coordinate for the point on the first surface in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point on the first surface in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point on the first surface in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point on the second surface in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point on the second surface in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point on the second surface in the document's default unit of measurement.

Note

- You must have two surfaces that intersect before using the .**IntersectingLine** command.
- The objects used can be either 2D or 3D; however, a 2D object must be a surface and a 3D object must be a solid.
- This command corresponds to the Intersecting Line command on the Line flyout of the Draw menu. Click Draw, Line, Intersecting Line.

Example

```
REM units in inches
.Rectangle -1, -8.5, 2.0, 0.0, 0.0, -2.5, 0.0
.Rectangle -1, -4.0, 0.5, 2.0, -4.0, -3.0, -1.5
.IntersectingLine -8.5, 2.0, 0.0, -4.0, 0.5, 2.0
```

This example draws two rectangles and then creates an intersecting line between the two.

{button ,AL(`CAD_IntersectingLine_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.JoinLines (CAD)

.JoinLines .X=*double*, .Y=*double*, .Z=*double*

This command joins the end points of previously selected lines.

Syntax	Description
.X	Specifies the X-coordinate of the point determining where the lines will be joined in the document's default unit measurement.
.Y	Specifies the Y-coordinate of the point determining where the lines will be joined in the document's default unit measurement.
.Z	Specifies the Z-coordinate of the point determining where the lines will be joined in the document's default unit measurement.

Note
- The lines must be drawn and selected before using this command.
- This command corresponds to the Join command on the Join Lines flyout of the Transform menu. Click Transform, Join Lines, Join.

Example
```
REM units in inches
.SetPointXYZ -103.535, 33.1608, 0.0
.SetPointXYZ -2.43662, -24.9533, 0.0
.LineSegment
.SetPointXYZ -38.6308, 60.9008, 0.0
.SetPointXYZ -62.3784, -42.9061, 0.0
.LineSegment
.JoinLines -51.7824, 3.41197, 0.0
```

This example creates two line segments and then joins them.

{button ,AL(`CAD_JoinLines_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.LayerSetColor (CAD)

.LayerSetColor .LayerName=*string*, .GroupName=*string***, .Red=*short*, .Green=*short*, .Blue=*short***

This command edits the color of a layer.

Syntax	Description
.LayerName	Specifies the name of the existing layer.
.GroupName	Specifies the name of the layer group. If this parameter is omitted, the default group is All Layers.
.Red	Specifies the red setting for layer color (RGB color model). Valid values range from 0 to 255. This parameter works in conjunction with **.Green** and **.Blue** to determine the color of the layer.
.Green	Specifies the green setting for layer color (RGB color model). Valid values range from 0 to 255. This parameter works in conjunction with **.Red** and **.Blue** to determine the color of the layer.
.Blue	Specifies the blue setting for layer color (RGB color model). Valid values range from 0 to 255. This parameter works in conjunction with **.Red** and **.Green** to determine the color of the layer.

Note
- The layer specified for **.LayerName** must exist and belong to the specified group.
- **.GroupName** is optional. If the **.GroupName** is provided, the name of the group must already exist.

Example
```
.LayerSetColor "mylayer", "mygroup", 255, 100, 100
```

The first example changes the color of the layer called "mylayer" in the group "mygroup.

```
.LayerSetColor "mylayer",, 255, 100,100
```

The second example changes the color of the same layer, but the layer is in the group "All Layers" because the group name is not specified.

{button ,AL(`CAD_LayerSetColor_Menu;cad_toolsmenu;;;;',0,"Defaultoverview",)} Related Topics

.LayerSetDescription (CAD)

.LayerSetDescription .LayerName=*string*, .Description=*string*

This command edits the description of a layer.

Syntax	Description
.LayerName	Specifies the name of the existing layer.
.Description	Specifies the description of the layer.

Note

- The layer specified for **.LayerName** must exist.

Example

```
.LayerSetDesciption "mylayer", "this is a description of mylayer"
```

{button ,AL(`CAD_LayerSetDescription_Menu;cad_toolsmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.LayerSetLineStyle (CAD)

.LayerSetLineStyle .LayerName=*string*, .GroupName=*string*, **.LineStyle=*short***

This command edits the line style of a layer.

Syntax	Description
.LayerName	Specifies the name of the existing layer.
.GroupName	Specifies the group name for the layer. If this parameter is omitted, the default group name is All Layers.
.LineStyle	Specifies the new line style used to draw objects on the layer. 1 Solid (1 point) 2 Dashes 3 Dots 4 Dash, Dot 5 Dash, Dot, Dot 6 Solid (2 points) 7 Solid (3 points) 8 Solid (4 points) 9 Solid (5 points) 10 Solid (6 points) 11 Solid (7 points)

Note

- The layer specified for **.LayerName** must exist and belong to the specified group.
- **.GroupName** is optional. If the **.GroupName** is provided, the name of the group must already exist.

Example
```
.LayerSetLineStyle "mylayer", "mygroup", 8
```

The above example sets the lines style for the layer called "mylayer" in the group "mygroup. The group name is specified.

```
.LayerSetLineStyle "mylayer",, 8
```

The above example sets the ine style for the layer called "mylayer" in the default group All Layers.

{button ,AL(`CAD_LayerSetLineStyle_Menu;cad_toolsmenu;;;;',0,"Defaultoverview",)} Related Topics

.LayerSetName (CAD)

.LayerSetName .LayerName=*string*, .NewName=*string*

This command edits the name of a layer.

Syntax	Description
.LayerName	Specifies the name of the existing layer.
.NewName	Specifies the new name set for the layer.

Note

- The layer specified for **.LayerName** must exist.
- If the new name already exists, this command is ignored.

Example
```
.LayerSetName "mylayer", "new name"
```

{button ,AL(`CAD_LayerSetName_Menu;cad_toolsmenu;;;;',0,"Defaultoverview",)} Related Topics

.LayerSetStatus (CAD)

.LayerSetStatus .LayerName=*string*, .GroupName=*string*, .Visible=*Boolean*, .Printable=*Boolean*, .Locked=*Boolean*, .Override=*Boolean*

This command edits the status of a layer.

Syntax	Description
.LayerName	Specifies the name of the existing layer.
.GroupName	Specifies the group name for the layer. The default is All Layers.
.Visible	Specifies whether the layer is visible. Set to TRUE (-1) to make the layer visible; set to FALSE (0) to make the layer invisible.
.Printable	Specifies whether objects on the layer are printable. Set to TRUE (-1) to print the layer; set to FALSE (0) to make the layer not printable.
.Locked	Specifies whether the layer is locked. Set to TRUE (-1) to lock the layer; set to FALSE to unlock the layer.
.Override	Specifies whether the color of the objects on the specified layer can be overriden. Set to TRUE (-1) to override the objects' color with the layer's color, set to FALSE (0) to keep the objects' color.

Note

- **.GroupName .Visible**, **.Printable**, **.Locked,** and **.Override** are optional. You can change only the properties that you want. For example, if you only want to make the layer invisible, set **.Visible** to FALSE and ignore all other properties.
- You cannot set the current layer as invisible and/or locked. If you do, the command is ignored.

Example
```
.LayerSetStatus "mylayer", "mygroup", 0, 0, -1, 0
```

The above example sets the all of the parameters for the layer called "mylayer".

```
.LayerSetStatus "mylayer", "mygroup", , ,-1, 0
```

The above example only locks the layer and the override color.

{button ,AL(`CAD_LayerSetStatus_Menu;cad_toolsmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.LineSegment (CAD)

.LineSegment

This command creates a line segment.

Note
- You must precede the **.LineSegment** command with either **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint** or **.SetPointXYZ**.
- This command corresponds to the Line Segments command on the Line flyout of the Draw menu. Click Draw, Line, Line Segments.

Example
```
REM units in inches
.SetPointXYZ -8.5, 8.5, 0.0
.SetPointXYZ -4.5, -4.0, 0.0
.LineSegment
```

{button ,AL(`CAD_LineUnconnected_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.LinearArray (CAD)

.LinearArray .DistanceMode=*integer*, .Copy=*integer*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command places copies of the selected object along a straight line.

Syntax	Description
.DistanceMode	Specifies the distance mode. Set to 0 for increment mode; set to 1 for overall mode.
.Copy	Specifies the number of copies.
.x1	Specifies the X-coordinate for one of two points that determine the linear direction and the distance between two copies.
.y1	Specifies the Y-coordinate for one of two points that determine the linear direction and the distance between two copies.
.z1	Specifies the Z-coordinate for one of two points that determine the linear direction and the distance between two copies.
.x2	Specifies the X-coordinate for one of two points that determine the linear direction and the distance between two copies.
.y2	Specifies the Y-coordinate for one of two points that determine the linear direction and the distance between two copies.
.z2	Specifies the Z-coordinate for one of two points that determine the linear direction and the distance between two copies.

Note
- An object must be selected before using this command.
- This command corresponds to the Linear Array command on the Array flyout of the Draw menu. Click Draw, Array, Linear Array.

Example
```
REM units in inches
.Box -1, -20.0, 9.0, -11.0, -17.5, 6.5, -13.5
.LinearArray 0, 3, -17.5, 8.5, -11.0, -17.5, 12.0, -11.0
```

This example creates a box and then places 20 copies of the box in a line. The distance mode is set to increment and the direction of the line and the distance between copies is set by the coordinates of the two points.

{button ,AL(`CAD_LinearArray_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.Loft (CAD)

.Loft

This command creates a smooth surface between two or more existing lines.

Note
- The lines must be drawn before using the **.Loft** command.
- The points on the lines must be created using **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint** and/or **.SetPointXYZ**.
- This command corresponds to the Loft command on the Surfaces flyout of the Draw menu. Click Draw, Surfaces, Loft.

Example
```
REM units in inches
.StartAddCmdPoint 3
.AddCmdPoint 20.0, 0.0, 0.0
.AddCmdPoint 25.0, 10.0, 0.0
.AddCmdPoint 40.0, 10.0, 0.0
.EndAddCmdPoint
.PolyLine
.SetPointXYZ 20.0, 0.0, 0.0
.SetPointXYZ 10.0, 0.0, 0.0
.SetPointXYZ 10.0, -10.0, 0.0
.PolyLine
.Loft
```

This example creates two polylines and then applies a smooth surface between the two lines.

{button ,AL(`CAD_Loft_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.Mirror (CAD)

.Mirror .LeaveOriginal=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command creates a mirror of the selected object.

Syntax	Description
.LeaveOriginal	Specifies whether a duplicate of the object is created when the object is mirrored. Set to TRUE (-1) to duplicate the object; set to FALSE (0) to not duplicate the object.
.x1	Specifies the X-coordinate for the first point that determines the plane for the mirrored object in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that determines the plane for the mirrored object in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that determines the plane for the mirrored object in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that determines the plane for the mirrored object in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that determines the plane for the mirrored object in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that determines the plane for the mirrored object in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the third point that determines the plane for the mirrored object in the document's default unit of measurement. This point is optional.
.y3	Specifies the Y-coordinate for the third point that determines the plane for the mirrored object in the document's default unit of measurement. This point is optional.
.z3	Specifies the Z-coordinate for the third point that determines the plane for the mirrored object in the document's default unit of measurement. This point is optional.

Note
- If only two points are specified, the mirror is two-dimensional. If three points are specified, the mirror is three-dimensional.
- An object must be selected before using the **.Mirror** command.
- This command corresponds to the Mirror command on the Transform menu. Click Transform, Mirror.

Example
```
REM units in inches
.Box -1, -25.5, 25.0, -23.0, -22.0, 21.5, -26.5
.Mirror -1, -20.5, 25.0, -26.5, -19.5, 22.5, -26.5
```

This example creates a box and then creates a duplicate of the box which is mirrored. Because only two points are specified, the mirror is two-dimensional.

{button ,AL(`CAD_Mirror_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.MirrorXY (CAD)

.MirrorXY .LeaveOriginal=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*

This command creates a mirror of the selected object with respect to the XY plane.

Syntax	Description
.LeaveOriginal	Specifies whether a duplicate of the object is created when the object is mirrored. Set to TRUE (-1) to duplicate the object; set to FALSE (0) to not duplicate the object.
.x1	Specifies the X-coordinate for the point that determines the plane for the mirrored object in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point that determines the plane for the mirrored object in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point that determines the plane for the mirrored object in the document's default unit of measurement.

Note
- Only the Z-coordinate is significant.
- An object must be selected before using the **.MirrorXY** command.

Example
```
REM units in inches
.Box -1, -25.5, 25.0, -23.0, -22.0, 21.5, -26.5
.MirrorXY -1, -18.0, 29.0, -26.5
```

This example creates a box, duplicates it, and mirrors the duplicate at the specified location along the Z axis.

{button ,AL(`CAD_MirrorXY_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.MirrorXZ (CAD)

.MirrorXZ .LeaveOriginal=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*

This command creates a mirror of the selected object with respect to the XZ plane.

Syntax	Description
.LeaveOriginal	Specifies whether a duplicate of the object is created when the object is mirrored. Set to TRUE (-1) to duplicate the object; set to FALSE (0) to not duplicate the object.
.x1	Specifies the X-coordinate for the point that determines the plane for the mirrored object in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point that determines the plane for the mirrored object in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point that determines the plane for the mirrored object in the document's default unit of measurement.

Note
- Only the Y-coordinate is significant.
- An object must be selected before using the **.MirrorXZ** command.

Example
```
REM units in inches
.Box -1, -25.5, 25.0, -23.0, -22.0, 21.5, -26.5
.MirrorXZ -1, -17.0, 19.5, -26.5
```

This example creates a box, duplicates it, and mirrors the duplicate at the specified location along the Y axis.

{button ,AL(`CAD_MirrorXZ_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.MirrorYZ (CAD)

.MirrorYZ .LeaveOriginal=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*

This command creates a mirror of the selected object with respect to the YZ plane.

Syntax	Description
.LeaveOriginal	Specifies whether a duplicate of the object is created when the object is mirrored. Set to TRUE (-1) to duplicate the object; set to FALSE (0) to not duplicate the object.
.x1	Specifies the X-coordinate for the point that determines the plane for the mirrored object in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point that determines the plane for the mirrored object in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point that determines the plane for the mirrored object in the document's default unit of measurement.

Note
- Only the X-coordinate is significant.
- An object must be selected before using the **.MirrorYZ** command.

Example
```
REM units in inches
.Box -1, -25.5, 25.0, -23.0, -22.0, 21.5, -26.5
.MirrorYZ -1, -20.5, 35.0, -26.5
```

This example creates a box, duplicates it, and mirrors the duplicate at the specified location along the X axis.

{button ,AL(`CAD_MirrorYZ_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.MirrorNormal (CAD)

.MirrorNormal .LeaveOriginal=_Boolean_**, .x1=**_double_**, .y1=**_double_**, .z1=**_double_**, x2=**_double_**, y2=**_double_**, z2=**_double_

This command creates a mirror of the selected object on the plane that is normal to the vector defined by two points.

Syntax	Description
.LeaveOriginal	Specifies whether a duplicate of the object is created when the object is mirrored. Set to TRUE (-1) to duplicate the object; set to FALSE (0) to not duplicate the object.
.x1	Specifies the X-coordinate for the first point that determines the vector for the mirrored object in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that determines the vector for the mirrored object in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that determines the vector for the mirrored object in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that determines the vector for the mirrored object in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that determines the vector for the mirrored object in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that determines the vector for the mirrored object in the document's default unit of measurement.

Note
- An object must be selected before using the **.MirrorNormal** command.

Example
```
REM units in inches
.Box -1, -25.5, 25.0, -23.0, -22.0, 21.5, -26.5
.MirrorNormal -1, -17.0, 25.0, -26.5, -15.0, 31.0, -26.5
```

This example creates a box, duplicates it and places the duplicate on the plane that is normal to the vector specifed by the coordinate of the two points.

{button ,AL(`CAD_MirrorXY_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.Move (CAD)

.Move .LeaveOriginal=*Boolean*, .MoveType=*integer*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*doubl e*, .y2=*double*, .z2=*double*

This command moves the selected object or its duplicate to a new location which is relative to the new base points.

Syntax	Description
.LeaveOriginal	Specifies whether a duplicate of the object is created when the object is moved. Set to TRUE (-1) to duplicate the object; set to FALSE (0) to not duplicate the object.
.MoveType	Specifies the type of move: 0 Absolute 1 Relative
.x1	If **.MoveType** is set to Absolute, specifies the X-coordinate for the first absolute point in the document's default unit of measurement. If **.MoveType** is set to Relative, specifies the X-coordinate for the base point in the document's default unit of measurement.
.y1	If **.MoveType** is set to Absolute, specifies the Y-coordinate for the first absolute point in the document's default unit of measurement. If **.MoveType** is set to Relative, specifies the Y-coordinate for the base point in the document's default unit of measurement.
.z1	If **.MoveType** is set to Absolute, specifies the Z-coordinate for the first absolute point in the document's default unit of measurement. If **.MoveType** is set to Relative, specifies the Z-coordinate for the base point in the document's default unit of measurement.
.x2	If **.MoveType** is set to Absolute, specifies the X-coordinate for the second absolute point in the document's default unit of measurement. If **.MoveType** is set to Relative, specifies the X-coordinate for the second point relative to the first point in the document's default unit of measurement.
.y2	If **.MoveType** is set to Absolute, specifies the Y-coordinate for the second absolute point in the document's default unit of measurement. If **.MoveType** is set to Relative, specifies the Y-coordinate for the second point relative to the first point in the document's default unit of measurement.
.z2	If **.MoveType** is set to Absolute, specifies the Z-coordinate for the second absolute point in the document's default unit of measurement. If **.MoveType** is set to Relative, specifies the Z-coordinate for the second point relative to the first point in the document's default unit of measurement.

Note
- You must have selected an object before using this command.
- This command corresponds to the Move command on the Transform menu. Click Transform, Move.

Example
```
REM units in inches
.Box -1, -25.5, 25.0, -23.0, -22.0, 21.5, -26.5
.Move -1, 0, -16.5, 32.5, -26.5, -11.0, 37.0, -26.5
```

This example creates a box, duplicates its, and places the duplicate at the location specified by the coordinates of two points. The type of move is absolute; therefore, the coordinates are relative to the origin (0,0,0). If **.MoveType** had been set to 1, the second point's coordinates would have been relative the first point set.

{button ,AL(`CAD_Move_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.MultiPlane (CAD)

.MultiPlane .SaveAsSurface=*Boolean*, .IsConnected=*Boolean*

This command draws a multi plane determined by points set around the perimeter of a surface.

Syntax	Description
.SaveAsSurface	Set to TRUE (-1) to save the multi plane as plane; set to FALSE (0) to save as a line vector.
.IsConnected	Set to TRUE (-1) to connect the points; set to FALSE (0) to leave the points unconnected.

Note
- You must precede the **.MultiPlane** command with either **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint** or **.SetPointXYZ**.
- This command corresponds to the Multiple Planes command on the Surfaces flyout of the Draw menu. Click Draw, Surfaces, Multiple Planes.

Example
```
REM units in inches
.SetPointXYZ -24.5, 30.5, -25.0
.SetPointXYZ -20.0, 27.5, -25.0
.SetPointXYZ -10.5, 26.5, -25.0
.SetPointXYZ -8.0, 33.5, -25.0
.MultiPlane -1, -1
```

{button ,AL(`CAD_MultiPlane_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.NewLayer (CAD)

.NewLayer .LayerName=*string*, .GroupName=*string*, .Visible=*Boolean*, .Printable=*Boolean*, .Locked=*Boolean*, .Override=*Boolean*, .Red=*short*, .Green=*short*, .Blue=*short*, .LineStyle=*short*, .Description=*string*

This command creates a new layer.

Syntax	Description
.LayerName	Specifies the name of the new layer. If you do not enter a name, the name of the layer is created automatically.
.GroupName	Specifies the name of the layer group.
.Visible	Specifies whether the layer is visible. Set to TRUE (-1) to make the layer visible; set to FALSE (0) to make the layer invisible. The default setting is TRUE (visible).
.Printable	Specifies whether objects on the layer are printable. Set to TRUE (-1) to print the layer; set to FALSE (0) to make the layer not printable. The default setting is TRUE (printable).
.Locked	Specifies whether the layer is locked. Set to TRUE (-1) to lock the layer; set to FALSE (0) to unlock the layer. The default setting is FALSE (unlocked).
.Override	Specifies whether the color of the objects on the specified layer can be overriden. Set to TRUE (-1) to override the objects' color with the layer's color, set to FALSE (0) to keep the objects' color.
.Red	Specifies the red setting for layer color (RGB color model). Valid values range from 0 to 255. This parameter works in conjunction with **.Green** and **.Blue** to determine the color of the layer. The default layer color is black (**.Red**=0, **.Green**=0, **.Blue**=0).
.Green	Specifies the green setting for layer color (RGB color model). Valid values range from 0 to 255. This parameter works in conjunction with **.Red** and **.Blue** to determine the color of the layer. The default layer color is black (**.Red**=0, **.Green**=0, **.Blue**=0).
.Blue	Specifies the blue setting for layer color (RGB color model). Valid values range from 0 to 255. This parameter works in conjunction with **.Red** and **.Green** to determine the color of the layer. The default layer color is black (**.Red**=0, **.Green**=0, **.Blue**=0).
.LineStyle	Specifies the line style used to draw objects on the layer. For example, if you select a solid line for the layer, all objects (rectangles, frustums, etc) are drawn with a solid line. 1 Solid (1 point) 2 Dashes 3 Dots 4 Dash, Dot 5 Dash, Dot, Dot 6 Solid (2 points) 7 Solid (3 points) 8 Solid (4 points) 9 Solid (5 points) 10 Solid (6 points) 11 Solid (7 points)
.Description	Text string describing the layer.

Note
- All of the parameters are optional.
- If you specify **.GroupName**, the name entered must exist as a group name.

Example
```
.NewLayer "mylayer", , -1, 0, 0, 0, 255, 255, 100, 3, "second layer"
```

This example does not specify a group name.

{button ,AL(`CAD_NewLayer_Menu;cad_toolsmenu;;;;',0,"Defaultoverview",)} Related Topics

.NewLayerGroup (CAD)

.NewLayerGroup .GroupName=*string*, .Description=*string***, .LayerName=*string***

This command creates a new layer group.

Syntax	Description
.GroupName	Specifies the name of the new layer group.
.Description	Text string describing the layer.
.LayerName	Specifies the name of the existing layer.

Note
- **.Desciption** is an optional parameter.
- You must specify an existing layer for the **.LayerName** parameter because an empty layer group cannot be created.
- You can add more layers to the group using the **.AddLayerToLayerGroup** command

Example
```
.NewLayerGroup "mygroup", "first layer group", "mylayer"
```

{button ,AL(`CAD_NewLayerGroup_Menu;cad_toolsmenu;;;;',0,"Defaultoverview",)} Related Topics

Offset (CAD)

.Offset .OffsetDist=*double*, .NumberofCopies=*integer*, .x1=double, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command draws a line(s) parallel to an existing line.

Syntax	Description
.OffsetDist	Specifies the distance between the existing line and the new line(s) in the document's default unit of measurement.
.NumberofCopies	Specifies the number of new parallel lines to be offset from the original.
.x1	Specifies the X-coordinate for the point on the original line in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point on the original line in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point on the original line in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point that determines on which side of the original line the new lines will be placed in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point that determines on which side of the original line the new lines will be placed in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point that determines on which side of the original line the new lines will be placed in the document's default unit of measurement.

Note
- You must have drawn a line before using this command.
- This command corresponds to the Offset command on the Transform menu. Click Transform, Offset.

Example
```
REM units in inches
.SetPointXYZ -12.0, 4.5, 0.0
.SetPointXYZ -2.0, 1.0, 0.0
.SetPointXYZ -0.5, 0.5, 0.0
.LineSegment
.Offset 1, 1, -7, 2.75, -5.0, 7.75, 0.0
```

This example creates a line segment and then offsets one copy one inch at a location specified by the coordinates of the two points.

{button ,AL(`CAD_Offset_Menu;cad_transmenu;;;',0,"Defaultoverview",)} Related Topics

.Parallel (CAD)

.Parallel .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command draws one line or curve parallel to an existing line or curve.

Syntax	Description
.x1	Specifies the X-coordinate for the point on the existing line in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point on the existing line in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point on the existing line in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point that determines the location of the new line or curve in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point that determines the location of the new line or curve in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point that determines the location of the new line or curve in the document's default unit of measurement.

Note

- You must have drawn a curve or line before using the **.Parallel** command.
- This command corresponds to the Parallel command on the Transform. Click Transform, Parallel.

Example
```
REM units in inches
.SetPointXYZ -14.0, 4.0, 0.0
.SetPointXYZ -11.0, 6.0, 0.0
.SetPointXYZ -8.0, 0.0, 0.0
.SetPointXYZ -4.5, 5.0, 0.0
.BezierCurve
.Parallel -11.0, 1.0, 0.0, -10.0, 4.0, 0.0
```

This example creates a Bezier Curve and draws another bezier curve parallel to the original at a location specified by the coordinates of two points.

{button ,AL(`CAD_Parallel_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.PerpendicularPlane (CAD)

.PerpendicularPlane .width=*double*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command draws a plane perpendicular to an existing line.

Syntax	Description
.width	Specifies the width of the plane.
.x1	Specifies the X-coordinate of the point on the existing line in the document's default unit of measurement. The plane will be perpendicular to the line specified by this point.
.y1	Specifies the Y-coordinate of the point on the existing line in the document's default unit of measurement. The plane will be perpendicular to the line specified by this point.
.z1	Specifies the Z-coordinate of the point on the existing line in the document's default unit of measurement. The plane will be perpendicular to the line specified by this point.
.x2	Specifies the X-coordinate for the second point that determines the location of the plane in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that determines the location of the plane in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that determines the location of the plane in the document's default unit of measurement.

Note
- You must have drawn a line before using the **.PerpendicularPlane** command.
- This command corresponds to the Perpendicular Plane command on the Surfaces flyout of the Draw menu. Click Draw, Surfaces, Perpendicular Plane.

Example
```
REM units in inches
.SetPointXYZ -10.0, -0.5, 0.0
.SetPointXYZ -2.5, 2.0, 0.0
.LineSegment
.PerpendicularPlane 30, -7.0, 0.5, 0.0, -2.5, 2.0, 0.0
```

This example creates a line segment and then creates a perpendicular plane 30 inches wide at a location specified the coordinates of two points.

{button ,AL(`CAD_PerpendicularPlane_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.Plane (CAD)

.Plane .SaveAsSurface=*Boolean*

This command creates a plane.

Syntax	Description
.SaveAsSurface	Specifies whether the plane is saved as a surface or as a wire. Set to TRUE (-1) to save the plane as surface; set to FALSE (0) to save as a wire.

Note
- You must precede the command with **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint** and/or **.SetPointXYZ** to set the command points.
- This command corresponds to the Free Form command on the Surfaces flyout of the Draw menu. Click Draw, Surfaces, Free Form

Example
```
REM units in inches
.SetPointXYZ -16.0, 4.5, 0.0
.SetPointXYZ -11.0, 9.0, 0.0
.SetPointXYZ -6.0, 4.5, 0.0
.SetPointXYZ -11.5, -4.0, 0.0
.SetPointXYZ -22.0, 8.0, 0.0
.Plane -1
```

This example sets 5 command points and then creates a surface plane.

{button ,AL(`CAD_Plane_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.PolygonCenter (CAD)

.PolygonCenter .SaveAsSurface=*Boolean***, .NumberEdges=***integer***, .x1=***double***, .y1=***double***, .z1=***double***, .x2=***double***, .y2=***double***, .z2=***double***

The command creates a polygon.

Syntax	Description
.SaveAsSurface	Specifies whether the rectangle is saved as a surface or a wire. Set to TRUE (-1) to save the rectangle as surface; set to FALSE (0) to save as a wire.
.NumberEdges	Specifies the number of sides.
.x1	Specifies the X-coordinate for the center point of the polygon in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center point of the polygon in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center point of the polygon in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the corner of two adjacent edges of the polygon in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the corner of two adjacent edges of the polygon in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the corner of two adjacent edges of the polygon in the document's default unit of measurement.

Note
▪ This command corresponds to the Center and Vertex command on the Polygon flyout of the Draw menu. Click Draw, Polygon, Center and Vertex.

Example
```
REM units in inches
.PolygonCenter -1, -9.5, 6.5, 0.0, -8.5, 2.0, 0.0, 6
```

{button ,AL(`CAD_PolygonCenter_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.PolygonEdge (CAD)

.PolygonEdge .SaveAsSurface=*Boolean*, .NumberEdges=*integer*, .x1=*double*, .y1=*double*, .z1=*doubl e*, .x2=*double*, .y2=*double*, .z2=*double*

This command creates a polygon based on the edge set by two points.

Syntax	Description
.SaveAsSurface	Specifies whether the rectangle is saved as a surface or a wire. Set to TRUE (-1) to save the rectangle as surface; set to FALSE (0) to save as a wire.
.NumberEdges	Specifies the number of sides.
.x1	Specifies the X-coordinate for the first point of the edge in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point of the edge in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point of the edge in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point of the edge in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point of the edge in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point of the edge in the document's default unit of measurement.

Note
- The center of the polygon is on the right of the first edge.
- This command corresponds to the 2 Points command on the Polygon flyout of the Draw menu. Click Draw, Polygon, 2 Points.

Example
```
REM units in inches
.PolygonEdge 0, -9.0, 2.0, 0.0, -4.5, -1.0, 0.0, 6
```

{button ,AL(`CAD_PolygonEdge_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.PolygonalCylinder (CAD)

.PolygonalCylinder .SaveAsSurface=*Boolean*, .NumberEdges=*integer*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command creates a polygonal cylinder.

Syntax	Description
.SaveAsSurface	Specifies whether the cylinder is saved as a solid or a surface. Set to TRUE (-1) to save the rectangle as cylinder; set to FALSE (0) to save as a surface.
.NumberEdges	Specifies the number of sides.
.x1	Specifies the X-coordinate for the center of the base in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the base in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the base in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point on the vertex in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point on the vertex in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point on the vertex in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the height in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the height in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the height in the document's default unit of measurement.

Example
```
REM units in inches
.PolygonalCylinder -1, 18, -6.5, -0.5, 0.0, -5.0, 0.0, 0.0, -6.5, 4.0, 0.0
```

{button ,AL(`CAD_PolygonalCylinder_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.PolygonalFrustum (CAD)

.PolygonalFrustum .SaveAsSurface=*Boolean***, .NumberEdges=***integer***, .x1=***double***, .y1=***double***, .z1=
double, .x2=***double***, .y2=***double***, .z2=***double***, .x3=***double***, .y3=***double***, .z3=***double***, .x4=***double***, .y4=
double, z4=***double***

This command creates a polygonal frustum.

Syntax	Description
.SaveAsSurface	Specifies whether the frustum is saved as a solid or a surface. Set to TRUE (-1) to save the frustum as solid; set to FALSE (0) to save as a surface.
.NumberEdges	Specifies the number of sides.
.x1	Specifies the X-coordinate for the center of the base in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the base in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the base in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point on the vertex in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point on the vertex in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point on the vertex in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the height in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the height in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the height in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the scale of the top in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the scale of the top in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the scale of the top in the document's default unit of measurement.

Example
```
REM units in inches
.PolygonalFrustum -1, 16, -6.0, 1.0, 0.0, -3.5, -0.5, 0.0, -3.5, 4.5, 0.0, -3.0, 4.5, 0.0
```

{button ,AL(`CAD_PolygonalFrustum_Menu;cad_solid;;;;',0,"Defaultoverview",)} **Related Topics**

.PolygonalPyramid (CAD)

.PolygonalPyramid .SaveAsSurface=*Boolean*, .NumberEdges=*integer*, .x1=*double*, .y1=*double*, .z1= *double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command creates a polygonal pyramid.

Syntax	Description
.SaveAsSurface	Specifies whether the pyramid is saved as a solid or a surface. Set to TRUE (-1) to save the rectangle as solid; set to FALSE (0) to save as a surface.
.NumberEdges	Specifies the number of sides.
.x1	Specifies the X-coordinate for the center of the base in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the base in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the base in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point on the vertex in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point on the vertex in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point on the vertex in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the height in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the height in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the height in the document's default unit of measurement.

Example
```
REM units in inches
.PolygonalPyramid -1, 10, -6.0, -1.5, 0.0, -3.0, -0.5, 0.0, -7.5, 9.0, 0.0
```

{button ,AL(`CAD_PolygonalPyramid_Menu;cad_solid;;;;',0,"Defaultoverview",)} Related Topics

.PolyLine (CAD)

.PolyLine

This command creates a polyline.

Note
- You must precede the **.PolyLine** command with either **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint** or **.SetPointXYZ**.

Example
```
REM units in inches
.SetPointXYZ -12.0, 6.0, 0.0
.SetPointXYZ -6.5, 5.5, 0.0
.SetPointXYZ -4.0, -0.5, 0.0
.SetPointXYZ 2.5, -1.5, 0.0
.SetPointXYZ 2.5, -1.5, 0.0
.PolyLine
```

This example creates a polyline using **.SetPointXYZ**.

```
REM units in inches
.StartAddCmdPoint 5
.AddCmdPoint -12.0, 6.0, 0.0
.AddCmdPoint -6.5, 5.5, 0.0
.AddCmdPoint -4.0, -0.5, 0.0
.AddCmdPoint 2.5, -1.5, 0.0
.AddCmdPoint 2.5, -1.5, 0.0
.EndAddCmdPoint
.PolyLine
```

This example creates a polyline using **.StartAddCmdPoint**, **AddCmdPoint**, and **.EndAddCmdPoint**.

{button ,AL(`CAD_Line_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.Rectangle (CAD)

.
Rectangle .SaveAsSurface=*Boolean*,.x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, y3=*double*, .z3=*double*

This command draws either a two-point rectangle or a three-point rectangle.

Syntax	Description
.SaveAsSurface	Specifies whether the rectangle is saved as a surface or a wire. Set to TRUE (-1) to save the rectangle as a surface; set to FALSE (0) to save it as a wire.
.x1	Specifies the X-coordinate for the starting point of the rectangle in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the starting point of the rectangle in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the starting point of the rectangle in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point of the rectangle in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point (opposite corner for a 2 point rectangle, width for a three point rectangle) of the rectangle in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point (opposite corner for a 2 point rectangle, width for a three point rectangle) of the rectangle in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the third point (length) of the rectangle in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the third point (length) of the rectangle in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the third point (length) of the rectangle in the document's default unit of measurement.

Note
- **.SaveAsSurface**, is optional. If this parameter is not specified, the default (TRUE) is used.
- The third point is optional. If you do not specify the third point, a 2 point rectangle is created. If you specify the third point, a 3 point rectangle is created.

Example
```
REM units in inches
.Rectangle 0, -12.0, 3.0, 0.0, -5.0, -1.0, 0.0
```

This example creates a two-point rectangle.

{button ,AL(`CAD_Rectangle_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

RectangularFrustum (CAD)

.RectangularFrustum .SaveAsSolid=*Boolean***, .x1=***double***, .y1=***double***, .z1=***double***, .x2=***double***, .y2=***double***, .z2=***double***, .x3=***double***, .y3=***double***, .z3=***double***, .x4=***double***, .y4=***double***, .z4=***double***,** *.x5=double,* *.y5=double, .z5=double*

This command creates a either a two-point or a three-point rectangular frustum.

Syntax	Description
.SaveAsSolid	Specifies whether the box is saved as a solid or a surface. Set to TRUE (-1) to save the frustum as a solid; set to FALSE (0) to save the frustum as a surface. The default setting is TRUE.
.x1	Specifies the X-coordinate for the starting point of the frustum in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the starting point of the frustum in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the starting point of the frustum in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point (opposite corner for a 2 point frustum, width for a three point frustum) in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point (opposite corner for a 2 point frustum, width for a three point frustum) in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point (opposite corner for a 2 point frustum, width for a three point frustum) in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the third point (height of a 2 point frustum, length of a 3 point frustum) in the document's default unit of measurement. If you are creating a 2 point frustum, the third point must be positioned above or below the center of the base.
.y3	Specifies the Y-coordinate for the third point (height of a 2 point frustum, length of a 3 point frustum) in the document's default unit of measurement. If you are creating a 2 point frustum, the third point must be positioned above or below the center of the base.
.z3	Specifies the Z-coordinate for the third point (height of a 2 point frustum, length of a 3 point frustum) in the document's default unit of measurement. If you are creating a 2 point frustum, the third point must be positioned above or below the center of the base.
.x4	Specifies the X-coordinate for the fourth point (width of the second plane of a 2 point frustum, height of a 3 point frustum) in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the fourth point (width of the second plane of a 2 point frustum, height of a 3 point frustum) in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the fourth point (width of the second plane of a 2 point frustum, height of a 3 point frustum) in the document's default unit of measurement.
.x5	Specifies the X-coordinate for the fifth point, which determines the width of the second plane for a 3 point frustum in the document's default unit of measurement.
.y5	Specifies the Y-coordinate for the fifth point, which determines the width of the second plane for a 3 point frustum in the document's default unit of measurement.
.z5	Specifies the Z-coordinate for the fifth point, which determines the width of the second plane for a 3 point frustum in the document's default unit of measurement.

Note
- The fifth point is optional. If only four points are specified, a 2 point frustum is created. If all five points are specified, a 3 point frustum is created.

- If you are creating a 2 point frustum, the fourth point must be positioned above or below the center of the base.
- **.SaveAsSolid** is optional. If not specified, the default setting (TRUE) is used.

Example

```
REM units in inches
.RectangularFrustum -1, -5.0, -9.0, 5.5, 0.5, -9.0, 5.5, 0.0, -5.0, 5.5, 0.0, -5.0, 13.0, -2.0,
-6.5, 13.0
```

This example creates a 3-point rectangular frustum.

{button ,AL(`CAD_RectangularFrustum_Menu;cad_solid;;;;',0,"Defaultoverview",)} Related Topics

RectangularPyramid (CAD)

.RectangularPyramid .SaveAsSolid=*Boolean***, .x1=***double***, .y1=***double***, .z1=***double***, .x2=***double***, .y2=***double***, .z2=***double***, .x3=***double***, .y3=***double***, .z3=***double***, .***x4=*double*, .y4=*double*, .z4=*double*

This command creates a two-point or three-point pyramid with four sides.

Syntax	Description
.SaveAsSolid	Specifies whether the box is saved as a solid or a surface. Set to TRUE (-1) to save the pyramid as a solid; set to FALSE (0) to save the pyramid as a surface. The default setting is TRUE.
.x1	Specifies the X-coordinate for the starting point of the pyramid in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the starting point of the pyramid in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the starting point of the pyramid in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point (opposite corner for a 2 point pyramid, width for a three point pyramid) in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point (opposite corner for a 2 point pyramid, width for a three point pyramid) in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point (opposite corner for a 2 point pyramid, width for a three point pyramid) in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the third point (height of a 2 point pyramid, length of a 3 point pyramid) in the document's default unit of measurement. If you are creating a 2 point pyramid, the third point must be positioned above or below the center of the base.
.y3	Specifies the Y-coordinate for the third point (height of a 2 point pyramid, length of a 3 point pyramid) in the document's default unit of measurement. If you are creating a 2 point pyramid, the third point must be positioned above or below the center of the base.
.z3	Specifies the Z-coordinate for the third point (height of a 2 point pyramid, length of a 3 point pyramid) in the document's default unit of measurement. If you are creating a 2 point pyramid, the third point must be positioned above or below the center of the base.
.x4	Specifies the X-coordinate for the fourth point (height of a 3 point pyramid) in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the fourth point (height of a 3 point pyramid) in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the fourth point (height of a 3 point pyramid) in the document's default unit of measurement.

Note

- The fourth point is optional. If only 3 points are specified, a 2 point pyramid is created. If all four points are specified, a 3 point pyramid is created.
- If you are creating a 2 point pyramid, the third point must be positioned above or below the center of the base.
- **.SaveAsSolid** is optional. If not specified, the default setting (TRUE) is used.

Example
```
REM units in inches
.RectangularPyramid -1, -7.5, -4.0, 0.0, -2.0, -1.0, 0.0, -2.0, -1.0, 9.5
```

This example creates a 2 point rectangular pyramid.

{button ,AL(`CAD_RectangularPyramid_Menu;cad_solid;;;;',0,"Defaultoverview",)} Related Topics

.Redo (CAD)

.Redo

This command restores changes reversed by the **.Undo** command.

Note
- This command corresponds to the Redo command on the Edit menu. Click Edit, Redo.

Example
`.Redo`

{button ,AL(`CAD_Redo_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} Related Topics

.Rotate (CAD)

.Rotate .LeaveOriginal=*Boolean*, .angle=*double*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2 =*double*, .z2=*double*

This command rotates the selected object through the specified angle along the axis determined by one two points.

Syntax	Description
.LeaveOriginal	Specifies whether a duplicate of the object is created when the object is rotated. Set to TRUE (-1) to duplicate the object; set to FALSE (0) to not duplicate the object.
.angle	Specifies the angle of the rotation in degrees.
.x1	Specifies the X-coordinate for the first point in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point in the document's default unit of measurement. This point is optional. If not specified, the rotation of the object is determined by one point.
.y2	Specifies the Y-coordinate for the second point in the document's default unit of measurement. This point is optional. If not specified, the rotation of the object is determined by one point.
.z2	Specifies the Z-coordinate for the second point the in document's default unit of measurement. This point is optional. If not specified, the rotation of the object is determined by one point.

Note
- An object must be selected before using this command.
- This command corresponds to the Rotate command on the Transform menu. Click Transform, Rotate.

Example
```
REM units in inches
.RectangularPyramid -1, -7.5, -4.0, 0.0, -2.0, -1.0, 0.0, -2.0, -1.0, 9.5
.Rotate -1, 90, 0.0, 0.0, 0.0
```

This example rotates the pyramid 90 degrees along the axis determined by one point.

```
REM units in inches
.RectangularPyramid -1, -7.5, -4.0, 0.0, -2.0, -1.0, 0.0, -2.0, -1.0, 9.5
.Rotate -1, 180, -1.5, -6.0, 0.0, -10.5, -2.5, 0.0
```

This example rotates the pyramid 180 degrees along an axis determined by two points.

{button ,AL(`CAD_Rotate_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.RotateNormal (CAD)

.RotateNormal .LeaveOriginal=*Boolean*, .angle=*double*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*, .x4=*double*, .y4=*double*, .z4=*double*

This command rotates the selected object through the specified angle along the axis defined by a normal to a plane.

Syntax	Description
.LeaveOriginal	Specifies whether a duplicate of the object is created when the object is rotated. Set to TRUE (-1) to duplicate the object; set to FALSE (0) to not duplicate the object.
.angle	Specifies the angle of the rotation in degrees.
.x1	Specifies the X-coordinate for the first point that defines the plane in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that defines the plane in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that defines the plane in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that defines the plane in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that defines the plane in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that defines the plane the in document's default unit of measurement.
.x3	Specifies the X-coordinate for the third point that defines the plane in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the third point that defines the plane in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the third point that defines the plane in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the fourth point that defines the location of the axis in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the fourth point that defines the location of the axis in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the fourth point that defines the location of the axis the in document's default unit of measurement.

Note
- An object must be selected before using this command.
- This command corresponds to the Rotate command on the Transform menu. Click Transform, Rotate.

Example
```
REM units in inches
.RectangularPyramid -1, -7.5, -4.0, 0.0, -2.0, -1.0, 0.0, -2.0, -1.0, 9.5
.RotateNormal -1, 180, -2.0, -1.0, 0.0, -1.5, 2.5, 0.0, 1.0, 0.0, 0.0, 2.5, -4.0, 0.0
```

This example creates a rectangular pyramid, and then creates a duplicate of the object. The duplicate is rotated 180 degrees and placed on the plane specified by the coordinates of the first three points at the location of the axis specified by the fourth point.

{button ,AL(`CAD_RotateNormal_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.RotateXAxis (CAD)

.RotateXaxis .LeaveOriginal=*Boolean*, .angle=*double*, .x1=*double*, .y1=*double*, .z1=*double*

This command rotates the selected object through the specified angle along the X-axis.

Syntax	Description
.LeaveOriginal	Specifies whether a duplicate of the object is created when the object is rotated. Set to TRUE (-1) to duplicate the object; set to FALSE (0) to not duplicate the object.
.angle	Specifies the angle of the rotation in degrees.
.x1	Specifies the X-coordinate for the point that determines the center of rotation in the document's default unit of measurement. This parameter is optional.
.y1	Specifies the Y-coordinate for the point that determines the center of rotation in the document's default unit of measurement. This parameter is optional.
.z1	Specifies the Z-coordinate for the point that determines the center of rotation in the document's default unit of measurement. This parameter is optional.

Note
- An object must be selected before using this command.
- The parameters for the coordinates of the point are optional. If you do not specify a point, the object is rotated about the X-axis at its current location. If you specify a point, the object is rotated about the X-axis in the new location.
- This command corresponds to the Rotate command on the Transform menu. Click Transform, Rotate.

Example
```
REM units in inches
.RectangularPyramid -1, -7.5, -4.0, 0.0, -2.0, -1.0, 0.0, -2.0, -1.0, 9.5
.RotateXAxis -1, 180, -0.5, 3.0, 0.0
```

This example creates a rectangular pyramid, and then creates a duplicate of the object. The duplicate is rotated 180 degrees around the X-axis. The center of rotation is determined by the coordinates of one point. If the center of rotation was not specified, the object would be rotated at its current location.

{button ,AL(`CAD_RotateXAxis_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.RotateYAxis (CAD)

.RotateYAxis .LeaveOriginal=*Boolean*, .angle=*double*, .x1=*double*, .y1=*double*, .z1=*double*

This command rotates the selected object through the specified angle along the Y-axis.

Syntax	Description
.LeaveOriginal	Specifies whether a duplicate of the object is created when the object is rotated. Set to TRUE (-1) to duplicate the object; set to FALSE (0) to not duplicate the object.
.angle	Specifies the angle of the rotation in degrees.
.x1	Specifies the X-coordinate for the point that determines the center of rotation in the document's default unit of measurement. This parameter is optional.
.y1	Specifies the Y-coordinate for the point that determines the center of rotation in the document's default unit of measurement. This parameter is optional.
.z1	Specifies the Z-coordinate for the point that determines the center of rotation in the document's default unit of measurement. This parameter is optional.

Note

- An object must be selected before using this command.
- The parameters for the coordinates of the point are optional. If you do not specify a point, the object is rotated about the Y-axis at its current location. If you specify a point, the object is rotated about the Y-axis in the new location.
- This command corresponds to the Rotate command on the Transform menu. Click Transform, Rotate.

Example
```
REM units in inches
.RectangularPyramid -1, -7.5, -4.0, 0.0, -2.0, -1.0, 0.0, -2.0, -1.0, 9.5
.RotateYAxis -1, 180, 0.0, -1.0, 0.0
```

This example creates a rectangular pyramid, and then creates a duplicate of the object. The duplicate is rotated 180 degrees around the Y-axis. The center of rotation is determined by the coordinates of one point. If the center of rotation was not specified, the object would be rotated at its current location.

{button ,AL(`CAD_YAxis_Menu;cad_transmenu;;;',0,"Defaultoverview",)} <u>**Related Topics**</u>

.RotateZAxis (CAD)

.RotateZAxis .LeaveOriginal=*Boolean*, .angle=*double*, .x1=*double*, .y1=*double*, .z1=*double*

This command rotates the selected object through the specified angle along the Z-axis.

Syntax	Description
.LeaveOriginal	Specifies whether a duplicate of the object is created when the object is rotated. Set to TRUE (-1) to duplicate the object; set to FALSE (0) to not duplicate the object.
.angle	Specifies the angle of the rotation in degrees.
.x1	Specifies the X-coordinate for the point that determines the center of rotation in the document's default unit of measurement. This parameter is optional.
.y1	Specifies the Y-coordinate for the point that determines the center of rotation in the document's default unit of measurement. This parameter is optional.
.z1	Specifies the Z-coordinate for the point that determines the center of rotation in the document's default unit of measurement. This parameter is optional.

Note

- An object must be selected before using this command.
- The parameters for the coordinates of the point are optional. If you do not specify a point, the object is rotated about the Z-axis at its current location. If you specify a point, the object is rotated about the Z-axis in the new location.

Example
```
REM units in inches
.RectangularPyramid -1, -7.5, -4.0, 0.0, -2.0, -1.0, 0.0, -2.0, -1.0, 9.5
.RotateZAxis -1, 180, -1.5, 1.0, 0.0
```

This example creates a rectangular pyramid, and then creates a duplicate of the object. The duplicate is rotated 180 degrees around the Z-axis. The center of rotation is determined by the coordinates of one point. If the center of rotation was not specified, the object would be rotated at its current location.

{button ,AL(`CAD_RotateZAxis_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.Scale (CAD)

.Scale .LeaveOriginal=*Boolean*, .Factor=*double*, .X=*double*, .Y=*double*, .Z=*double*

This command scales the selected object(s).

Syntax	Description
.LeaveOriginal	Specifies whether a duplicate of the object is created when the object is scaled. Set to TRUE (-1) to duplicate the object; set to FALSE (0) to not duplicate the object.
.Factor	Specifies the scaling factor in the document's default unit of measurement.
.X	Specifies the X-coordinate for the base point in the document's default unit of measurement.
.Y	Specifies the Y-coordinate for the base point in the document's default unit of measurement.
.Z	Specifies the Z-coordinate for the base point in the document's default unit of measurement.

Note

- This command corresponds to the Scale command on the Transform menu. Click Transform, Scale.

Example
```
REM units in inches
.Torus -1, -9.0, -0.5, 0.0, -7.0, -2.0, 0.0, -8.0, -1.0, 0.0, -7.5, -1.5, 0.0
.Scale -1, 1.5, -8.0, -13.0, 0.0
```

This example creates a torus, creates a duplicate and then scales it by a factor of 1.5. The duplicate is placed at the location specified by the coordinates of one point.

{button ,AL(`CAD_Scale_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.SelectAll (CAD)

.SelectAll

This command selects all of the objects in the current document.

Note
- This command corresponds to the Select All command on the Edit menu. Click Edit, Select All.

Example
```
REM units in inches
.Torus -1, -9.0, -0.5, 0.0, -7.0, -2.0, 0.0, -8.0, -1.0, 0.0, -7.5, -1.5, 0.0
.Scale -1, 1.5, -8.0, -13.0, 0.0
.SelectAll
```

This example creates a torus, creates a scaled version of the torus in another location and then selects both objects.

{button ,AL(`CAD_SelectAll_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} Related Topics

.SelectNext (CAD)

.SelectNext

This command selects the next object in the current document.

Note

▪ The objects are selected in the order they are created; therefore this command selects the object created after the currently selected object.

Example
```
REM units in inches
.Rectangle 0, -65.5365, 28.402, -6.31773, -58.35, 23.0614, -6.31773
.Sphere -1, -49.9314, 31.7953, -6.31773, -48.1257, 28.334, -6.31773
.Box -1, -53.0743, 21.6508, -6.31773, -45.9882, 17.2436, -10.2424
.SelectPointAt -54.7808, 39.875, -10.2424, 0
.SelectNext
```

This example creates three objects and then selects the sphere using **.SelectPointAt**. The **.SelectNext** command selects the box which is the object created after the sphere. To see this command work, you should create two scripts. The first script should contain the commands to create the objects and select the sphere. The second script should contain only the **.SelectNext** command. This will demonstrate the order of selection used by the **.SelectNext** command.

{button ,AL(`CAD_SelectNext_Menu;cad_select;;;;',0,"Defaultoverview",)} **Related Topics**

.SelectObject (CAD)

.SelectObject .Name=*string*

This command selects all of the objects in the current document with the same name.

Syntax	Description
.Name	Specifies the name of the objects to select.

Note

- You must have named objects before using this command.

Example

```
REM units in inches
.Torus -1, -9.0, -0.5, 0.0, -7.0, -2.0, 0.0, -8.0, -1.0, 0.0, -7.5, -1.5, 0.0
.ApplyName "Canada"
.RectangularPyramid -1, -7.5, -4.0, 0.0, -2.0, -1.0, 0.0, -2.0, -1.0, 9.5
.ApplyName "Canada"
.SelectObject "Canada"
```

This example creates a torus and a pyramid and applies the same name to both. All objects with the name "Torusname" are selected.

{button ,AL(`CAD_SelectObject_Menu;cad_name;;;;',0,"Defaultoverview",)} <u>**Related Topics**</u>

.SelectPointAt (CAD)

Command:
.SelectPointAt .X=*double*, .Y=*double*, .Z=*double,* Flag=*integer*

Function:
ReturnValue =.SelectPointAt .X=*double*, .Y=*double*, .Z=*double,* Flag=*integer*

This command selects the object at the specified point.

Syntax	Description
ReturnValue	**.SelectPointAt** can also be used as a function. This parameter specifies the variable that is passed the return value corresponding to whether the command has selected a point. It is passed TRUE (-1) if a point has been selected; otherwise FALSE (0). This variable must be declared using a <u>DIM</u> statement.
.X	Specifies the X-coordinate for the selected point in the document's default unit of measurement.
.Y	Specifies the Y-coordinate for the selected point in the document's default unit of measurement.
.Z	Specifies the Z-coordinate for the selected point in the document's default unit of measurement.
.Flag	Specifies whether 1 or more points is to be selected. Set <=0 to do a single selection (any previous selection is canceled). Set >0 to do a multiple selection (previous selections are still selected).

Example
```
REM units in inches
.SetPointXYZ -14.0, 5.5, 0.0
.SetPointXYZ -8.0, 4.5, 0.0
.SetPointXYZ -9.0, -1.5, 0.0
.SetPointXYZ -1.5, 1.0, 0.0
.SetPointXYZ -1.5, -3.0, 0.0
.LineSegment
.SelectPointAt -8.0, 4.5, 0.0. 0
```

TThis example creates a line segment and then selects it by selecting a point on the line segment. To see how the **.SelectPointAt** command works, you should create two scripts. The first script should create the line segment. Ensure that after you have created the line segment, it is deselected. Then create a second script that selects a point on the line segment.

```
REM units in inches
.SetPointXYZ -14.0, 5.5, 0.0
.SetPointXYZ -8.0, 4.5, 0.0
.SetPointXYZ -9.0, -1.5, 0.0
.SetPointXYZ -1.5, 1.0, 0.0
.SetPointXYZ -1.5, -3.0, 0.0
.LineSegment
ReturnValue=.SelectPointAt -8.0, 4.5, 0.0. 0
```

This example demonstrates **.SelectPointAt** being used as a function.

{button ,AL(`CAD_SelectPointAt_Menu;cad_select;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.SelectPrevious (CAD)

.SelectPrevious

This command selects the previous object in the current document.

Note
- The objects are selected in the order they are created.

Example
```
REM units in inches
.Torus -1, -9.0, -0.5, 0.0, -7.0, -2.0, 0.0, -8.0, -1.0, 0.0, -7.5, -1.5, 0.0
.Box -1, -14.5, 4.0, -1.5, -13.5, 3.0, -2.5
.Box -1, -15.0, -5.5, -10.0, 7.5, -5.5, -11.0, 4.0, -5.5, -11.0, 4.0, -10.0
.SelectPrevious
```

This example creates a torus and two boxes and selects the first box.

{button ,AL(`CAD_SelectPrevious_Menu;cad_select;;;;',0,"Defaultoverview",)} Related Topics

.SelectWithinRegion (CAD)

Command:
.SelectWithinRegion .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

Function:
ReturnValue
=.SelectWithinRegion .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command selects all objects within the specified region.

Syntax	Description
ReturnValue	**.SelectWithinRegion** can also be used as a function. This parameter specifies the variable that is passed the return value corresponding to whether the command has selected an object. It is passed TRUE (-1) if objects have been selected; otherwise FALSE (0). This variable must be declared using a <u>DIM</u> statement.
.x1	Specifies the X-coordinate for the first point (starting corner) of the region in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point (starting corner) of the region in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point (starting corner) of the region in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point (opposite corner) of the region in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point (opposite corner) of the region in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point (opposite corner) of the region in the document's default unit of measurement.

Example
```
REM units in inches
.Cone -1, -12.5, 4.0, 0.0, -10.5, 4.5, 0.0, -12.5, 9.0, 0.0
.Box -1, -14.0, 0.5, 0.0, -10.0, -2.0, -4.0
.SelectWithinRegion -19.0, 11.5, -4.0, -8.0, -2.0, -4.0
```

This example creates a cone and a box and then selects both by determining a region specified by the coordinates of two points.

```
REM units in inches
.Cone -1, -12.5, 4.0, 0.0, -10.5, 4.5, 0.0, -12.5, 9.0, 0.0
.Box -1, -14.0, 0.5, 0.0, -10.0, -2.0, -4.0
ReturnValue = .SelectWithinRegion -19.0, 11.5, -4.0, -8.0, -2.0, -4.0
```

This example demonstrates **.SelectWithinRegion** being used as a function.

{button ,AL(`CAD_SelectWithinRegion_Menu;cad_select;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.SetCurrentLayer (CAD)

.SetCurrentLayer .LayerName=*string*, .GroupName=*string*

This command sets the current layer and layer group.

Syntax	Description
.LayerName	Specifies the name of the layer.
.GroupName	Specifies the name of the layer group. The default layer group is "All Layers".

Note

- **.GroupName** is an optional parameter.
- If **.LayerName** refers to a non-existing layer and **.GroupName** refer to non-existing group, nothing occurs.
- This command corresponds to the Set Current Layer and Set Current Group commands on the Layers flyout of the Tools menu.

Example
```
.NewLayer "mylayer", , -1, 0, 0, 0, 0, 0, 1, "second layer"
.NewLayer "my2layer", , -1, 0, 0, 0, 0, 0, 1, "third layer"
.SetCurrentLayer "mylayer"
```

This example creates two new layers and sets the current layer to "mylayer" and sets the layer group to "All Layers".

{button ,AL(`CAD_NewLayerGroup_Menu;cad_toolsmenu;;;;',0,"Defaultoverview",)} Related Topics

.SetEdgeRefinement (CAD)

.SetEdgeRefinement .nView=*integer*, .nValue=*integer*

This command sets the edge refinement for a specified view.

Syntax	Description
.nView	Specifies view type. 0 rendered 1 hidden line 2 model
.nValue	Specifies edge refinement. The values are 0 to 10. Smaller values cause coarser edge refinement.

Note
- This command corresponds to the Refinements command on the View menu. Click View, Refinements.

Example
```
.SetEdgeRefinement 0, 8
```

{button ,AL(`CAD_SetEdgeRefinement_Menu;cad_viewmenu;;;;',0,"Defaultoverview",)} Related Topics

.SetLightOff (CAD)

.SetLightOff .LightNo=*integer*

This command turns off the specified light.

Syntax	Description
.LightNo	Specifies the light to be turned off. The lights are numbered 0 to 7.

Note

- This command corresponds to the Shade command on the View menu. Click View, Shade, Lights.

Example
```
.SetLightOff 3
```

{button ,AL(`CAD_SetLightOff_Menu;cad_viewmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.SetLightOn (CAD)

.SetLightOn .LightNo=*integer*, .Intensity=*double*, .HorizontalAngle=*double*, .VerticalAngle=*double*, . Red=*integer*, .Green=*integer*, .Blue=*integer*

This command turns the specified light on and determines the intensity, the horizontal and vertical angles, and color of the light. You can only turn one light on each time you use the command. If you want to turn another light on, use **.SetLightOn** again.

Syntax	Description
.LightNo	Specifies the location of the light source. The lights are numbered 0 to 7.
.Intensity	Specifies the intensity of the light source. The values are 0 to 10.
.HorizontalAngle	Specifies the horizontal angle of the light source relative to the object. The values are -180 to 180.
.VerticalAngle	Specifies the vertical angle of the light source relative to the object. The values are -90 to 90.
.Red	Specifies the red setting for light color (RGB color model). Valid values range from 0 to 255. This parameter works in conjunction with **.Green** and **.Blue** to determine the color of the light.
.Green	Specifies the green setting for light color (RGB color model). Valid values range from 0 to 255. This parameter works in conjunction with **.Red** and **.Blue** to determine the color of the light.
.Blue	Specifies the blue setting for light color (RGB color model). Valid values range from 0 to 255. This parameter works in conjunction with **.Red** and **.Green** to determine the color of the light.

Note
- This command corresponds to the Shade command on the View menu. Click View, Shade, Lights.

Example
```
.SetLightOn 3, 25.0, -70, -45. 255, 0, 255
```

{button ,AL(`CAD_SetLightOn_Menu;cad_viewmenu;;;;',0,"Defaultoverview",)} Related Topics

.SetPointXYZ (CAD)

.SetPointXYZ .X=*double*, .Y=*double*, .Z=*double*

This command sets a point at the specified X, Y, and Z-coordinates.

Syntax	Description
.X	Specifies the X-coordinate for the point in the document's default unit of measurement.
.Y	Specifies the Y-coordinate for the point in the document's default unit of measurement.
.Z	Specifies the Z-coordinate for the point in the document's default unit of measurement.

Example
```
REM units in inches
.SetPointXYZ -12.0, 4.5, 0.0
.SetPointXYZ -2.0, 1.0, 0.0
.SetPointXYZ -0.5, 0.5, 0.0
.ArrowLine 10, 4
```

This example sets three points and then creates a line with an arrow.

{button ,AL(`CAD_SetPointXYZ_Menu;cad_point;;;;',0,"Defaultoverview",)} Related Topics

.SetSurfaceRefinement (CAD)

.SetSurfaceRefinement .nView=*integer*, .nValue=*integer*

This command sets the surface refinement for a specified view.

Syntax	Description
.nView	Specifies view type. 0 rendered 1 hidden line 2 model
.nValue	Specifies surface refinement. The values are 0 to 10. Smaller values cause coarser surface refinement.

Note

- This command corresponds to the Refinements command on the View menu. Click View, Refinements.

Example
```
.SetSurfaceRefinement 2, 5
```

{button ,AL(`CAD_SetSurfaceRefinement_Menu;cad_viewmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.SetUnits (CAD)

.SetUnits .Units=*integer*

This command sets the active document's default unit of measurement and sets the measurement unit used in CorelCAD application commands. You should include a **.SetUnits** command at the beginning of all of your CorelCAD scripts. This command ensures that your CorelCAD scripts are executed using the correct unit of measurement for those commands that use measurement parameters.

Syntax	Description
.Units	Specifies the unit of measurement as follows
	0 inches
	1 millimeters
	2 centimeters
	3 feet
	4 miles
	5 meters
	6 kilometers

Note:
- This command changes only the active document's default unit of measurement settting.
- This command cannot be recorded.

Example
```
.SetUnits 2
```

{button ,AL(`CAD_SetUnits_Menu;cad_toolsmenu;;;;',0,"Defaultoverview",)} Related Topics

.SetVisible (CAD)

.SetVisible .Visible=*Boolean*

This command makes the CorelCAD application visible or hidden on your Windows desktop. When CorelCAD is hidden, it runs in the Windows background and is not visible on screen. Running as a hidden application can make a CorelCAD script run faster since it does not have to redraw a diagram when changes are applied to it.

Syntax	Description
.Visible	Specifies whether CorelCAD is visible or hidden. Set to TRUE (-1) to show the CorelCAD application. Set to FALSE (0) to hide the application.

Note
- By clicking CTRL+ALT+DELETE on the Windows desktop, you can find all of the applications Windows is running, both visible and hidden.
- This command is cannot be recorded.

Example
```
.SetVisible -1
```

The above example shows the CorelCAD application.

{button ,AL(`CAD_SetUnits_Menu;cad_viewmenu;;;;',0,"Defaultoverview",)} Related Topics

.ShadeEntireView (CAD)

.ShadeEntireView .ShowText=*Boolean*, .ShowDimension=*Boolean*, .ShadeType=*integer*, .IsAllViews= *Boolean*

This command shades the entire view.

Syntax	Description
.ShowText	Specifies whether the text is displayed. If set to TRUE (-1), the text is displayed; if set to FALSE (0) the text is not displayed.
.ShowDimension	Specifies whether dimensions are displayed. If set to TRUE (-1), the dimensions are displayed; if set to FALSE (0), the dimensions are not displayed.
.ShadeType	Specifies the shading type. The values are as follows: 0 Flat Shading 1 Gouraud Shading 2 Phong Shading 3 Preview 4 Full Render 5 Ray-traced preview 6 Ray-traced full render
.IsAllViews	Specifies whether all views are shaded. If set to TRUE (-1), all views are shaded; if set to FALSE (0), only the active view is shaded.

Note
- This command corresponds to the Shade command on the View menu. Click View, Shade.

Example
```
.ShadeEntireView -1, -1, 1, -1
```

{button ,AL(`CAD_ShadeEntireView_Menu;cad_viewmenu;;;;',0,"Defaultoverview",)} Related Topics

.ShadeSectionView (CAD)

.ShadeSectionView .ShowText=*Boolean*, .ShowDimensions=*Boolean*, .ShadeType=*integer*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command shades a view determined by the coordinates of two points.

Syntax	Description
.ShowText	Specifies whether the text is displayed. If set to TRUE (-1), the text is displayed; if set to FALSE (0) the text is not displayed.
.ShowDimension	Specifies whether dimensions are displayed. If set to TRUE (-1), the dimensions are displayed; if set to FALSE (0), the dimensions are not displayed.
.ShadeType	Specifies the shading type. The values are as follows: 0 Flat Shading 1 Gouraud Shading 2 Phong Shading 3 Preview 4 Full Render 5 Ray-traced preview 6 Ray-traced full render
.x1	Specifies the X-coordinate for the first point (starting corner) of the region in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point (starting corner) of the region in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point (starting corner) of the region in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point (opposite corner) of the region in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point (opposite corner) of the region in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point (opposite corner) of the region in the document's default unit of measurement.

Note

▪ This command corresponds to the Shade command on the View menu. Click View, Shade.

Example
```
.Box -1, -14.0, 7.0, -5.0, -8.5, 5.0, -10.0
.ShadeSectionView -1, -1, 1, -18.0, 13.0, -10.0, -6.0, 4.0, -10.0
```

This example creates a box and then shades the section that contains the box. Text and dimension lines are displayed. The shade type is Flat Shading. The region is specified by the coordinates of two points.

{button ,AL(`CAD_ShadeSectionView_Menu;cad_viewmenu;;;;',0,"Defaultoverview",)} Related Topics

.ShadeSelected (CAD)

.ShadeSelected .ShowText=*Boolean*, .ShowDimension=*Boolean*, .ShadeType=*integer*

This command shades the currently selected object.

Syntax	Description
.ShowText	Specifies whether the text is displayed. If set to TRUE (-1), the text is displayed; if set to FALSE (0), the text is not displayed.
.ShowDimension	Specifies whether dimensions are displayed. If set to TRUE (-1), the dimensions are displayed; if set to FALSE (0), the dimensions are not displayed.
.ShadeType	Specifies the shading type. The values are as follows:
	0 Flat Shading
	1 Gouraud Shading
	2 Phong Shading
	3 Preview
	4 Full Render
	5 Ray-traced preview
	6 Ray-traced full render

Note
- You must select the object before running this command.

Example
```
REM units in inches
.Box -1, -14.5, 13.0, -10.0, -10.5, 10.0, -14.5
.ChangeColor 255, 0, 204
.ShadeSelected -1, -1, 4
```

This example creates a box, changes the color of the box and then shades it. Text and Dimension lines are displayed. The shade type is Full Render.

{button ,AL(`CAD_ShadeSelected_Menu;cad_viewmenu;;;;',0,"Defaultoverview",)} Related Topics

.Skin (CAD)

.Skin

This command creates a surface between two or more existing lines.

Note
- The lines must be drawn before using the Skin command.
- The points on the lines must be created using **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint** and/or **.SetPointXYZ**.
- This command corresponds to the Skin command on the Surfaces flyout of the Draw menu. Click Draw, Surfaces, Skin.

Example
```
REM units in inches
.SetPointXYZ -17.5, 19.0, -11.5
.SetPointXYZ -17.5, 19.0, -22.5
.LineSegment
.SetPointXYZ -16.0, 27.0, -22.5
.SetPointXYZ -16.0, 27.0, -33.0
.LineSegment
.SetPointXYZ -16.0, 43.0, -33.0
.SetPointXYZ -16.0, 43.0, -43.5
.LineSegment
.SelectAll
.Skin
```

This example creates three line segments, selects all of the lines, and then creates a surface between the lines.

{button ,AL(`CAD_Skin_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.SlicePlane (CAD)

.SlicePlane .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command slices a plane with a line or curve whose two end points intersect with the plane's edge.

Syntax	Description
.x1	Specifies the X-coordinate for the point lying on the cutting line in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point lying on the cutting line in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point lying on the cutting line in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point which lies on the plane in the document's unit of measurement.
.y2	Specifies the Y-coordinate for the point which lies on the plane in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point which lies on the plane in the document's default unit measurement.

Note

- The plane and the line must already exist before using the **.SlicePlane** command.
- This command corresponds to the Slice Plane command on the Slice flyout of the Transform menu. Click Transform, Slice, Slice Plane.

Example
```
REM units in inches
.SetPointXYZ -24.5, 30.5, -25.5
.SetPointXYZ -20.0, 27.5, -25.5
.SetPointXYZ -10.5, 26.5, -25.5
.SetPointXYZ -8.0, 33.5, -25.5
.Multiplane -1, -1
.SetPointXYZ -16.5, 32.0, -25.5
.SetPointXYZ -21.5, 21.0, -25.5
.LineSegment
.SlicePlane -19.62, 25.635, -25.5, -17.635, 28.264, -25.5
```

This example creates a multiplane and a line segment. The multiplane is then sliced with the line segment.

{button ,AL(`CAD_SlicePlane_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.SolidAdd (CAD)

.SolidAdd

This command adds solid objects to other solid objects to create a new object.

Note
- You must precede this command with either **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint** or **.SetPointXYZ** to select the objects.
- This command corresponds to the Add command on the Boolean flyout of the Transform menu. Click Transform, Boolean, Add.

Example
```
REM units in inches
.Box -1, 115.0, 43.8, -42.5, -94.6, 21.1, -72.5
.Box -1, -112.0, 81.3, -72.5, -83.0, 48.3, -99.7
.SetPointXYZ -104.8, 43.8, -42.6
.SetPointXYZ -112.0, 81.3, -72.5
.SolidAdd
```

This example creates two boxes. Both boxes are then selected. To select both boxes, the first box is selected using **.SetPointXYZ**. The last box is already selected because it has just been created. The first box is then added to the second box to create a new object.

{button ,AL(`CAD_SolidAdd_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.SolidDefine (CAD)

.SolidDefine

This command creates a solid object from the selected objects.

Note

- You must have objects selected before using this command.
- This command corresponds to the Define Object command on the Transform menu. Click Transform, Define Object.

Example
```
REM units in inches
.Rectangle 0, 128.9, 91.6, -72.5, -69.8, 54.2, -72.5
.Rectangle 0, -95.9, 105.4, -72.5, -39.7, 41.5, -72.5
.SelectWithinRegion -171.6, 126.8, -72.5, 23.3, 32.0, -72.5
.SolidDefine
```

This example creates two rectangles.Both rectangles are then selected using **.SelectWithinRegion**. Then the objects are combined to create a new solid object

---

{button ,AL(`CAD_SolidDefine_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.SolidExplode (CAD)

.SolidExplode

This command breaks the selected object into its component parts (planes, surface meshes, and lines).

Note
- You must have objects selected before using this command.
- This command corresponds to the Explode Object command on the Transform menu. Click Transform, Explode Object.

Example
```
REM units in inches
.Box -1, -151.4, 95.9, -72.5, -108.9, 63.8, -105.0
.SolidExplode
```

This example creates a box and then breaks the box into its component parts.

_____

{button ,AL(`CAD_SolidExplode_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.SolidIntersect (CAD)

.SolidIntersect

This command creates a new solid object from the overlapped section of existing solid objects.

Note
- You must precede this command with either **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint** or **.SetPointXYZ** to select the objects.
- This command corresponds to the Intersect command on the Boolean flyout of the Transform menu.

Example
```
REM units in inches
.Box -1, -156.6, 3.9, 0.0, -105.3, 40.4, -26.6
.Sphere -1, -156.6, 22.1, -26.6, -153.9, 16.1, -26.6
.SetPointXYZ -130.954, 3.86937, 0.0
.SetPointXYZ -156.6, 22.1127, -26.6011
.SolidIntersect
```

This example creates a box and a sphere and then selects both. A new object is created from the overlapping section of the two objects.

{button ,AL(`CAD_SolidIntersect_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.SolidSubtract (CAD)

.SolidSubtract

This command subtracts objects from the first selected object.

Note

- This command corresponds to the Intersect command on the Boolean flyout of the Transform menu. Click Transform, Boolean, Intersect.
- The objects must be selected using either **.StartAddCmdPoint**, **.AddCmdPoint**, and **.EndAddCmdPoint** or **.SetPointXYZ** commands to select the objects.

Example

```
REM units in inches
.Box -1, -214.6, 166.5, -160.6, -176.0, 144.0, -205.3
.Box -1, -214.6, 166.5, -160.6, -191.6, 113.6, -205.5
.SetPointXYZ -191.6, 113.6, -160.6
.SetPointXYZ -176.0, 155.276, -160.6
.SolidSubtract
```

This example creates two boxes and then selects both. The second box is subtracted from the first.

{button ,AL(`CAD_SolidSubtract_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.Sphere (CAD)

.Sphere .SaveAsSolid=Boolean, .x1=double, .y1=double, .z1=double, .x2=double, .y2=double, .z2=double

This command creates a sphere based on coordinates which specify the center and the radius of the sphere.

Syntax	Description
.SaveAsSolid	Specifies whether the sphere is saved as a solid or a surface. Set to TRUE (-1) to save the sphere as a solid; set to FALSE (0) to save the sphere as a surface.
.x1	Specifies the X-coordinate for the center of the sphere in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the sphere in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the sphere in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the radius of the sphere in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the radius of the sphere in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the radius of the sphere in the document's default unit of measurement.

Note
- This command corresponds to the Sphere, Center and Radius command on the Solids flyout of the Draw menu. Click Draw, Solids, Sphere, Center and Radius.

Example
```
REM units in inches
.Sphere -1, -191.3, 168.0, -160.6, -179.4, 151.7, -160.0
```

{button ,AL(`CAD_Sphere_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.SphereDiameter (CAD)

.SphereDiameter .SaveAsSolid=*Boolean*, **.x1=***double*, **.y1=***double*, **.z1=***double*, **.x2=***double*, **.y2=***double*, **.z2=***double*

This command creates a sphere based on coordinates that specify the diameter of the sphere.

Syntax	Description
.SaveAsSolid	Specifies whether the sphere is saved as a solid or a surface. Set to TRUE (-1) to save the sphere as a solid; set to FALSE (0) to save the sphere as a surface.
.x1	Specifies the X-coordinate for the first point that determines the diameter in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that determines the diameter in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that determines the diameter in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that determines the diameter in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that determines the diameter in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that determines the diameter in the document's default unit of measurement.

Example
```
REM units in inches
.SphereDiameter -1, -200.0, 141.6, -160.6, -175.2, 150.0, -160.6
```

{button ,AL(`CAD_SphereDiameter_Menu;cad_solid;;;;',0,"Defaultoverview",)} Related Topics

.SphericalArray (CAD)

**.SphericalArray .CopiesEquator=*long*, .CopiesPoles=*long*, .
Rows=*long*, .RotateObjects=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*,** .x2=*double*, .y2=*double*, .z
2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command duplicates the selected object(s) along a sphere.

Syntax	Description
.CopiesEquator	Specifies the number of copies of the selected object created at the equator.
.CopiesPoles	Specifies the number of copies of the selected object created at the poles (north and south).
.Rows	Specifies the number of rows.
.RotateObjects	Specifies whether the object(s) is rotated. Set to TRUE (-1) to rotate the object; set to FALSE (0) to not rotate the object. If **.RotateObjects** is set to TRUE, the number of points required is one. The second point is optional. If **.RotateObjects** is set to FALSE, the number of points required is two and the third point is optional.
.x1	If **.RotateObjects**=TRUE, specifies the X-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
	If **.RotateObjects**=FALSE, specifies the X-coordinate for the base point in the document's default unit of measurement.
.y1	If **.RotateObjects**=TRUE, specifies the Y-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
	If **.RotateObjects**=FALSE, specifies the Y-coordinate for the base point in the document's default unit of measurement.
.z1	If **.RotateObjects**=TRUE, specifies the Z-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
	If **.RotateObjects**=FALSE, specifies the Z-coordinate for the base point in the document's default unit of measurement.
.x2	If **.RotateObjects**=TRUE, specifies the X-coordinate for the point that defines the direction of the axis in the document's default unit of measurement. If you do not specify the second point, it is assumed to be perpendicular to the current view. For example, if you are in Top View, the second point is perpendicular to the z axis.
	If **.RotateObjects**=FALSE, specifies the X-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
.y2	If **.RotateObjects**=TRUE, specifies the Y-coordinate for the point that defines the direction of the axis in the document's default unit of measurement. If you do not specify the second point, it is assumed to be perpendicular to the current view. For example, if you are in Top View, the second point is perpendicular to the z axis.
	If **.RotateObjects**=FALSE, specifies the Y-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
.z2	If **.RotateObjects**=TRUE, specifies the Z-coordinate for the point that defines the direction of the axis in the document's default unit of measurement. If you do not specify the second point, it is assumed to be perpendicular to the current view. For example, if you are in Top View, the second point is perpendicular to the z axis.
	If **.RotateObjects**=FALSE, specifies the Z-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the point that defines the direction of the axis in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the point that defines the direction of the axis in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the point that defines the direction of the axis in the document's default unit of measurement.

Note

- You must have drawn an object before using this command.

▪   If **.RotateObjects**=FALSE and you do not select a base point (**.x1**, **.y1**, **z2**) on the body of the object, the object is rotated relative to the base point.

▪   This command corresponds to the Spherical Array command on the Array flyout of the Edit menu. Click Edit, Array, Spherical Array.

Example
```
REM units in inches
.Cylinder -1, 98.2, -0.6, -151.3, --90.3, -0.6, -151.3, -88.9, 53.7, -151.3
.SphericalArray 10, 3, 5, -1, -90.3, -0.6, -151.3, -90.3, 53.7, -151.3
```

This command creates a cylinder and then duplicates the cylinder along a sphere. Ten copies of the object are placed in the middle of the sphere (equator). Three copies of the object are placed at the top and bottom (poles) of the sphere. **.RotateObjects** is set to TRUE; however the second optional point is used. The first point determines the center of the axis of rotation and the second point determines the direction of the axis.

{button ,AL(`CAD_SphericalArray_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.SpiralArray (CAD)

.SpiralArray .Copies=_long_**, .Revolutions=**_double_**,** .Offset=_Boolean_**, .RotateObjects=**_Boolean_**, .x1=**_double_**, .y1=**_double_**, .z1=**_double_**,** .x2=_double_, .y2=_double_, .z2=_double_, .x3=_double_, .y3=_double_, .z3=_double_

This command duplicates the selected object(s) along the path of a spiral.

Syntax	Description
.Copies	Specifies the number of copies.
.Revolutions	Specifies the number of revolutions. The revolutions are the number of times the copies of the object revolve around the axis.
.Offset	Specifies the amount each copy of the object is offset. The value entered will be the measurement of the entire array. If **.Offset** is not specified, this parameter is not used and the objects are not offset in the spiral array.
.RotateObjects	Specifies whether the object(s) is rotated. Set to TRUE (-1) to rotate the object; set to FALSE (0) to not rotate the object. If **.RotateObjects** is set to TRUE, the number of points required is one. The second point is optional. If **.RotateObjects** is set to FALSE, the number of points required is two and the third point is optional.
.x1	If **.RotateObjects**=TRUE, specifies the X-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement. If **.RotateObjects**=FALSE, specifies the X-coordinate for the base point in the document's default unit of measurement.
.y1	If **.RotateObjects**=TRUE, specifies the Y-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement. If **.RotateObjects**=FALSE, specifies the Y-coordinate for the base point in the document's default unit of measurement.
.z1	If **.RotateObjects**=TRUE, specifies the Z-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement. If **.RotateObjects**=FALSE, specifies the Z-coordinate for the base point in the document's default unit of measurement.
.x2	If **.RotateObjects**=TRUE, specifies the X-coordinate for the point that defines the direction of the axis in the document's default unit of measurement. If you do not specify the second point, it is assumed to be perpendicular to the current view. For example, if you are in Top View, the second point is perpendicular to the z axis. If **.RotateObjects**=FALSE, specifies the X-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
.y2	If **.RotateObjects**=TRUE, specifies the Y-coordinate for the point that defines the direction of the axis in the document's default unit of measurement. If you do not specify the second point, it is assumed to be perpendicular to the current view. For example, if you are in Top View, the second point is perpendicular to the z axis. If **.RotateObjects**=FALSE, specifies the Y-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
.z2	If **.RotateObjects**=TRUE, specifies the Z-coordinate for the point that defines the direction of the axis in the document's default unit of measurement. If you do not specify the second point, it is assumed to be perpendicular to the current view. For example, if you are in Top View, the second point is perpendicular to the z axis. If **.RotateObjects**=FALSE, specifies the Z-coordinate for the point that is the center of the axis of rotation in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the point that defines the direction of the axis in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the point that defines the direction of the axis in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the point that defines the direction of the axis in the document's default unit of measurement.

Note

- You must have drawn an object before using this command.
- If **.RotateObjects**=FALSE and you do not select a base point (**.x1**, **.y1**, **z1**) on the body of the object, the object is rotated relative to the base point.
- This command corresponds to the Spiral Array command on the Array flyout of the Edit menu. Click Edit, Array, Spiral Array.

Example
```
REM units in inches
.Sphere -1, -103.8, 1.4, 0.0, -97.9, -2.0, 0.0
.SpiralArray 20, 5, 1, -1, -62.9, -13.0, 0.0, -64.2, 26.8, 0.0
```

This command creates a sphere and then places twenty copies of the sphere along a the path of a spiral. The copies of the sphere are offset by 5 inches. **.RotateObjects** is set to TRUE; however the second optional point is used. The first point determines the center of the axis of rotation and the second point determines the direction of the axis.

{button ,AL(`CAD_SpiralArray_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.StartAddCmdPoint (CAD)

.StartAddCmdPoint .pointNumber=*integer*

This commands sets the number of command points to be added. The **.StartAddCmdPoint** command is followed by **.AddCmdPoint** and **.EndAddCmdPoint** commands.

Syntax	Description
.pointNumber	Specifies the number of command points.

Note
- The **.AddCmdPoint** command is repeated according to the number specified by the **.pointNumber** parameter.

Example
```
REM units in inches
.StartAddCmdPoint 3
.AddCmdPoint 20.0, 0.0, 0.0
.AddCmdPoint 25.0, 10.0, 0.0
.AddCmdPoint 40.0, 10.0, 0.0
.EndAddCmdPoint
.PolyLine
```

This example adds 4 command points and creates a polyline.

{button ,AL(`CAD_StartAddCmdPoint_Menu;cad_point;;;;',0,"Defaultoverview",)} Related Topics

.Stretch2D (CAD)

.Stretch2D .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*, .x4=*double*, .y4=*double*, .z4=*double*

This command performs a 2D stretch on an object.

Syntax	Description
.x1	Specifies the X-coordinate for the first point that determines the bounding box in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that determines the bounding box in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that determines the bounding box in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that determines the bounding box in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that determines the bounding box in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that determines the bounding box in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the first point that determines the displacement in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the first point that determines the displacement in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the first point that determines the displacement in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the second point that determines the displacement in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the second point that determines the displacement in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the second point that determines the displacement in the document's default unit of measurement.

Note
- An object must be selected before using this command.
- This command corresponds to the 2D Stretch command on the Transform menu. Click Transform, 2D Stretch.

Example
```
REM units in inches
.Rectangle 0.0, -94.1, 9.1, 0.0, -43.6, -18.0
.Stretch2D -43.6, 9.1, 0.0, -43.7, 9.2, 0.0-43.6, 9.1, 0.0, -43.7, -68.7, 0.0
```

This example creates a box and then stretches it two-dimensionally.

{button ,AL(`CAD_Stretch2D_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.Stretch3D (CAD)

.Stretch3D .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*, .x4=*double*, .y4=*double*, .z4=*double*, .distance=*double*

This command stretches an object starting from a plane determined by three points.

Syntax	Description
.x1	Specifies the X-coordinate for the first point that determines the plane in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that determines the plane in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that determines the plane in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that determines the plane in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that determines the plane in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that determines the plane in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the third point that determines the plane in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the third point that determines the plane in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the third point that determines the plane in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the point that determines which side of the object is being stretched in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the point that determines which side of the object is being stretched in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the point that determines which side of the object is being stretched in the document's default unit of measurement.
.distance	Specifies the distance the object is being stretched in the document's default unit of measurement.

Note
- An object must be selected before using this command.
- This command corresponds to the 3D Stretch command on the Transform menu. Click Transform, 3D Stretch.

Example
```
REM units in inches
.Box -1, 126.7, 106.6, -101.9, -93.9, 74.8, -125.4
.Stretch3D -110.3, 106.6, -101.9, -110.3, 74.8, -125.4, -110.3, 74.8, -101.9, -71.4, 74.1, -101.9, 32.5
```

This example creates a box and then stretches three-dimensionally.

{button ,AL(`CAD_Stretch3D_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.SpiralSweep (CAD)

.SpiralSweep .Revolutions=*double*, .Offset=*double*, .Steps=*long*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command sweeps a 2D object around an axis with an offset creating a 3D object.

Syntax	Description
.Revolutions	Specifies the number of revolutions. The revolutions are the number of times the copies of the object (**.steps**) revolve around the axis at the specified distance apart (**.offset**).
.Offset	Specifies the distance between the copies of the object.
.Steps	Specifies the number of copies of the object.
.x1	Specifies the X-coordinate for the first point defining the axis in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point defining the axis in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point defining the axis in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point defining the axis in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point defining the axis in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point defining the axis in the document's default unit of measurement.

Note
- An object must be selected before using this command.
- The object selected must be a 2D object.
- This command corresponds to the Spiral Sweep command on the Extrude flyout of the Draw menu. Click Draw, Extrude, Spiral Sweep.

Example
```
REM units in inches
.Rectangle 0, -117.9, 77.5, -68.3, -79.8, 46.6, -68.3
.SpiralSweep 1, 0.35, 20, -49.6, 44.9, -68.3, -14.0, 66.9, -68.3
```

This example creates a rectangle, duplicates it and then sweeps the copies around the specified axis. The copies are offset by 0.35 inches. The coordinate of the two points define the axis.

{button ,AL(`CAD_SpiralSweep_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.Text2D (CAD)

.Text2D .Text=*string*, .Font=*string*, .Size=*double*, .Justification=*integer*, .Style=*integer*, .Angle=*double*, **.x1=*double*, .y1=*double*, .z1=*double*,** x2=*double*, y2=*double*, z2=*double*

This command draws 2D text at the location specified by the coordinates of one point.

Syntax	Description
.Text	Specifies the text string.
.Font	Specifies the font. The font must be installed on your system. For a list of fonts installed on your system, see the font drop-down list box in your CorelCAD application.
.Size	Specifies the text size in inches.
.Justification	Specifies the type of justification to use: 0 right 1 center 2 left
.Style	Specifies the style of the text: 0 regular 1 bold 2 italic 3 bold italic 4 underline 5 bold underline 6 italic underline
.Angle	Specifies the angle of the text in degrees. This parameter is optional. If you do not enter a value for .angle, you must set two points.
.x1	Specifies the X-coordinate for the point, that determines the location of the text in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point, that determines the location of the text in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point, that determines the location of the text in the document's default unit of measurement.
.x1	Specifies the X-coordinate for the angle of the text in the document's default unit of measurement. This parameter is optional and used only if a value is not set for the .angle parameter.
.y2	Specifies the Y-coordinate for the angle of the text in the document's default unit of measurement. This parameter is optional and used only if a value is not set for the .angle parameter.
.z2	Specifies the Z-coordinate for the angle of the text in the document's default unit of measurement. This parameter is optional and used only if a value is not set for the .angle parameter.

Note

■   This command corresponds to the Text2D command on the Text menu. Click Text, Text2D.

Example
```
REM units in inches
.Text2D "yourname" "Arial", 1, 0, 0, 45, -7.0, 0.5, 0.0
REM units in inches
.Text2D "yourname" "Arial", 1,0,4,, 1.0, -3.0, 0.0, 0.0, -3.0, 2.0
```

The first example demonstrates the command syntax when the **.Angle** parameter has a value and the second example demonstrates the command syntax when the **.Angle** parameter does not have a value.

{button ,AL(`CAD_Text2D_Menu;cad_textmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.Text3D (CAD)

.Text3D .Text=*string***, .Font=***string***, .Size=***double***, .Justification=***integer***, .Style=***integer***,** .Angle=*double***, .x1=***double***, .y1=***double***, .z1=***double***,** .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, y3=*double*, z3=*double*

This command draws 3D text at a location specified by the coordinates of one point.

Syntax	Description
.Text	Specifies the text string.
.Font	Specifies the font. The font must be installed on your system. For a list of fonts installed on your system, see the font drop-down list box in your CorelCAD application.
.Size	Specifies the text size in inches.
.Justification	Specifies the type of justification to use: 0 right 1 center 2 left
.Style	Specifies the style of the text: 0 regular 1 bold 2 italic 3 bold italic 4 underline 5 bold underline 6 italic underline
.Angle	Specifies the angle of the text in degrees. This parameter is optional. If you do not enter a value for .angle, you must set three points.
.x1	Specifies the X-coordinate for the point that determines the location of the text in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point that determines the location of the text in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point that determines the location of the text in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point that determines the angle of the text in the document's default unit of measurement. This parameter is optional and used only if a value is not set for the .angle parameter.
.y2	Specifies the Y-coordinate for the point that determines the angle of the text in the document's default unit of measurement. This parameter is optional and used only if a value is not set for the .angle parameter.
.z2	Specifies the Z-coordinate for the point that determines the angle of the text in the document's default unit of measurement. This parameter is optional and used only if a value is not set for the .angle parameter.
.x3	Specifies the X-coordinate for the point that determines the orientation of the text in the document's default unit of measurement. This parameter is optional and used only if a value is not set for the .angle parameter.
.y3	Specifies the Y-coordinate for the point that determines the orientation of the text in the document's default unit of measurement. This parameter is optional and used only if a value is not set for the .angle parameter.
.z3	Specifies the Z-coordinate for the point that determines the orientation of the text in the document's default unit of measurement. This parameter is optional and used only if a value is not set for the .angle parameter.

Note

- If you use the .Angle parameter and one point, the text is only placed on the XY plane.
- This command corresponds to the Text3D command on the Text menu. Click Text, Text3D.

Example
```
REM units in inches
```

```
.Text3D "yourname", "Arial", 1, 0, 4, 90, -1.5, 1.0, 0.0
REM units in inches
.Text3D "yourname", "Arial", 1, 0, 0,, -8.0, 1.5, 0.0, -1.0, 4.0, 0.0, -1.5, 5.5, 0.0
```

The first example demonstrates the command syntax when the **.Angle** parameter has a value and the second example demonstrates the command syntax when the **.Angle** parameter does not have a value.

{button ,AL(`CAD_Text3D_Menu;cad_textmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.TextLeader (CAD)

.TextLeader .Text=*string*, .Font=*string*, .Size=*double*, .Style=*integer*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, y3=*double*, z3=*double*

This command draws 3D text with a leader line.

Syntax	Description
.Text	Specifies the text string.
.Font	Specifies the font. The font must be installed on your system. For a list of fonts installed on your system, see the font drop-down list box in your CorelCAD application.
.Size	Specifies the text size in inches.
.Style	Specifies the style of the text: 0 regular 1 bold 2 italic 3 bold italic 4 underline 5 bold underline 6 italic underline
.x1	Specifies the X-coordinate for the point that determines the position of the leader line's arrow in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point that determines the position of the leader line's arrow in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point that determines the position of the leader line's arrow in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point that determines the end of the leader line in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point that determines the end of the leader line in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point that determines the end of the leader line in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the point that determines the location and orientation of the text in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the point that determines the location and orientation of the text in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the point that determines the location and orientation of the text in the document's default unit of measurement.

Note

- This command corresponds to the Leader command on the Text menu. Click Text, Leader.

Example
```
REM units in inches
.TextLeader "yourname", "Arial", 1, 0, -8.0, 1.5, 0.0, -1.0, 4.0, 0.0, -1.5, 5.5, 0.0
```

{button ,AL(`CAD_Text3D_Menu;cad_textmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.ThreeDArray (CAD)

.ThreeDArray .DistanceMode=*integer*, .Copy1=*integer*, .Copy2=*integer*, .Copy3=*integer*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*, .x4=*double*, .y4=*double*, .z4=*double*

This command creates copies of the selected object along three directional paths.

Syntax	Description
.DistanceMode	Specifies the distance mode. Set to 0 for increment mode; set to 1 for overall mode.
.Copy1	Specifies the number of copies along the first path.
.Copy2	Specifies the number of copies along the second path.
.Copy3	Specifies the number of copies along the third path.
.x1	Specifies the X-coordinate for the point that, with the second, third, and fourth points, determines the direction and the distance between copies
.y1	Specifies the Y-coordinate for the point that, with the second, third, and fourth points, determines the direction and the distance between copies
.z1	Specifies the Z-coordinate for the point that, with the second, third, and fourth points, determines the direction and the distance between copies
.x2	Specifies the X-coordinate for the second point that, with the first point, determines the direction of the first path and the distance between copies.
.y2	Specifies the Y-coordinate for the second point that, with the first point, determines the direction of the first path and the distance between copies.
.z2	Specifies the Z-coordinate for the second point that, with the first point, determines the direction of the first path and the distance between copies.
.x3	Specifies the X-coordinate for the third point that, with the first point, determines the direction of the second path and the distance between copies.
.y3	Specifies the Y-coordinate for the third point that, with the first point, determines the direction of the second path and the distance between copies.
.z3	Specifies the Z-coordinate for the third point that, with the first point, determines the direction of the second path and the distance between copies.
.x4	Specifies the X-coordinate for the fourth point that, with the first point, determines the direction of the third path and the distance between copies.
.y4	Specifies the Y-coordinate for the fourth point that, with the first point, determines the direction of the third path and the distance between copies.
.z4	Specifies the Z-coordinate for the fourth point that, with the first point, determines the direction of the third path and the distance between copies.

Note

- An object must be selected before using this command.
- This command corresponds to the 3 Directional command on the Array flyout of the Edit menu. Click Edit, Array, 3 Directional.

Example
```
REM units in inches
.Box 1.0, 8.5 -34.0, 10.0, 6.0,-40.0, 30.0
.ThreeDArray 1, 2, 2, 2, 23.5, 26.0, -41.0, 28.0, 37.0, -41.0, 36.0, 28.0, -41.0, 30.0, 15.0, -41.0
```

This example creates a box, duplicates it and places the copies along three paths.

{button ,AL(`CAD_ThreeDArray_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} Related Topics

.TwoDArray (CAD)

.TwoDArray .DistanceMode=*integer*, .Copy1=*integer*, .Copy2=*integer*, .x1=*double*, .y1=*double*, .z1= *double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command creates copies of the selected object along two directional paths.

Syntax	Description
.DistanceMode	Specifies the distance mode. Set to 0 for increment mode; set to 1 for overall mode.
.Copy1	Specifies the number of copies along the first path.
.Copy2	Specifies the number of copies along the second path.
.x1	Specifies the X-coordinate for the point that, with the second and third points, determines the direction and the distance between copies
.y1	Specifies the Y-coordinate for the point that, with the second and third points, determines the direction and the distance between copies
.z1	Specifies the Z-coordinate for the point that, with the second and third points, determines the direction and the distance between copies
.x2	Specifies the X-coordinate for the second point that, with the first point, determines the direction of the first path and the distance between copies.
.y2	Specifies the Y-coordinate for the second point that, with the first point, determines the direction of the first path and the distance between copies.
.z2	Specifies the Z-coordinate for the second point that, with the first point, determines the direction of the first path and the distance between copies.
.x3	Specifies the X-coordinate for the third point that, with the first point, determines the direction of the second path and the distance between copies.
.y3	Specifies the Y-coordinate for the third point that, with the first point, determines the direction of the second path and the distance between copies.
.z3	Specifies the Z-coordinate for the third point that, with the first point, determines the direction of the second path and the distance between copies.

Note
- An object must be selected before running this command.
- This command corresponds to the 2 Directional command on the Array flyout of the Edit menu. Click Edit, Array, 2 Directional.

Example
```
REM units in inches
.Box 1.0, 8.5 -34.0, 10.0, 6.0,-40.0, 30.0
.TwoDArray 1, 2, 2, 24.0, -40.0, 40.0, 38.5, -41.0, 38.0, 1.5, -41.0
```

This example creates a box, duplicates it and places the copies along two paths.

{button ,AL(`CAD_TwoDArray_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} Related Topics

.ThreeDSection (CAD)

.ThreeDSection .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command draws the section of a 3D object at the plane determined by three points.

Syntax	Description
.x1	Specifies the X-coordinate for the first point that determines the plane in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that determines the plane in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that determines the plane in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that determines the plane in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that determines the plane in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that determines the plane in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the third point that determines the plane in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the third point that determines the plane in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the third point that determines the plane in the document's default unit of measurement.

Note
- A 3D object has to be selected before using this command.
- This command corresponds to the Section command on the Slice flyout of the Transform menu. Click Transform, Slice, Section.

Example
```
REM units in inches
.Box -1, -17.0, 73.5, -46.1, -11.7, 70.4, -50.7
.ThreeDSection -14.4, 73.5, -46.1, -14.4, 70.4, -46.1, -14.4, 70.4, -50.7
```

{button ,AL(`CAD_ThreeDSection_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.ThreeDSlice (CAD)

.ThreeDSlice .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*

This command slices a 3D object at the plane determined by 3 points.

Syntax	Description
.x1	Specifies the X-coordinate for the first point that determines the plane in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that determines the plane in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that determines the plane in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that determines the plane in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that determines the plane in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that determines the plane in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the third point that determines the plane in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the third point that determines the plane in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the third point that determines the plane in the document's default unit of measurement.

Note

- A 3D object has to be selected before using this command.
- This command corresponds to the 3D Slice command on the Slice flyout of the Transform menu. Click Transform, Slice, 3D Slice.

Example
```
REM units in inches
.Box -1, -96.1, 10.0, 0.0, -66.4, -17.5, -25.9
.ThreeDSlice -81.3, 10.0, -81.3, -17.5, 0.0, -81.3, -17.5, -25.9
```

{button ,AL(`CAD_ThreeDSlice_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.ThreeDTrim (CAD)

.ThreeDTrim .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*, .x4=*double*, .y4=*double*, .z4=*double*

This command trims a 3D object at the plane determined by three points

Syntax	Description
.x1	Specifies the X-coordinate for the first point that determines the plane in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first point that determines the plane in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first point that determines the plane in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second point that determines the plane in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second point that determines the plane in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second point that determines the plane in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the third point that determines the plane in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the third point that determines the plane in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the third point that determines the plane in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the point that determines the side to be removed in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the point, that determines the side to be removed in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the point that determines the side to be removed in the document's default unit of measurement.

Note

- A 3D object has to be selected before using this command.
- This command corresponds to the 3D Trim command on the Slice flyout of the Transform menu. Click Transform, Slice, 3D Trim.

Example
```
REM units in inches
.Box -1, -111.0, 41.2, -25.9, -73.4, 12.2, -53.7
.ThreeDTrim -92.2, 41.2, -25.9, -92.2, 41.2, -53.7, -73.4, 12.2, -53.7, -73.4, 26.7, -25.9
```

{button ,AL(`CAD_ThreeDTrim_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.Torus (CAD)

.Torus .SaveAsSolid=*Boolean*, .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*, .x3=*double*, .y3=*double*, .z3=*double*, .x4=*double*, .y4=*double*, .z4=*double*

This command creates a torus object.

Syntax	Description
.SaveAsSolid	Specifies whether the torus is saved as a solid or a surface. Set to TRUE (-1) to save the torus as a solid; set to FALSE (0) to save the torus as a surface.
.x1	Specifies the X-coordinate for the center of the torus in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the center of the torus in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the center of the torus in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the center of the torus' tube in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the center of the torus' tube in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the center of the torus' tube in the document's default unit of measurement.
.x3	Specifies the X-coordinate for the plane of the torus in the document's default unit of measurement.
.y3	Specifies the Y-coordinate for the plane of the torus in the document's default unit of measurement.
.z3	Specifies the Z-coordinate for the plane of the torus in the document's default unit of measurement.
.x4	Specifies the X-coordinate for the height of the torus' tube in the document's default unit of measurement.
.y4	Specifies the Y-coordinate for the height of the torus' tube in the document's default unit of measurement.
.z4	Specifies the Z-coordinate for the height of the torus' tube in the document's default unit of measurement.

Note

▪ This command corresponds to the Torus, Center and Radius command on the Solids flyout of the Draw menu. Click Draw, Solids, Torus, Center and Radius.

Example
```
REM units in inches
.Torus -1, -82.5, 32.0, -25.9, -58.2, 18.9, -25.9, -75.6, 30.6, -25.9, -62.3, 28.5, -25.9
```

{button ,AL(`CAD_Torus_Menu;cad_drawmenu;;;;',0,"Defaultoverview",)} Related Topics

.Trim (CAD)

.Trim .bSlice=*Boolean*, .X=*double*, .Y=*double*, .Z=*double*

This command trims one line from another selected line.

Syntax	Description
.bSlice	Specifies the trim type. Set to TRUE (-1) to choose slice (Keep Segments on the dialog box); set to FALSE (0) to choose TRIM (Remove Segments on the dialog box).
.X	Specifies the X-coordinate for the point located on the line to be trimmed in the document's default unit of measurement.
.Y	Specifies the Y-coordinate for the point located on the line to be trimmed in the document's default unit of measurement.
.Z	Specifies the Z-coordinate for the point located on the line to be trimmed in the document's default unit of measurement.

Note

▪ This command corresponds to the Trim command on the Transform menu. Click Transform, Trim.

Example

```
REM units in inches
.SetPointXYZ -144.311, 150.821, -114.535
.SetPointXYZ -60.0345, 100.522, -114.535
.SetPointXYZ -119.904, 81.3809, -114.535
.LineSegment
.SetPointXYZ -60.0347, 100.522, -114.535
.SetPointXYZ -83.4631, 185.01, -114.535
.LineSegment
.Trim 0, -60.5399, 100.354, -114.535
```

This example creates two line segments and then trims the second line segment from the first line segment.

{button ,AL(`CAD_Trim_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.TrimBoth (CAD)

.TrimBoth .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command trims two intersecting lines.

Syntax	Description
.x1	Specifies the X-coordinate for the point that selects the first line in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the point that selects the first line in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the point that selects the first line in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the point that selects the second line in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the point that selects the second line in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the point that selects the second line in the document's default unit of measurement.

Note

- This command corresponds to the Trim Both command on the Transform menu. Click Transform, Trim Both.

Example
```
REM units in inches
.SetPointXYZ -156.394, 154.973, -114.535
.SetPointXYZ -116.512, 125.763, -114.535
.SetPointXYZ -127.596, 64.9594, -114.535
.SetPointXYZ -75.7392, 76.9482, -114.535
.SetPointXYZ -28.5687, 49.116, -114.535
.LineSegment
.SetPointXYZ -138.117, 184.39, -114.535
.SetPointXYZ -98.0945, 156.415, -114.535
.SetPointXYZ -59.3899, 182.809, -114.535
.SetPointXYZ -75.7392, 76.9482, -114.535
.SetPointXYZ -29.9464, 125.64, -114.535
.LineSegment
.TrimBoth -52.9764, 101.244, -114.535, -53.3414, 62.8308, -114.535
```

This example creates two line segments and then trims them both.

{button ,AL(`CAD_TrimBoth_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} **Related Topics**

.Undo (CAD)

.Undo

This command cancels the last change and returns the document to its former state.

Note
- This command corresponds to the Undo command on the Edit menu. Click Edit, Undo.

Example
```
.Undo
```

{button ,AL(`CAD_Undo_Menu;cad_editmenu;;;;',0,"Defaultoverview",)} Related Topics

.Ungroup (CAD)

.Ungroup

This command breaks up the selected group into its individual objects

Note
- A grouped object must be created and selected before using this command.
- This command corresponds to the Ungroup command on the Transform menu. Click Transform, Ungroup.

Example
```
REM units in inches
.Box -1, -138.9, 130.5, -114.5, -118.9, 154.6, -137.8
.Box -1, -119.3, 208.0, -137.8, -87.6, 180.5, -172.0
.Box -1, -118.6, 155.4, -172.0, -92.3, 136.4, -199.2
.SelectWithinRegion -198.4, 260.4, -199.2, -34.8, 143.0, -199.2
.Group
.Ungroup
```

This examples creates three boxes, groups them and then ungroups them.

{button ,AL(`CAD_Ungroup_Menu;cad_transmenu;;;;',0,"Defaultoverview",)} Related Topics

.WireFrame (CAD)

.WireFrame

This command changes the view to wireframe.

Note
- This command corresponds to the Wireframe button on the Standard toolbar.
- To change the view to Hidden, use **.HideEntireView**, **.HideSectionView** or **.HideSelected**. To change the view to Shade, use **.ShadeEntireView**, **.ShadeSectionView** or **.ShadeSelected**.

Example
```
.Wireframe
```

{button ,AL(`CAD_WireFrame_Menu;cad_viewmenu;;;;',0,"Defaultoverview",)} Related Topics

.ZoomToAll (CAD)

.ZoomToAll

This command adjusts the view to fit all objects on the screen.

Note
- This command corresponds to the Zoom To All Objects tool of the Zoom flyout on the toolbox.

Example
```
.ZoomToAll
```

{button ,AL(`CAD_ZoomToAll_Menu;cad_zoom;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.ZoomToSelected (CAD)

.ZoomToSelected

This command adjusts the view to fit all selected objects on the screen.

Note
- This command corresponds to the Zoom To Selected tool of the Zoom flyout on the toolbox.
- An object must be selected before using this command.

Example
```
REM units in inches
.Box -1, -110.508, 24.7329, 0.0, -95.6924, 9.78302, -20.3729
.Box -1, -75.1914, 72.2257, -20.3729, -24.6292, 11.3801, -44.1854
.SelectPointAt -121.157, 68.7571, -44.1854, 0
.ZoomToSelected
```

This example creates two boxes, selects the first box created (The second is already selected because it has just been created.) and then zooms in to the selected boxes.

{button ,AL(`CAD_ZoomToSelected_Menu;cad_zoom;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.ZoomPrevious (CAD)

.ZoomPrevious

This command zooms to the previous zoom level.

Note

- This command corresponds to the Zoom Previous tool of the Zoom flyout on the toolbox.

Example
```
.ZoomPrevious
```

{button ,AL(`CAD_ZoomPrevious_Menu;cad_zoom;;;;',0,"Defaultoverview",)} **Related Topics**

.ZoomIn (CAD)

.ZoomIn .x1=*double*, .y1=*double*, .z1=*double*, .x2=*double*, .y2=*double*, .z2=*double*

This command zooms in to a region determined by two points.

Syntax	Description
.x1	Specifies the X-coordinate for the first of two points that define the region in the document's default unit of measurement.
.y1	Specifies the Y-coordinate for the first of two points that define the region in the document's default unit of measurement.
.z1	Specifies the Z-coordinate for the first of two points that define the region in the document's default unit of measurement.
.x2	Specifies the X-coordinate for the second of two points that define the region in the document's default unit of measurement.
.y2	Specifies the Y-coordinate for the second of two points that define the region in the document's default unit of measurement.
.z2	Specifies the Z-coordinate for the second of two points that define the region in the document's default unit of measurement.

Note

- This command corresponds to the Zoom In tool of the Zoom flyout on the toolbox.

Example
```
REM units in inches
.Box -1, -110.508, 24.7329, 0.0, -95.6924, 9.78302, -20.3729
.Box -1, -75.1914, 72.2257, -20.3729, -24.6292, 11.3801, -44.1854
.ZoomIn -143.619, 75.7336, -44.1854, -92.0959, 30.6536, -44.1854
```

{button ,AL(`CAD_ZoomIn_Menu;cad_zoom;;;;',0,"Defaultoverview",)} <u>Related Topics</u>

.ZoomOut (CAD)

.ZoomOut

This command decreases the magnification of the current view by 30%.

Note
- This command corresponds to the Zoom Out tool of the Zoom flyout on the toolbox.

Example
```
.ZoomOut
```

{button ,AL(`CAD_ZoomOut_Menu;cad_zoom;;;;',0,"Defaultoverview",)} Related Topics

Crystal Blue
Crystal Clear
Crystal Green
Crystal Red
Glass Blocks
Glass Clear
Glass Frosted
Glass Green
Glass Orange
Glass Red
Glass Tinted
Glass Violet
Glass Yellow
Mirror

Ambrite
Amethyst
Diamond
Emerald
Mother of Pearl
Opal
Pearl
Rhine Stone
Ruby
Sapphire
Tiger's eye

Aluminum Anodized
Aluminum Brushed
Aluminum Polished
Brass
Bronze
Bronze Wrought
Chrome
Copper
Copper Wrought
Gold
Gold Wrought
Iron
Iron Cast
Iron Rusty
Silver
Steel
Steel No Skid

Blue Marble
Bricks
Cement Rough
Cement Smooth
Coal
Granite
Gravel
Marble
Pavement
Sand
Slate
Stone

None

Banded Malachite
Checkers
Cherry
Cubes
Fabric
Felt
Insulation
Leather
Rubber
Sky

Bumpy Black
Bumpy Blue
Bumpy Gray
Bumpy Green
Bumpy Orange
Bumpy Red
Bumpy White
Bumpy Yellow
Clear Black
Clear Blue
Clear Gray
Clear Green
Clear Orange
Clear Red
Clear White
Clear Yellow
Matte Black
Matte Blue
Matte Gray
Matte Green
Matte Orange
Matte Red
Matte White
Matte Yellow
Shiny Black
Shiny Blue
Shiny Gray
Shiny Green
Shiny Orange
Shiny Red
Shiny White
Shiny Yellow

Cedar
Dark Grain
Heavy Grain
Lacquered
Light Grain
Low Grain
Mahogany
Oak
Pine
Rosewood