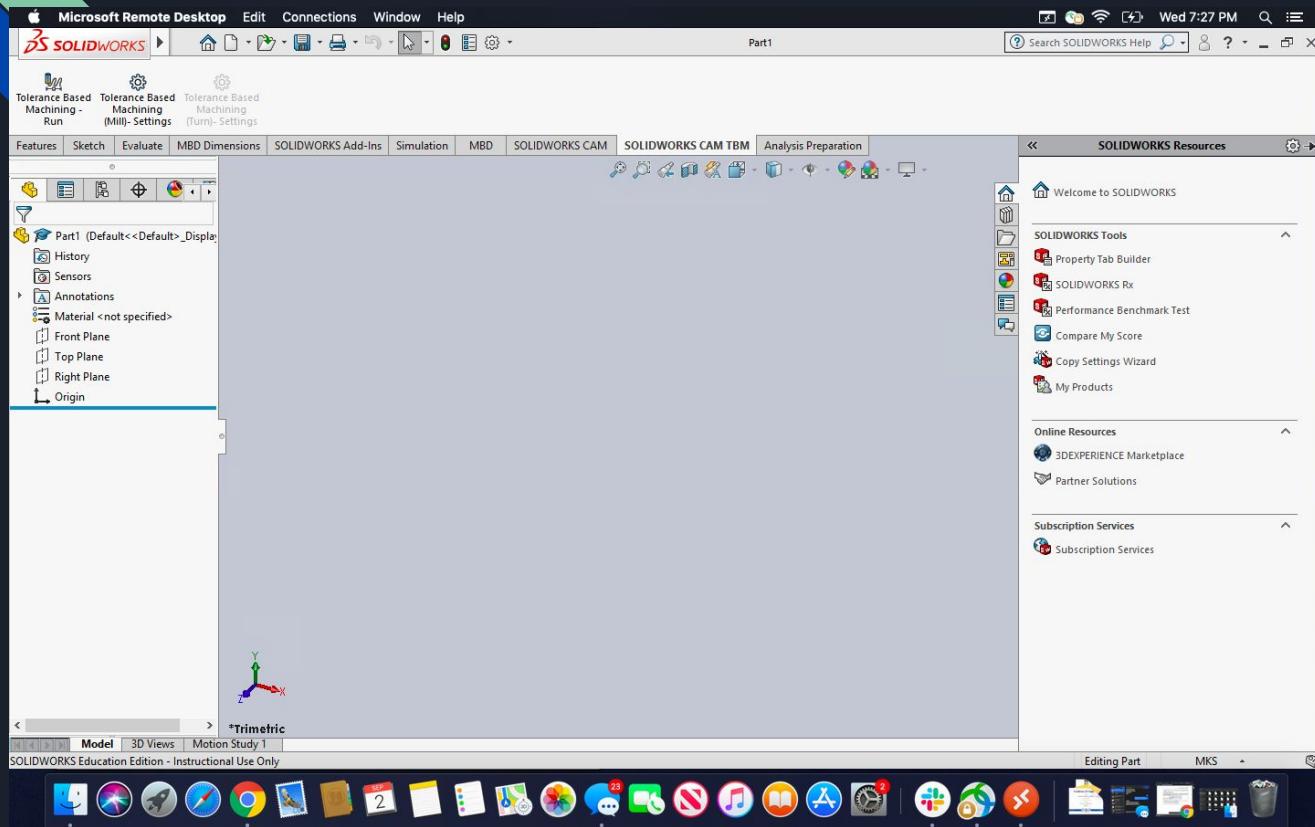


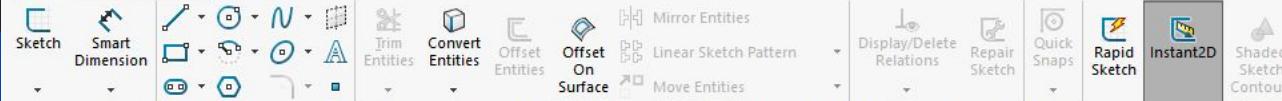


Solidworks Modeling

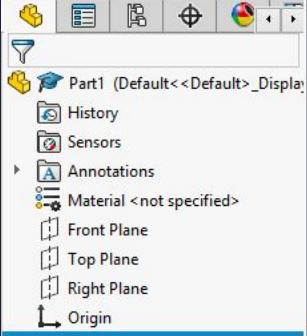
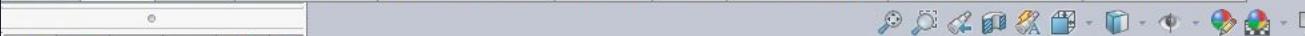
Sean Liam McCarthy
UCLA Transfer Bridge 2020



After Pressing
“NEW PART” on
the first page, you
should see the
following screen



Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation



*Trimetric

Model 3D Views Motion Study 1

SOLIDWORKS Education Edition - Instructional Use Only

Editing Part MKS



Microsoft Remote Desktop

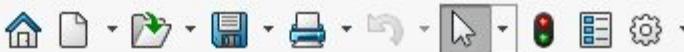
Edit

Connections

Window

Help

DS SOLIDWORKS

 Sketch
 Smart Dimension Trim Entities
 Convert Entities
 Offset Entities
 Offset On Surface Mirror Entities
 Linear Sketch Pattern
 Move Entities

Features

Sketch

Evaluate

MBD Dimension

SOLIDWORKS Add-Ins

Simulation

MBD

SOLIDWORKS CA



Part1 (Default<<Default>>_Display)

- History
- Sensors
- Annotations
 - Material <not specified>
- Front Plane
- Top Plane
- Right Plane
- Origin

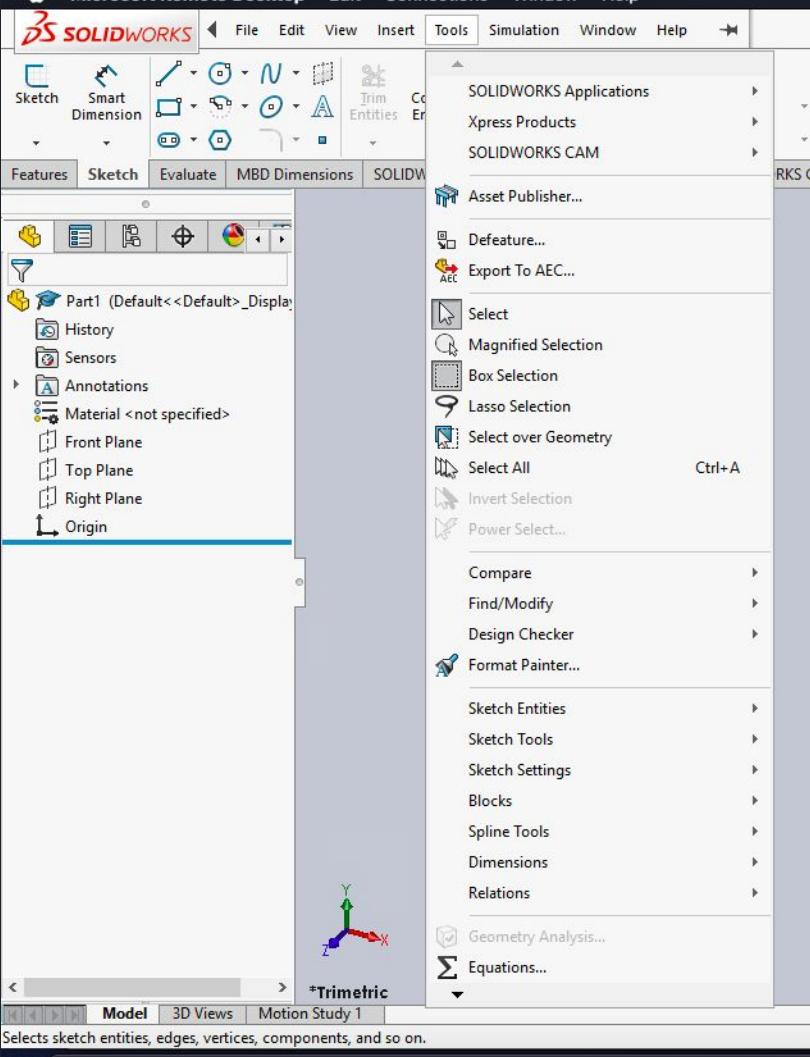
Command Bar

Feature Tree

On the far left, you should see this window.

The upper command bar is where we will make changes to our part

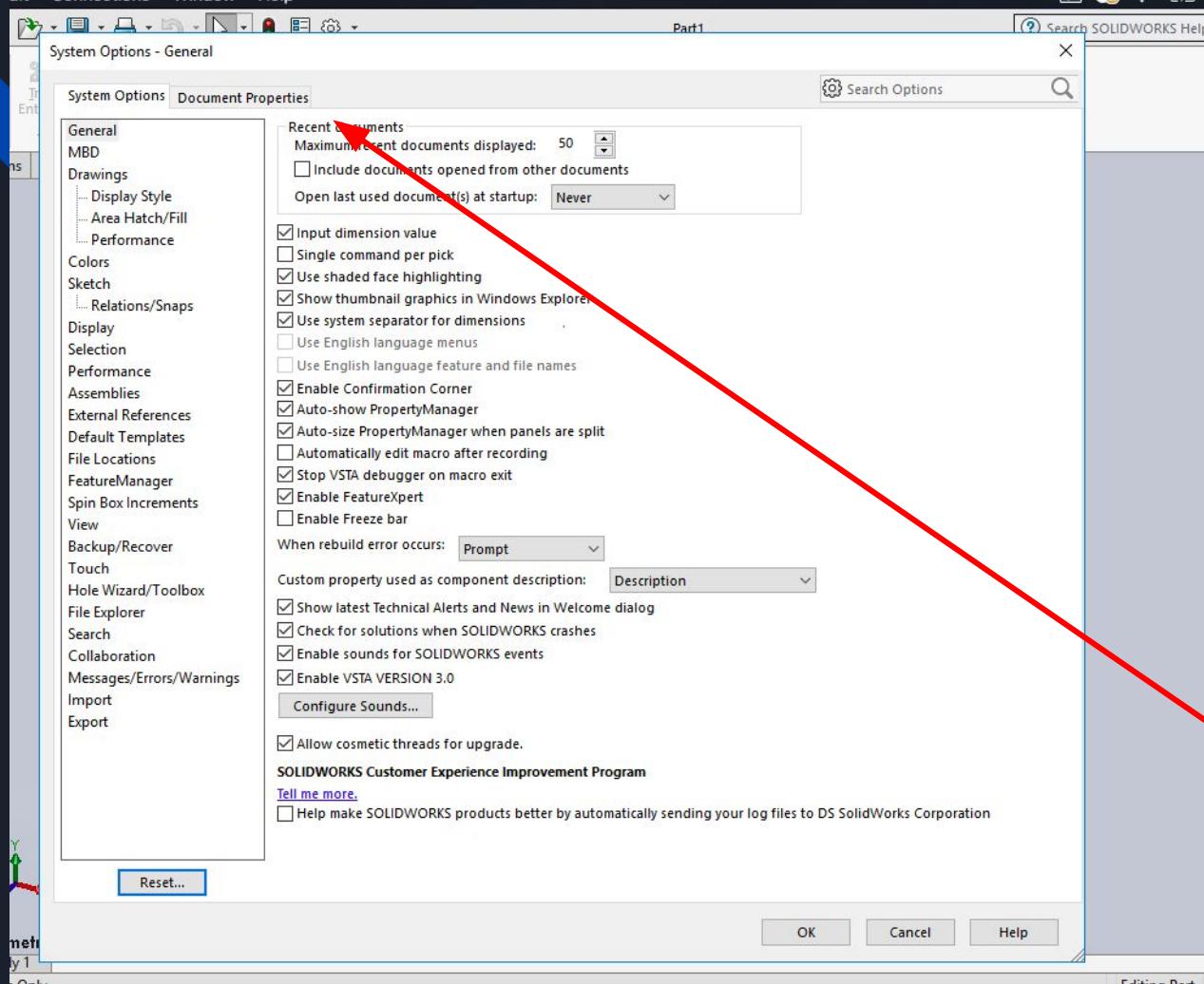
The feature tree will allow you to see all of the part's organized features



Additionally, the upper-most bar can be used for managing files.

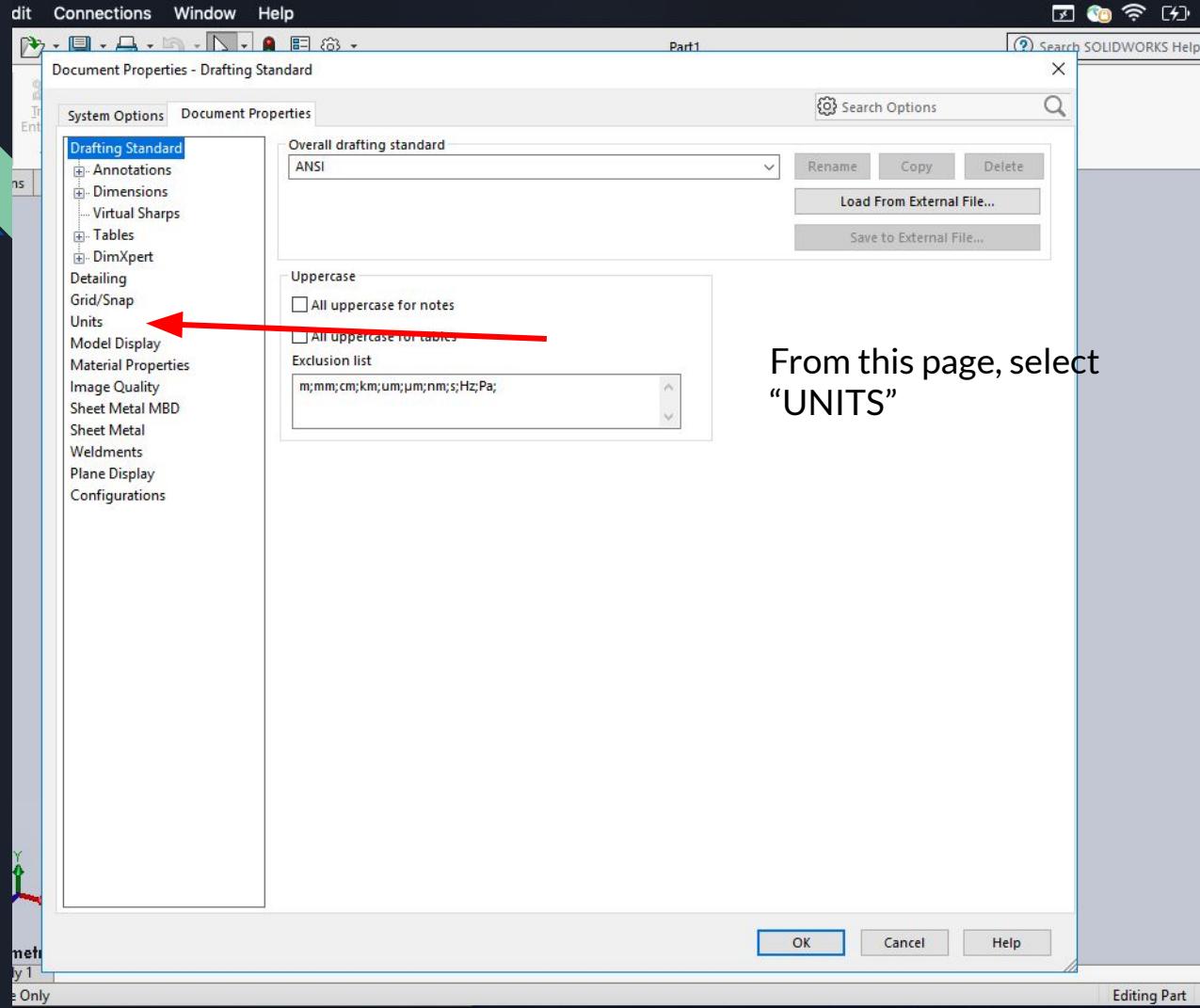
We will begin setting up this part file by selecting tools, which will create the dropdown menu to the right.

Scroll to the bottom, and select options. It will have a gear icon next to it.

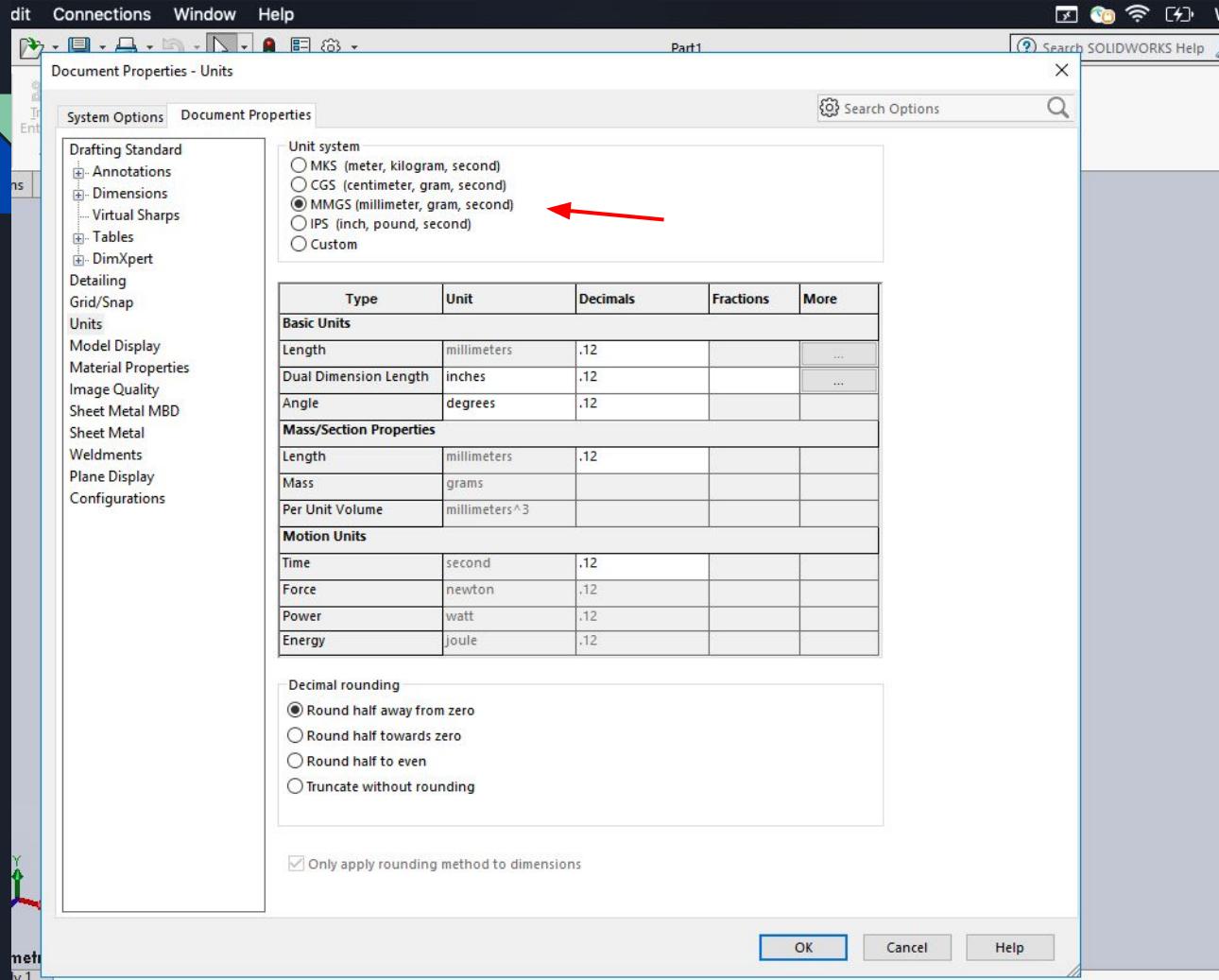


Having clicked options, you should see the window to the right. There are many different options which we will not worry about for now.

At the very top, select the document properties tab.

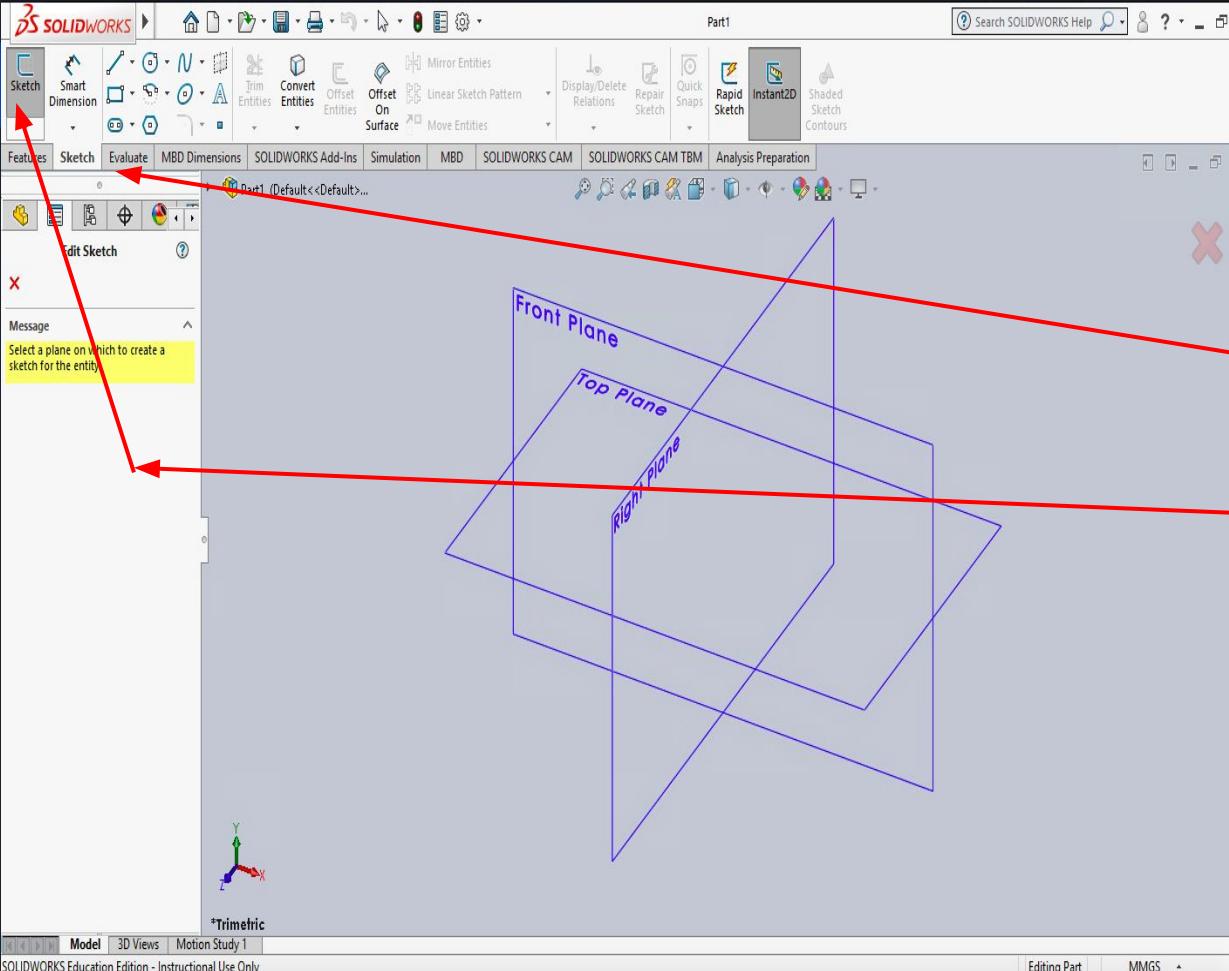


From this page, select
“UNITS”



This display allows you to control how your part is going to be described. We will be selecting the MMGS option. Press OK.

EXTRA: You will note that weight is included in these dimensions - solidworks modeling allows for more than just dimensions. It allows you to create the part from a given material, and will calculate the density, weight, and other properties.



After pressing OK, you will be brought back to the modeling window. Make sure the Sketch tab is selected.

Select “Sketch” from the command bar.

The following planes will be displayed. Click on one - I prefer to work on the top plane.

SOLIDWORKS Sketch1 of Part1 *

Search SOLIDWORKS Help

Exit Sketch Smart Dimension Trim Entities Convert Entities Offset Entities Mirror Entities Linear Sketch Pattern Display/Delete Relations Repair Sketch Quick Snaps Rapid Sketch Instant2D Shaded Sketch Contours

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Part1 (Default<<Default>>...)

View Orientation Changes the current view orientation or number of viewports.

Circle Type

Parameters

x: 0.00
y: 0.00
z: 0.00

*Top

After selecting a plane, you will be brought to a “head-on” view. If you wish to return to the 3rd angle view, press **ctrl+7** at any point.

Changes the current view orientation or number of viewports.

12.84mm 99.83mm 0mm Under Defined Editing Sketch1 MMGS



SOLIDWORKS

Sketch1 of Part1 *

Search SOLIDWORKS Help

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Circle

Circle Type

Existing Relations

Under Defined

Add Relations

Fix

Options

For construction

Parameters

88.28193858
51.37586113
39.37032614

*Top

Model 3D Views Motion Study 1

SOLIDWORKS Education Edition - Instructional Use Only

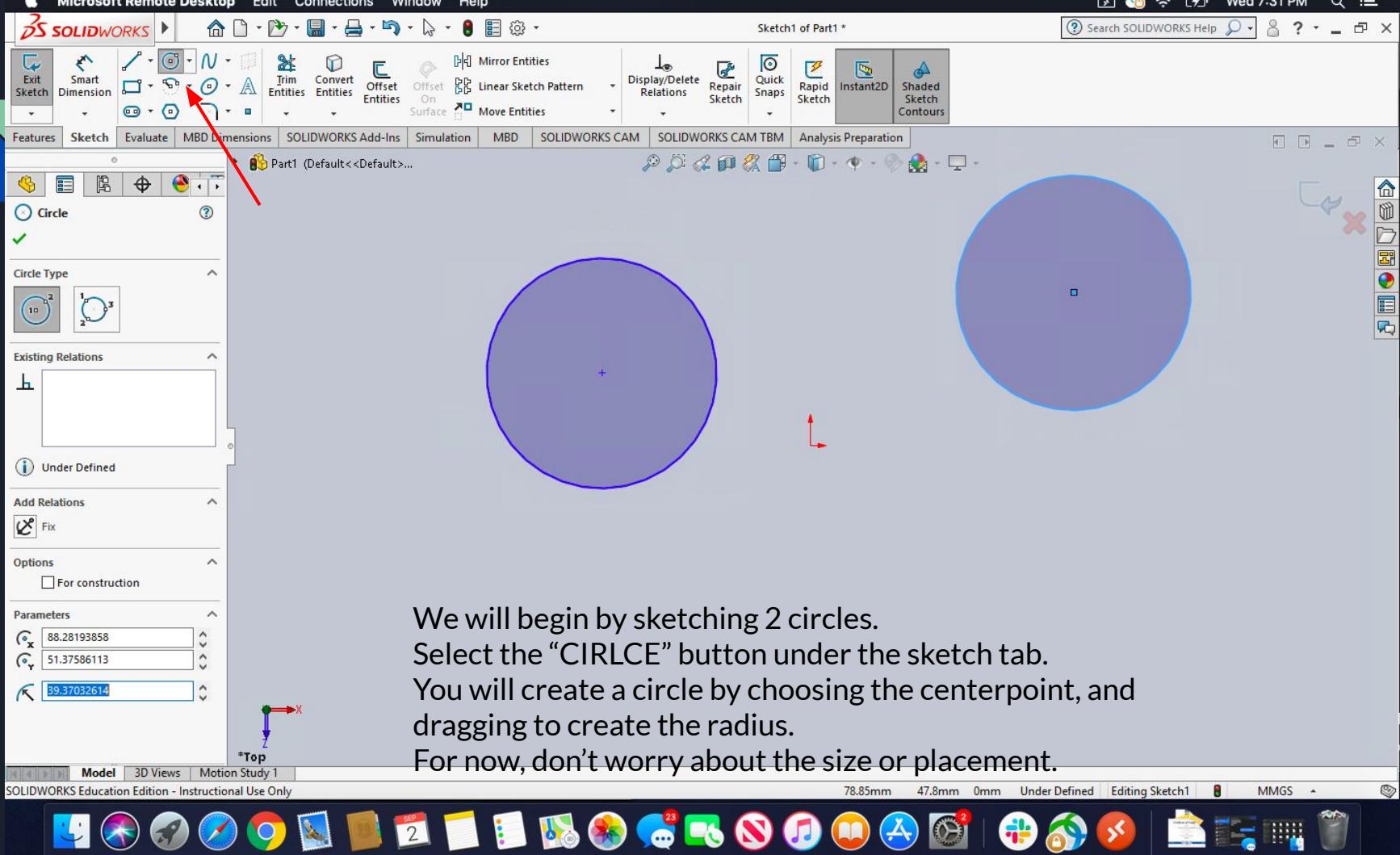
78.85mm 47.8mm 0mm Under Defined Editing Sketch1 MMGS

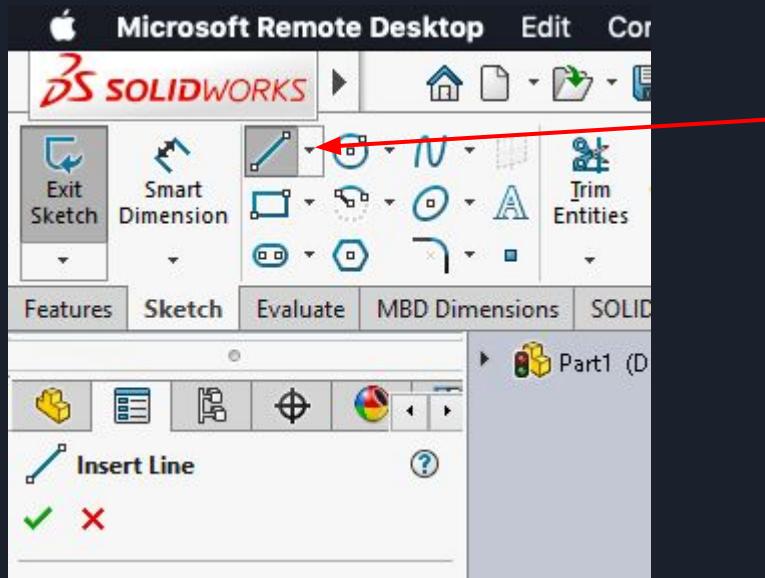
Smart Dimension Trim Entities Convert Entities Offset Entities Offset On Surface Linear Sketch Pattern Move Entities Mirror Entities Display/Delete Relations Repair Sketch Quick Snaps Rapid Sketch Instant2D Shaded Sketch Contours

Sketch1 of Part1 *

Part1 (Default<<Default>>...)

We will begin by sketching 2 circles.
Select the "CIRCLE" button under the sketch tab.
You will create a circle by choosing the centerpoint, and dragging to create the radius.
For now, don't worry about the size or placement.





Next we will be drawing lines.

Select the line feature from the command window.

Microsoft Remote Desktop Edit Connections Window Help Sketch1 of Part1 * ? Search SOLIDWORKS Help User ? X

Exit Sketch Smart Dimension Trim Entities Convert Entities Offset Entities Mirror Entities Offset On Surface Linear Sketch Pattern Move Entities Display/Delete Relations Repair Sketch Quick Snaps Rapid Sketch Instant2D Shaded Sketch Contours

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Line Properties Existing Relations Horizontal Under Defined Add Relations Horizontal Vertical Fix Options For construction Infinite length Parameters 421.918856 0.00° Additional Parameters

*Top

Part1 (Default<<Default>>...)

Create the lines from the following several slides.
Click to create the start point, click again to create an endpoint
At any time, you can press ESC to stop drawing the line.
Do not worry about dimensioning for now.

SOLIDWORKS Education Edition - Instructional Use Only 152.84mm 148.63mm 0mm Under Defined MMGS

The screenshot shows the SOLIDWORKS interface with a sketch in progress. Two circles are already drawn. A sketch toolbar is at the top, and a properties manager is open on the left. A callout box points to the 'Under Defined' status in the properties manager. A text overlay provides instructions for creating lines.

Microsoft Remote Desktop Edit Connections Window Help

SOLIDWORKS Sketch1 of Part1 *

Search SOLIDWORKS Help

Exit Sketch Smart Dimension Trim Entities Convert Entities Offset Entities Mirror Entities Linear Sketch Pattern Move Entities Display/Delete Relations Repair Sketch Quick Snaps Rapid Sketch Instant2D Shaded Sketch Contours

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Line Properties

Message: Edit the settings of the current line, sketch a new line, or select OK to change the settings for the next new line.

Existing Relations

Under Defined

Add Relations: Horizontal, Vertical, Fix

Options: For construction, Infinite length

Parameters: 200.79306348, 85.93663401°

*Top

Sketch1 of Part1 *

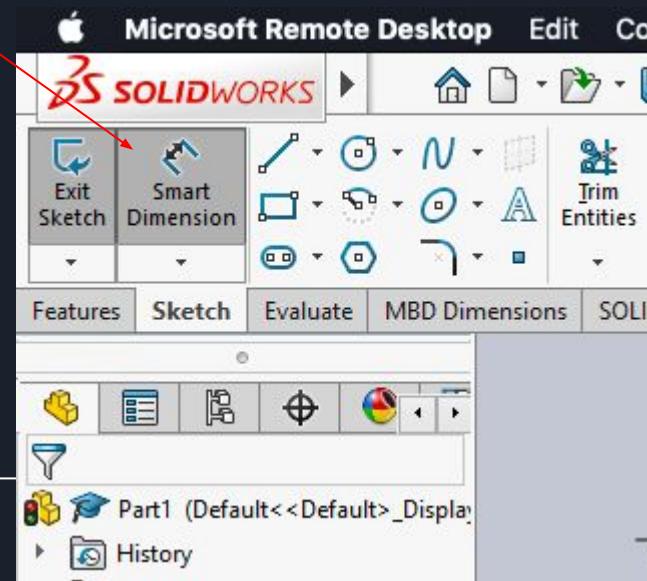
Part1 (Default<<Default>>...)

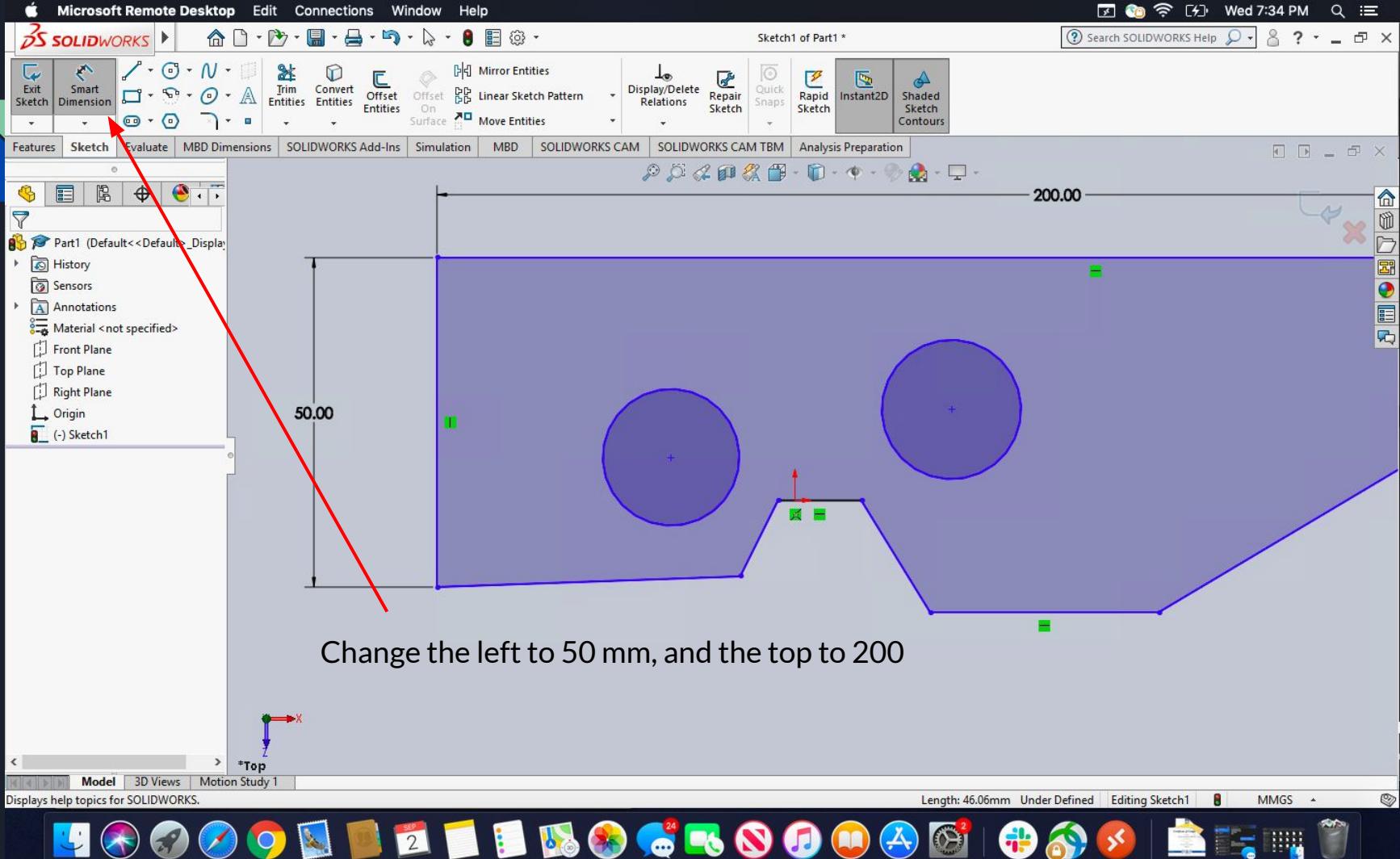
Create something similar to the following.

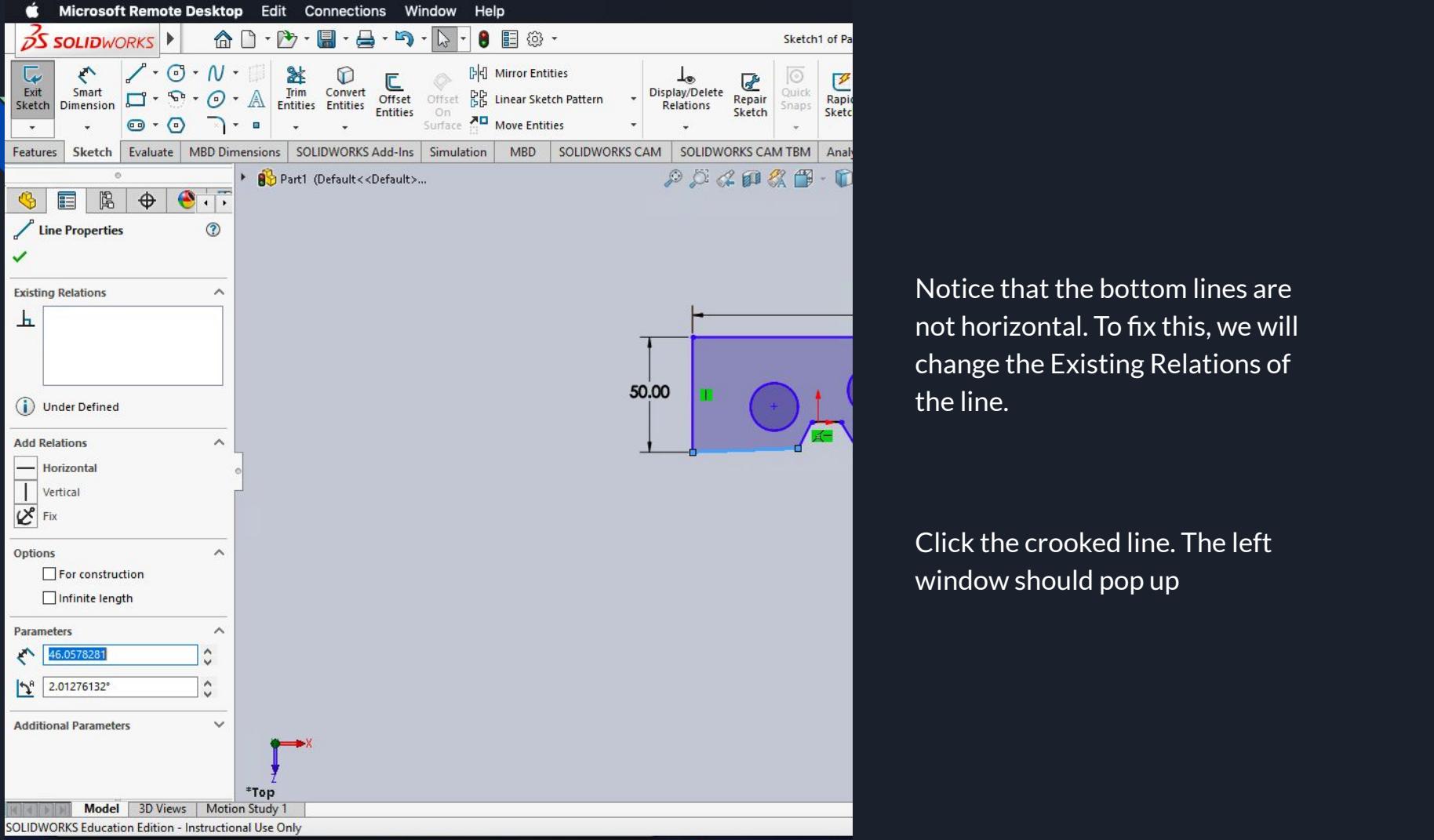
SOLIDWORKS Education Edition - Instructional Use Only

116.72mm 141.51mm 0mm Under Defined MMGS

The most common ways to dimension a part are through Smart Dimension and Relations.
Click the smart dimension tab

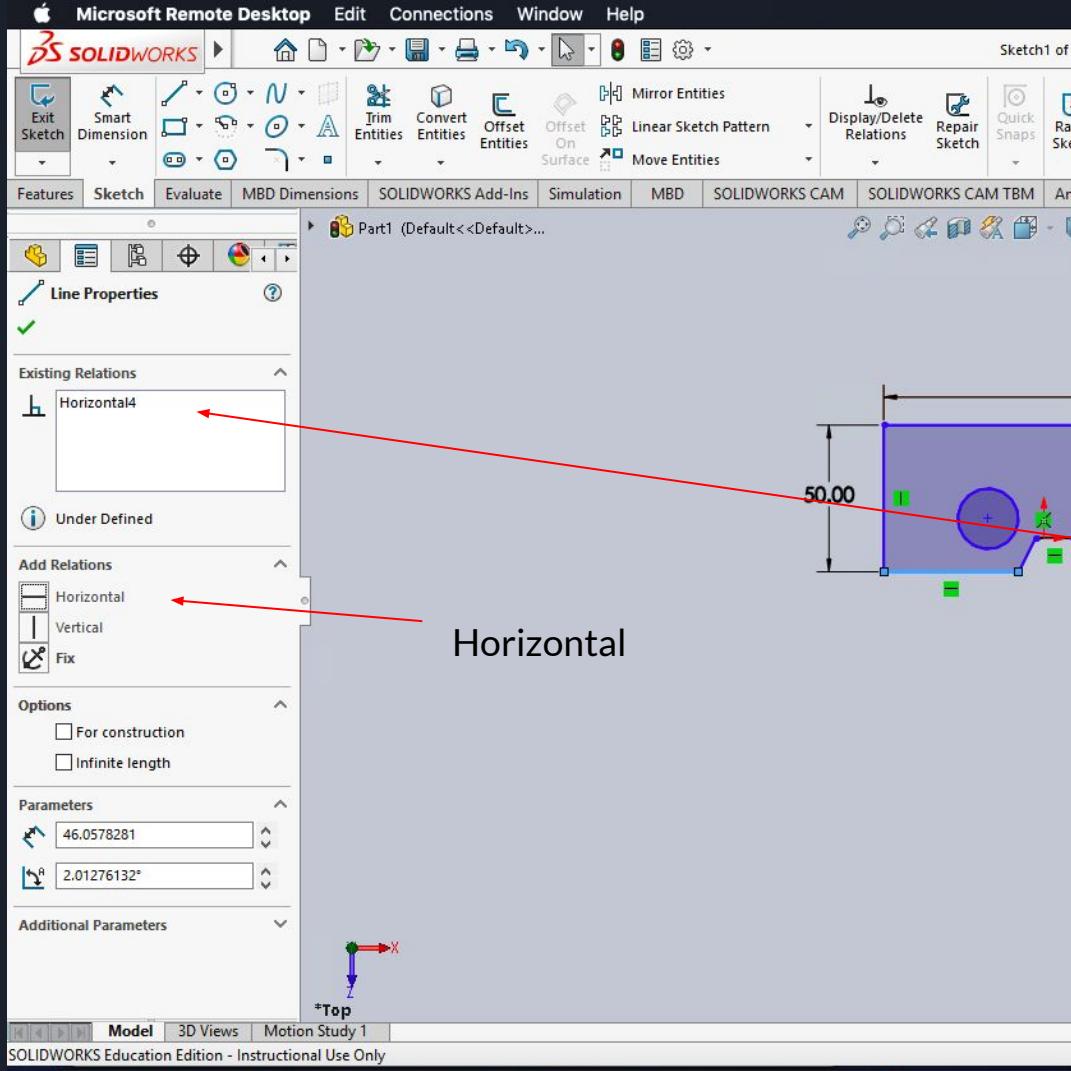






Notice that the bottom lines are not horizontal. To fix this, we will change the Existing Relations of the line.

Click the crooked line. The left window should pop up



Select the horizontal relation.

Once this has been selected, it should appear in the top left of the side window

Microsoft Remote Desktop Edit Connections Window Help

SOLIDWORKS Sketch1 of Part1 *

Search SOLIDWORKS Help

Exit Sketch Smart Dimension Trim Entities Convert Entities Offset Entities Mirror Entities Linear Sketch Pattern Move Entities Display/Delete Relations Repair Sketch Quick Snaps Instant2D Shaded Sketch Contours

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Line Properties

Existing Relations Vertical Under Defined

Add Relations Horizontal Vertical Fix

Options For construction Infinite length

Parameters 106.05749028 31.51665886°

Additional Parameters

*Top

Part1 (Default<<Default>>...)

200.00 50.00

Similarly, set the bottom right line and the middle line to horizontal.

Model 3D Views Motion Study 1

SOLIDWORKS Education Edition - Instructional Use Only

Length: 106.06mm Under Defined Editing Sketch1 MMGS

25

Microsoft Remote Desktop Edit Connections Window Help

Sketch1 of Part1 *

Search SOLIDWORKS Help

Exit Sketch Smart Dimension Trim Entities Convert Entities Offset Entities Mirror Entities Linear Sketch Pattern Offset On Surface Move Entities Display/Delete Relations Repair Sketch Quick Snaps Instant2D Shaded Sketch Contours

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Part1 (Default<<Default>>...)

Dimension

Value Leaders Other

Style <NONE>

Tolerance/Precision None

.12 (Document)

Primary Value D4@Sketch1 125.00mm

Dimension Text <DIM>

Smart Dimension

200.00 125.00 25.00 75.00

*Top

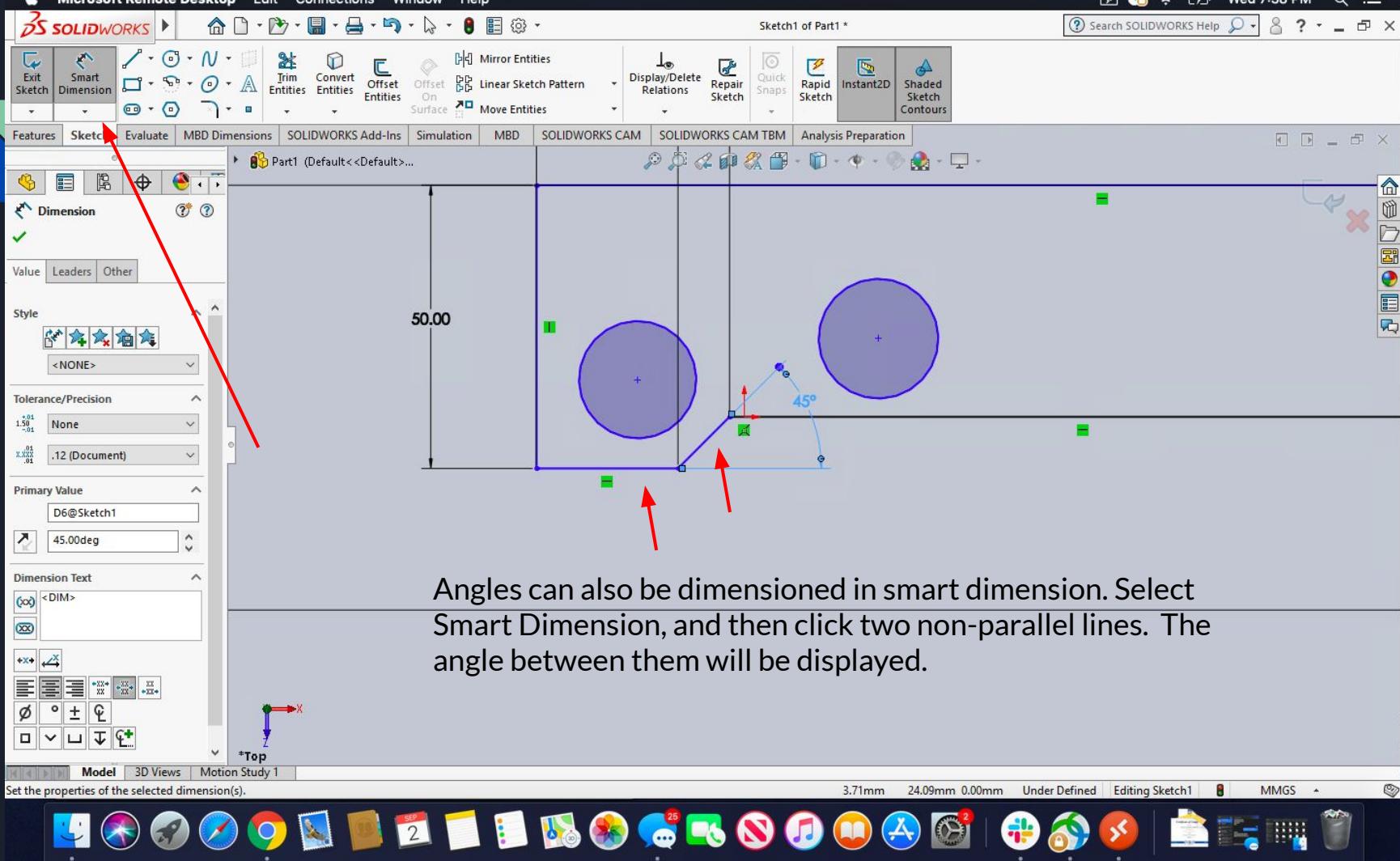
Model 3D Views Motion Study 1

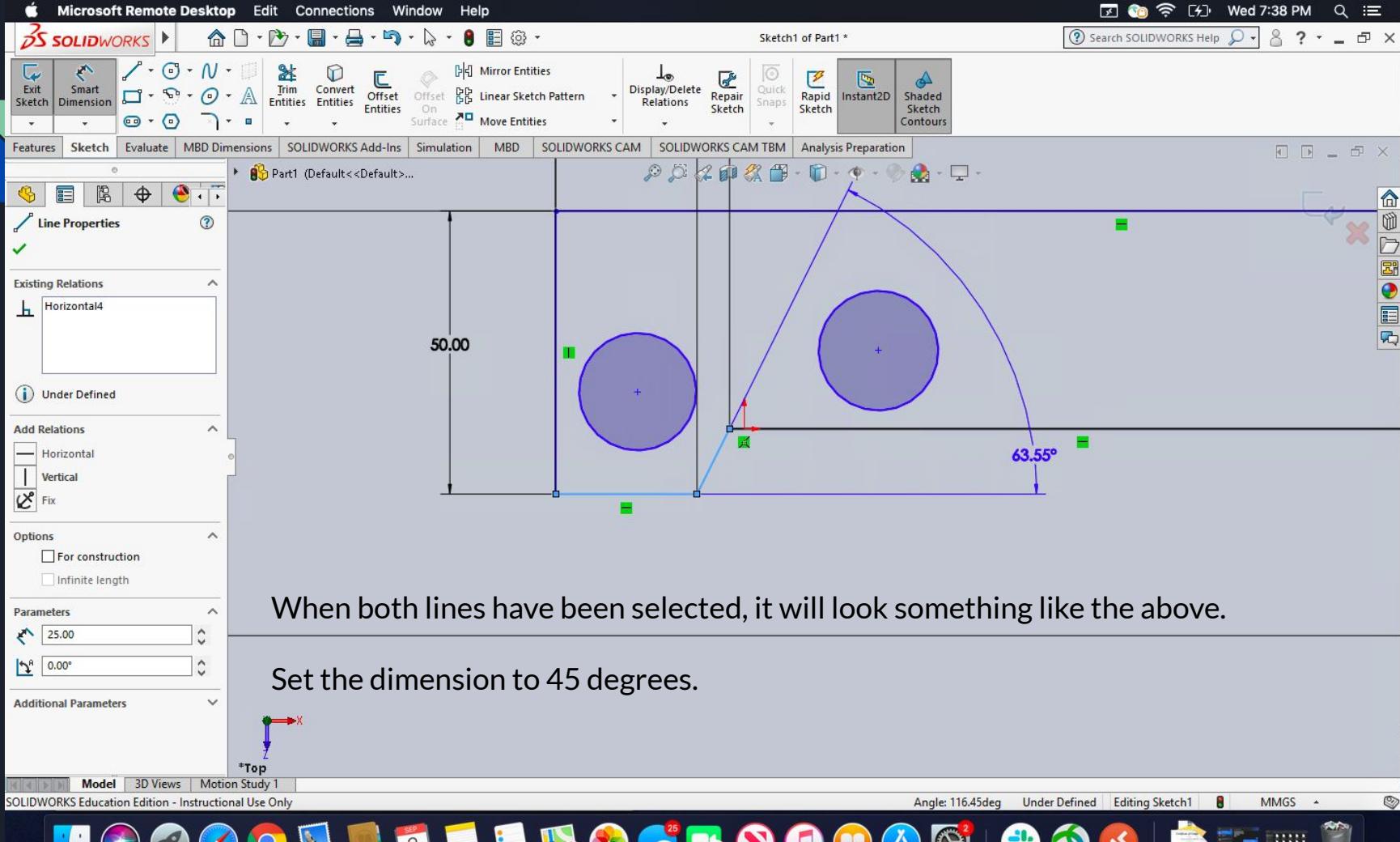
Set the properties of the selected dimension(s).

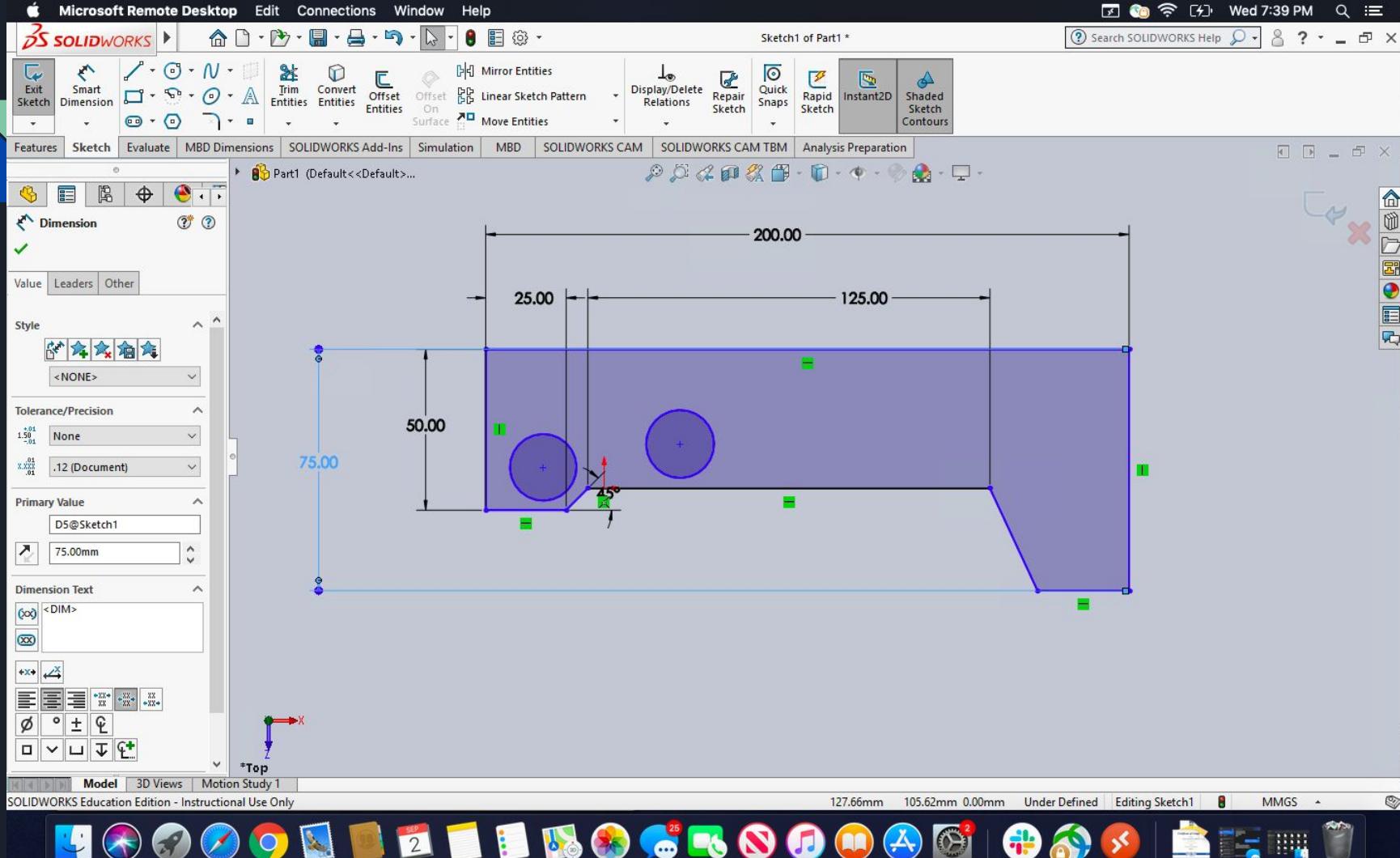
130.23mm 111.96mm 0.00mm Under Defined Editing Sketch1 MMGS

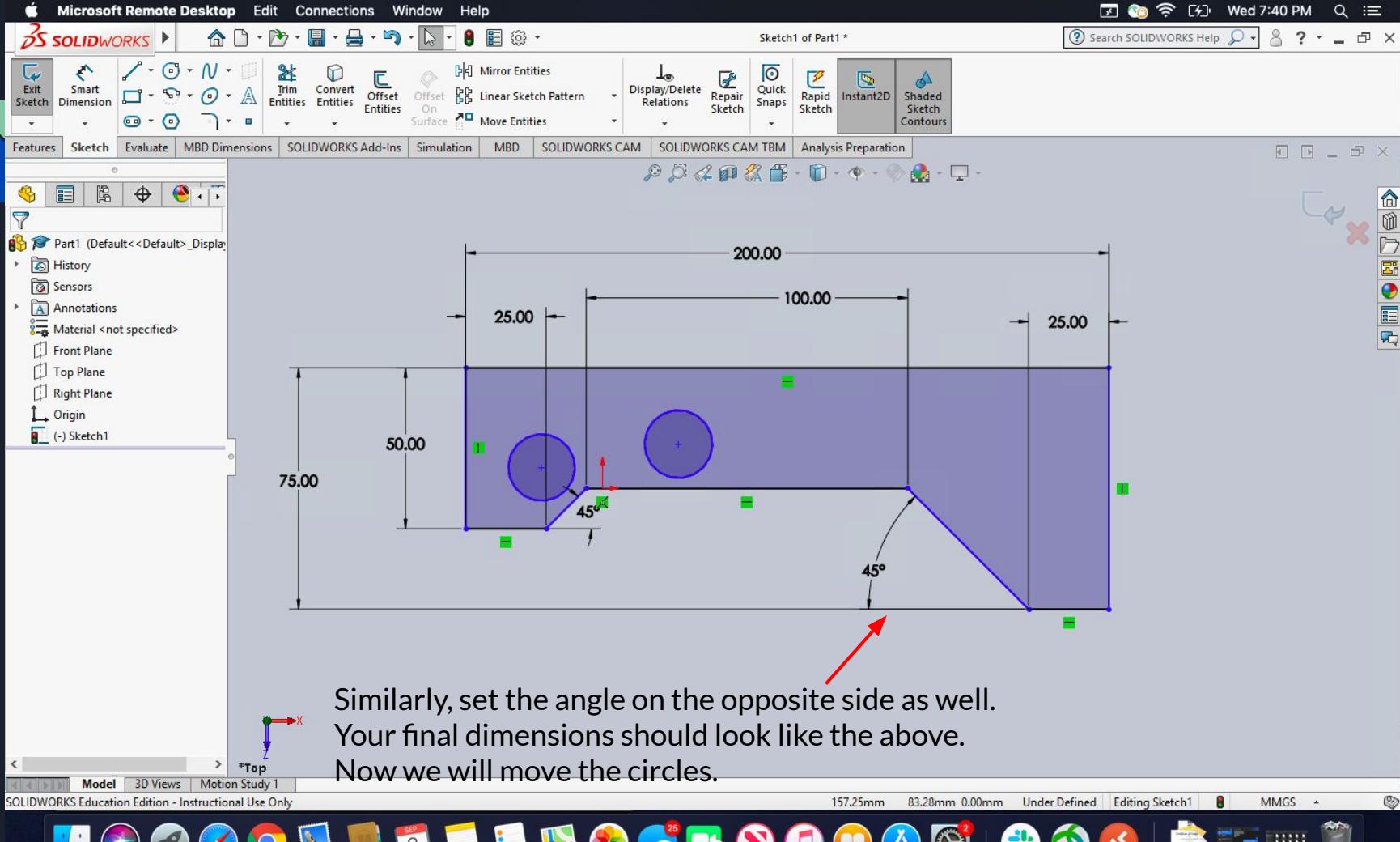
Using smart dimension, we will now add the above dimensions.
Don't worry if your sketch does not look exactly the same.
We will correct the right overlap in the following slides.

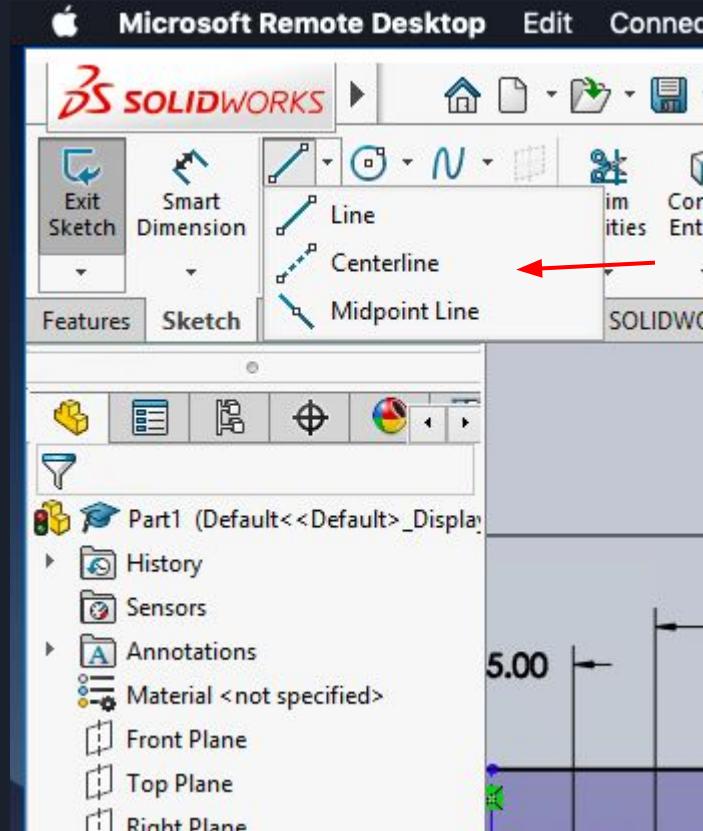






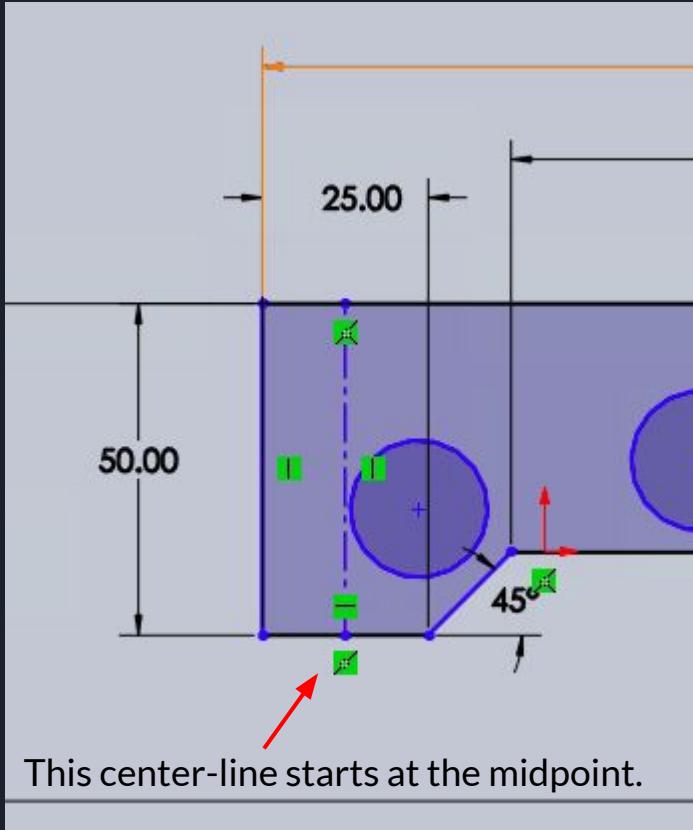






From the command bar, click the dropdown menu on LINE.

Select CENTERLINE

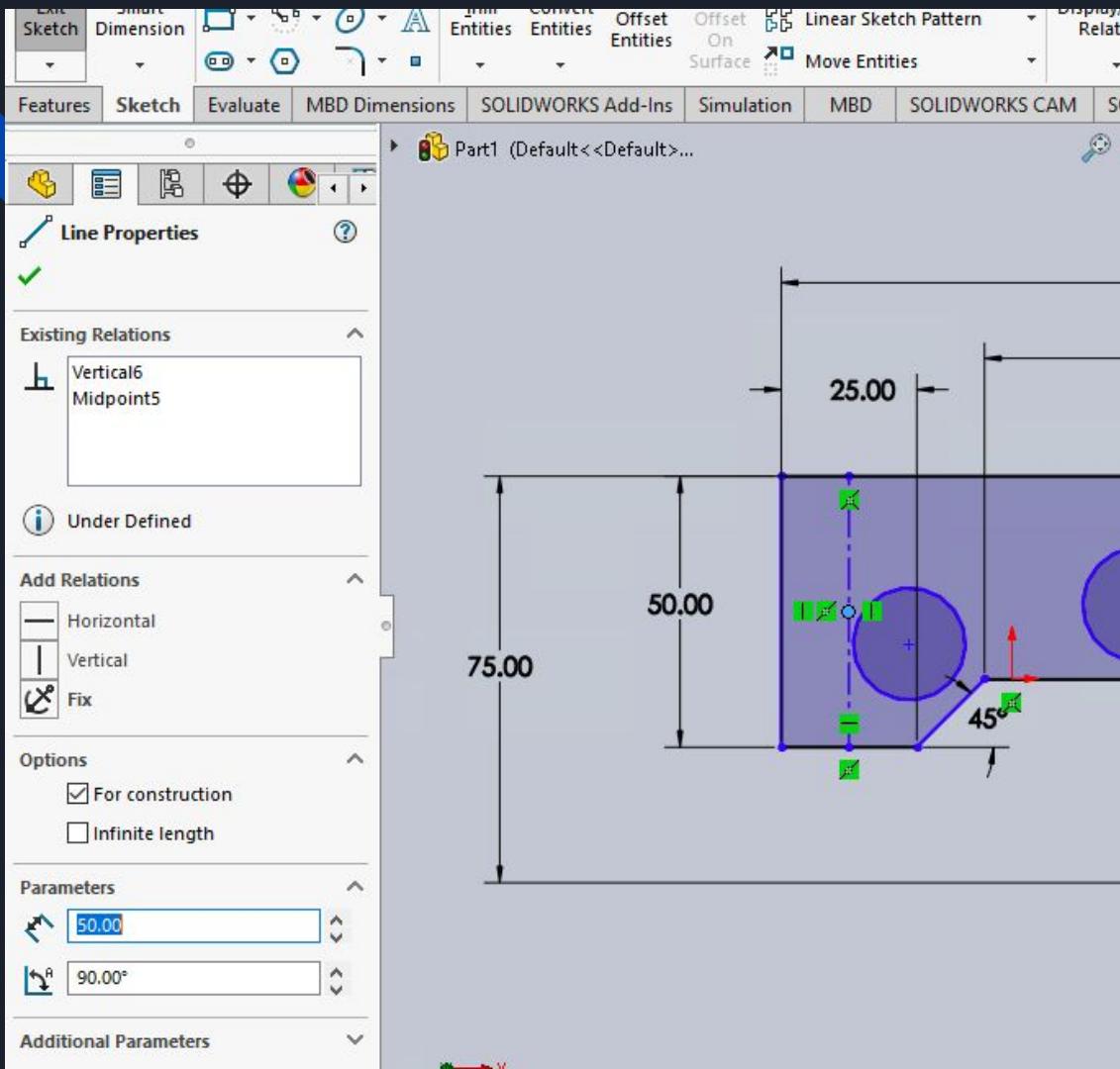


Hover over the bottom right line. An orange dot should appear at the center of this line.

Click that dot and then create a line to the top of our sketch.

The final result should look like this.

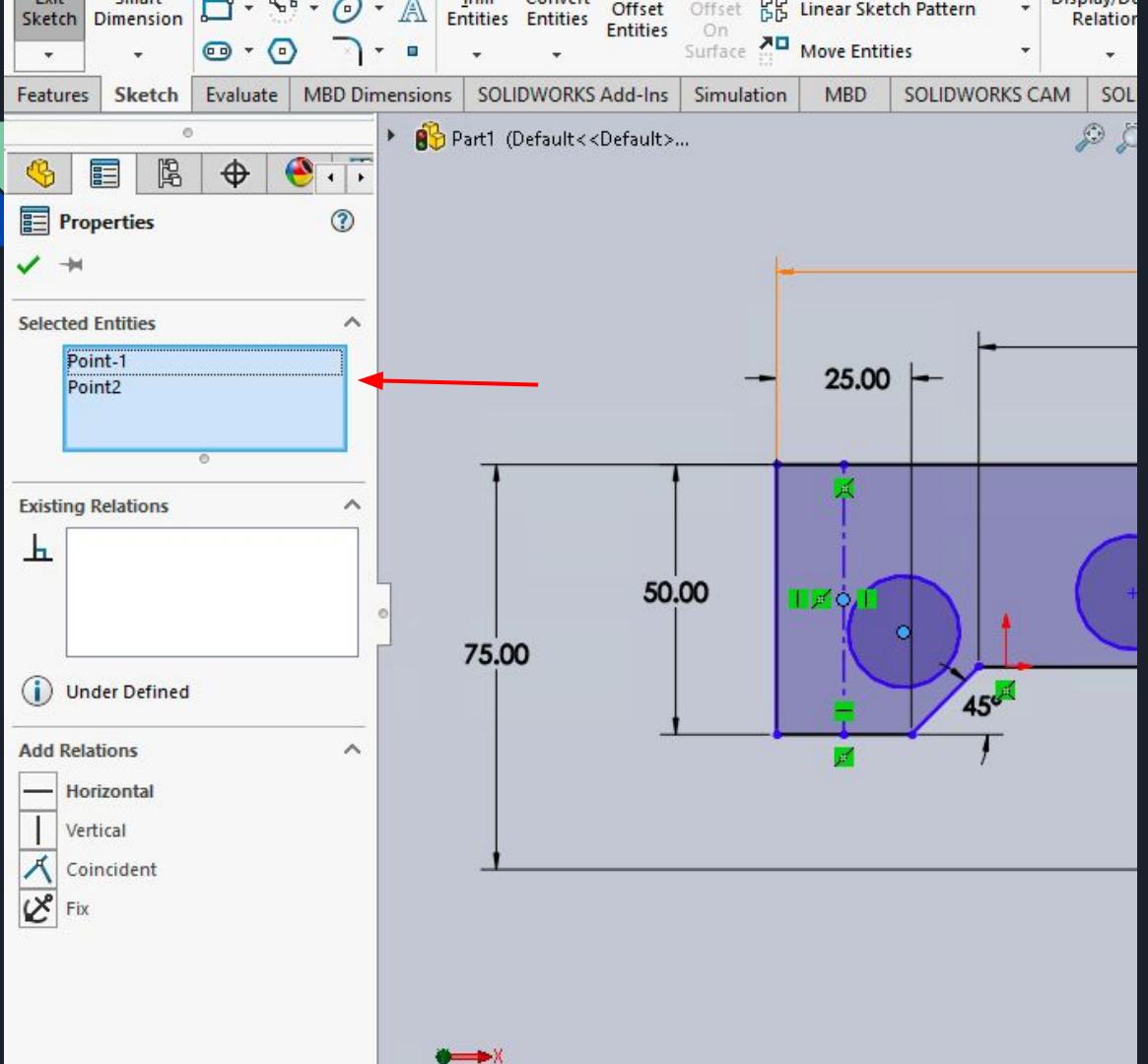
The centerline will be dotted - it is for construction.



Hover over this center line until a blue dot appears at the midpoint of our centerline.

Click this blue dot, and the left window will be opened.

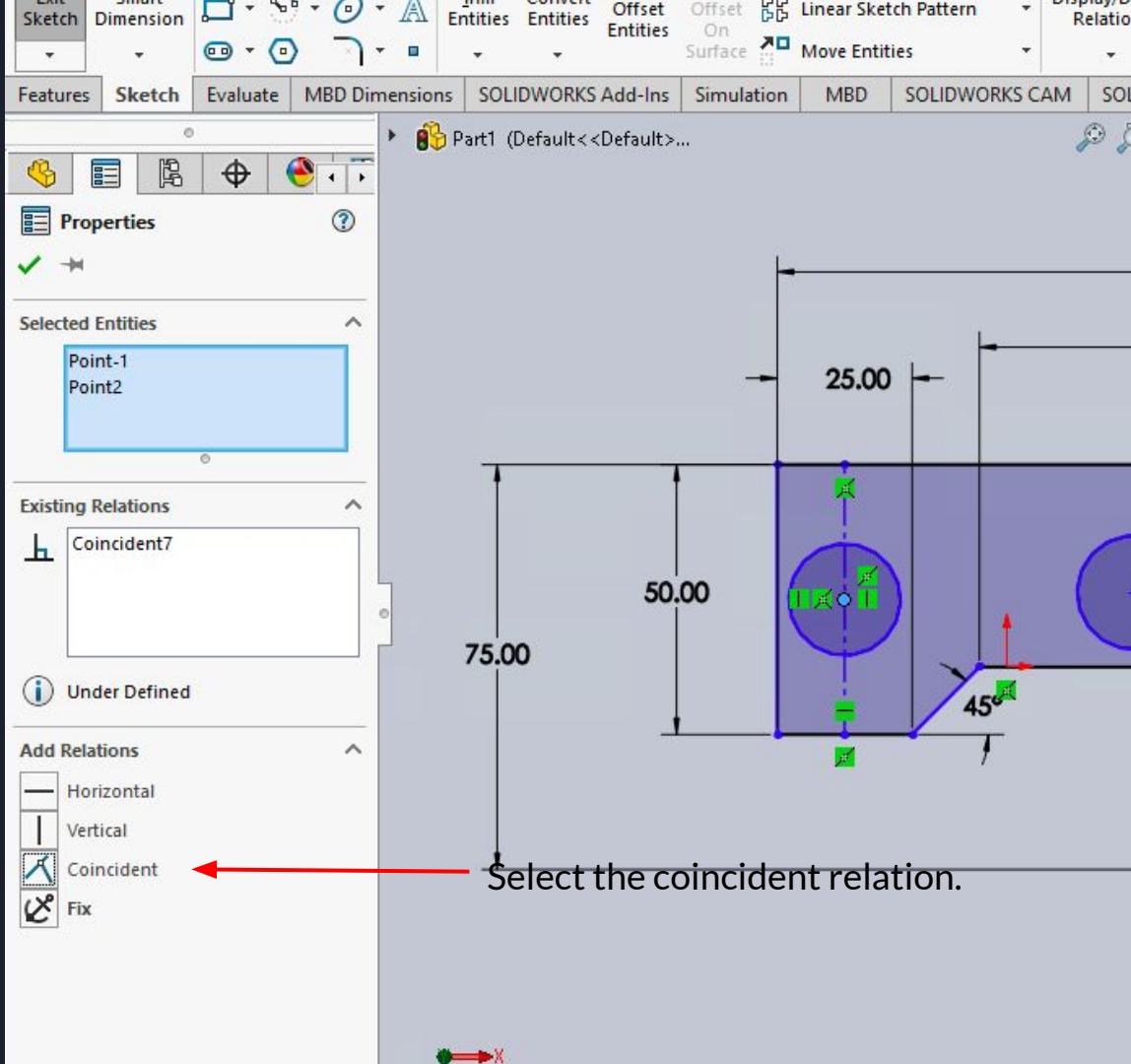
EXTRA: Notice that under existing relations, the point you have selected is a midpoint. This is another relation in addition to horizontal and vertical.



While the midpoint is still selected, CTRL+Click the vertex of the circle.

The selected entities box will pop up in the properties table on the left.

There should be two points in this blue box.



The Microsoft Remote Desktop window showing the SOLIDWORKS interface.

SolidWorks ribbon tabs: Microsoft Remote Desktop, Edit, Connections, Window, Help, Sketch1 of Part1 *, Search SOLIDWORKS Help, User Account.

Toolbar icons: Exit Sketch, Smart Dimension, Trim Entities, Convert Entities, Offset Entities, Mirror Entities, Linear Sketch Pattern, Display/Delete Relations, Repair Sketch, Quick Snaps, Rapid Sketch, Instant2D, Shaded Sketch Contours.

FeatureManager: Features, Sketch, Evaluate, MBD Dimensions, SOLIDWORKS Add-Ins, Simulation, MBD, SOLIDWORKS CAM, SOLIDWORKS CAM TBM, Analysis Preparation.

Properties panel: Selected Entities (Point-1, Point4), Existing Relations (Coincident11), Under Defined, Add Relations (Horizontal, Vertical, Coincident, Fix).

Sketch1 of Part1 *: A 2D sketch with dimensions 200.00, 100.00, 25.00, 50.00, 75.00. It features two circles, one at [50, 50] and another at [150, 50], both with a radius of 25.00. A 45° angle is shown between the vertical line and the bottom line.

Bottom status bar: Model, 3D Views, Motion Study 1, SOLIDWORKS Education Edition - Instructional Use Only, Distance: 122.1mm dX: -121.32mm dY: 0mm dZ: -13.81mm Under Defined Editing Sketch1 MMGS, system icons.

The circle should now be centered over the base like in the above figure, both horizontally and vertically.
Please repeat this process for the circle on the right.

SOLIDWORKS

Sketch1 of Part1*

Search SOLIDWORKS Help

Exit Sketch Smart Dimension Trim Entities Convert Entities Offset Entities Mirror Entities Linear Sketch Pattern Move Entities Display/Delete Relations Repair Sketch Quick Snaps Instant2D Shaded Sketch Contours

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Part1 (Default<<Default>>...)

Properties

Selected Entities Arc1 Arc2

Existing Relations

Under Defined

Add Relations

- Coradial
- Tangent
- Concentric
- Equal
- Fix
- Equal Curve Length

Options

For construction

*Top

200.00

25.00

100.00

25.00

50.00

75.00

45°

45°

Ø20.74

We will now make the two circles the same radius.
Using smart dimension, set the circle diameter and set it to 20.74.
Then, click and CTRL+click the two circle diameters.
When selected, both circles should glow and the properties tab will show

Distance between circle centers: 175.45mm Under Defined Editing Sketch1 MMGS

Model 3D Views Motion Study 1

SOLIDWORKS Education Edition - Instructional Use Only

Distance between circle centers: 175.45mm Under Defined Editing Sketch1 MMGS

Microsoft Remote Desktop Edit Connections Window Help Sketch1 of Part1 * ? Search SOLIDWORKS Help X

Exit Sketch Smart Dimension Trim Entities Convert Entities Offset Entities Mirror Entities Linear Sketch Pattern Offset On Surface Move Entities Display/Delete Relations Repair Sketch Quick Snaps Instant2D Shaded Sketch Contours

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Part1 (Default<<Default>>...)

Properties

Selected Entities Arc1 Arc2

Existing Relations Equal radius12

Under Defined

Add Relations Coradial Tangent Concentric Equal Fix Equal Curve Length

Options For construction

*Top

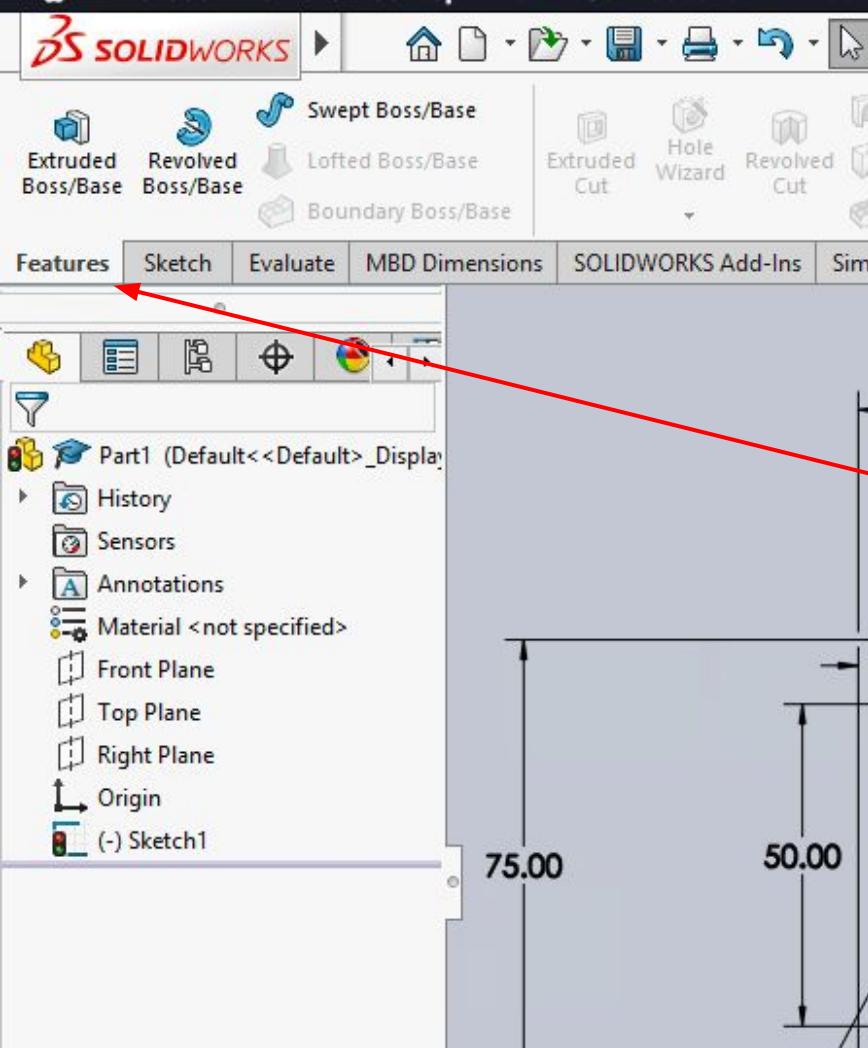
Sketch dimensions:

- Horizontal top edge: 200.00
- Vertical left edge: 75.00
- Vertical right edge: 25.00
- Vertical center line: 50.00
- Bottom horizontal line: 100.00
- Bottom-left circle: Ø 20.74
- Bottom-right circle: Ø 20.74
- Angle between bottom-left circle and bottom-right circle: 45°

A red arrow points from the text "Select the EQUAL Relation." to the "Equal" option in the "Add Relations" list.

Select the EQUAL Relation.
As before, the equal property should show up in the Existing Relations Tab.

SOLIDWORKS Education Edition - Instructional Use Only Distance between circle centers: 175.45mm Under Defined Editing Sketch1 MMGS



We have now finished our sketch.

Below the command bar, select the FEATURES tab.

Then, click Extruded Boss/Base

SOLIDWORKS Sketch1 of Part1*

Search SOLIDWORKS Help

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Boss-Extrude

From Sketch Plane

Direction 1

- Blind
- 10.00mm

Draft outward

Direction 2

Thin Feature

Selected Contours

*Trimetric

This function will allow us to make our sketch 3D.
Make sure that the circles are left out of the extrusion.

Select a handle to modify parameters

90.19mm 246.04mm 0.00mm Under Defined

MMGS



SOLIDWORKS Sketch1 of Part1*

Search SOLIDWORKS Help

Instant3D

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Part1 (Default<<Default>>...)

Boss-Extrude

From Sketch Plane

Direction 1

- Blind
- 30.00mm

Draft outward

Direction 2

Thin Feature

Selected Contours

75.00 50.00 25.00 200.00 100.00 25.00

30.00mm

45°

20.74

Extrude the sketch 30mm.

*Trimetric

Model 3D Views Motion Study 1

Applies a specific scene to your model.

-28.11mm 281.02mm 0.00mm Under Defined Editing Sketch1

MMGS



Microsoft Remote Desktop Edit Connections Window Help

Part1 *

SOLIDWORKS Search SOLIDWORKS Help ? X

Extruded Boss/Base Revolved Boss/Base Lofted Boss/Base Extruded Cut Hole Wizard Revolved Cut Lofted Cut Boundary Cut Fillet Linear Pattern Draft Intersect Shell Mirror Instant3D Reference Geometry Curves

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Home Insert Tools References Assembly Display

Part1 (Default<<Default>>_Display)

History Sensors Annotations Solid Bodies(1) Material <not specified>

Front Plane Top Plane Right Plane Origin

Boss-Extrude1

*Trimetric

Model 3D Views Motion Study 1

SOLIDWORKS Education Edition - Instructional Use Only MMGS

Having pressed enter, the part will now look like the above.

The screenshot shows a SOLIDWORKS 3D model of a mechanical part. The part is a rectangular block with two circular holes and a stepped base. It is displayed in a trimetric view. The left side of the screen shows the FeatureManager design tree, which includes the part name 'Part1', history, sensors, annotations, solid bodies (1), material, and a selected feature 'Boss-Extrude1'. At the bottom left, there is a 3D coordinate system. The bottom of the screen displays the SOLIDWORKS ribbon tabs (Model, 3D Views, Motion Study 1) and the status bar (SOLIDWORKS Education Edition - Instructional Use Only, MMGS). The top right corner shows the system tray with icons for signal strength, battery, and date/time (Wed 7:46 PM).

Microsoft Remote Desktop Edit Connections Window Help

Wed 7:47 PM

DS SOLIDWORKS Search SOLIDWORKS Help

Swept Boss/Base
Extruded Boss/Base Revolved Boss/Base Lofted Boss/Base Extruded Cut Hole Wizard Revolved Cut Lofted Cut Boundary Cut
Swept Cut Fillet Linear Pattern Rib Wrap
Boundary Geometry Curves Instant3D
Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Part1 *

Extruded Cut Hole Wizard Revolved Cut Lofted Cut Boundary Cut
Fillet Linear Pattern Rib Wrap
Boundary Shell Mirror

Instant3D

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Home Insert Tools Evaluate View Window Help

Part1 (Default<<Default>>_Display)
History
Sensors
Annotations
Solid Bodies(1)
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
Boss-Extrude1

Front Plane Top Plane Right Plane Origin Boss-Extrude1

Rotate so that you are looking at the part from the rear.
Pressing CTRL+(1 through 6) will take you to look at each face.
For me, this was ctrl+4. However, it depends which plane you started on.

Model 3D Views Motion Study 1

SOLIDWORKS Education Edition - Instructional Use Only

MMGS

Back

The image shows a screenshot of the SOLIDWORKS 2014 software interface on a Mac OS X desktop. The window title is 'DS SOLIDWORKS'. The menu bar includes 'Edit', 'Connections', 'Window', 'Help', and system status like 'Wed 7:47 PM'. The toolbar has icons for various features like Swept Boss/Base, Extruded Cut, Hole Wizard, etc. The ribbon tabs include 'Features', 'Sketch', 'Evaluate', 'MBD Dimensions', 'SOLIDWORKS Add-Ins', 'Simulation', 'MBD', 'SOLIDWORKS CAM', 'SOLIDWORKS CAM TBM', and 'Analysis Preparation'. The left pane is the FeatureManager, showing the part name 'Part1' and its history: History, Sensors, Annotations, Solid Bodies(1), Material <not specified>, Front Plane, Top Plane, Right Plane, and Origin. The main workspace shows a 3D model of a rectangular block with a circular hole. A callout arrow points to the 'Back' view orientation button on the bottom toolbar. The bottom taskbar shows other applications like Finder, Mail, and Safari. The status bar at the bottom says 'SOLIDWORKS Education Edition - Instructional Use Only'.

SOLIDWORKS

Sketch Smart Dimension Trim Entities Convert Entities Offset Entities Offset On Surface Mirror Entities Linear Sketch Pattern Move Entities Display/Delete Relations Repair Sketch Quick Snaps Instant2D Shaded Sketch Contours

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Part1 *

Search SOLIDWORKS Help

Edit Sketch

X

Message

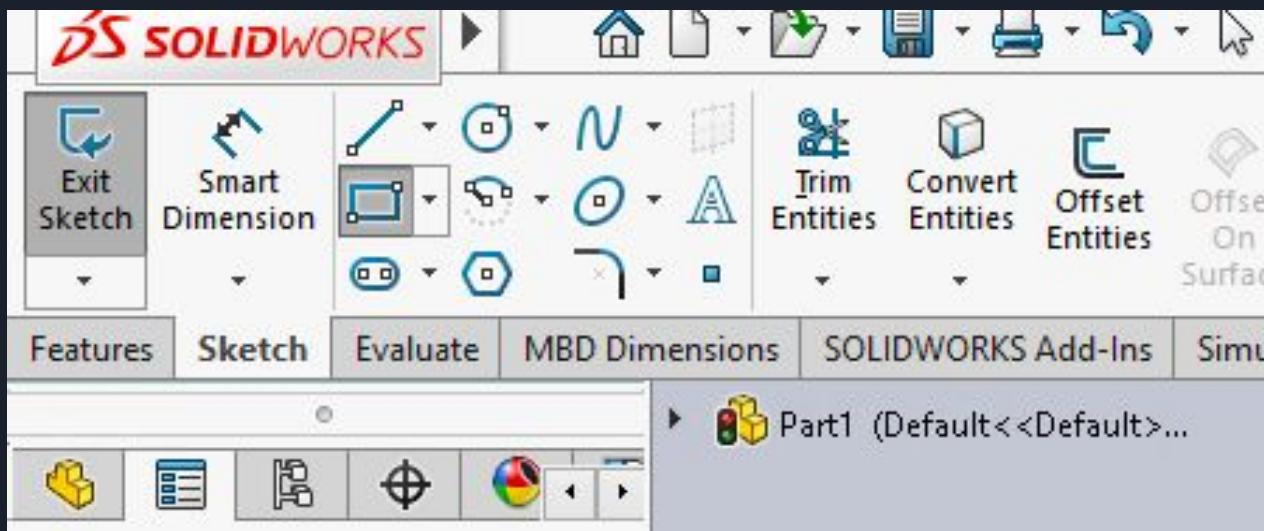
Select:
1) a plane, a planar face, or an edge on which to create a sketch for the entity
or
2) an existing sketch to add the entity to that sketch.

*Back

Model 3D Views Motion Study 1

SOLIDWORKS Education Edition - Instructional Use Only





Microsoft Remote Desktop Edit Connections Window Help

SOLIDWORKS Sketch2 of Part1 *

Search SOLIDWORKS Help

Exit Sketch Smart Dimension Trim Entities Convert Entities Offset Entities Mirror Entities Linear Sketch Pattern Move Entities Display/Delete Relations Repair Sketch Quick Snaps Rapid Sketch Instant2D Shaded Sketch Contours

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Part1 (Default<<Default>>...)

Rectangle Type

Add construction lines

Existing Relations

Static

Horizontal

Vertical

Fix

For construction

Parameters

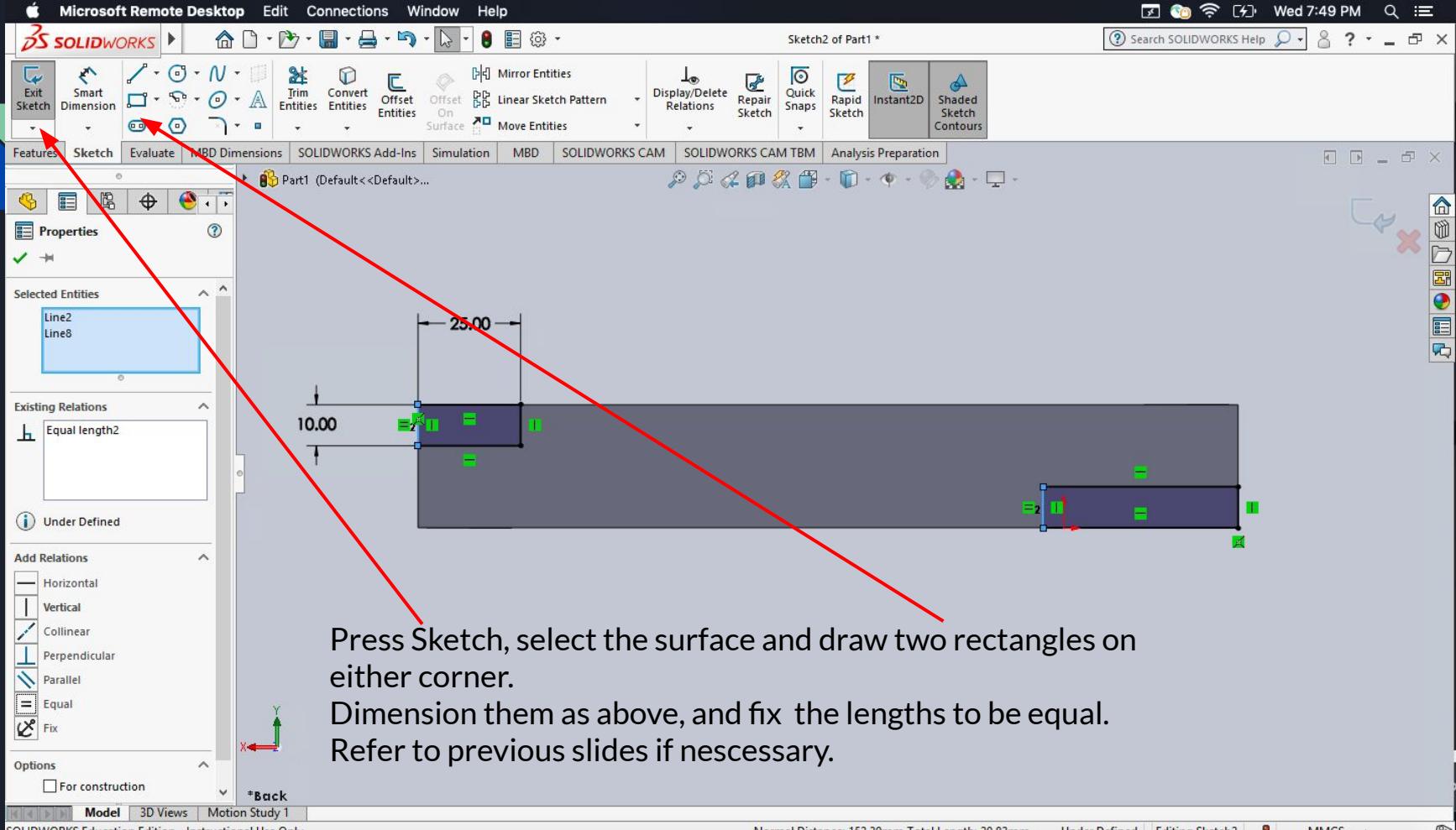
Model 3D Views Motion Study 1

SOLIDWORKS Education Edition - Instructional Use Only

25.32mm 82.34mm 0.00mm Under Defined Editing Sketch2 MMGS

Mac OS X Dock Icons

The screenshot shows the SOLIDWORKS interface with a sketch of a rectangular part. The sketch consists of two horizontal lines and two vertical lines forming a rectangle. There are several green dimension lines and green constraint points indicating relationships between the vertices and midpoints of the lines. A blue dimension line indicates a width of 82.34mm. A red dimension line indicates a height of 25.32mm. A green dimension line indicates a thickness of 0.00mm. The sketch is labeled "Sketch2 of Part1 *". The "Sketch" tab is selected in the ribbon. The left panel displays the "Rectangle" tool with options for "Add construction lines" and "Existing Relations". The "Parameters" panel shows a parameter named "47 45403407". The bottom status bar indicates "Under Defined" and "Editing Sketch2". The Mac OS X dock icons are visible at the bottom.



Microsoft Remote Desktop Edit Connections Window Help Sketch2 of Part1 * ? Search SOLIDWORKS Help User X

Exit Sketch Smart Dimension Trim Entities Convert Entities Offset Entities Mirror Entities Offset On Surface Linear Sketch Pattern Move Entities Display/Delete Relations Repair Sketch Quick Snaps Rapid Sketch Instant2D Shaded Sketch Contours

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Part1 (Default<<Default>>...)

Properties Selected Entities Line1 Line5 Existing Relations Equal length3 Fully Defined Add Relations Horizontal Vertical Collinear Perpendicular Parallel Equal Fix Options For construction *Back

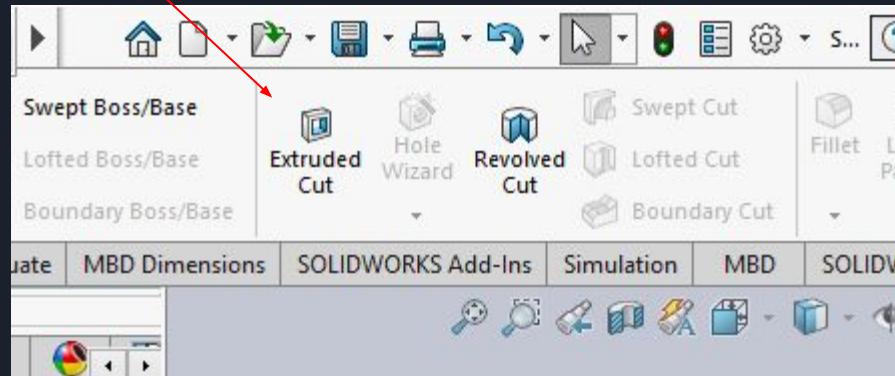
Normal Distance: 30mm Total Length: 72.61mm Fully Defined Editing Sketch2 MMGS

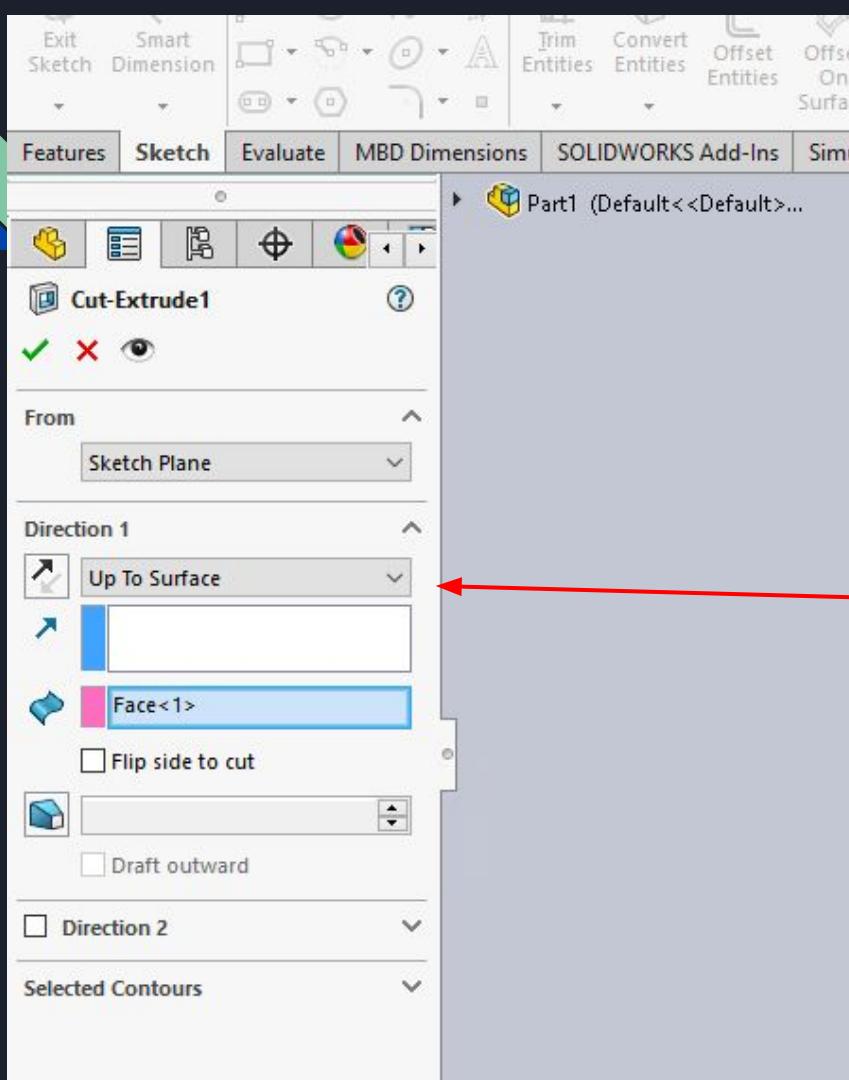
Model 3D Views Motion Study 1 SOLIDWORKS Education Edition - Instructional Use Only

Once the rectangles are identical, the face should look like the above.

Under the command window, go back to the features tab. We will now select

EXTRUDED CUT





The left window should pop up.

In the Direction 1 section, choose the dropdown window next to the arrow.

Select UP TO SURFACES

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SOLIDWORKS Sketch2 of Part1 *

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Cut-Extrude1

From Sketch Plane

Direction 1

- Up To Surface
- Face<1>

Flip side to cut

Draft outward

Direction 2

Selected Contours

25.00 10.00

*Front

Without clicking on the part, turn it over (Ctrl+3 for me). Select the lower face. Press enter.

The part should now look like this.
We will now fillet and chamfer the part

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Sketch Smart Dimension Trim Entities Convert Entities Offset Entities Offset On Surface Mirror Entities Linear Sketch Pattern Move Entities Display/Delete Relations Repair Sketch Quick Snaps Instant2D Shaded Sketch Contours

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Part1 *

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Model 3D Views Motion Study 1

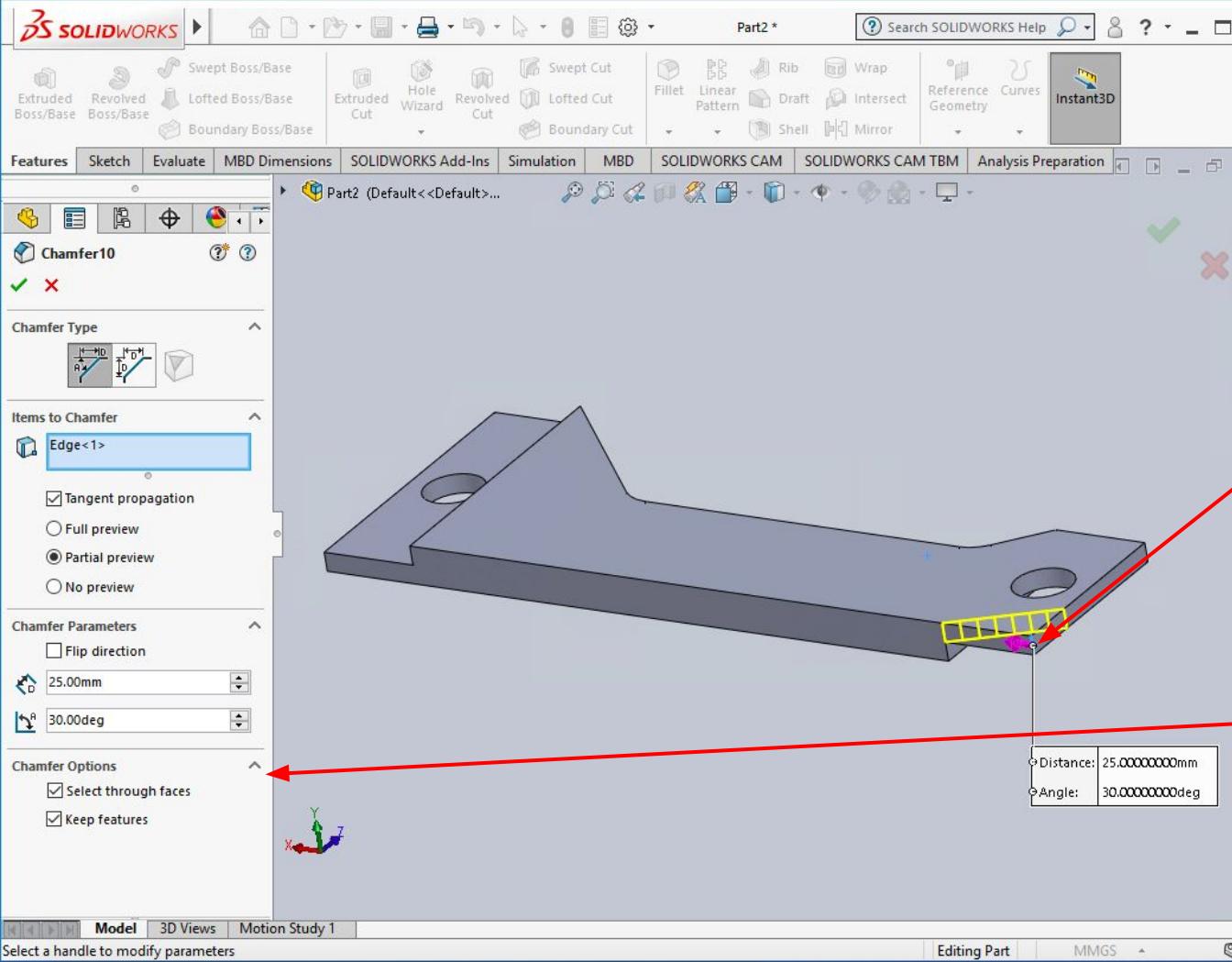
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MMGS



In the features tab, select the dropdown window under Filler.

Select Chamfer.



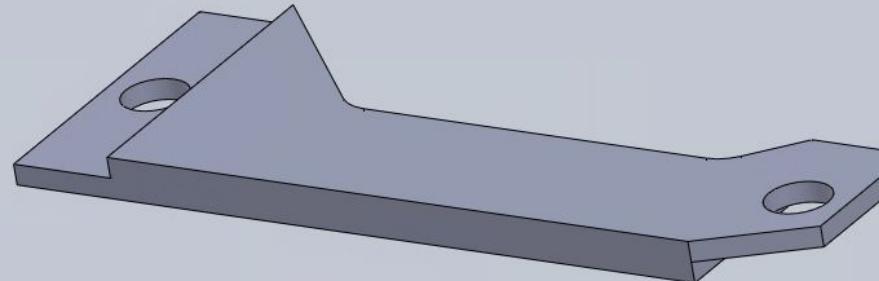
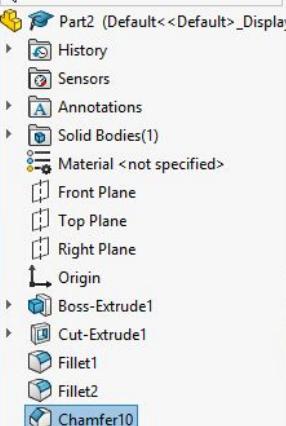
Once the chamfer window has popped up, select only the closer corner, as shown.

Change the dimension in CHAMFER PROPERTIES to 25mm, and the angle to 30 degrees.

You should have the following preview.



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Once chamfered,
the part should
look like this.



Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

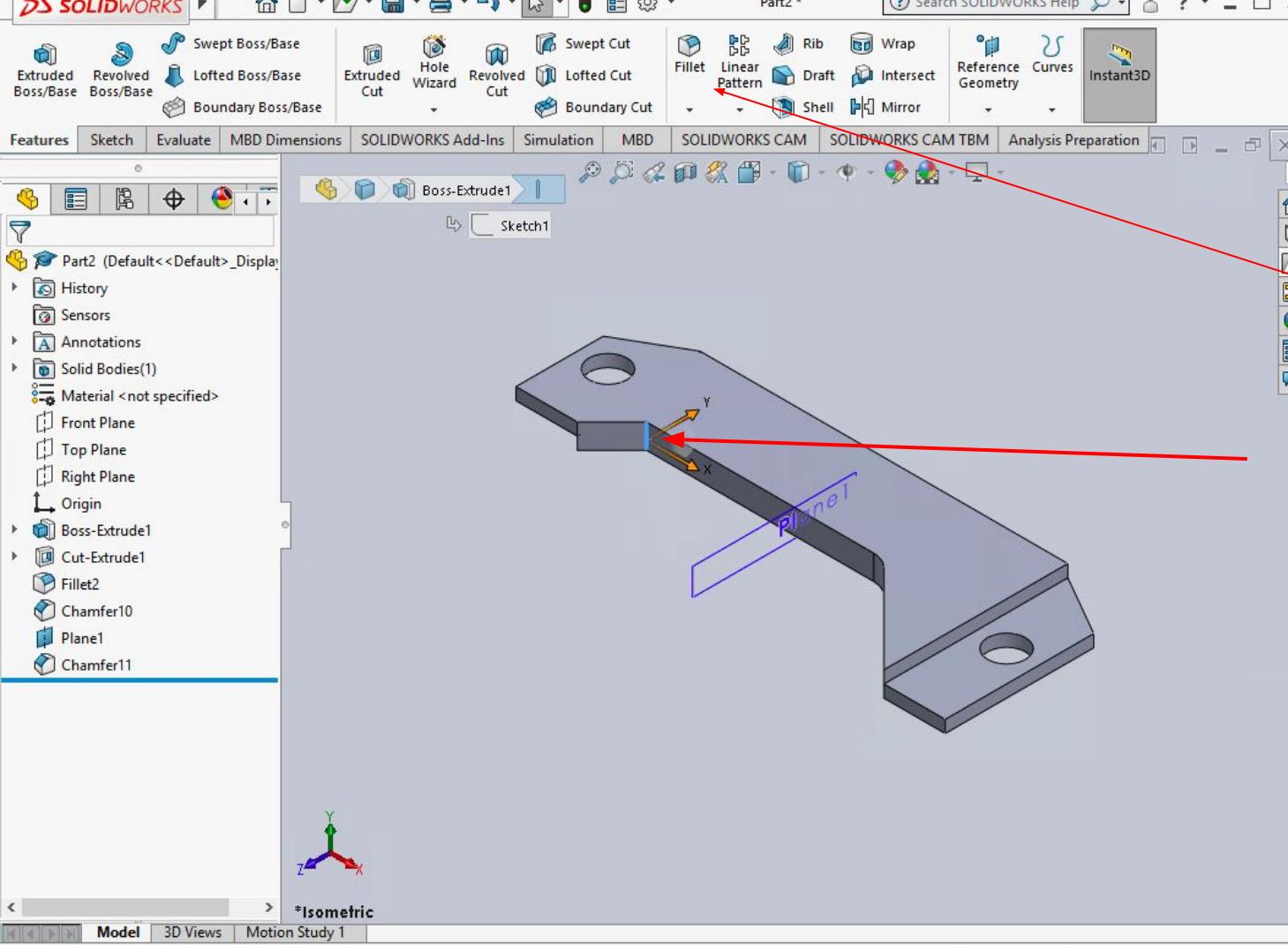
The 3D model shows a rectangular base with two circular holes. A sketch plane, labeled "Plane1", is highlighted in blue and positioned near the bottom edge of the base. A red arrow points from the text below towards this sketch plane. The model is shown in an isometric view with a coordinate system at the bottom left.

Sketch1

Part2 (Default<>Default>_Display)

- > History
- > Sensors
- > Annotations
- > Solid Bodies(1)
 - Material <not specified>
 - Front Plane
 - Top Plane
 - Right Plane
 - Origin
 - Boss-Extrude1
 - Cut-Extrude1
 - Fillet2
 - Chamfer10
 - Plane1
 - Chamfer11

Repeat this process on the opposite side.

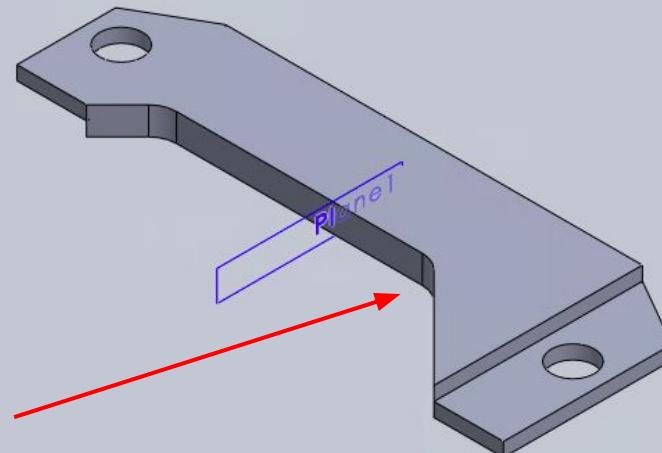
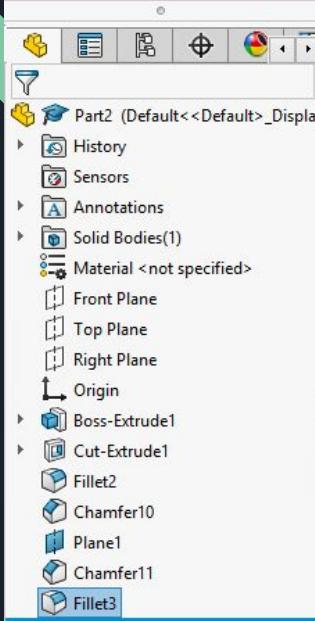


On the features tab,
select FILLET

Click the corner
where the 45 degree
angle meets the body,
and then press enter.



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Repeat this process on the other edge



*Isometric

Model 3D Views Motion Study 1

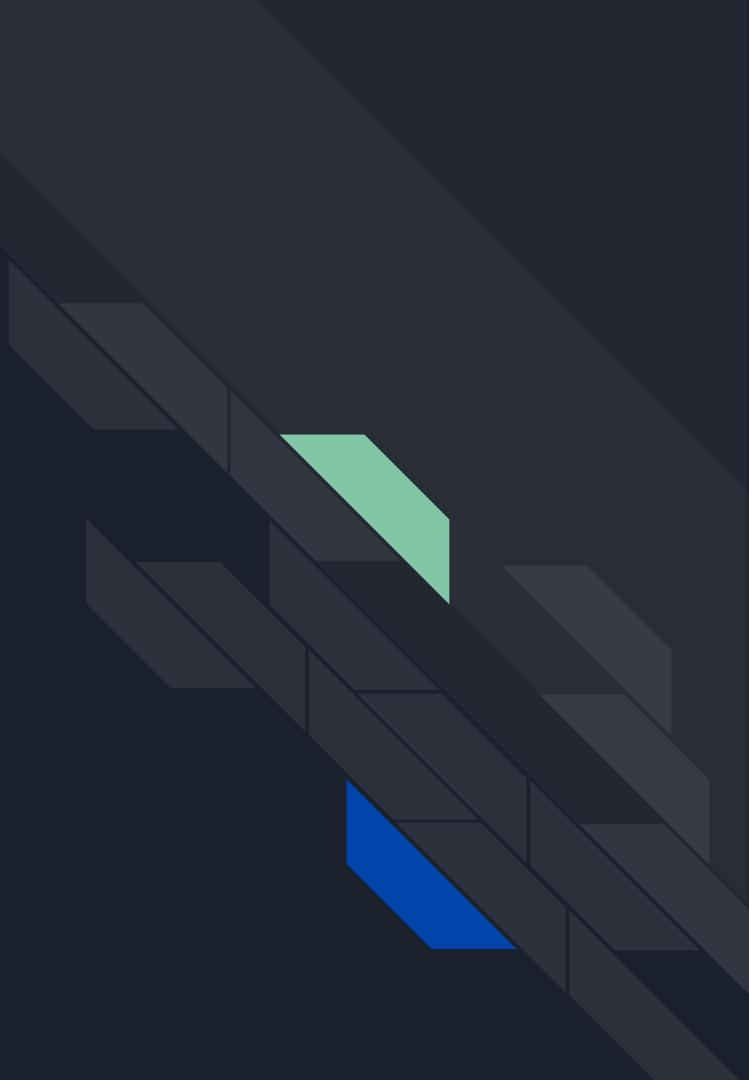


Breakout Rooms - 10 Min

Please ask the mentor in your group any questions you may have about solidworks (Gabriel, Dennis, Chris, Olivia, SeanLiam)

- Sketch - line, circle, rectangle
- Smart Dimension, Relations
- Extruded Boss/Base, Cut
- Chamfer, Fillet
- Why?

On your Own:



SOLIDWORKS

Swept Boss/Base
Revolved Boss/Base
Lofted Boss/Base
Extruded Cut
Hole Wizard
Revolved Cut
Lofted Cut
Boundary Boss/Base
Swept Cut
Rib
Wrap
Filler
Linear Pattern
Draft
Intersect
Shell
Mirror
Reference Geometry
Curves
Instant3D

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Part1 *

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Extruded Boss/Base Revolved Boss/Base Lofted Boss/Base Extruded Cut Hole Wizard Revolved Cut Lofted Cut Boundary Boss/Base Swept Boss/Base Revolved Boss/Base Lofted Boss/Base Extruded Cut Hole Wizard Revolved Cut Lofted Cut Boundary Boss/Base Swept Cut Rib Wrap Filler Linear Pattern Draft Intersect Shell Mirror Reference Geometry Curves Instant3D

Features Sketch Evaluate MBD Dimensions SOLIDWORKS Add-Ins Simulation MBD SOLIDWORKS CAM SOLIDWORKS CAM TBM Analysis Preparation

Part1 (Default<<Default>>_Display)

History Sensors Annotations Solid Bodies(1) Material <not specified>

- Front Plane
- Top Plane
- Right Plane
- Origin

Boss-Extrude1

Cut-Extrude1

- Sketch2

Cut-Extrude2

- Sketch4

Cut-Extrude5

- Sketch6

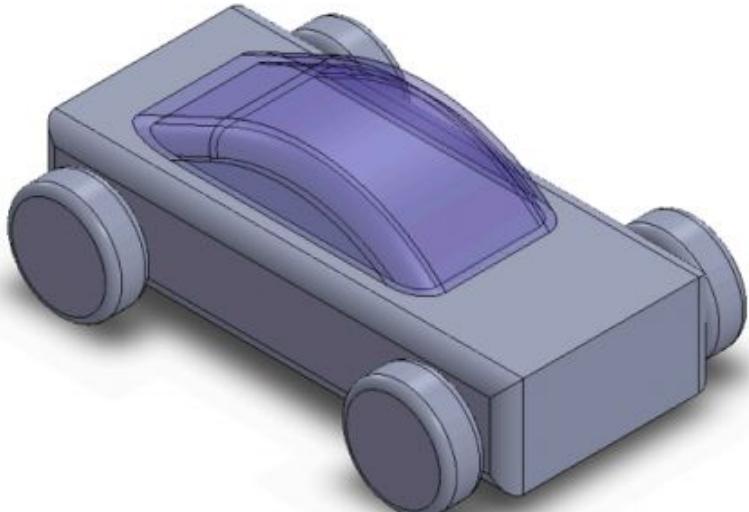
*Isometric

Model 3D Views Motion Study 1

SOLIDWORKS Education Edition - Instructional Use Only



SolidWorks Basics



The screenshot shows the SOLIDWORKS Student Edition software interface. A green arrow points from the top-left towards the home button in the toolbar. Another green arrow points from the left side of the screen towards the 'Learn' tab in the top navigation bar. The 'Learn' tab is highlighted in blue, indicating it is selected. The main content area displays several learning options:

- Introducing SOLIDWORKS**:
 - SOLIDWORKS Fundamentals
 - Parts, Assemblies and Drawings
- Tutorials**:
 - Step-by-step, interactive instruction
 - Learn at your own pace
- MySolidWorks Training**:
 - Unlimited 24/7 access
 - Follow along using example files

Below these sections, there is a 'Samples' section with four categories:

- Introducing SOLIDWORKS Samples**: Includes icons for a wrench and a bolt.
- Tutorials Samples**: Includes icons for a yellow 3D model and a purple assembly.
- What's New Samples**: Includes icons for a key and a small structure.
- 3D CONTENTCENTRAL Community Favorites**: A dark blue button.

At the bottom of the window, there are four navigation links:

- Online Tutorials**
- My Training**
- Certification**
- Curriculum**

On the right side of the interface, there is a vertical sidebar titled "SOLIDWORKS Resources" with the following sections:

- SOLIDWORKS Tools**: Includes links for Property Tab Builder, SOLIDWORKS Rx, Performance Benchmark Test, Compare My Score, Copy Settings Wizard, and My Products.
- Online Resources**: Includes links for 3DEXPERIENCE Marketplace and Partner Solutions.
- Subscription Services**: Includes a link for Subscription Services.

Click the home button

Navigate to learn and select one of the options