1 Constraining Geometry

Available in TurboCAD Professional only

TurboCAD includes several tools that let you constrain 2D drawing objects. Depending on the tool you select, you can place a geometric constraint or a dimensional constraint between any two drawing objects, or place a single constraint to control the size of an object. The constraints you place create relationships between drawing objects that are maintained as other constraints are added. When you place constraints on one or more drawing objects, you create a constrained sketch.

In general, geometric constraints create positional relationships between two objects, and dimensional constraints create either angular relationships or size relationships. You can use any combination of geometric and dimensional constraints to control geometry in a constrained sketch. Together, TurboCAD’s geometric and dimensional constraint tools provide:

• **Significant functionality**: You can draw quickly, without concern for accuracy, and then use constraints to refine the sketch. For dynamic designs, you can create sketches before you know their exact size or position, and then use constraints and adjust them later. Also, you will use fewer drawing tools because you can achieve the same results with 2D constraints.

• **Sketch intelligence**: You can use constraints to build intelligent behavior into a sketch. For example, you can use geometric constraints to control how a linkage moves, or you can use dimensional constraints to define a proportional relationship where the length of an enclosure is always twice its width.

• **Persistent behavior**: Constraints are not a temporary drawing aid. Instead, they maintain their relationships with the geometry to which they are attached unless you delete them.

You can use constrained sketches to:

• Create fully adjustable 2D drawings or profiles for creating 3D features.

• Create drawings or models of part families (parts with similar features that are available in different sizes).

• Capture design intent and enforce design rules.

• Create dynamic assembly layouts, which control the size and shape of several parts as constrained sketches of their mating parts change. For example, a layout of a belt and pulley would determine the size of its mounting brackets and positions of mounting holes.

• Develop designs with constraint schemes. For example, you can start with fewer constraints in a “flexible scheme” to determine the length of a link. Once the length is determined, you can freeze the design with a “fully constrained scheme”.

• Verify motion in mechanisms. Use the Select tool for precise control, or use the Edit Tool to dynamically drag geometry in assembly layout sketches that have not been fully constrained. You can also move or drag geometry to test sketch constraints. For updating options while dragging objects, see "Constraint Preferences" on page 1-2.

**NOTE**: For additional constraint considerations, see "Constraint Rules & Limitations" on page 1-15.
Valid Objects for Constraints
You can apply constraints to all types of 2D objects except for the following:

- Construction objects
- Ellipses
- Spline objects
- Objects created with the Sketch tool
- Walls
- Symbols
- Planar edges and features of 3D objects.

NOTE: You can use the Auto Constraint tool with some compound objects such as rectangles, polylines, and polygons. In other cases, you must first explode objects such as multilines one or more times before placing constraints.

Constraint Preferences
Menu: Options / Constraints
Provides settings and controls for geometric constraint tools and constrained dimensions.

- Auto Add Constraints: Activates or deactivates the Auto Add Constraints tool 🛠️.
- Update Connected Objects While Dragging: Dynamically updates the position, shape, and size of constrained objects as you drag them with the Edit tool 📐.
- DCM Default: The Constraints Manager is used in default mode. Changes to any part of a set of constrained objects can affect all objects equally.
- Use Priority Level: The Constraints Manager Changes to any part of a set of constrained objects affect that part first, with the minimum possible changes to the rest of the objects.
- Incremental Evaluation: The Constraint Manager constantly rechecks the results of any changes as the changes are being made. If this option is off, the Constraint Manager will only check the results of changes after the change has been made. If large scale changes are necessary it is best this option should be turned on.
- Place Marker to Same Layer with Object: Inserts constraint markers on the same layer as the object to which the constraint has been applied. Constraint markers inherit the same layer properties applied to the object and are subject to the same layer controls. For example, turning off the visibility of the object’s layer also turns off the visibility of the constraint markers.
- Show Variable Name in Dimension Text: Displays the name of the variable associated with each constrained dimension in parentheses after the dimension value.
- Print Constraint Markers: Includes the constraint markers in the printed version of your sketch.
- Print Constraint Dimensions: Includes constrained dimensions in the printed version of your sketch.
Geometric Constraints
You can apply geometric constraints to create positional relationships of 2D sketch objects.

Geometric constraints work in conjunction with dimensional constraints to provide flexible control of sketch objects. See "Dimensional Constraints" on page 1-11

Placing Geometric Constraints
After you create 2D objects, applicable geometric constraint tools are enabled. You can then place constraints by selecting the appropriate drawing object or vertex. A constraint indicator is displayed on constrained geometry.

As you place constraints, degrees of freedom are removed from the sketch objects. When you remove all degrees of freedom, the sketch is fully constrained.

**TIP:** Use the Edit Tool to drag sketch entities and check the impact constraints will have on any element in the drawing. For updating options while dragging objects, see "Constraint Preferences" on page 1-2.

Editing Geometric Constraints
You can double-click constraint indicators and edit their display properties, but you cannot change the type of constraint. When you need to change a constraint type, delete the incorrect constraint by selecting its indicator and pressing the Delete key. You can then apply the desired constraint.

Points Coincident
**Menu:** Format / Constraints / Points Coincident

Moves a point or endpoint of a line or arc to the same location as another point in your drawing and defines their coordinates as equal to one another at all times. When an endpoint of a line or arc is selected first, the entire object moves and its size, shape, and orientation are preserved.

1. Select a point or an endpoint of a line or arc that you wish to move in order to make it coincident with another point.
2. Select the point or endpoint on another line or arc to which your first selection will be made coincident.
3. The first point or object is moved such that the selected points share the same location and the Points Coincident symbol is displayed.

Coincident
**Menu:** Format / Constraints / Coincident

Creates either a collinear alignment or a direct connection between the closest vertex of the first object selected and the geometry of the second object selected. The first vertex maintains this relationship to the second object at all times.

While the first selection must be a vertex, the second selection can be anywhere on an object. If two vertices are selected, the first object moves such that the selected vertices share the same coordinates. If the second selection is not a vertex, the first selection moves such that it connects to the closest point on the second object. This may be a theoretical point that would only exist if the line or arc were extended. A circle is only valid as a second object selection.

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1. Select a point or a vertex of a line or arc that you wish to move in order to make it coincident with another object.

2. Select a point or vertex on another line or arc to which your first selection will be made coincident.

3. Or, select anywhere other than a vertex to move your first selection to the closest point on the second object.

4. The first object is moved such that its selected vertex is coincident with the second vertex or object selected and the Coincident constraint symbol is displayed.

**Parallel**  
**Menu:** Format / Constraints / Parallel

Changes the orientation of a line you select such that it is equidistant at all points to another line in your drawing. The first line maintains this relationship to the second line at all times.

1. Select a line that you wish to make parallel to another line in your drawing.

2. Select an existing line to which your first selection will be made parallel.

3. The first line is made equidistant at all points to the second line and the Parallel constraint symbol is displayed.

**Perpendicular**  
**Menu:** Format / Constraints / Perpendicular

Rotates a line you select such that it maintains a ninety degree angle to another line in your drawing at all times.

1. Select a line that you wish to make perpendicular to another line in your drawing.

2. Select an existing line to which your first selection will be made perpendicular.

3. The first line is rotated to a ninety degree angle to the second line and the Perpendicular constraint symbol is displayed.

For updates and additional information,
Tangent

Menu: Format / Constraints / Tangent

Changes the position of an arc or circle you select such that it meets the selected line at the point where they share the same slope. Depending on the initial positions of the geometry you select, this may be a theoretical point that would only exist if the line, arc (when an arc is selected), or both were extended. The arc or circle then maintains this relationship to the line at all times.

1. Select a line in your drawing to which an arc or circle will be made tangent.

2. Select an existing arc or circle to make tangent to the line.

3. The arc or circle is moved into a position tangent to the selected line and the Tangent constraint symbol is displayed.

4. Due to the initial positions of the geometry, the arc or circle may move to where it would be tangent to the line:

Connect

Menu: Format / Constraints / Connect

Moves, rotates, and trims or extends any two lines or arcs as necessary in order to make the two endpoints you select coincident and tangent. Since lines cannot be made tangent the tangency aspect is ignored when the tool is used between two lines. The selected objects then maintain this relationship to one another at all times.

1. Select an endpoint on the first line or arc you wish to connect.

2. Select an endpoint on the second line or arc you wish to connect.
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3. The geometry is reoriented and modified as necessary to connect the selected endpoints. The Connect constraint symbol is displayed.

The following figure shows an example of the Connect constraint applied between two arcs:

Concentric
Menu: Format / Constraints / Concentric

To set two circles or arcs concentric:
1. Select the circle or arc you wish to move into a concentric position with another circle or arc.
2. Select the circle or arc to which the first selection will be made concentric.
3. The first selection is repositioned such that its center point shares the same location as the center point of the second selection and the Concentric constraint symbol is displayed.

To constrain a point or an endpoint of a line to the center point of an arc or circle:

1. Select the point you wish to constrain to the center of the arc or circle.
2. Select the arc or circle to which it will be constrained.
3. The point is constrained to the center point of the selected arc or circle, and the Concentric constraint symbol is displayed.

Symmetric
Menu: Format / Constraints / Symmetric

The Symmetric constraint tool creates relationships between objects on either side of an axis of symmetry. The selected objects then maintain their relationship to one another at all times:

- When you select two lines or arcs and an axis of symmetry, the objects are set equidistant from the axis and to inverse angles in relation to the axis.
- When you select endpoints of two objects, the objects are moved such that the selected endpoints are aligned and set equidistant from the axis.
- Applying the constraint to both sets of corresponding endpoints will also resize the objects, thereby making them mirror images of one another.
- When you select two circles and an axis, the circles are automatically moved and resized to become mirror images of each other.

To set two arcs or lines to equal distances and inverse angles to an axis of symmetry:

For updates and additional information,
1. Select the first arc or line.

2. Select the second arc or line.

3. Select the axis of symmetry. The two arcs or lines are set to equal distances from the axis and at inverse angles to the axis. The Symmetric constraint symbol is displayed.

To make two arcs, lines, or circles mirror images of one another:

1. For lines and arcs, select an endpoint from one line or arc, then select the corresponding endpoint from the other line or arc. Select the axis of symmetry to make the endpoints symmetric.

2. Repeat this process for the other set of endpoints to make the objects mirror one another.

3. To create mirror images with circles, select each of them and the axis. Circles are automatically moved and resized to become mirror images of one another.

**Equal Radius**

*Menu: Format / Constraints / Equal Radius*

Sets the radii of two arcs or circles in your drawing to the same value. When the radius of one of the arcs or circles is modified, the other one automatically updates to match.

1. Select the circle or arc whose radius will be modified.

2. Select another circle or arc in your drawing to set their radii equal to one another.

3. The first selection assumes the same radius as the second selection, and the Equal Radius constraint symbol is displayed.
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**Equal Length**
**Menu:** Format / Constraints / Equal Length

Sets the length of two lines in your drawing to the same value. When the length of one of the lines is modified, the other one automatically updates to match.

1. Select the line whose length will be modified.

2. Select another line in your drawing to set their lengths equal to one another.

3. The first selection assumes the same length as the second selection, and the Equal Length constraint symbol is displayed.

**Equal Distance**
**Menu:** Format / Constraints / Equal Distance

Sets the distance between two lines or points in your drawing to the same value as the distance between two other lines or points you specify. When one of the constrained distances is modified, the other automatically updates to match.

1. Select a line or snap to a point in your drawing. Then select a second line or point to define the distance between the two objects.

2. Select two additional lines or points to identify the distance between them.

3. Objects are moved as necessary to make the distances defined in the two selection sets equal to one another, and the Equal Distance constraint symbol is displayed.

4. The Equal Distance constraint can also be applied to the distances between lines or points in a row:
**Change Chirality**

**Menu:** Format / Constraints / Change Chirality

This tool offers two options to help you correct tangent sketched arcs that have become inadvertently skewed when other constraints are applied, or during drag-testing of your constrained sketch. The Flip option replaces the selected arc with its complement (the segment that when added to the initial arc would form a complete circle). The Change Object Chirality option changes the tangency of the arc to the opposite-hand, on the other side of the tangent line. It will also move the arc from the inside of an arc or circle to the outside, or outside to inside.

*NOTE: The word chirality is derived from the Greek χειρ (cheir), the hand. So chirality refers to the orientation or "handedness" of an arc. Is it left or right, top or bottom, on side A or Side B. Since there can be several correct ways in which an arc can be tangent to something else, the correct orientation is not always selected and this tool allows you to modify the results of tangency assignment.*

1. Suppose your sketch was intended to look like the figure to the left, but when constrained dimensions were placed on other sketch objects (not shown), the arc fillet became skewed as shown in the figure to the right.

2. With the Change Chirality tool active, click the **Flip** option. Select the arc to replace it with its complement.

3. Select the **Change Object Chirality** option. Select the arc, and then select the vertical line to switch the arc's tangency to the opposite side. The arc is returned to its intended position and tangency.

**Fix Geometry**

**Menu:** Format / Constraints / Fix Geometry

Locks the selected object or point at its current coordinates in the drawing, preventing it from moving when additional constraints are applied to the sketch or when the other constrained objects are drag-tested. Use this constraint to lock the position of one object and then constrain your other sketch geometry in relation to it.

*NOTE: The Fix Geometry constraint can only be placed on objects which are already constrained.*

1. Select a line, arc, circle, or point in your drawing to lock it at its current position. The Fix Geometry constraint symbol is displayed.

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2. As you apply additional constraints, the fixed object maintains its position and other sketch geometry moves as necessary.

3. You can also apply the Fix Geometry constraint to a vertex to lock one endpoint of an object in its current position.

**Auto Constraint**

*Menu: Format / Constraints / Auto Constraint*

Automatically applies multiple constraints to a selection set of sketch geometry. Available constraints include Points Coincident, Parallel, Perpendicular, Concentric, Tangent, and Equal Radius. Each constraint type can be turned off individually to prevent it from being applied to the selection set.

You can also use the Auto Constraint tool to apply constraints to compound objects such as polygons and polylines. The compound objects are automatically exploded before the constraints are applied. After using the Auto Constraint tool, you can create constraint relationships between the exploded compound objects and other sketch geometry.

1. Create a selection set of sketch geometry.

2. Adjust the types of constraints that will be applied as necessary, and then click **Finish**.

3. Constraints are automatically applied to geometry that falls within the preset valid range for each active constraint type.

4. Add additional constraints as necessary to finish constraining your sketch.

*TIP: The Auto Constraint tool can create constraints faster than using the various constraint tools one at a time. Therefore is used to best effect on old drawings which do not have constraints.*
Dimensional Constraints

You can apply dimensional constraints to create angular and positional relationships between 2D sketch objects. Dimensional constraints are created using the standard TurboCAD dimension tools and are similar to associative dimensions, with the following additional benefits:

- Their values are controlled by variables, which are displayed in the Calculator palette. You can assign meaningful names to variables and change their numerical values directly. You can also drive them by creating equations with any combination of mathematical expressions and other variables. See "Variables" on page 1-12.

- Instead of updating when geometry is changed, constrained dimensions directly control the size and position of sketch objects to which they are attached. When you change a constrained dimension’s variable value, the sketch geometry updates accordingly.

- They work in conjunction with geometric constraints to provide flexible control of sketch objects. See "Geometric Constraints" on page 1-3.

- A new Distance dimension tool lets you specify a parallel distance between two lines. See "Distance Dimension" on page 1-14.

You can create dimensional constraints with the following dimension tools:

- Orthogonal
- Parallel
- Distance (NEW in 10.5)
- Angular
- Radius
- Diameter

You can use dimensional constraints to:

- Create sketches for 3D features in nested Boolean operations that are easy to modify, without having to enter the Part Tree for access to sketch objects.
- Create meaningful names for dimension values.
- Create fully dimensioned 2D drawings (without any 3D objects) that are easy to modify.

To apply dimensional constraints, you enable the Auto Add Constraints option, and then you use TurboCAD’s standard dimension tools and object selection options. The Auto Adds Constraint option can be toggled on and off in the Inspector Bar. If the option is on the dimension created will be a constraint. If the option is off the dimension will be a standard associative dimension.

Try the following Constrained Dimensions Exercise

1. Click New to open a new drawing.
2. From the Line flyout toolbar, click Rectangle. Draw a rectangle similar to the one shown in the following figure.
3. From the Constraints toolbar, click Auto Constraint. Make sure all the available constraint types are turned on, and then select the rectangle and then click Finish. The rectangle is exploded and constrained as shown.
4. Click the Circle Center and Point tool. Draw a circle inside the rectangle approximately as shown.
5. Click the **Dot** tool. Place a dot (point) anywhere inside the circle. From the Constraints toolbar, click the **Concentric** tool. Select the circle and then select the dot to constrain the dot to the center point of the circle. This dot will enable you to dimension to the center of the circle.

6. Press F2 to open the Calculator palette. From the Dimension flyout toolbar, click the **Diameter** tool. Click the **Auto Add Constraints** tool to enable dimensional constraints. Add the diameter dimension to the circle as shown.

7. In the Calculator palette, notice the new variable that was added. Click the variable name and change it to **HoleDia**.

8. Click the **Orthogonal Dimension** tool. Dimension between the lower edge of the sketch and the center of the circle as shown.

9. Notice that another variable was added for this constrained dimension. Click the variable name and change it to **Hole2Edge**. Then click the **Formula** field and type \(1.5 \text{HoleDia}\) to tie its value to the hole diameter value.

10. Use the **Orthogonal Dimension** tool to place another constrained dimension between the left edge of the sketch and the center of the hole. Click the **Formula** field and type **Hole2Edge** to equate its value to the distance between the center of the hole and the lower edge.

11. Keep your file open for the next exercise, “Variables”.

**Variables**

After you apply dimensional constraints to your sketches, you can access and control them in the Calculator palette. Each constrained dimension is assigned a variable name and appears in the Calculator palette in the order in which it was created.

In the Calculator palette, you can:

- Identify the variable associated with a constrained dimension by selecting the dimension in the drawing. The associated variable highlights in the Calculator palette.

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For updates and additional information,
- Click in the Variable field to modify the automatically assigned variable name for a constrained dimension.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Var1</td>
<td>0.61</td>
<td>Var4</td>
<td>0.42</td>
</tr>
<tr>
<td>Var2</td>
<td>0.44</td>
<td>Var5</td>
<td>0.53</td>
</tr>
<tr>
<td>Var3</td>
<td>0.35</td>
<td>Var6</td>
<td>0.53</td>
</tr>
</tbody>
</table>

- Click the Value field to directly edit the value of a constrained dimension.

- Click the Formula field and create an equation that controls the value of a constrained dimension.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Var1</td>
<td>0.61</td>
<td>11*1.12/22</td>
</tr>
<tr>
<td>HoleRadius</td>
<td>0.5</td>
<td>11*1.12/22</td>
</tr>
<tr>
<td>Var3</td>
<td>0.35</td>
<td></td>
</tr>
<tr>
<td>Var4</td>
<td>0.42</td>
<td></td>
</tr>
<tr>
<td>Var5</td>
<td>0.53</td>
<td></td>
</tr>
</tbody>
</table>

- Define a relational formula that calculates the value of a constrained dimension based on the values of other variables.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Var1</td>
<td>0.61</td>
<td>0.61</td>
</tr>
<tr>
<td>HoleRadius</td>
<td>0.500000</td>
<td>11*1.12/22</td>
</tr>
<tr>
<td>HoleRadius2</td>
<td>1.120000</td>
<td>11*1.12/22</td>
</tr>
<tr>
<td>Var4</td>
<td>0.42</td>
<td></td>
</tr>
<tr>
<td>Var5</td>
<td>0.53</td>
<td></td>
</tr>
</tbody>
</table>

- Access and insert predefined functions into formulas by clicking the \( \text{ } \) button in the Calculator palette. Units conversion functions are also available.

<table>
<thead>
<tr>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>abs(x)</td>
<td>Return absolute value</td>
</tr>
<tr>
<td>max(x, y)</td>
<td>Return larger of two values</td>
</tr>
<tr>
<td>min(x, y)</td>
<td>Return smaller of two values</td>
</tr>
<tr>
<td>ceil(x)</td>
<td>Find integer ceiling</td>
</tr>
<tr>
<td>floor(x)</td>
<td>Find largest integer less than or equal</td>
</tr>
<tr>
<td>mod(x, y)</td>
<td>Find floating-point remainder</td>
</tr>
<tr>
<td>sqrt(x)</td>
<td>Find square root</td>
</tr>
<tr>
<td>pow(x, y)</td>
<td>Calculate value raised to a power</td>
</tr>
<tr>
<td>hrev(Rev)</td>
<td>Calculate function of right triangle</td>
</tr>
</tbody>
</table>

- Create a user-defined variable by entering the name, value, and formula (if desired) in the appropriate field. You can then use these variables to control others when you include them in formula equations.

**Try the following Variables Exercise**

1. Return to the drawing you started in the previous exercise, “Constrained Dimensions”. In the Calculator palette, change the value of the HoleDia variable to .3 and watch as its size and position update in the drawing.

2. With the Orthogonal Dimension tool active, select the Segment Dimensioning option \( \text{ } \). Add constrained dimensions to the upper edge of the sketch and the right edge of the sketch as shown.
3. In the drawing, select the vertical dimension you placed on the rectangle. Notice that it highlights in the Calculator palette.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>HoleDia</td>
<td>.3</td>
<td></td>
</tr>
<tr>
<td>Hole2Edge</td>
<td>0.450000</td>
<td>1.5*HoleDia</td>
</tr>
<tr>
<td>Var3</td>
<td>0.450000</td>
<td>Hole2Edge</td>
</tr>
<tr>
<td>Var4</td>
<td>2.37</td>
<td></td>
</tr>
<tr>
<td>Var6</td>
<td>1.65</td>
<td></td>
</tr>
</tbody>
</table>

4. Change the value for the vertical dimension to 2 and notice how the sketch updates. Then change the value of the horizontal dimension to 2.5 and notice how the circle maintains its position relative to the edges of the sketch.

5. In the Calculator palette, click the empty variable field below the last variable. Type the name Pin and enter .3 in the value field.

6. Change the value for the HoleDia variable to Pin+.025 to tie the size of the hole to the pin diameter plus the minimum clearance required.

7. Next, change the value for the Pin variable to .475 and notice how the size and position of the hole change to reflect the new value.

Distance Dimension
Menu: Insert / Dimension / Distance

Automatically sets two lines parallel to each other and places a constrained dimension between them. You can select the line objects themselves or snap to vertices to place the dimension. The Distance Dimension tool is only available when the Auto Add Constraints tool is active.

1. Select the line (or snap to a vertex on the line) that will maintain its current orientation.

2. Select the line (or snap to one of its vertices) that will be made parallel to the first selection.

For updates and additional information,
3. Click to place the
dimension in the drawing
area.

Constraint Rules & Limitations

Keep in mind the following when using geometric or
dimensional constraints in your drawings:

- You can create reference dimensions in a constrained
  profile by turning off the Auto Add Constraints
  option, and creating associative dimensions. These
dimensions will update when their associated objects
are moved or resized by other constraints, but you
cannot use them to modify sketch geometry.

- If you select an invalid object for a constrained
dimension, a standard dimension will be created, but
the dimension will not be constrained and a variable
will not be created.

- Your sketches may distort when you apply
  constraints or edit variable values. You can minimize
distortion by drawing sketch objects closer to their
intended size and shape.

- In many cases, you can prevent sketch distortion by
  connecting objects together with geometric
  constraints before you apply dimensional
  constraints.

- For predicable sketch behavior, avoid placing
duplicate geometric constraints.

- If you place duplicate dimensional constraints, you
  will lose the ability to edit their variable values. If
  this happens, delete all duplicates.

- IMPORTANT: The only way to copy constraints and
  maintain their associativity is to use the Make Copy
  option in the Inspector Bar. You cannot use CTRL+C
  and CTRL+V to copy and paste, and you cannot use
  the rubber stamp option. The Copy tools such as,
  Array Copy and Radial Copy will not work on
  constrained objects.

- Constraints cannot be made into groups, blocks, or
  symbols.

- Many of the standard 2D Edit tools will not work on
  constrained objects. The exceptions are: Line
  Length, Shrink/Extend Line, Multi Shrink/Extend
  Line, Meet 2 Lines, Stretch, Fillet, and Chamfer

- If you intend to create 3D solid objects from
  constrained sketch profiles, keep the sketches as
  simple as possible. For example, place fillet features
  on the edges of a 3D solid (do not draw them in the
  constrained sketch profile).

- When you create 3D objects from constrained sketch
  profiles, the same object creation and editing rules
  apply, except that you have more editing options and
  sketch flexibility compared with unconstrained
  profiles.

- Geometric and dimensional constraints exported
  from TurboCAD may be displayed when imported
  into other CAD programs, but they will lose all
  functionality.

- Geometric and dimensional constraints imported
  into a drawing from another drawing will be
  displayed, but they will lose all associativity and
  functionality.
Constraint FAQs

Q: Why don’t my dimensions have variables in the Calculator palette?
A: If a dimension does not have a variable in the Calculator palette it is not a constraint. To make sure that the dimension becomes a constrained dimension make sure that the Auto Add Constraints button is turned on in the Inspector Bar. In some cases the dimension be a constraint because you used a dimension tool that does not create constraints. See “Dimensional Constraints” on page 1-11 for a list of dimension tools that create constraints.

Q: When I use constraints on a profile sketch for a 3D object the constraints change the profile but they don’t change the 3D object, why?
A: There are two reasons this might be happening. First, the Part Tree must be activated before the 3D object is created or changes in the profile will not be reflected in the 3D object. Second, to make 3D objects compliant to 2D constraints the profile objects must be compound profiles. The compound profile is a new feature in TurboCAD 10.5 and is available for all 3D tools that use profiles. The compound profile allows you to select a series of lines and arcs that are connected so that there is no distance between the end of one element and the start of the next. The series that makes up a compound profile can open or closed at the ends of the series. If your are extruding along a path the path must be a compound profile as well. You must use the compound profile feature even if the profile is a circle, polyline, or polygon, or the 3D object will not associate with any constraints added to the profile.

To use the compound profile feature:

1. Draw a profile.
2. Select a 3D tool that uses profiles (such as Normal Extrude).
3. Select the Use Compound Profile option in the Inspector Bar.
4. Select all of the element that will be a part of the profile.
5. Click the Finish selection of profile button.
6. Proceed using the tool as you would with a regular profile.

Q: I use a compound profile to create a 3D object, and I TurboCAD won’t let me use that profile again, why?
A: TurboCAD will only allow you to use a compound profile, or any element of a compound profile once. In essence the 3D object consumes the profile into itself so that it cannot be used again.
2 Constraint Tutorial

**Geometric Constraints**
This short tutorial will walk you through the creation of a part from start to finish. The first steps are to create our drawing and add geometric constraints.

1. First, go to Constraints in the Options menu, and select the Priority Level option.

2. Using the arc and line tools to draw a sketch like the following:

3. Use the circle tool to add four circles as shown:

4. Select the Connect constraint tool and close the outer part of the sketch.
5. Using the edit tool make sure that the top line of the sketch parallel with the x-axis.
6. Use the fix geometry constraint tool to lock the top lint of the sketch.
7. Use the parallel constraint tool to make the bottom line of the sketch parallel to the top line, and the line on the left parallel to the line on the right.
8. Use the perpendicular constraint tool to make the line on the right perpendicular to the top line.
9. Use the Coincident constraint tool to make the circles coincident to the outer arcs.

10. Use the Cross point tool + from the Point toolbar to make center points for each of the circles.

11. Use the Coincident constraint tool to make the center points coincident to their respective circles.
TurboCAD 2D Constraints Guide

12. Use the equal radius constraint tool to make all of the arcs share the same radius.

13. Use the equal radius constraint tool again to make all of the circles have the same radius.

14. Use the equal length constraint tool on the top and bottom lines to make them equal.

15. Use the equal length constraint tool to make the lines on the right and left side lines equal.

16. Using the polyline tool draw add the brace shape seen on the right side of the drawing:

17. Use the Auto Constraint tool to make the polyline constrained.

For updates and additional information,
18. Use the point coincident constraint tool to make the top point of the brace coincident with the top point of the right line.

19. Use the point coincident constraint tool to make the bottom point of the brace coincident with the bottom point of the right line.

20. Use the equal length tool to on the top and bottom lines of the brace.

Dimensional Constraints
To really make our drawing powerful and dynamic we need to add dimensional constraints. Follow the next section of the tutorial to set up all the constraints we will need.


22. Use the Diameter dimension tool to set a dimensional constraint on the top left circle.
23. Use the Radius dimension tool to set a dimensional constraint on the bottom right arc.

24. Select the circle dimension in the drawing.

25. Press the F2 key to open the Calculator palette.

26. In the Calculator palette select the highlighted variable and rename it **Hole**.

![Image of Circle Dimension]

27. Select the other variable and rename it **OuterRad**.

![Image of Hole and OuterRad variables]

28. In the formula area for the OuterRad variable set the value as **Hole/2+0.5**

![Image showing formula calculation]

29. In the value area for the Hole variable set the value as **1**.

30. Use the Parallel dimension tool to constrain the distance between the two top circles by snapping to the center points you created earlier. Use the vertex snap to make the selection of the points accurate.

31. In the calculator palette rename the variable for the new parallel dimension to “Length” and set its value to **5**.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hole</td>
<td>0.86</td>
<td></td>
</tr>
<tr>
<td>OuterRad</td>
<td>0.93000</td>
<td>Hole/2+0.5</td>
</tr>
<tr>
<td>Length</td>
<td>5</td>
<td></td>
</tr>
</tbody>
</table>

32. Use the Parallel dimension tool to constrain the distance between the to left side holes.

For updates and additional information,
33. In the Calculator palette rename the variable for the new parallel dimension to “Width” and set its value to 4.

<table>
<thead>
<tr>
<th>Variable</th>
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<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hole</td>
<td>0.86</td>
<td></td>
</tr>
<tr>
<td>OuterRad</td>
<td>0.930000</td>
<td>Hole/2+0.5</td>
</tr>
<tr>
<td>Length</td>
<td>5</td>
<td></td>
</tr>
<tr>
<td>Width</td>
<td>3.48</td>
<td></td>
</tr>
</tbody>
</table>

We now have a nicely constrained sketch.

34. Save the drawing.

Please visit the TurboCAD Web site where you find a tutorial on how to take this drawing into 3D.