

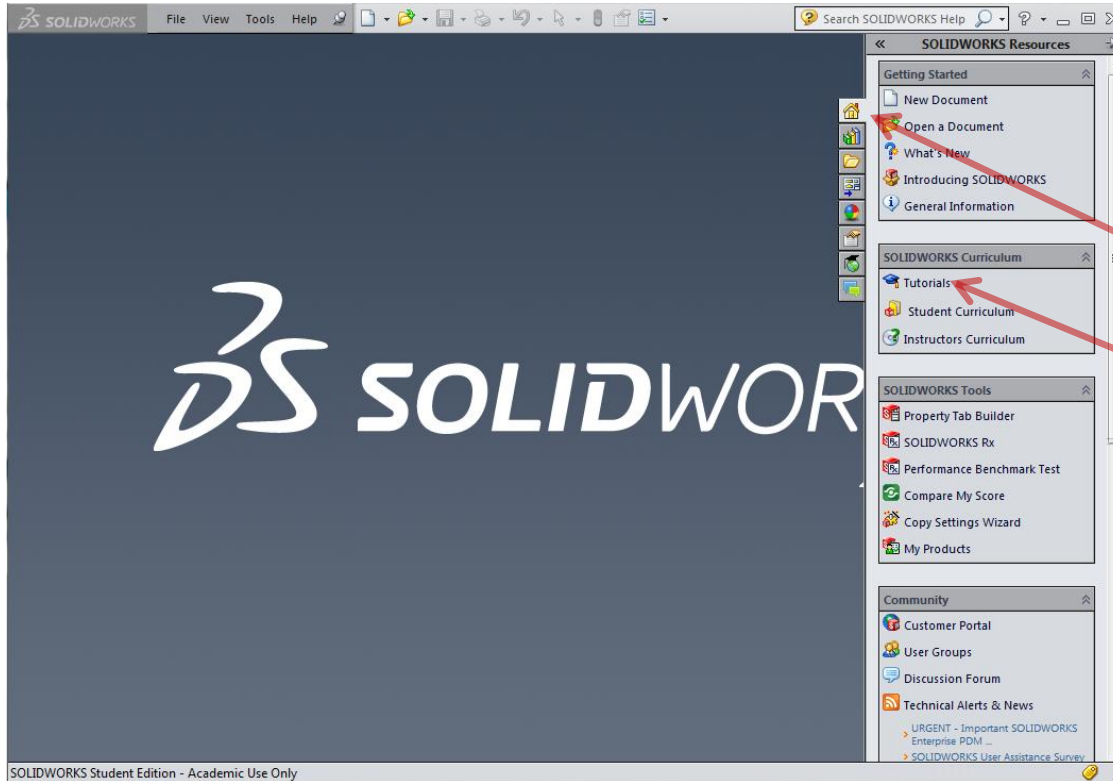
# SolidWorks Part 6

Prof. Steven S. Saliterman  
Introductory Medical Device Prototyping

Department of Biomedical Engineering, University of Minnesota

<http://saliterman.umn.edu/>

# Starting The Tutorials



- Launch SolidWorks
- Select Resources to open the Task Pane.
- Select Tutorials

# Select Design Evaluations

The screenshot shows the SOLIDWORKS Tutorials interface. At the top, there is a navigation bar with 'Show', 'Back', and 'Print' buttons. Below this is a main menu with several categories: 'Getting Started', 'Basic Techniques', 'Advanced Techniques', 'Productivity Tools', 'Design Evaluation' (highlighted in blue), 'CSWP/CSWA Preparation', 'What's New Examples', 'All SOLIDWORKS Tutorials', and 'Go to SOLIDWORKS Simulation Tutorials'. The main content area is divided into several tiles, each representing a different tutorial category: 'Animation', 'SOLIDWORKS SimulationXpress', 'SOLIDWORKS FloXpress', 'SOLIDWORKS Costing', 'SustainabilityXpress', 'DimXpert Tutorials', and 'TolAnalyst Tutorials'. Each tile contains a representative image of the tutorial's content.

Getting Started	Basic Techniques	Advanced Techniques
Productivity Tools	<b>Design Evaluation</b>	CSWP/CSWA Preparation
What's New Examples	All SOLIDWORKS Tutorials	Go to SOLIDWORKS Simulation Tutorials

**Animation**

**SOLIDWORKS SimulationXpress**

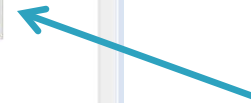
**SOLIDWORKS FloXpress**

**SOLIDWORKS Costing**

**SustainabilityXpress**

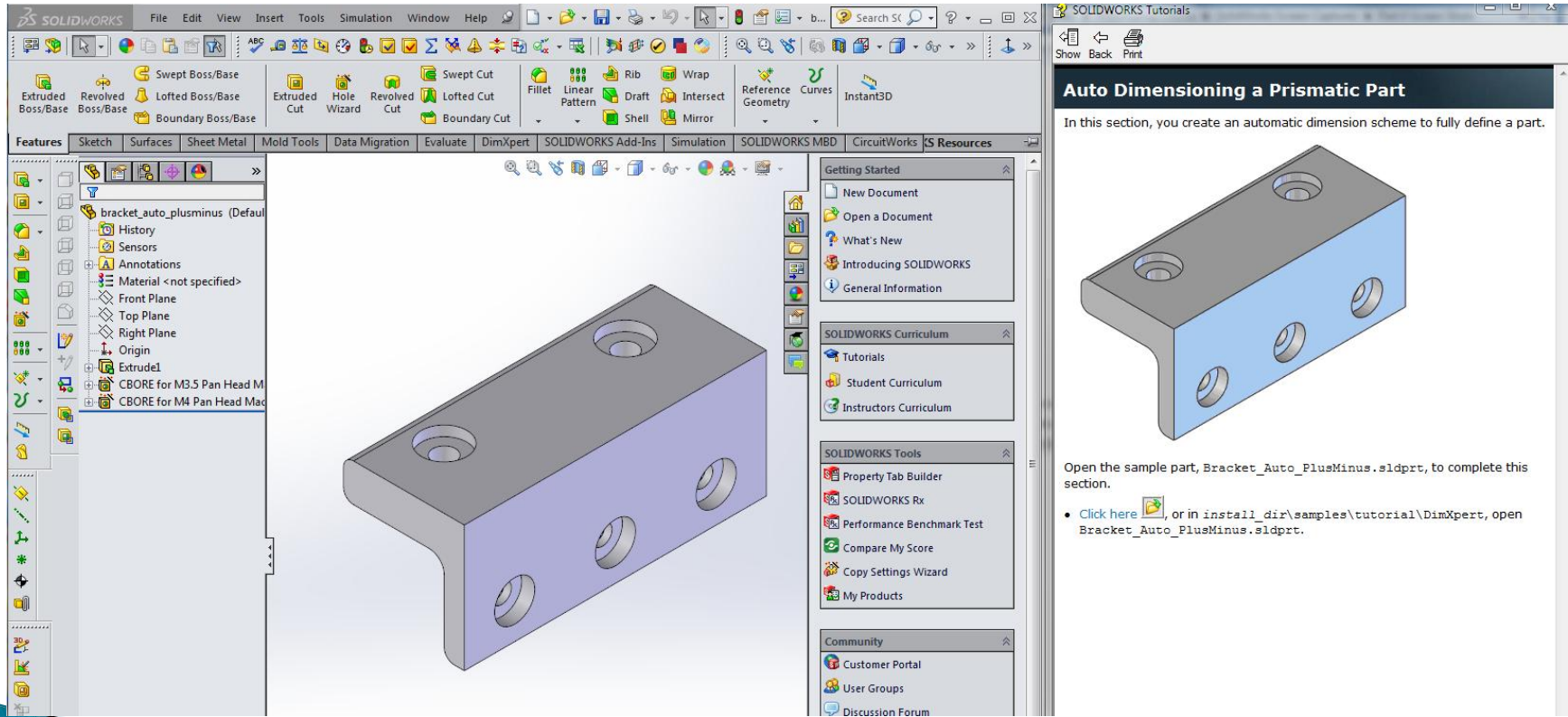
**DimXpert Tutorials**

**TolAnalyst Tutorials**



DimXpert

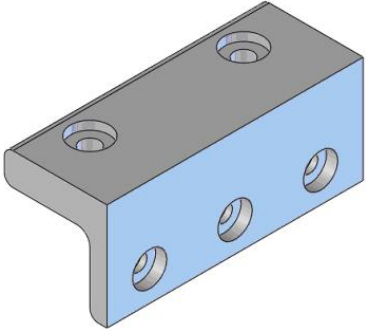
# Fully Defining a Part




The image displays the SolidWorks CAD software interface. The main window shows a 3D model of a bracket part, which is a rectangular block with a chamfered end and four circular holes. The left-hand side contains the Feature Tree, showing the part's history and structure. The top of the interface features a menu bar and a ribbon with various tool icons. On the right side, a tutorial window titled "Auto Dimensioning a Prismatic Part" is open. This window contains text explaining the purpose of the section and includes a 3D model of the same bracket part. Below the model, there is a list of instructions and a link to a sample part file.

**Auto Dimensioning a Prismatic Part**

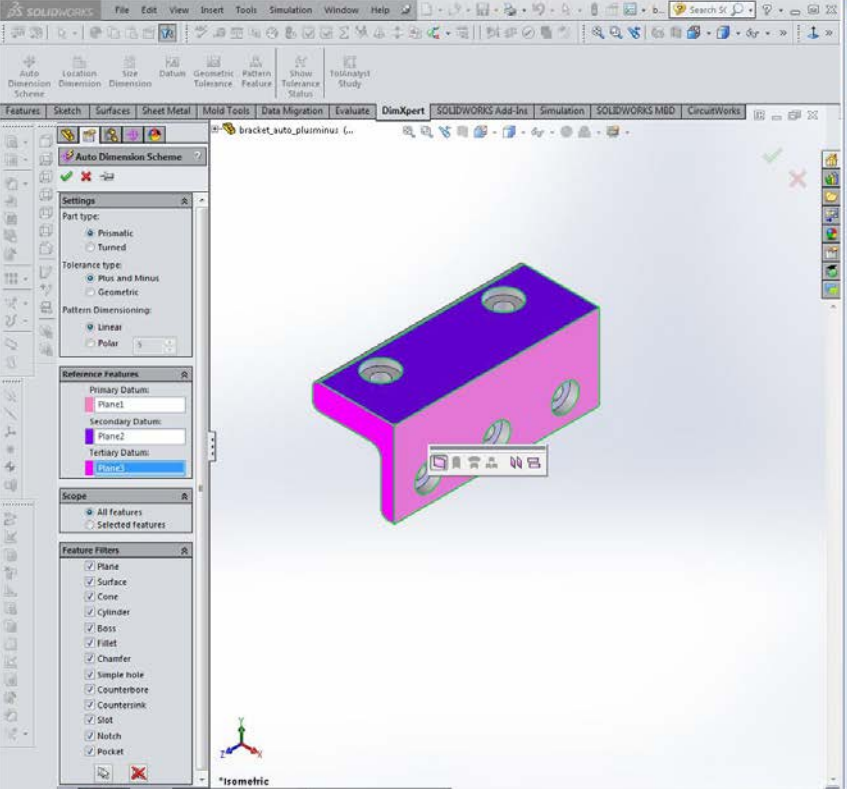
In this section, you create an automatic dimension scheme to fully define a part.



Open the sample part, Bracket\_Auto\_PlusMinus.sldprt, to complete this section.

- [Click here](#) , or in `install_dir\samples\tutorial\DimXpert`, open Bracket\_Auto\_PlusMinus.sldprt.


# Selecting the Reference Features



**Selecting the Reference Features**

You select reference features to use as origins for location dimensions. Manufacturing and inspection use these features for part setup or to position the part within an assembly.

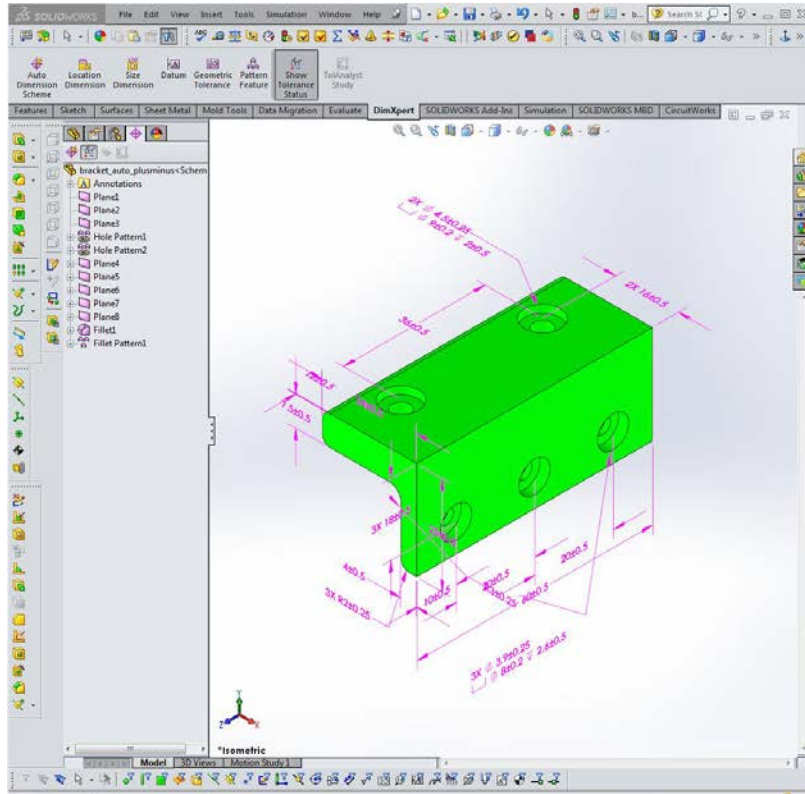
1. In the PropertyManager, under **Reference Features**, click **Primary Datum**.
2. Select the right face as shown.

Right-click when this pointer appears  to advance the selection to the next datum.

3. Click **Secondary Datum**.
4. Select the face of the top plane.

5. Click **Tertiary Datum**.
6. Select the face of the front plane.

# Chain Dimensions



### Creating the Dimension Scheme

1. Click
2. Inspect the dimensions by annotation view:

**Front view** The instance count (3x) is part of the R2 radius dimension.

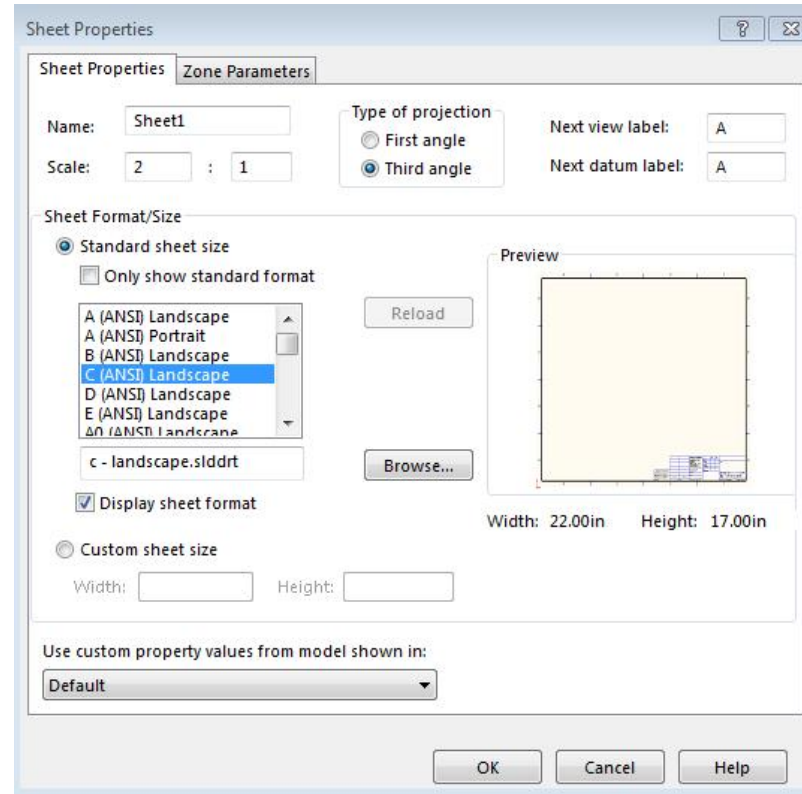
**Right view** The instance count and chain dimensions are part of the 3X counterbore hole pattern.

**Top view** The instance count and chain dimensions are part of the 2X counterbore hole pattern.

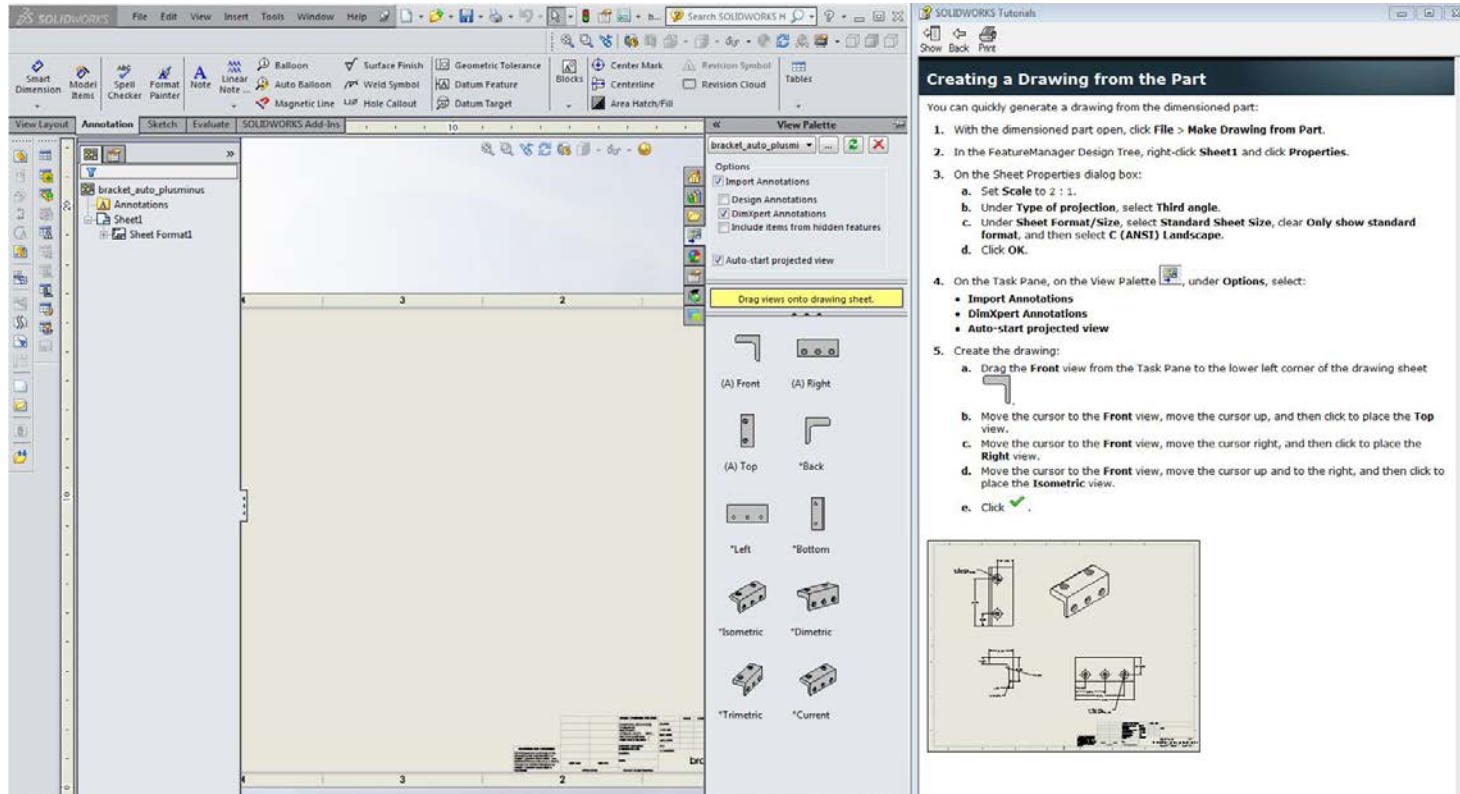
To choose between baseline and chain dimension schemes, click **Options** (Standard toolbar). On the Document Properties tab, go to **DimXpert** > **Chain Dimension**.

PREVIOUS TOPIC      NEXT TOPIC

# Drawing Sheet Properties






# View Palette Under Task Pane



The screenshot displays the SolidWorks interface. The main window shows a 3D model of a bracket part. The View Palette is open, showing various view options. A tutorial window titled "Creating a Drawing from the Part" is overlaid on the right side of the screen. The tutorial provides a step-by-step guide on how to generate a drawing from a dimensioned part.

**Creating a Drawing from the Part**

You can quickly generate a drawing from the dimensioned part:

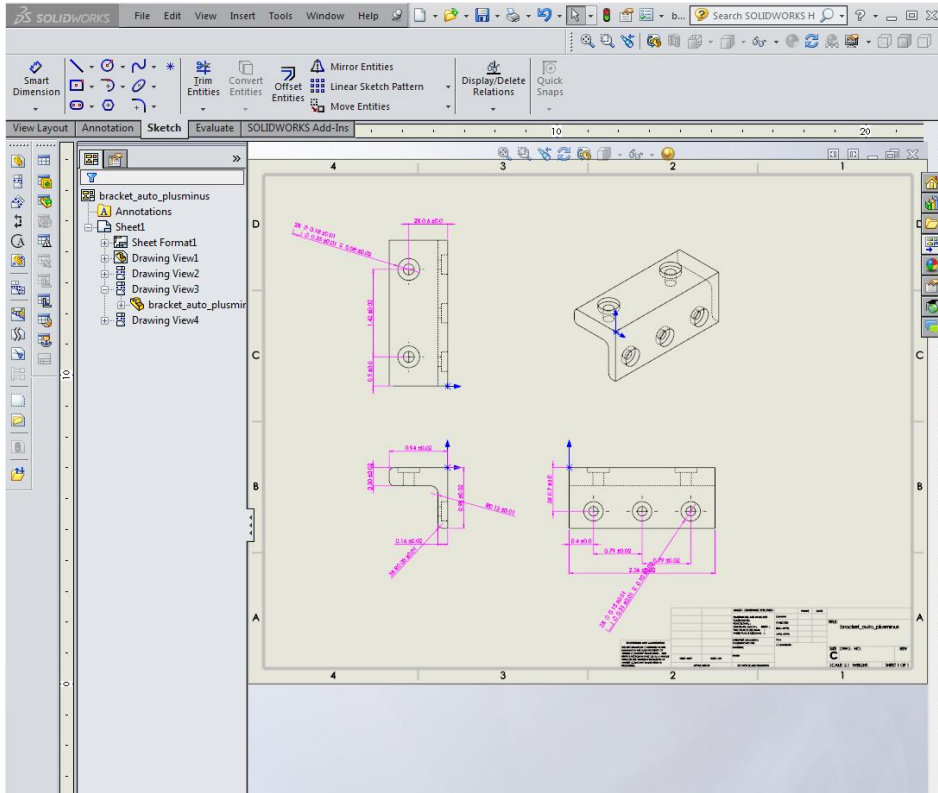
1. With the dimensioned part open, click **File > Make Drawing from Part**.
2. In the FeatureManager Design Tree, right-click **Sheet1** and click **Properties**.
3. On the Sheet Properties dialog box:
  - a. Set **Scale** to 2 : 1.
  - b. Under **Type of projection**, select **Third angle**.
  - c. Under **Sheet Format/Size**, select **Standard Sheet Size**, clear **Only show standard format**, and then select **C (ANSI) Landscape**.
  - d. Click **OK**.
4. On the Task Pane, on the View Palette , under **Options**, select:
  - **Import Annotations**
  - **DimXpert Annotations**
  - **Auto-start projected view**
5. Create the drawing:
  - a. Drag the **Front** view from the Task Pane to the lower left corner of the drawing sheet .
  - b. Move the cursor to the **Front** view, move the cursor up, and then click to place the **Top** view.
  - c. Move the cursor to the **Front** view, move the cursor right, and then click to place the **Right** view.
  - d. Move the cursor to the **Front** view, move the cursor up and to the right, and then click to place the **Isometric** view.
  - e. Click .

The View Palette options shown in the screenshot are:

- (A) Front
- (A) Right
- (A) Top
- \*Back
- \*Left
- \*Bottom
- \*Isometric
- \*Dimetric
- \*Trimetric
- \*Current






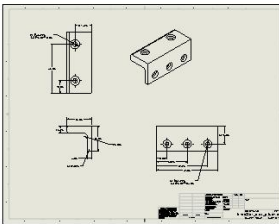
# Adding Front, Top (Up), Right and Isometric Views



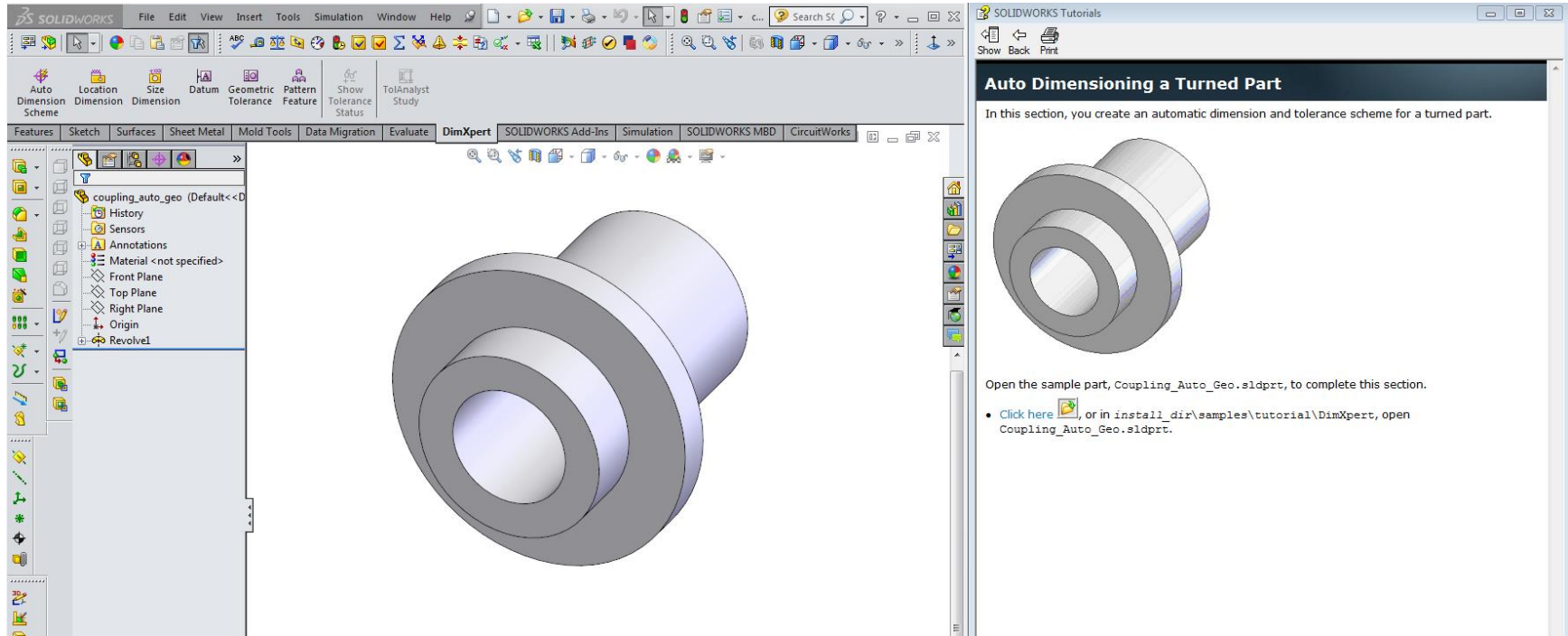
### Creating a Drawing from the Part

You can quickly generate a drawing from the dimensioned part:

1. With the dimensioned part open, click **File > Make Drawing from Part**.
2. In the FeatureManager Design Tree, right-click **Sheet1** and click **Properties**.
3. On the Sheet Properties dialog box:
  - a. Set **Scale** to 2 : 1.
  - b. Under **Type of projection**, select **Third angle**.
  - c. Under **Sheet Format/Size**, select **Standard Sheet Size**, clear **Only show standard format**, and then select **C (ANSI) Landscape**.
  - d. Click **OK**.
4. On the Task Pane, on the View Palette , under **Options**, select:
  - **Import Annotations**
  - **DimXpert Annotations**
  - **Auto-start projected view**
5. Create the drawing:
  - a. Drag the **Front** view from the Task Pane to the lower left corner of the drawing sheet .
  - b. Move the cursor to the **Front** view, move the cursor up, and then click to place the **Top** view.
  - c. Move the cursor to the **Front** view, move the cursor right, and then click to place the **Right** view.
  - d. Move the cursor to the **Front** view, move the cursor up and to the right, and then click to place the **Isometric** view.
  - e. Click .



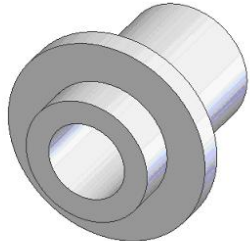
# Auto-Dimensioning a Turned Part



The image displays a SolidWorks software interface. The main window shows a 3D model of a turned part, which is a cylindrical component with a larger diameter section on the left and a smaller diameter section on the right. The interface includes a menu bar at the top with options like File, Edit, View, Insert, Tools, Simulation, Window, and Help. Below the menu bar is a toolbar with various icons. The left sidebar contains a Feature Tree with the following items: coupling\_auto\_geo (Default<<D), History, Sensors, Annotations, Material <not specified>, Front Plane, Top Plane, Right Plane, Origin, and Revolve1. The right sidebar is titled "SOLIDWORKS Tutorials" and contains a section titled "Auto Dimensioning a Turned Part".

**Auto Dimensioning a Turned Part**

In this section, you create an automatic dimension and tolerance scheme for a turned part.



Open the sample part, Coupling\_Auto\_Geo.sldprt, to complete this section.

- [Click here](#), or in `install_dir\samples\tutorial\DimXpert`, open Coupling\_Auto\_Geo.sldprt.

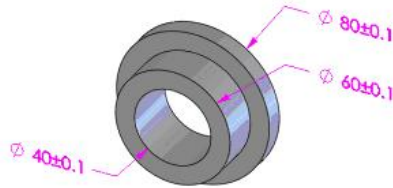
# Prismatic Dimensions

## Prismatic vs. Turned

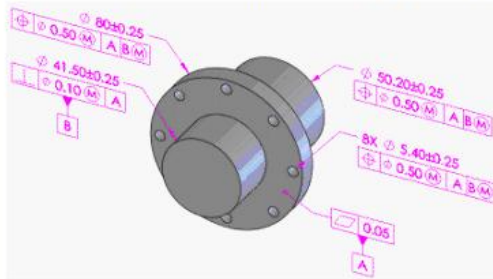
Using **Auto Dimension Scheme**, you can specify one of two part types, **Prismatic** or **Turned**. This option affects the annotation views and leader options used to display diameter dimensions. With geometric tolerancing, the option also affects the type of tolerances used to locate features that are concentric to a datum axis.

### Prismatic Parts

Diameter dimensions are displayed with a directed leader in an annotation view that is perpendicular to the feature's axis.



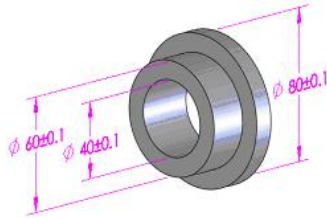
Position tolerances are used to locate features that are concentric with a datum axis. The tolerances applied to the 41.50, 50.20, and 80 diameters are position tolerances. They are concentric with the axis of datum B.



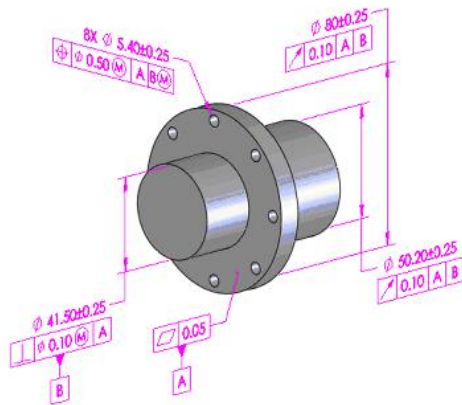
# Turned Dimensions

## Turned

Diameter dimensions are displayed in an annotation view as linear dimensions that are parallel to the feature's axis.



Runout tolerances are used to locate features that are concentric with a datum axis. The tolerances applied to the 41.50, 50.20, and 80 diameters are runout tolerances. They are concentric with the axis of datum **B**.



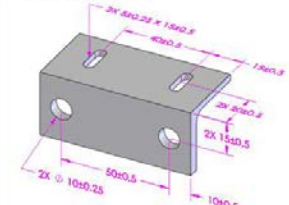
# Plus-Minus Tolerances

**Plus-Minus vs. Geometric**

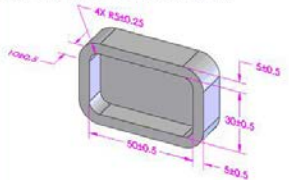
Using **Auto Dimension Scheme**, you can specify one of two tolerance types, **Plus and Minus** or **Geometric**. This option affects the tolerance types used to control the location of bosses, cones, holes, notches, slots, and widths, as well as pockets and profile features. With geometric tolerancing, tolerances also control the relationships between the datum features.

**Plus and Minus Tolerancing**

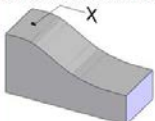
Features-of-size are located with linear dimensions if they are parallel to one of the reference features.



Pockets are located with linear dimensions if the wall features are parallel to one of the reference features. The fillets, chamfers, and depth of the pocket are dimensioned for size.



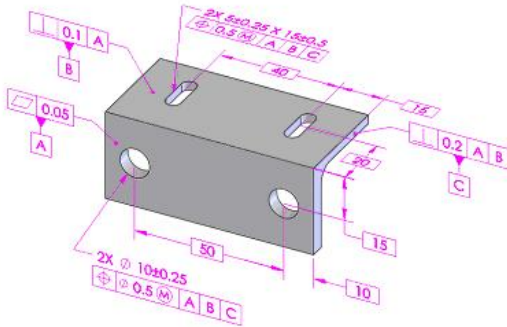
Profile features are not dimensioned.



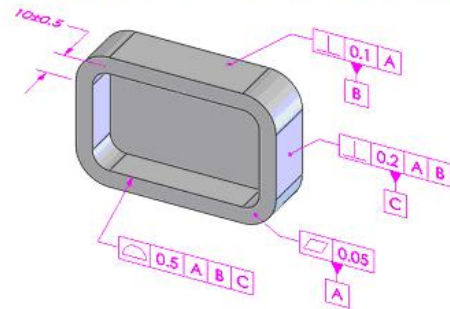
# Geometric Tolerances

## Geometric Tolerancing

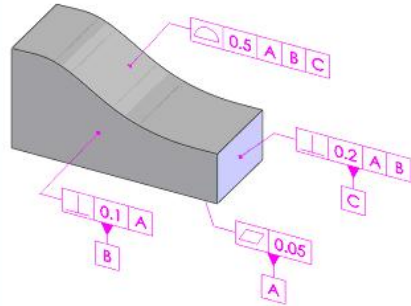
Features-of-size are located with position or runout tolerances as defined by the part type (prismatic or turned).



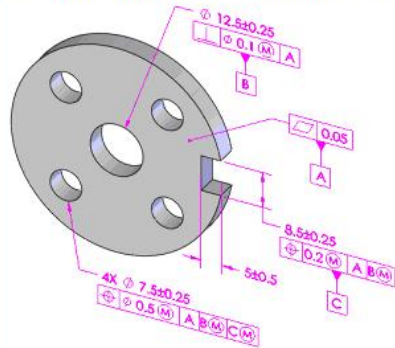
Pockets are located with surface profile tolerances, including the fillets and chamfers applied along the walls of the pocket. The depth of the pocket is dimensioned separately.



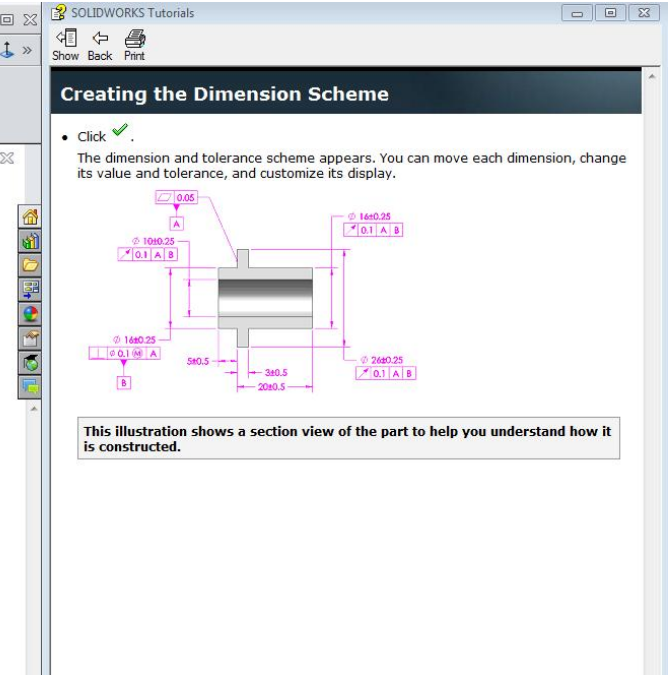
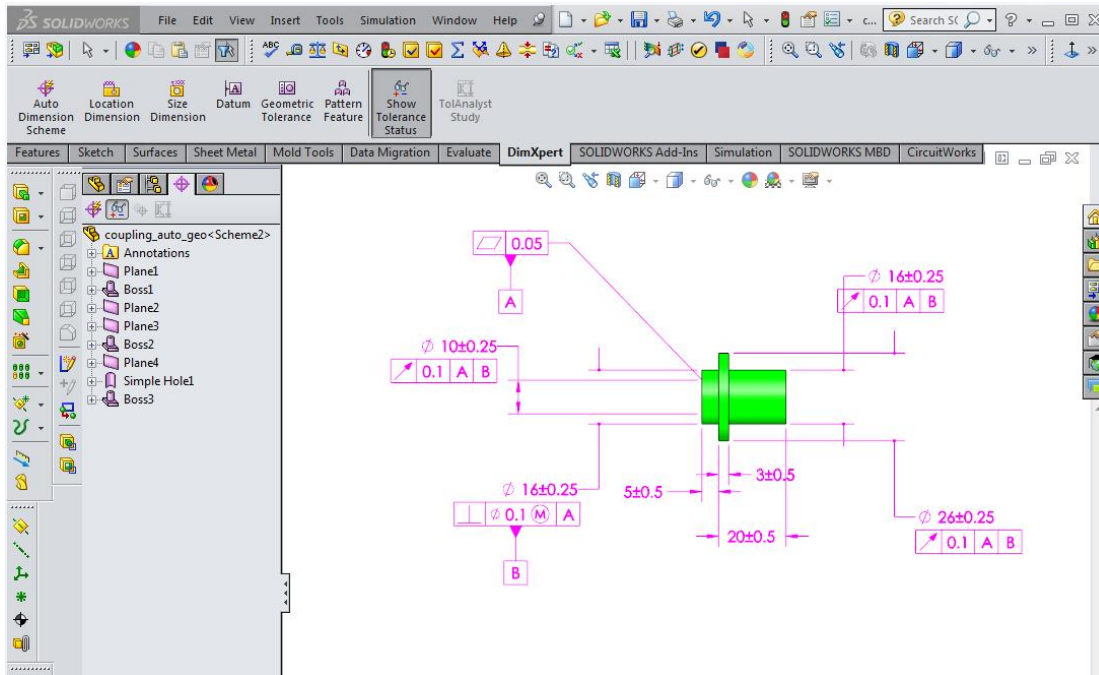
Profile features are located with surface profile tolerances.



The datum selections are tolerated in order to control their interrelationships. The following is the result where the front plane (A) is chosen as primary, the center hole (B) as secondary, and the notch (C) as tertiary, for controlling the 4X pattern of holes.

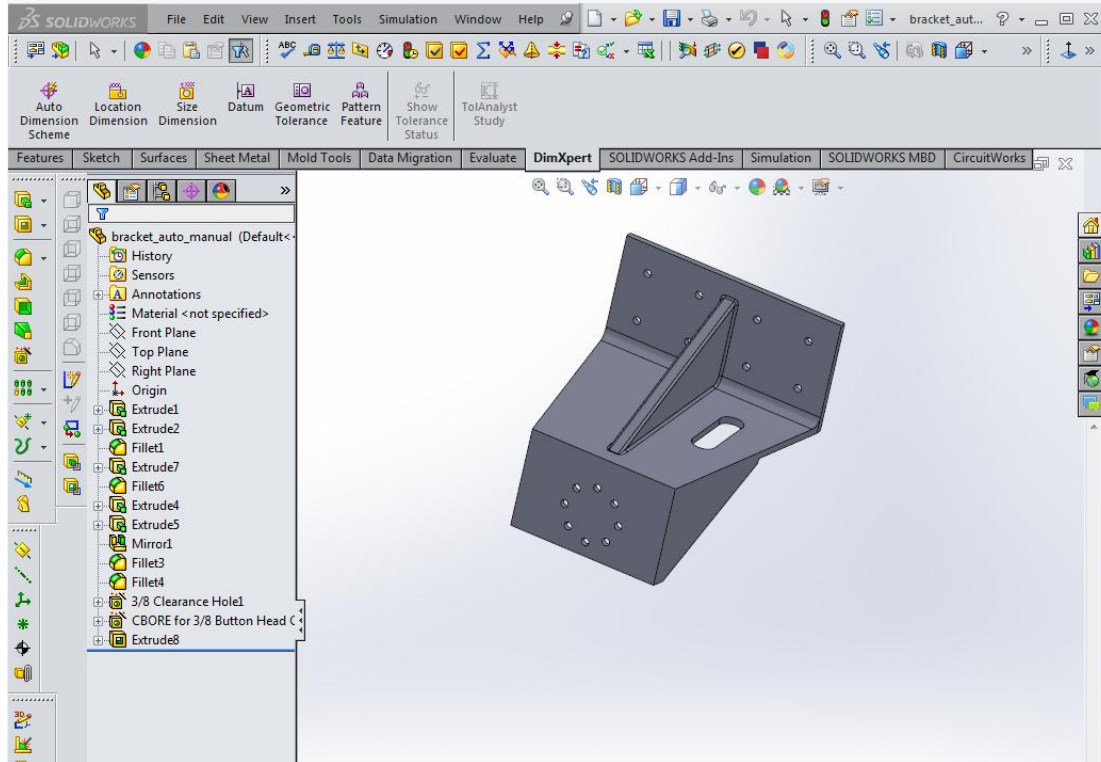


# Dimension Scheme





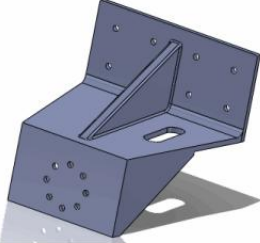
# Combining Dimension Schemes




SOLIDWORKS Tutorials

## Combining Dimension Schemes

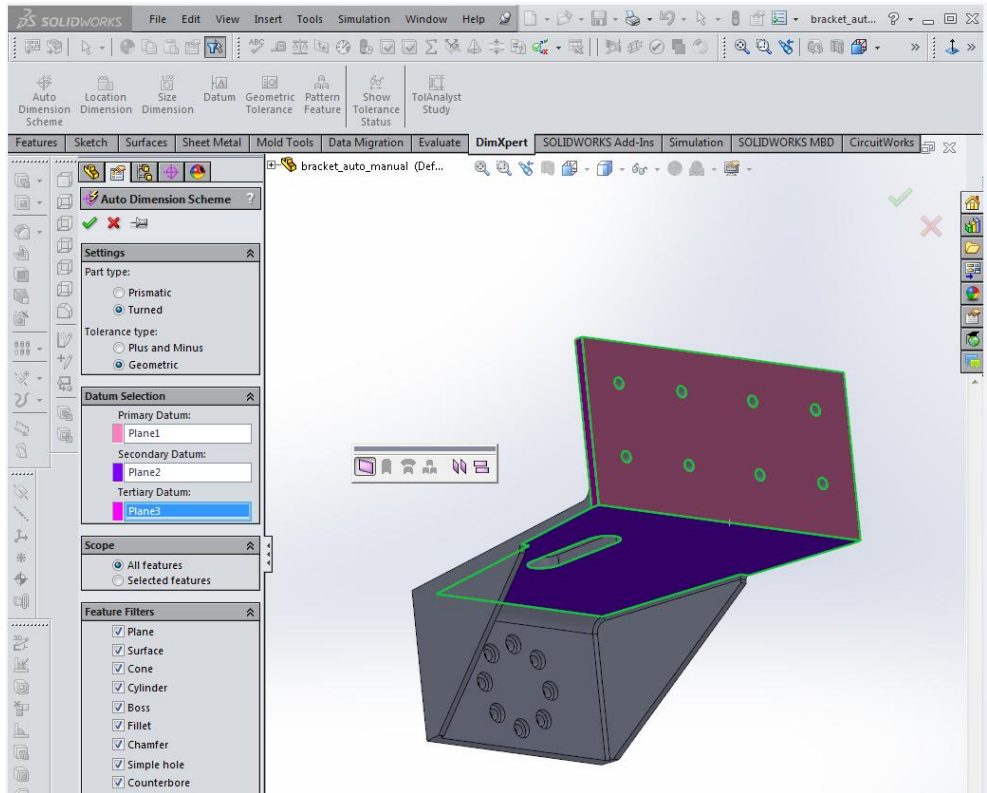
In this section, you use a combined automatic and manual scheme to fully define a part. When you have finished, the part will be ready for use in TolAnalyst.



Open the sample part, Bracket\_Auto\_Manual.sldprt, to complete this section.

- [Click here](#) , or in `install_dir\samples\tutorial\DimXpert`, open Bracket\_Auto\_Manual.sldprt.

# Select Primary, Secondary and Tertiary Datums



bracket\_auto\_manual (Def...)

Auto Dimension Scheme

Settings

Part type:

- Prismatic
- Turned

Tolerance type:

- Plus and Minus
- Geometric

Datum Selection

Primary Datum: Plane1

Secondary Datum: Plane2

Tertiary Datum: Plane3

Scope

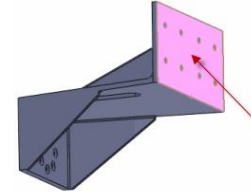
- All Features
- Selected Features


Feature Filters

- Plane
- Surface
- Cone
- Cylinder
- Boss
- Fillet
- Chamfer
- Simple hole
- Counterbore

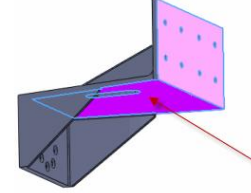
SOLIDWORKS Tutorials

- In the PropertyManager, under **Datum Selection**, click **Primary Datum**.
- Select the right face as shown.

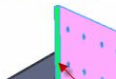


Right-click when this pointer appears  to advance the selection to the next datum.

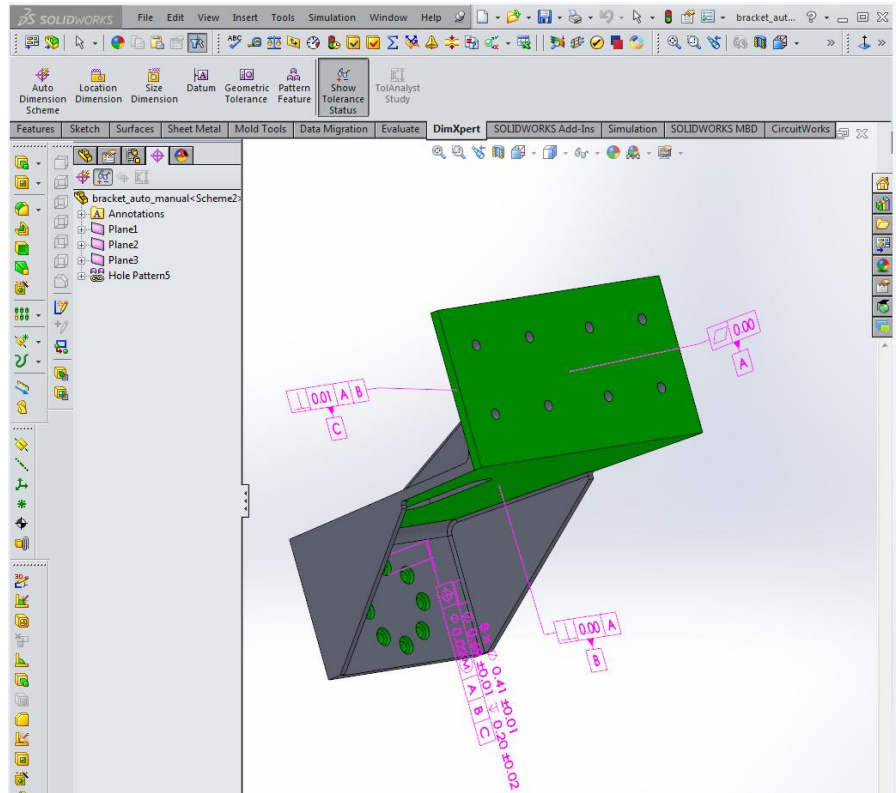
- Click **Secondary Datum**.
- Select the face containing the slot.



- Click **Tertiary Datum**.
- Select the side face as shown.



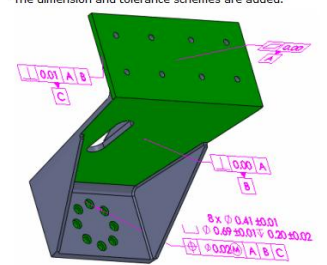
# Green Areas are Fully Toleranced



The screenshot shows the SolidWorks interface with a 3D model of a bracket. The top surface of the bracket is highlighted in green. Several dimension lines are applied to the model, with some dimensions having green boxes around them. The feature tree on the left shows the model's structure, including annotations, planes, and a hole pattern.

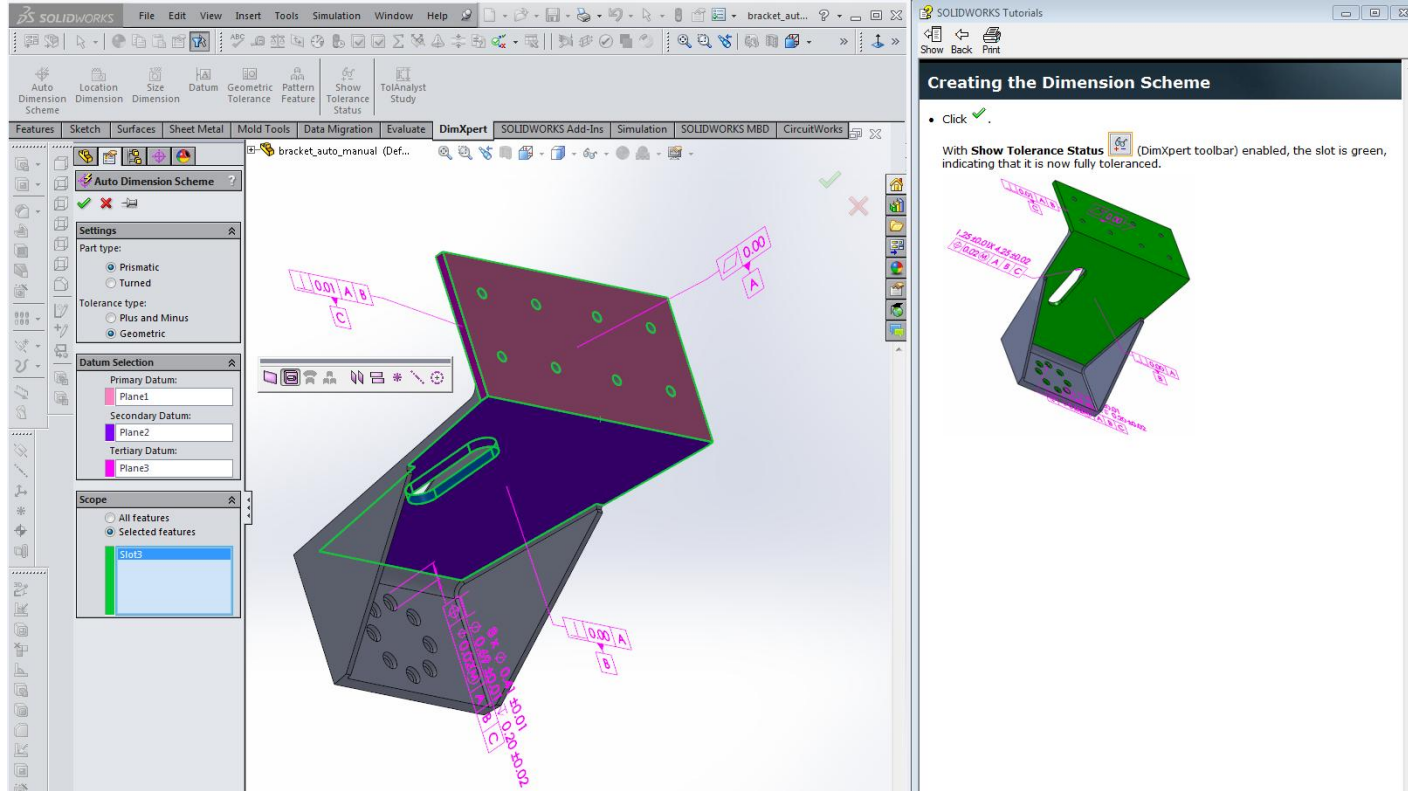
**SOLIDWORKS Tutorials**  
Creating the Dimension Scheme

- Click ✓  
The dimension and tolerance schemes are added.



Notice that **Show Tolerance Status** (DimXpert toolbar) is enabled. The three datum faces and the counterbore holes are green, indicating that those entities are fully toleranced.

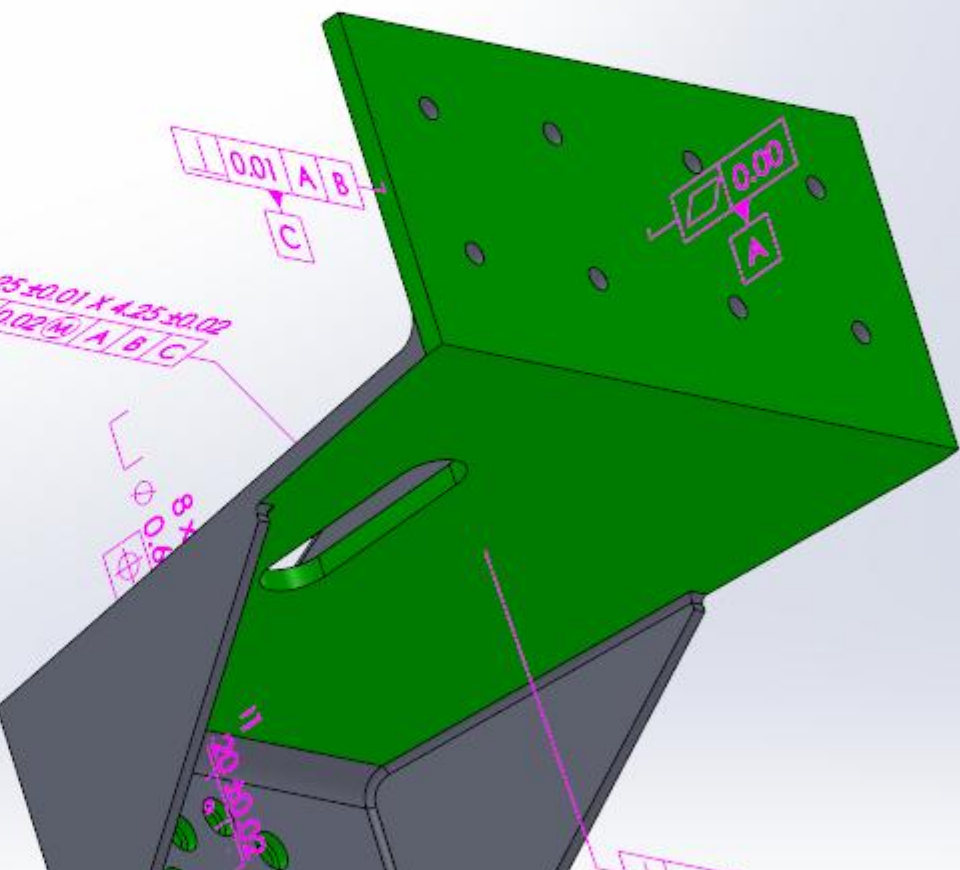
# Adding the Slot




The image displays the SolidWorks software interface. The main window shows a 3D model of a bracket with various dimensions and datums. The left sidebar contains the 'Auto Dimension Scheme' settings, including 'Part type' (Prismatic), 'Tolerance type' (Geometric), 'Datum Selection' (Plane1, Plane2, Plane3), and 'Scope' (Slot3). The right sidebar shows a tutorial window titled 'Creating the Dimension Scheme' with the following content:

- Click ✓
- With **Show Tolerance Status** (DimXpert toolbar) enabled, the slot is green, indicating that it is now fully tolerated.

The 3D model shows a bracket with a slot. Dimensions are labeled with 'A', 'B', 'C', and 'D'. The slot is highlighted in green, indicating it is fully tolerated. The tutorial window shows a 3D model of the bracket with dimensions and a green highlight on the slot.

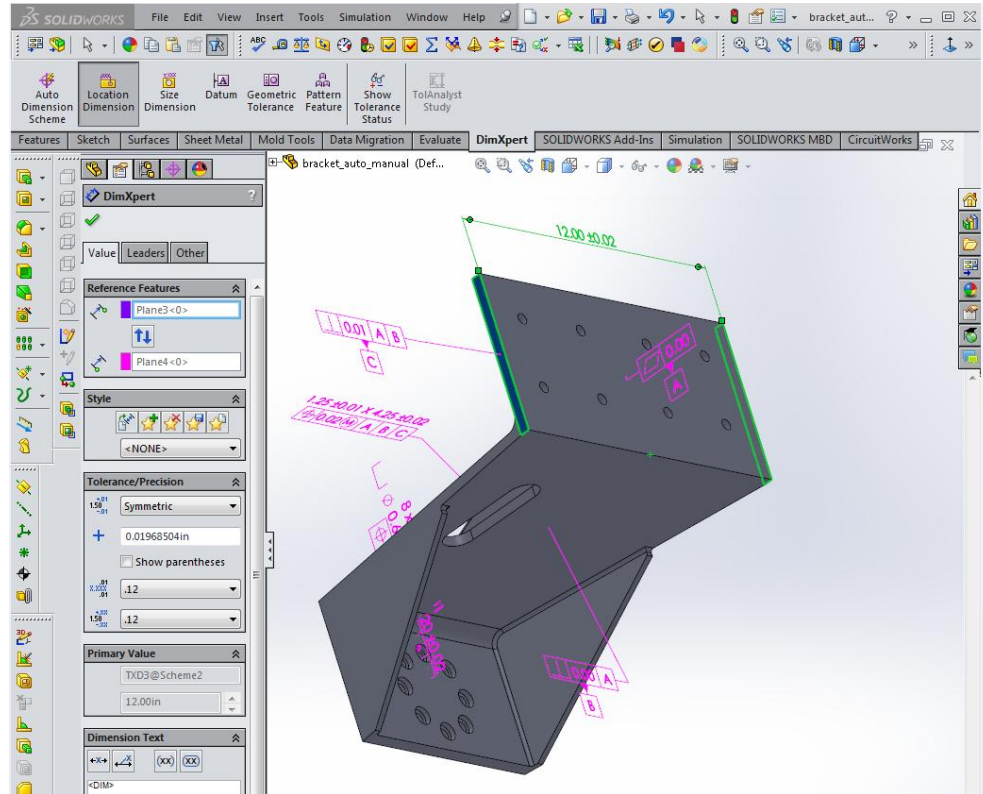


- Click  .

With **Show Tolerance Status**  indicating that it is now fully tolera




# Dimensioning Manually



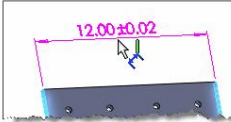
## Dimensioning Manually

In this section, you create some dimensions manually.

1. Click **Location Dimension** (DimXpert toolbar).
2. Hover over the upper edge of the bracket.

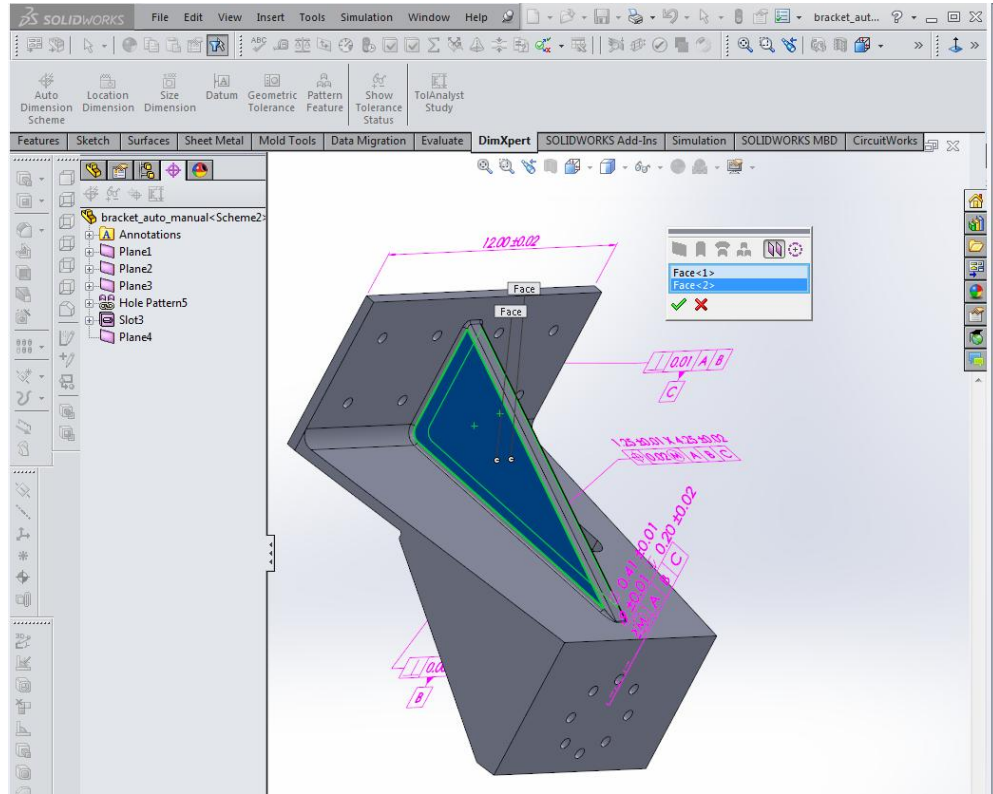


3. Drag up and then click to place the dimension.



4. In the PropertyManager, click .


# Size Dimension –Create Width Feature



The screenshot shows the SolidWorks interface with a 3D model of a bracket. A width dimension of  $12.00 \pm 0.02$  is applied to the top surface. A rib width dimension of  $0.75 \pm 0.01$  is applied to the rib. The PropertyManager shows the 'Create Width Feature' dialog with 'Face-1' and 'Face-2' selected. The Feature Tree on the left shows 'Annotations', 'Plane1', 'Plane2', 'Plane3', 'Hole Pattern5', 'Slot3', and 'Plane4'. The Command Bar shows 'Auto Dimension Scheme', 'Location Dimension', 'Size Dimension', 'Datum', 'Geometric Tolerance', 'Pattern Feature', 'Show Tolerance Status', and 'ToAnalyst Study'. The DimXpert toolbar is visible at the top.

### Dimensioning the Width of the Rib

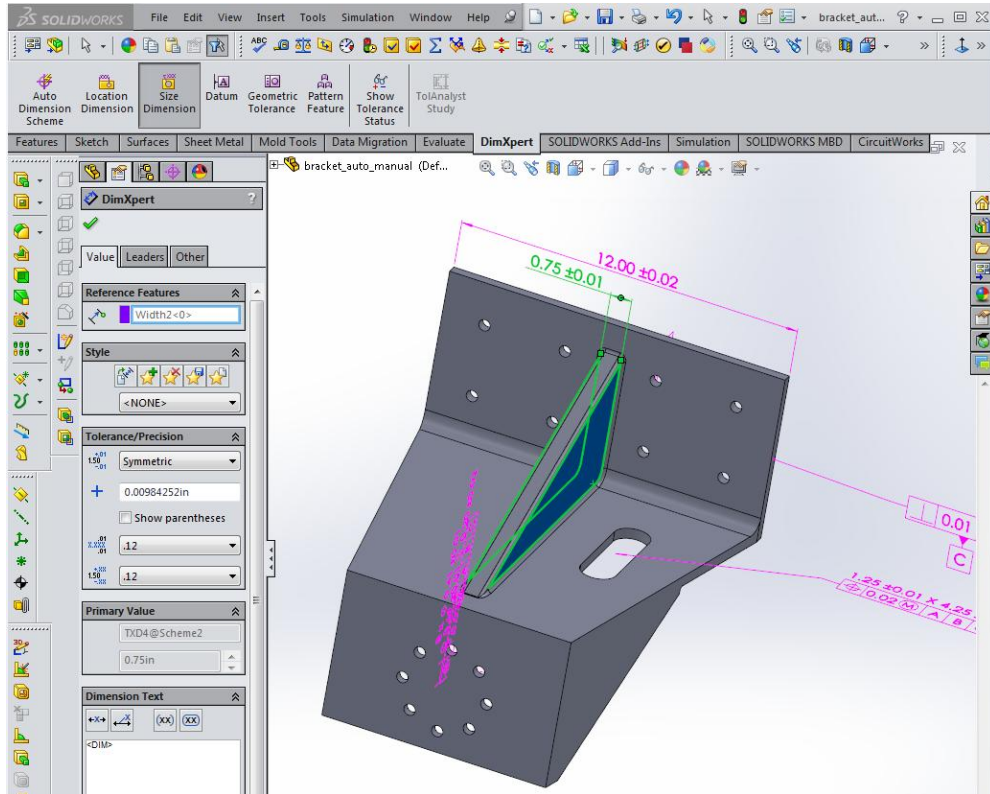
1. Click **Size Dimension** (DimXpert toolbar).
2. Turn the model to the front so that you can see the rib.
3. Select a triangular side of the rib.
4. On the feature selector, select **Create Width Feature**.
5. Turn the model and select the other side of the rib.
6. On the feature selector, click **✓**.
7. Drag up from the rib to place the dimension.



8. In the PropertyManager, click **✓**.

With **Show Tolerance Status** (DimXpert toolbar) enabled, the rib sides are yellow, indicating that it is partly tolerated.

# Showing Width Dimension



### SOLIDWORKS Tutorials

#### Dimensioning the Width of the Rib

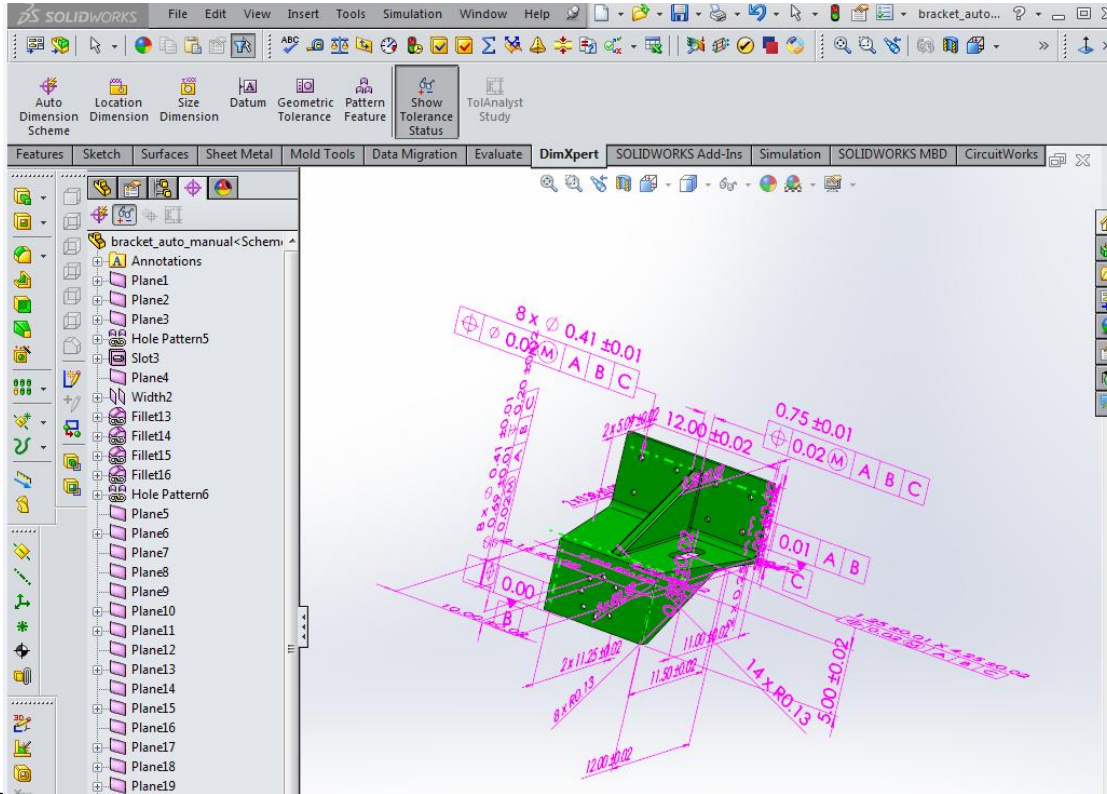
1. Click **Size Dimension** (DimXpert toolbar).
2. Turn the model to the front so that you can see the rib.
3. Select a triangular side of the rib.
4. On the feature selector, select **Create Width Feature**.
5. Turn the model and select the other side of the rib.
6. On the feature selector, click **✓**.
7. Drag up from the rib to place the dimension.

8. In the PropertyManager, click **✓**.

With **Show Tolerance Status** (DimXpert toolbar) enabled, the rib sides are yellow, indicating that it is partly tolerated.




# Final Dimensioning



SOLIDWORKS Tutorials

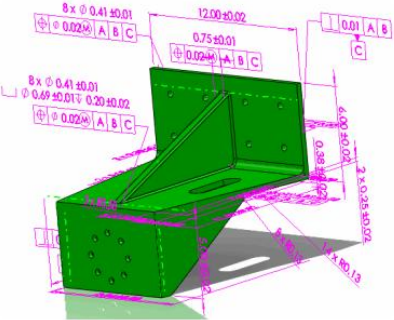
## Creating the Dimension Scheme

1. Click  .

With **Show Tolerance Status** (DimXpert toolbar) enabled, the entire model is green, indicating that the whole part is now fully tolerated.

2. Optionally, drag the dimensions to rearrange them for readability.

The part is now ready for use in TolAnalyst.



**Congratulations!** You have completed this tutorial.  
Return to the tutorials overview page.

# Summary

## ▶ Topics Covered

- Selecting datum reference features from a part.
- Making a drawing from the part.
- Drawing sheet properties.
- Adding views to the drawing.
- Auto-dimensioning a turned part.
- Prismatic vs turned dimensions.
- Plus-minus vs Geometric tolerances.
- Selecting primary, secondary and tertiary datums.
- Automatic and manual dimensioning.
- Using Create Width feature.