

# General FreeCAD instructions

Alternative Instructions:

[http://www.freecadweb.org/wiki/index.php?title=Getting\\_started](http://www.freecadweb.org/wiki/index.php?title=Getting_started)



*Technology Training that Works*

# FreeCAD Software

We shall cover:

- Getting familiar with your workspace
- Knowing your design tools
- Designing a 3D Model

You can download your personal copy by clicking here:

<http://www.freecadweb.org/>

Also note that FreeCAD is easy to install all you are required to do is click install and follow the instructions.

FreeCAD is also installed on REMOTE LAB 12



*Technology Training that Works*

# Using FreeCAD

On opening the software, you would be shown the FreeCAD start page which contains the needed instructions on how to start.

The start page introduces you to the different kind of designs that can be done on FreeCAD namely:

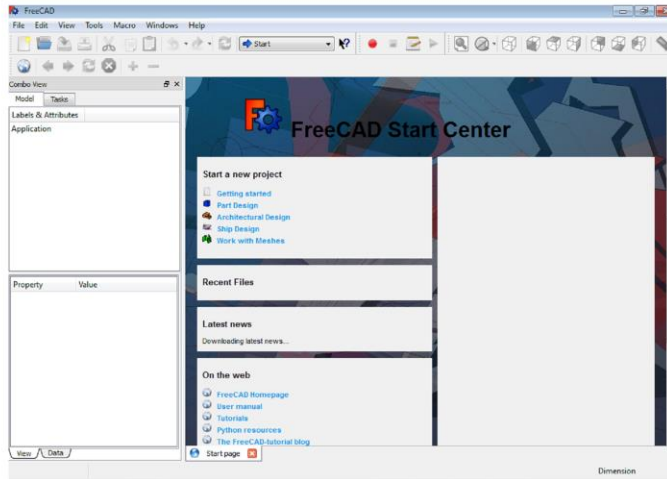
- Part Design
- Architectural Design
- Ship Design and
- Working with meshes

Simply hovering your mouse on any of this design types brings out a summary of what can be done there



*Technology Training that Works*

# FreeCAD Start Page



*Technology Training that Works*

# Understanding the Workspace

The workspace of every CAD software is quite similar in terms of the arrangement of the drawing area/drafting tools and FreeCAD is no different. From the image below, we can see:

1. The drawing area—this is the space where every sketch, modeling or drafting done is shown or previewed
2. The Tools Menu—the top right corner holds all the needed tools for drafting, viewing, coloring, texturing and designing a model



*Technology Training that Works*

## Understanding Your Work Space

3. Customizable Window 1—freeCAD gives you the choice of customizing this window to your personal specification. You can make use of it to view: model dimensions, color types, textures etc.

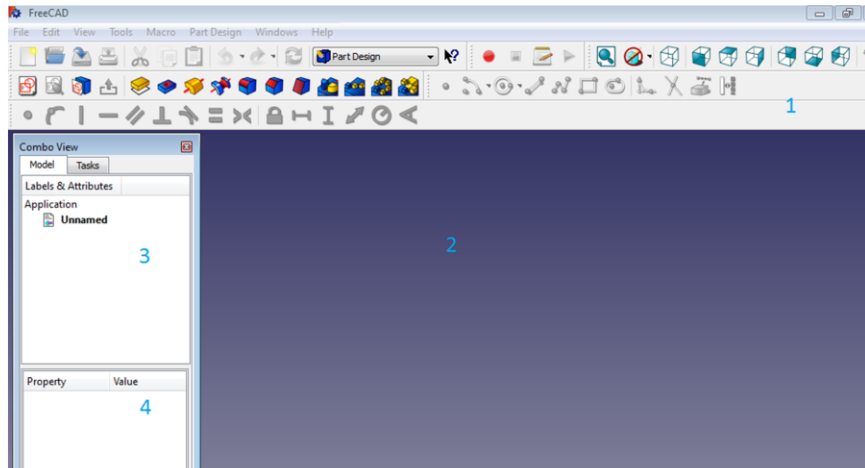
4. Customizable Window 2—the second window can also be customized to show certain properties of the object being designed as well as the workspace. You can view the statistics of the designed model view, model properties in different format, model component dimensions etc.

And as you become more proficient, customizing your workspace becomes an extension of your preferences



*Technology Training that Works*

# Understanding Your Workspace



*Technology Training that Works*

# Creating Preferences

As stated earlier, it is quite important to set some design parameters that make use of FreeCAD's intuitive workspace to simplify your modeling activities. So here are some preferences that must be fine-tuned before drafting begins:

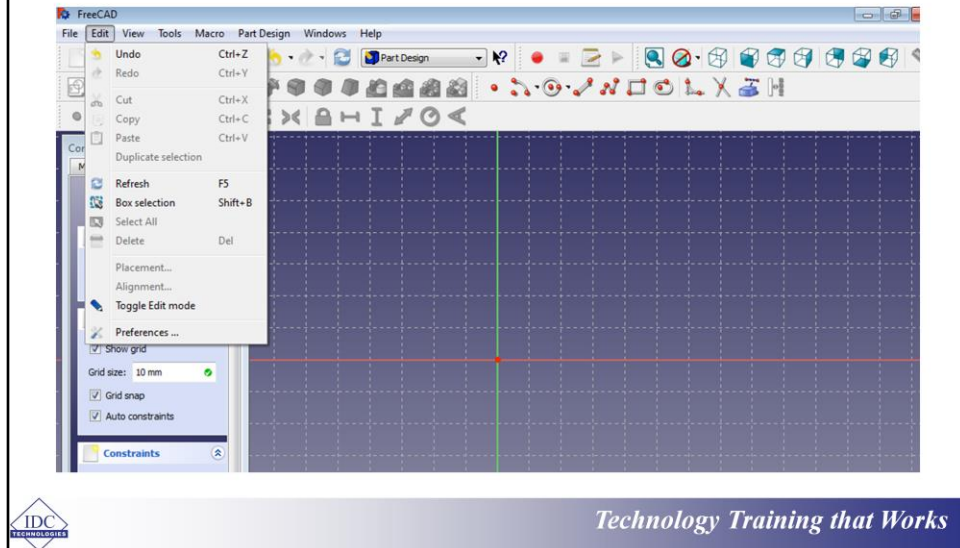
- Refine sketch—setting this parameter to 'on' ensures that FreeCAD produces well defined lines as you sketch
- Refine Booleans—ticking the refine Boolean box ensures that you work with highly refined objects while using the Boolean feature
- To do these follow the instructions below



*Technology Training that Works*



# Refining Sketch and Boolean Preferences

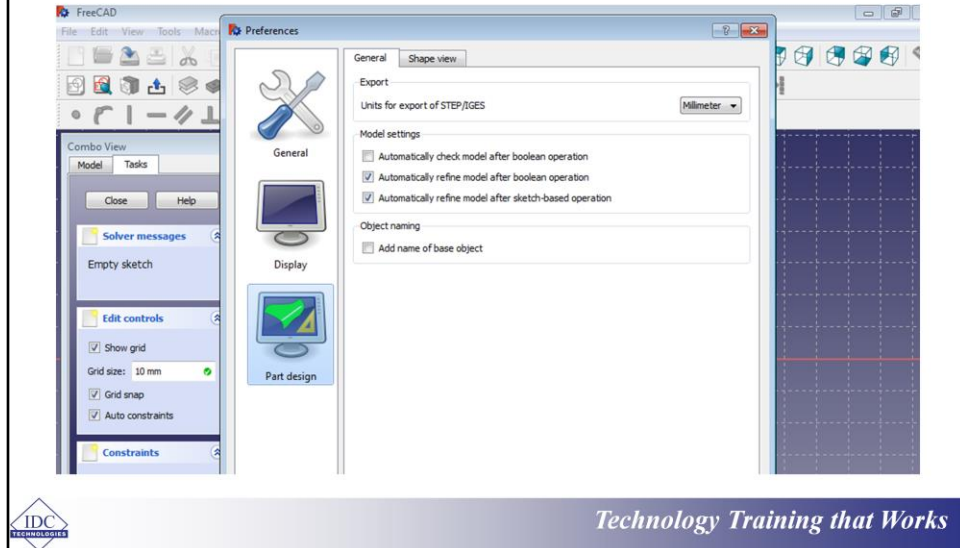


*Technology Training that Works*

To change the sketch and Boolean refining parameters, these are the steps:

1. Navigate your mouse cursor to the top right corner of the tool section and click on the edit drop down menu.
2. This action brings out the drop down menu and the next step is navigating to 'preferences' and clicking on it.

# Refining Sketch and Boolean Preferences



After 'preferences' has been clicked, it opens your 'preferences' in a new menu and you should proceed by clicking the part design icon on the left—as seen in the image. This in turn opens its own menu and you should click on;

1. The Automatically refine model after Boolean operation
2. And the automatically refine model after sketch-based operation.

These actions complete the refine preference setting and prepares your workspace for designing.

# Chapter 1: Creating A Model

For this tutorial, a simple 3D model will be constructed, designed or modeled to create the perfect environment for developing your 3D modeling skills. The model to be designed is a simple bow tie having four components:

- The bi-triangular shaped body of the bow tie
- A cylindrical support for the bow tie's knots
- Two knots situated at both ends of the cylindrical supporting beam.



*Technology Training that Works*

# Outlining The Steps to Designing The Bow Tie

The bow tie will be designed using the 'Part design' workbench due to its very intuitive sketch feature that is easy to use. The steps to creating the bow tie are:

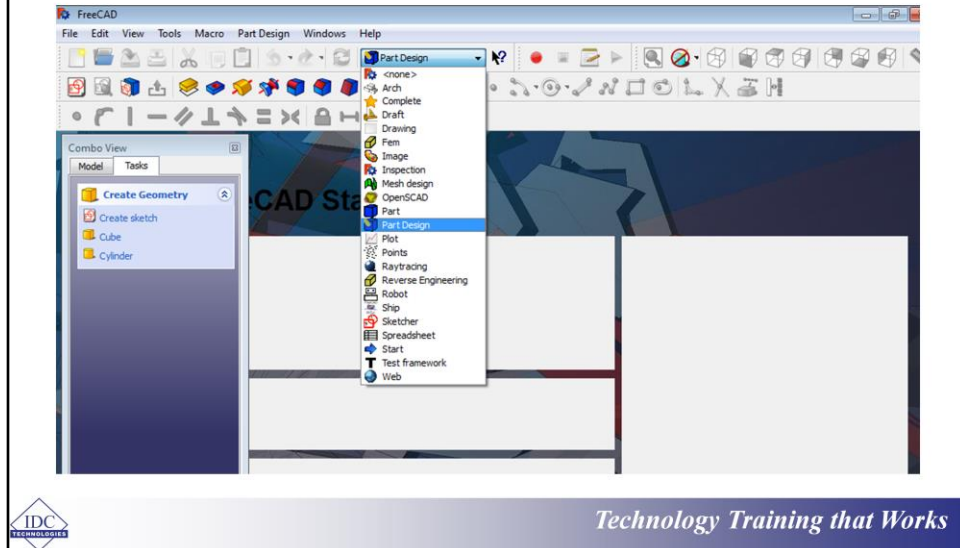
1. Designing the main triangular body
2. Creating a cylindrical hole for the support beam
3. Designing knots that fit

Therefore, there will be three components to be designed that will form the complete object at the end of this exercise.



*Technology Training that Works*

# Designing the Bow: Choosing Design Technique



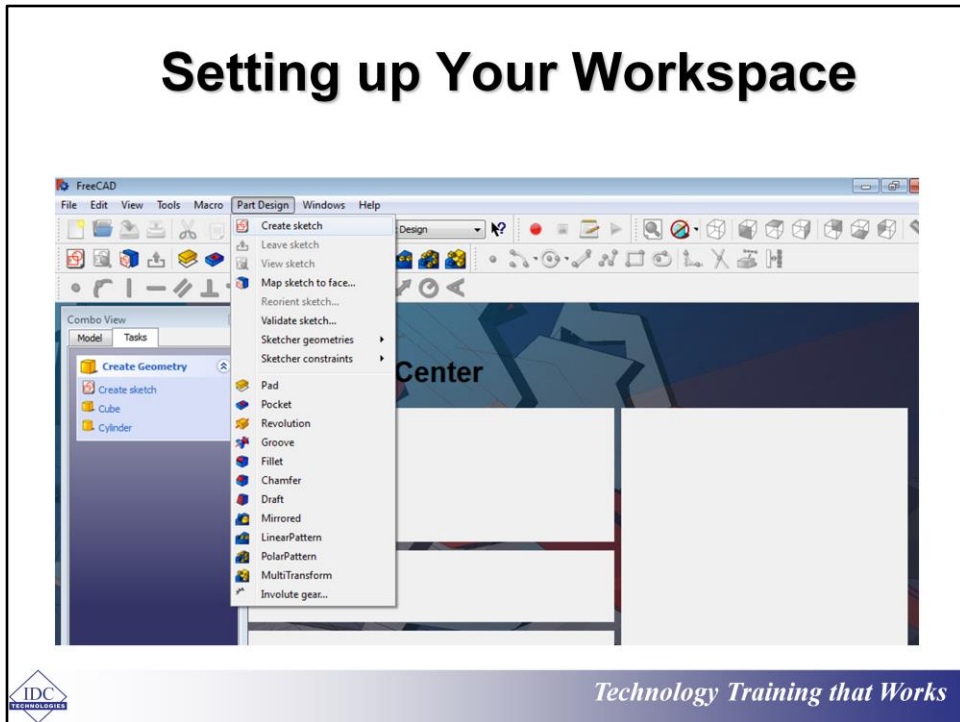
The first step to kick starting the design of the bow tie is getting your workspace and the 'workbench—which is a design technique with its own set of tools ready. To do this simply;

1. Navigate your mouse cursor to the 'tool section' and click on 'Start' section.
2. This brings out a drop down menu, showing the different drafting techniques that can be used
3. Simply click on 'Part design' to activate the design technique to be used in this tutorial.



*Technology Training that Works*

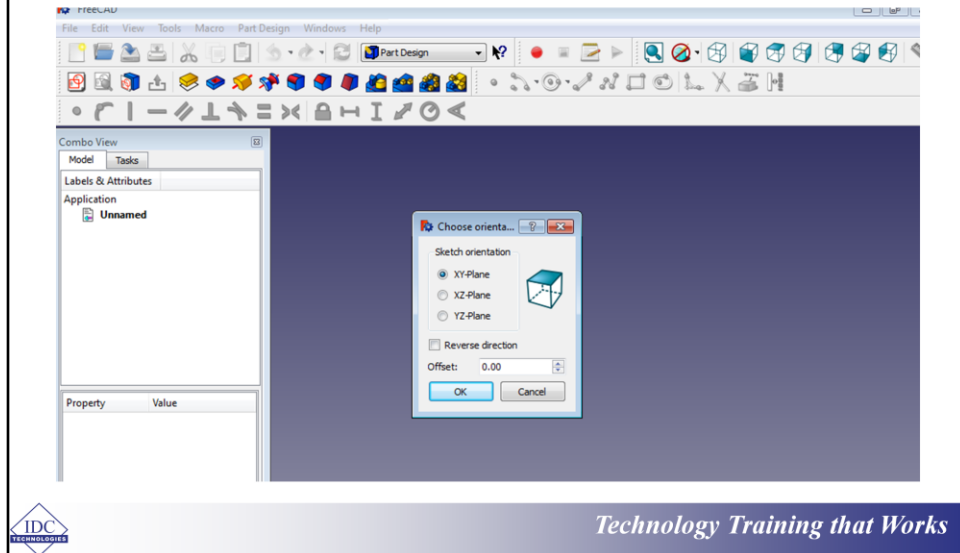
# Setting up Your Workspace



The next step is preparing your workspace for the actual sketching and design process. Here are the steps to get this done:

1. On selecting 'Part design'—which was shown in the last slide—a new menu titled 'Part Design' opens up at the top left corner of the tools section.
2. With your mouse cursor, click on the new 'Part Design' drop down menu, and proceed to click the 'create sketch' icon located on the drop down menu. This creates a blank page with a 'choose orientation menu' which can be used to adjust your workspace to the drawing orientation you prefer.

# Setting the Work Space Orientation

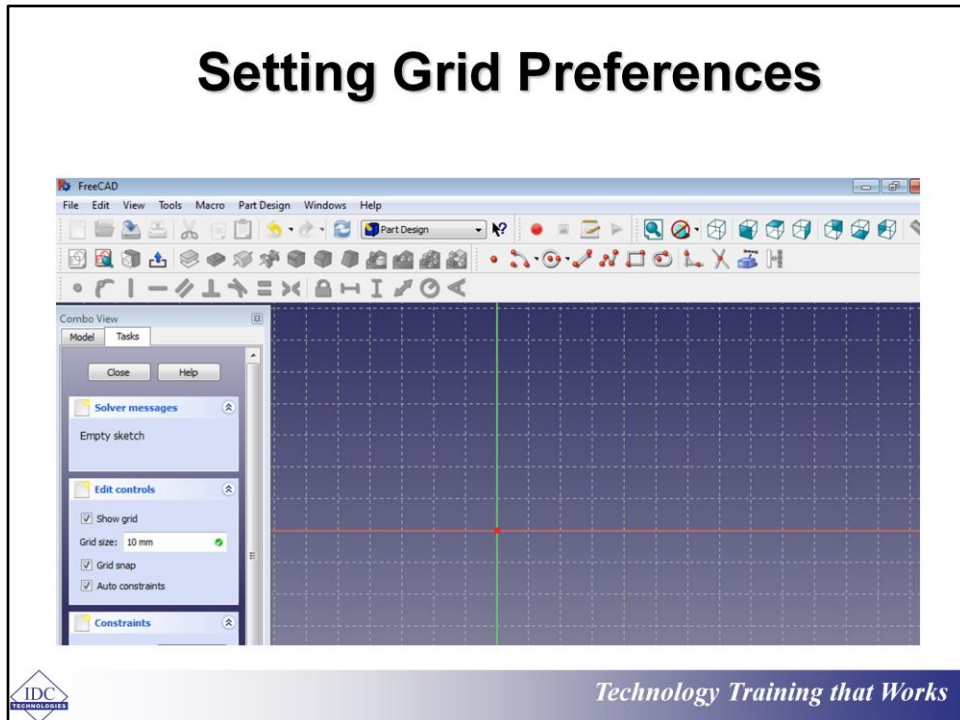


*Technology Training that Works*

To set the orientation or drawing axis for designing the bow tie, we make use of the time-tested XY axis which can easily be chosen by selecting the 'XY axis' option and clicking 'OK'.

Doing this and setting your 'grid preferences' will create fine lines that aid the drawing and dimensioning process of the drafted objects. To set the Grid preferences, simply move your mouse to the new menu opened at the left side of your screen and do the following as shown below/

# Setting Grid Preferences



*Technology Training that Works*

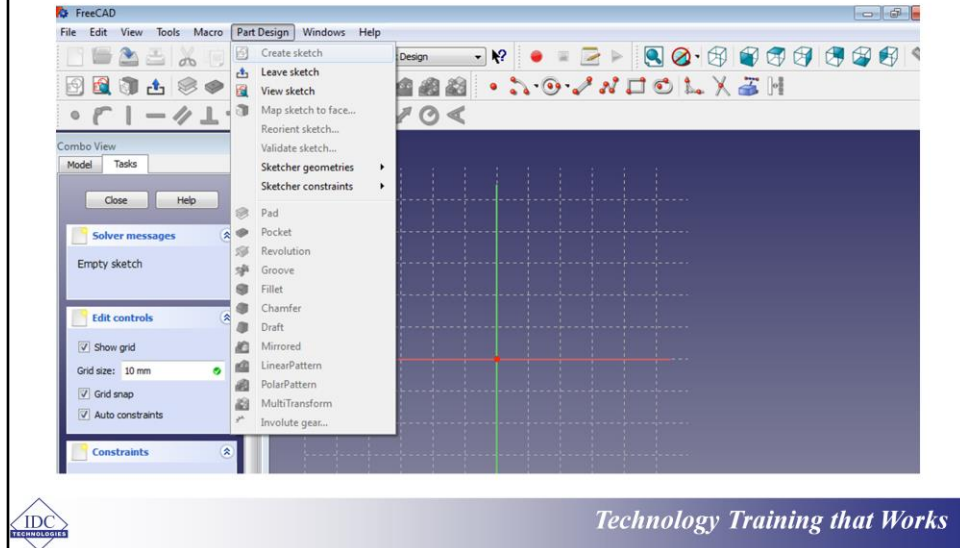
After navigating to the menu at the left, ensure you do the following;

1. Click the Show Grid—a tick means its activated
2. Set your Grid size to 10mm
3. Also tick the Auto constraints to yes

Once these are done a fine grid in the XY plane is generated to aid designing.



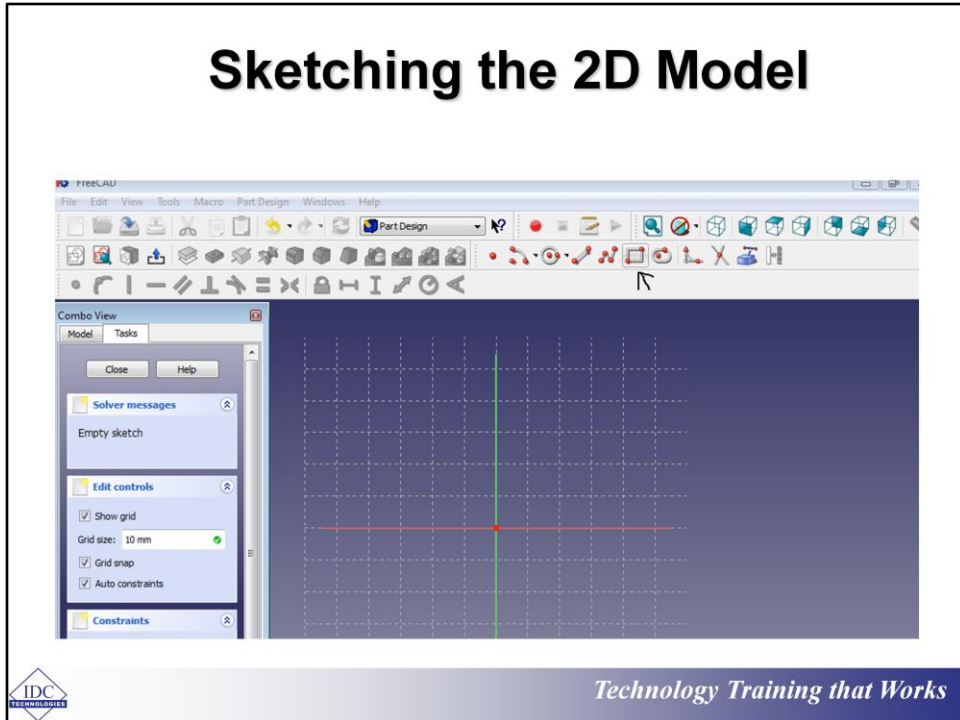
# Designing The Bow Tie: Drafting the 2D Model



The first step to drafting our bow tie is creating a 2 dimensional sketch which can be developed into a proper 3D object and to start a sketch simply;

1. Move your mouse to the tool section at the right left corner of your workspace, and click 'Part design'.
2. A drop down menu will then come up, click the 'Create sketch' option on the menu and this selects the drafting tool to be used

# Sketching the 2D Model

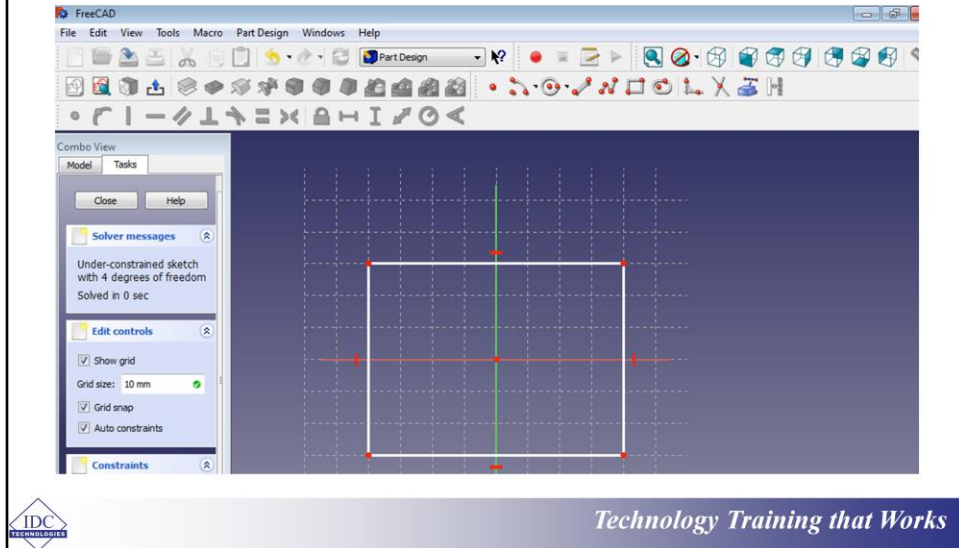


*Technology Training that Works*

Now that the sketch option has been selected, we can proceed to sketch by using the preset sketch tools 'create a point, create a line, create a rectangle etc.' to start sketching.

For our bow tie, the 'sketch a rectangle' drafting tool is what will be used to start.

# Designing the Body of the Bow Tie

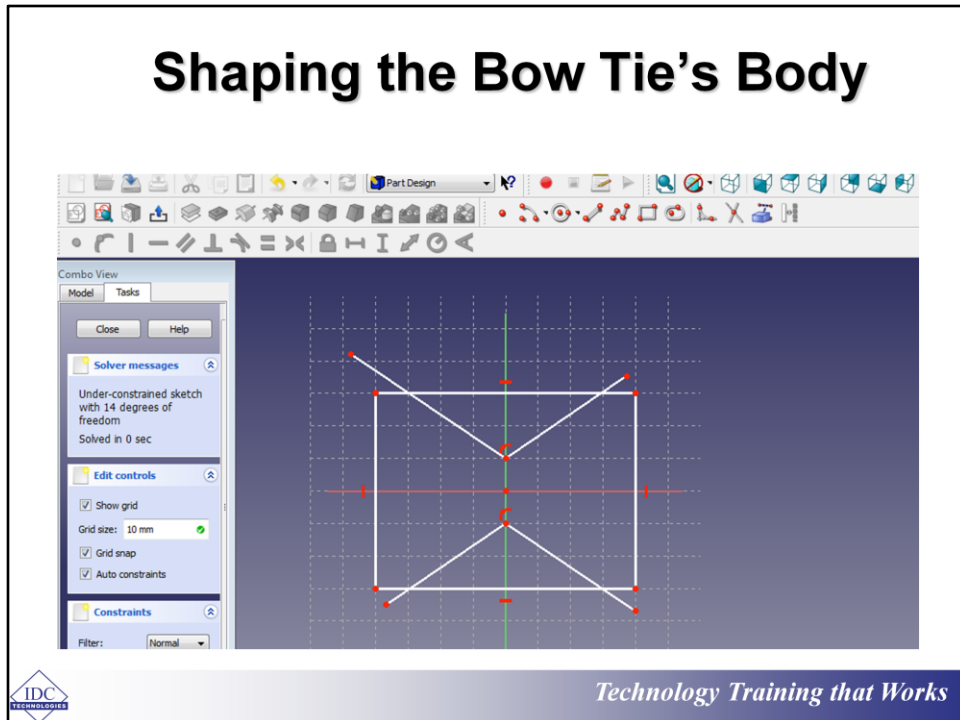


The next step after selecting the 'create a rectangle tool' is drawing a rectangle, using the mid point of your XY axis as the pivot. Once this is done, the 'Combo view' parameters which can be seen on the left menu, shows;

1. The 2D model's constraints
2. And once you click on 'close' you lock down whatever diagram you have drawn.
3. You can then view the diagrams' dimensions by clicking on the model option also on the left menu.

Now we have drawn the box, the next task is shaping it to look like a bow tie and this is done with the use of the 'create lines tool'

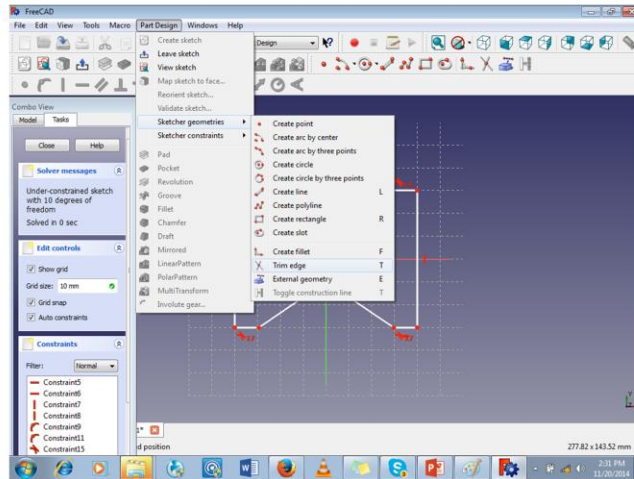
# Shaping the Bow Tie's Body



With the 'create line tool' -which is selected from the sketch tools at the top right corner of the tool section-we proceed to cut the rectangle in strategic places thereby forming the two extreme triangular ends of a bow tie. To do this;

1. Move your mouse 10mm on the +x axis off the rectangle's center point, click here to form your line's base and connect it to the rectangle's breadth.
2. Proceed to do these for the remaining 3 lines as seen in the picture

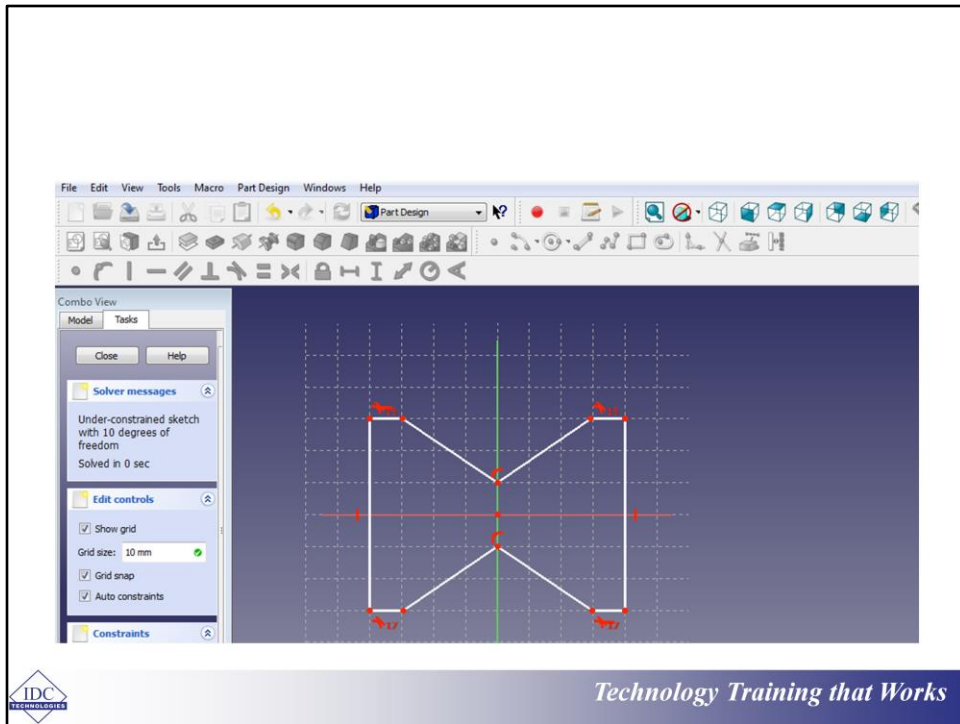
# Cutting the 2D Bow Tie to Shape



*Technology Training that Works*

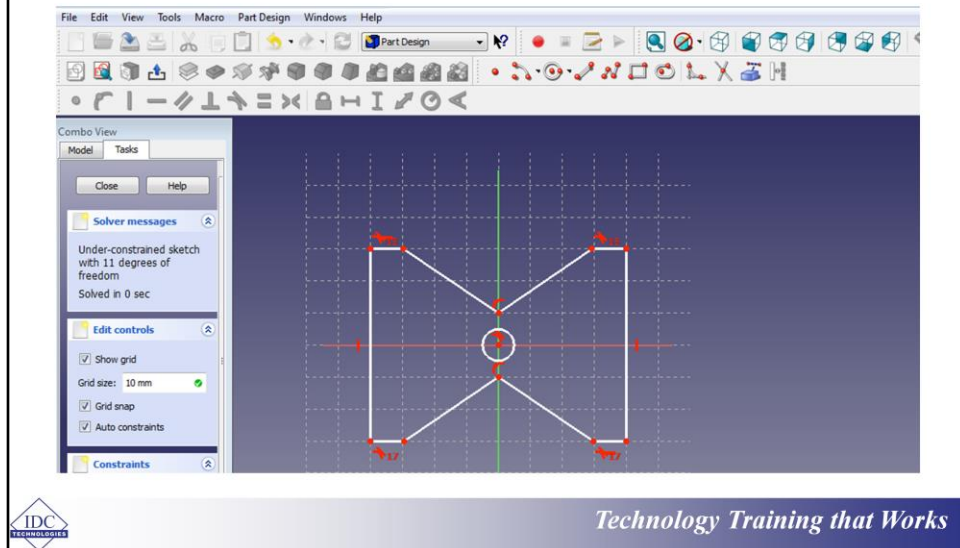
Now, to cut the rectangle into shape, the excess regions decimated by our 4 lines must be removed, leaving us with the body of the bow tie. To do this;

1. You have to cut out these lines with the 'Trim tool' –which can be selected by navigating with your mouse to the 'part Design' menu located at the top of your design page.
2. Clicking on it reveals a drop down menu, move your cursor to 'Sketcher geometries', and clicking on it brings out another dropdown menu, then proceed to select 'Trim edge'
3. Now you are ready to trim the edges created by the lines
4. Alternatively you can simply click on the letter 'T' which automatically activates the Trim edge feature.



With the trim tool selected, proceed to click on all erroneous lines till these shape is gotten.

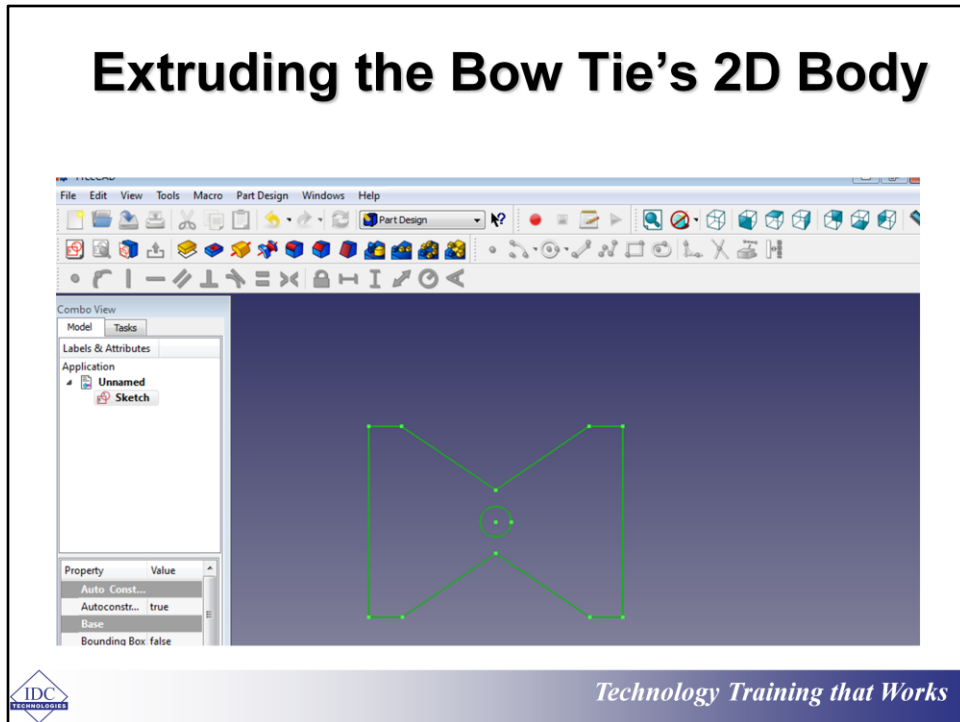
# Creating A Hole for Our Cylinder



To create the whole for the cylinder, we first have to insert a 2D circle into the body of our bow tie and this is done with the aid of the 'Create a circle I sketcher' tool. To do this;

1. Pay a visit to the tool section and click the ' Create a circle tool'—located where we found the 'create rectangle and lines tools'.
2. Next, move the cursor to the center of the bow tie, click at this center and extend your mouse till you get the required dimension. Then click to stop.
3. Ensure you keep the value of the hole's dimension in mind for future references. This is 4.5mm

# Extruding the Bow Tie's 2D Body

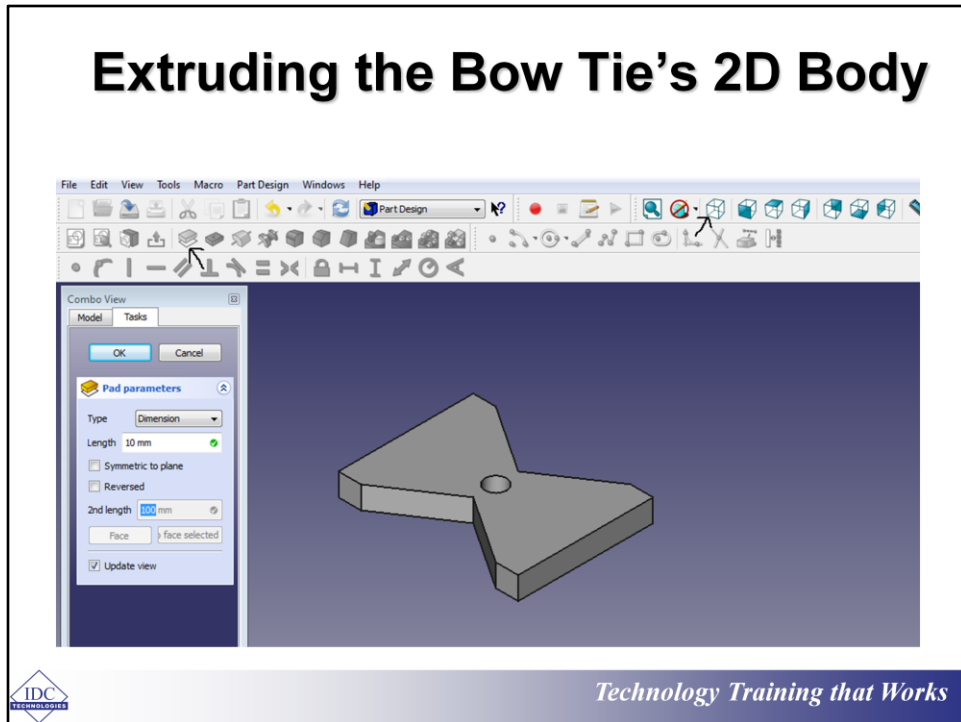


The next step in getting our project on the way is turning the 2 dimensional model we have designed so far into a 3 dimensional model that brings out the shape of our bow tie tastefully. To do this here are the steps;

1. The first step is changing the sketch tools we have used so far for they will not be needed for now. To do this simply navigate your cursor to the 'Combo menu' at the left and click 'close'
2. This closes the sketch tools.
3. Next, on the 'Combo menu' click on 'Model' which in turn brings out a menu, then proceed to select sketch under the title of our model—currently known as unnamed.
4. On doing this, the entire 2D diagram of our bowtie will be highlighted in green colored lines.



# Extruding the Bow Tie's 2D Body

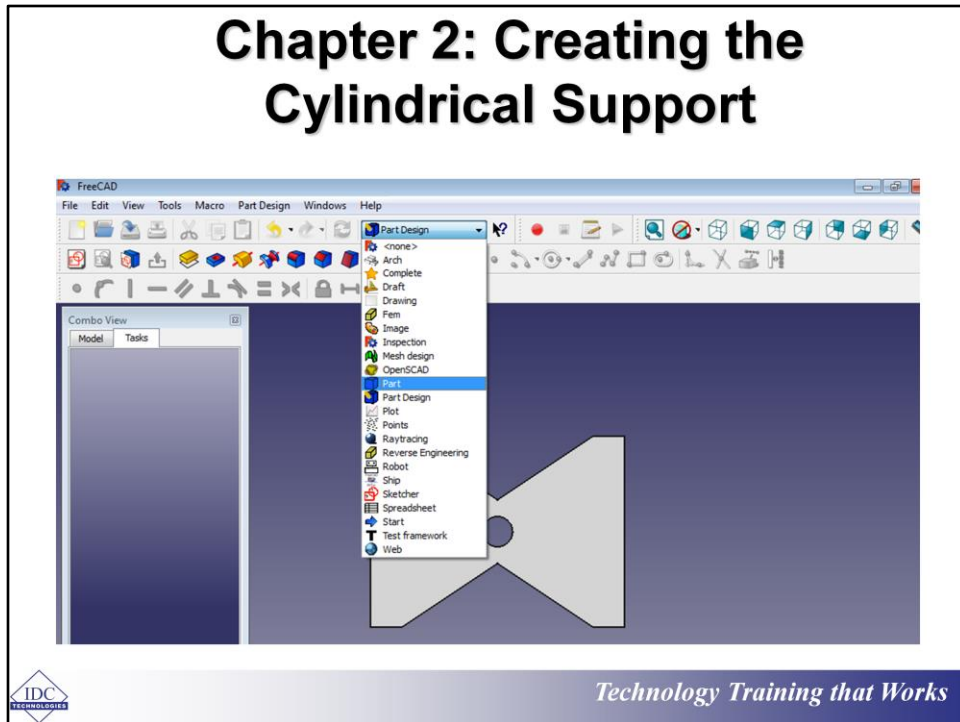


To extrude the 2D bow tie we designed in the previous stages, we have to make use of the 'Pad' tool which can be found at the tool section—as shown by the arrow.

These are the steps to extrude our 2D diagram

1. Navigate to the 'Pad' tool—at the top left corner of the tools section—and click it. This immediately extrudes the 2d diagram
2. next, move over to the 'Combo view' menu to change the length from 1mm to the dimension you prefer—here, 10mm was used.
3. Click ok and this changes will be applied.
4. To better view the 3D object, navigate to the set view menu at the top right corner of your tools section as shown by the arrow.
5. This angles the 3D model as seen in the image below

## Chapter 2: Creating the Cylindrical Support



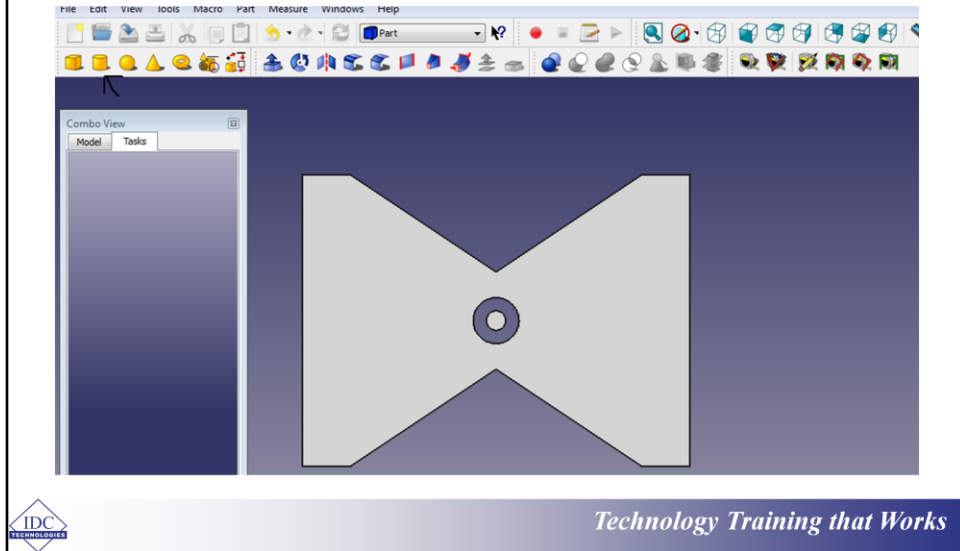
Now to create our cylindrical support beam which will hold the bow tie's knots at the two ends, we will need to change the current work bench 'Part Design' been used to 'Part' for the following reasons:

1. Part design immediately creates a stock object without the need for sketching and extruding
2. It is easier to dimension and position an object designed with 'Part' than with 'Part Design'

So to make this change –from 'Part Design' to 'Part' simply;

1. Move your mouse cursor to the top of your tool section and click on the design menu currently showing 'Part Design'
2. It brings down a drop down menu where you can simply click on 'Part' to select it.
3. On doing this, 'Part Design' changes to 'Part' and a new set of drafting features appear on the tool section

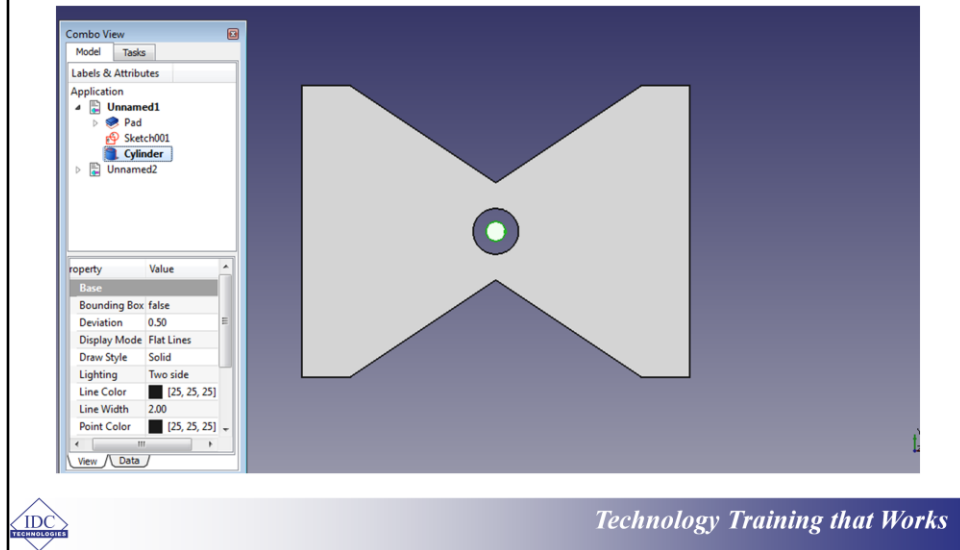
# Creating the Cylindrical Support



The black arrow shows the new set of tools that choosing the design workbench or feature 'Part' provides. Which leaves us with the next step of;

1. Choosing the 'Create a Cylinder' tool by clicking on it
2. This tool intuitively creates a cylinder in the middle of the incomplete bow tie as seen in the picture
3. On doing this, you will note that the cylinder's dimension is quite smaller than that of our hole and to remedy this situation, the dimensions must be tweaked

# Creating the Cylindrical Support

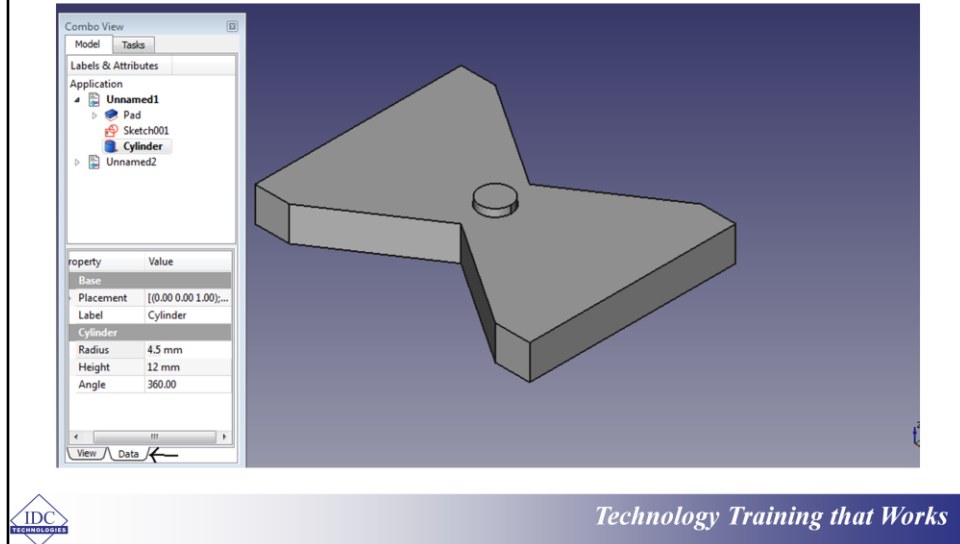


*Technology Training that Works*

To tweak the cylinder's dimensions, you must first access its overall data and this is done by navigating to the 'Combo View' menu at the left side of your workspace. Next, click on model and on doing so, you would see;

1. The hierarchy of your model—which tells you what model was first created and allows you to select individual components. For our bow tie, the hierarchy seen is: 'Unnamed'—which is the name of the entire project, Pad—which is the name of our bow tie's triangular body, and Cylinder—represents the cylinder to be designed
2. You can experiment by selecting any of these components to see how it becomes highlighted on the original object.
3. For this design, we will be selecting 'Cylinder'
4. Notice that on selecting Cylinder, the lower window on the combo page changes to show the parameters of the designed cylinder

# Creating the Cylindrical Support



Now, to change the dimensions of the cylindrical support, we need to access its data of dimensions to accomplish the task. To do this;

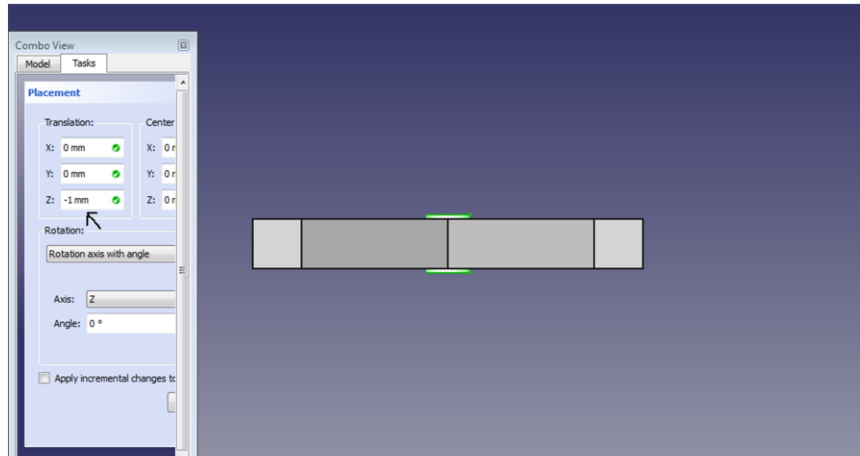
1. Simply move your cursor to the 'Combo View' menu and click on data as shown by the arrow in the image.
2. On the data window, you will see certain parameters which can be used to change the dimension of the cylinder: 'Placement' allows you change the objects location in the XYZ plane, Radius allows you change the radius/circumference of the object, angle allows you set its angle of inclination or declination etc.
3. So here, we set the radius to 4.5mm which is the radius of the hole and the height to 12mm which is 2mm more than the bow ties 10mm width.

This 2mm clearance space is for us to attach the knots to the cylindrical support



*Technology Training that Works*

# Creating the Cylindrical Support

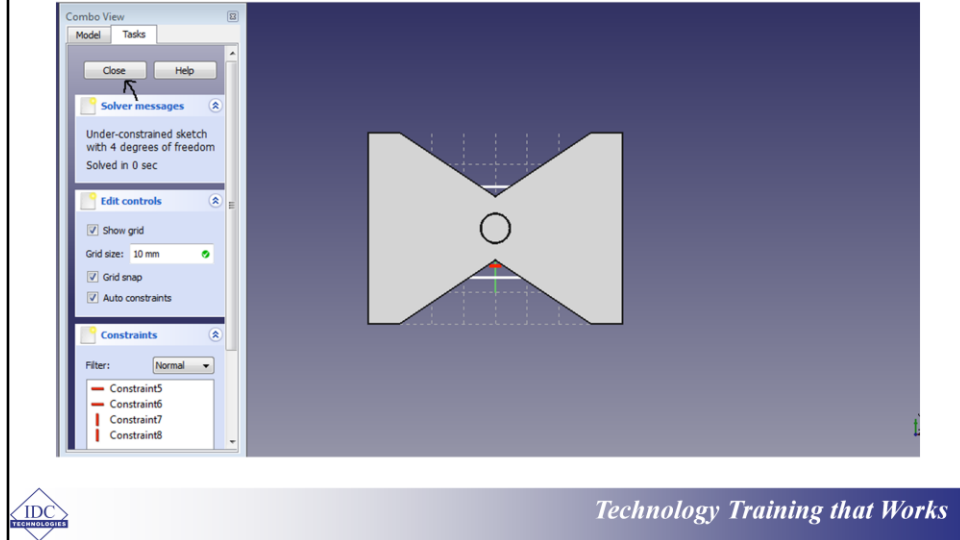


*Technology Training that Works*

Next, we have to place the cylindrical support strategically so it can hold knots at both bases of the bow tie and to do this, we need to tweak the 'Placement' data so that an inch will be visible at each base-end of the bow tie. This is done by:

1. Clicking on the placement tab—it has a '...' sign to bring out its menu box.
2. Now, since we plan to move the cylinder vertically, the only change to be done is on the 'z' axis
3. Click on the 'z' axis and change the dimension to -1mm and the cylinder becomes perfectly placed as can be seen in the image above
4. To get this view of your object simply navigate to the top left corner of your window and click the view boxes for the view of your choice
5. Finally, click OK to end the design of the cylindrical base

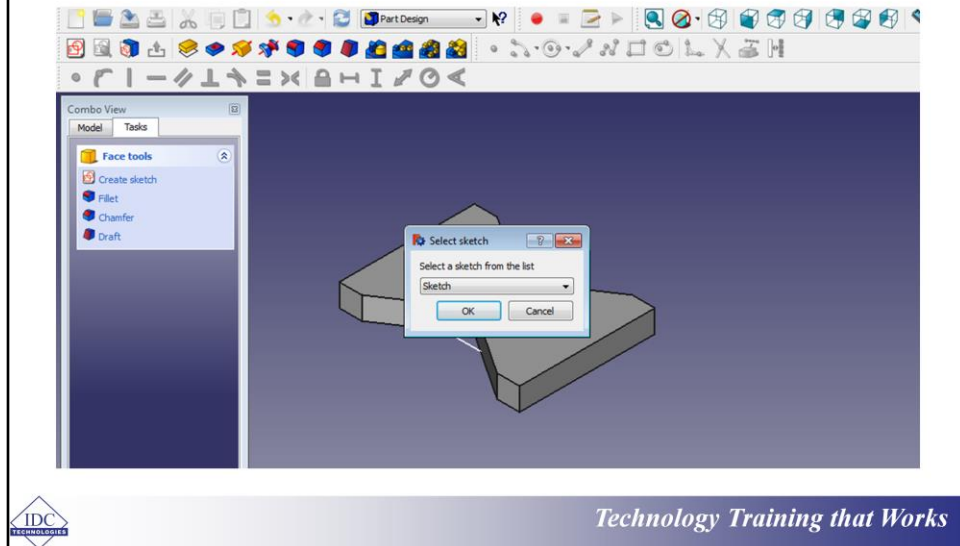
# Chapter3: Creating the Knots



The steps to creating the knot consists of using the 'Part Design' workbench—not 'Part'—to design the components that will become the two knots of the bow tie. To do this we;

1. Select the 'Part design' from the top of the tool section
2. Select 'Create Sketch' which can now be found on the 'Combo view' menu under tasks—due to it been used recently
3. Select the 'create a rectangle option' and draw a small rectangle across the cylindrical base.
4. Lastly click on the close tab to end the sketch task

# Creating Knots By Mapping a Sketch to a Solid Surface

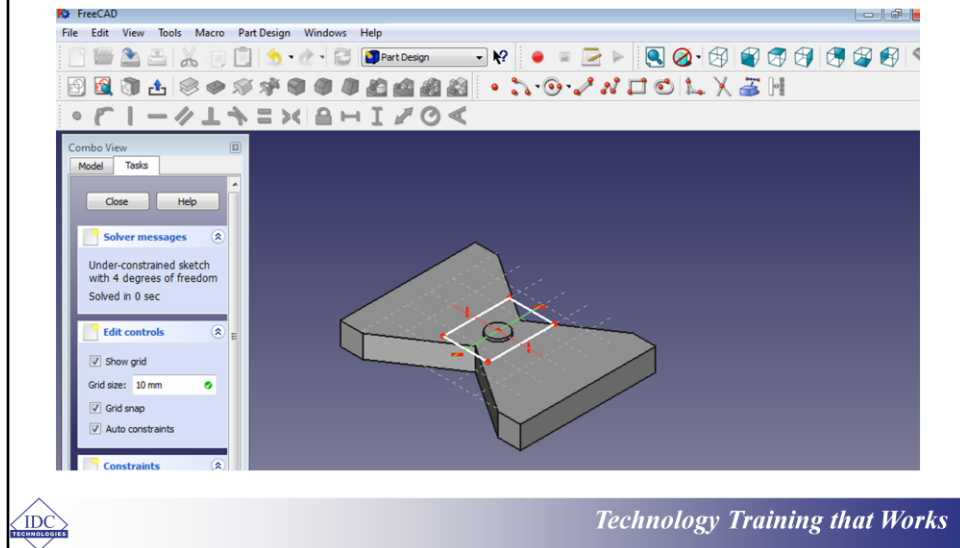


If you observe the new sketch just drawn, you would note that it is resting on the XY plane instead of on one of the cylindrical surfaces which makes the next step that of getting the sketched object on a cylindrical surface. Here are the steps to achieve this.

1. Select a surface of the cylindrical support—which gives it a yellow cast
2. Next move your cursor to the left side of your tools section to select the 'map a sketch to the face' and click on it
3. A 'select sketch' menu drops down and you select the sketch 001—which is the rectangular knot.
4. Click the ok button and the rectangle sketch will be aligned to the chosen surface



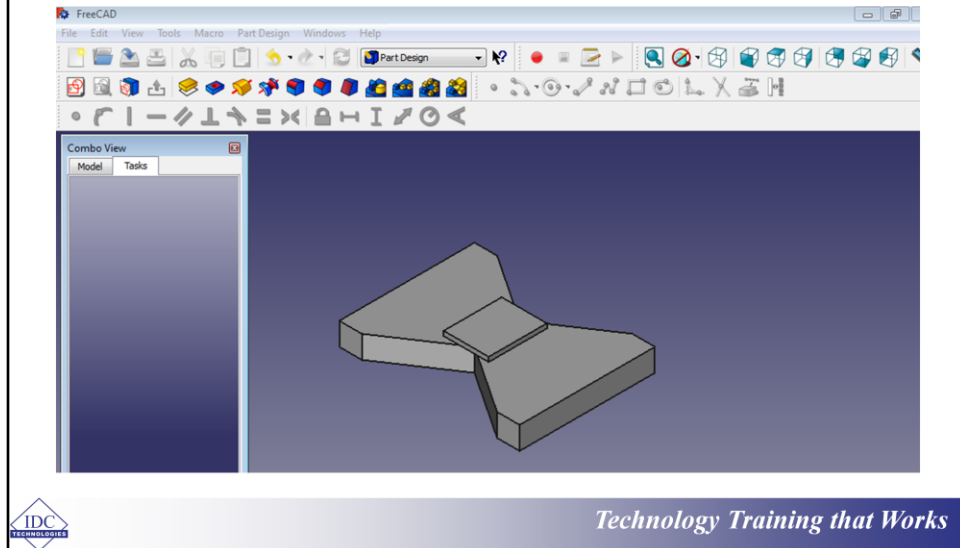
# Creating the Knot through Extrusion



Clicking the ok button aligns the sketch to the cylindrical surface as seen in the image above. The next step is extruding the 2D sketch to create a 3D model that serves as the first knot of our bow tie. These are the steps:

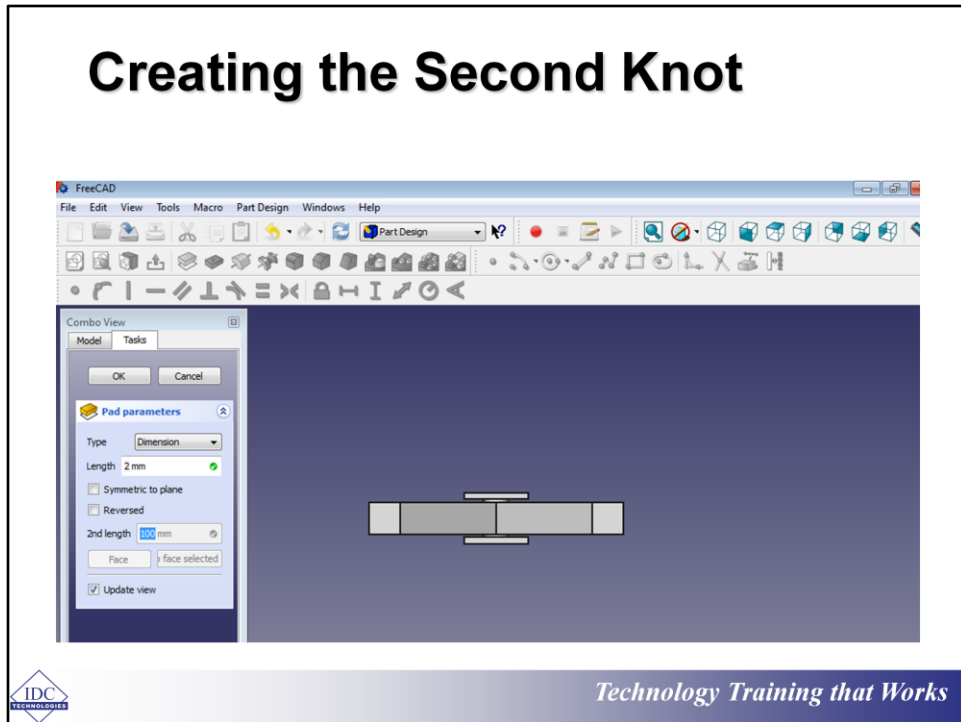
1. next, is extruding the sketch to form a rectangular knot and to do this, simply navigate to the 'Pad' and click it
2. This creates a 3D model from the 2D sketch.
3. You can then proceed to adjust the knot's width according to your preferences through the data section on the 'Combo view' menu.

# Creating the Knot through Extension



On clicking ok, we get the 3D shape of our model.

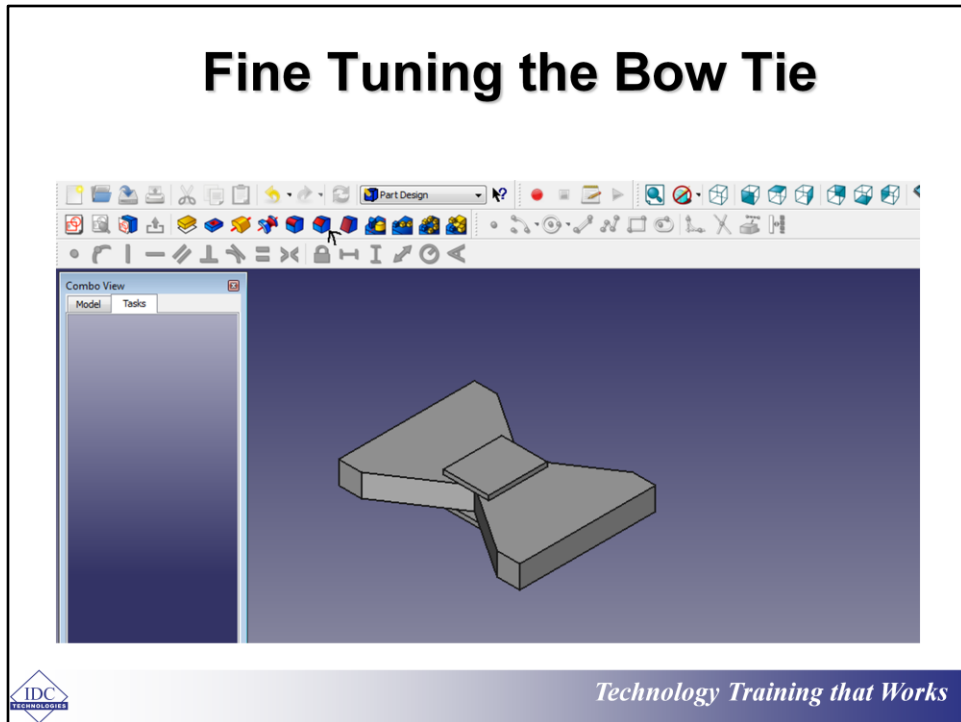
# Creating the Second Knot



The steps to creating the second knot are exactly similar to that taken in designing the first knot in the previous slides. The steps are:

1. Activating 'Part design'
2. Choosing the sketch tool and sketching a rectangle
3. Aligning the rectangular sketch on the second cylindrical surface
4. Extruding the sketch using 'Pad'
5. Click 'OK' to complete the process

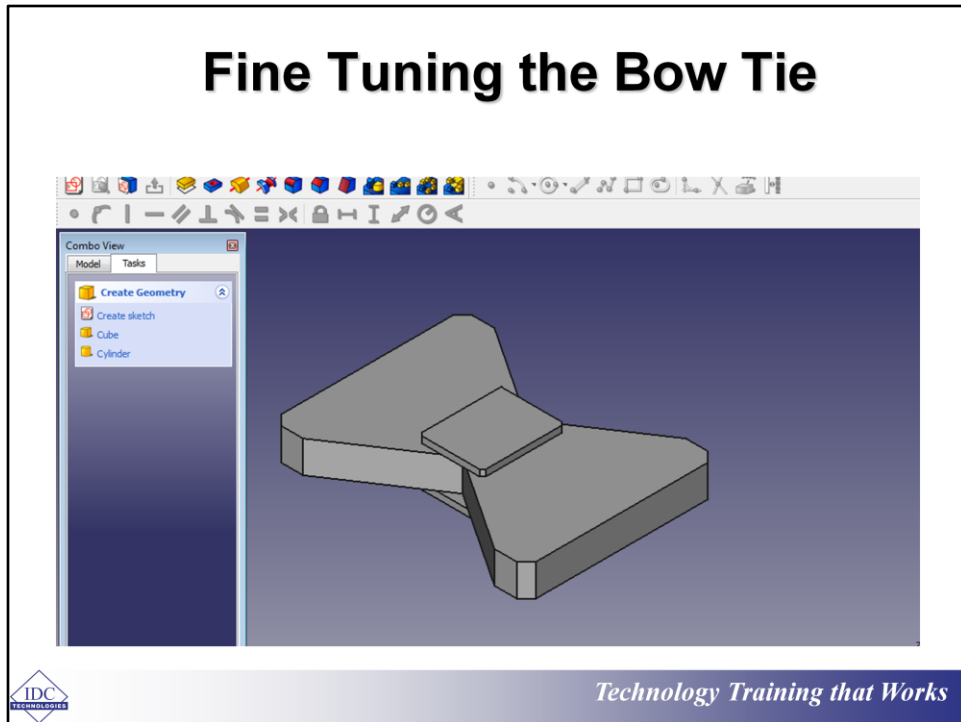
# Fine Tuning the Bow Tie



I believe no one has ever knotted a perfectly angled bow tie without some creases at its edge, so in this tutorial, some aesthetic work will be done to create a more rounded bow tie and to do this we will make use of the 'Chamfer' tool that can be found at the top left corner of the drawing tool's section. The arrow shows us the chamfer tool.

Note: Chamfer is a tool or feature that turns straight edges into rounded arcs.

# Fine Tuning the Bow Tie

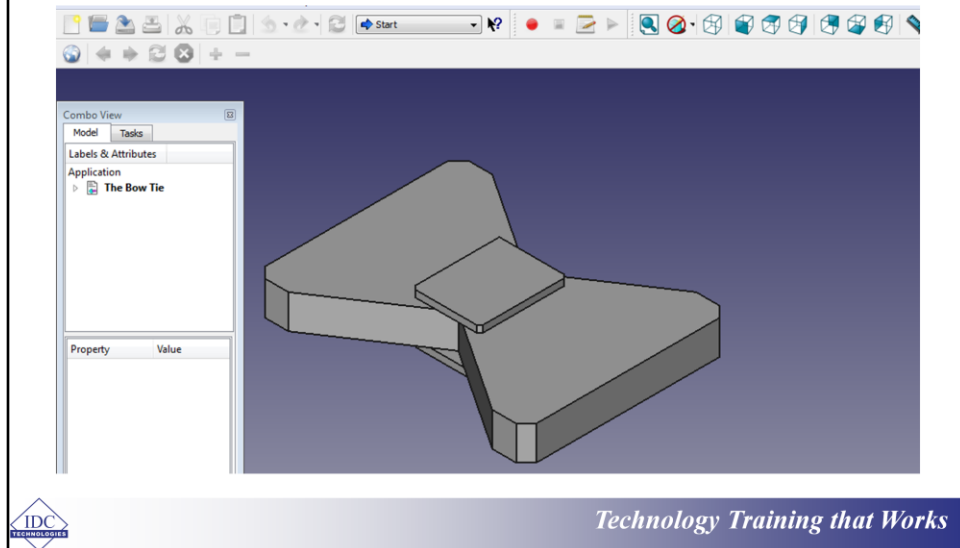


*Technology Training that Works*

Once the 'Chamfer' tool has been selected, simply:

1. Pick the edges you need to fine tune and apply it by clicking 'ok' from the 'Combo view' menu
2. You can also change the width of the chamfer by either increasing or decreasing the size.
3. Lastly, click ok to confirm
4. As you can see, certain edges are still un-chamfered, so the responsibility is lies on you to complete them.

# Saving and Exporting the Designed 3D File



The fact that this is a hands-on tutorial led to the overlooking of the importance and need to save every step of the way as you design your model. But since we have come to the end of the design phase now is the time to save and to do this, simply;

1. Navigate with your cursor to the left side of your workspace and click on file which brings down a menu
2. On that menu click on the save button and name our file 'The Bow Tie'
3. After saving, take a look at the 'Combo View' menu and you will see that 'unnamed' becomes 'The Bow Tie'.