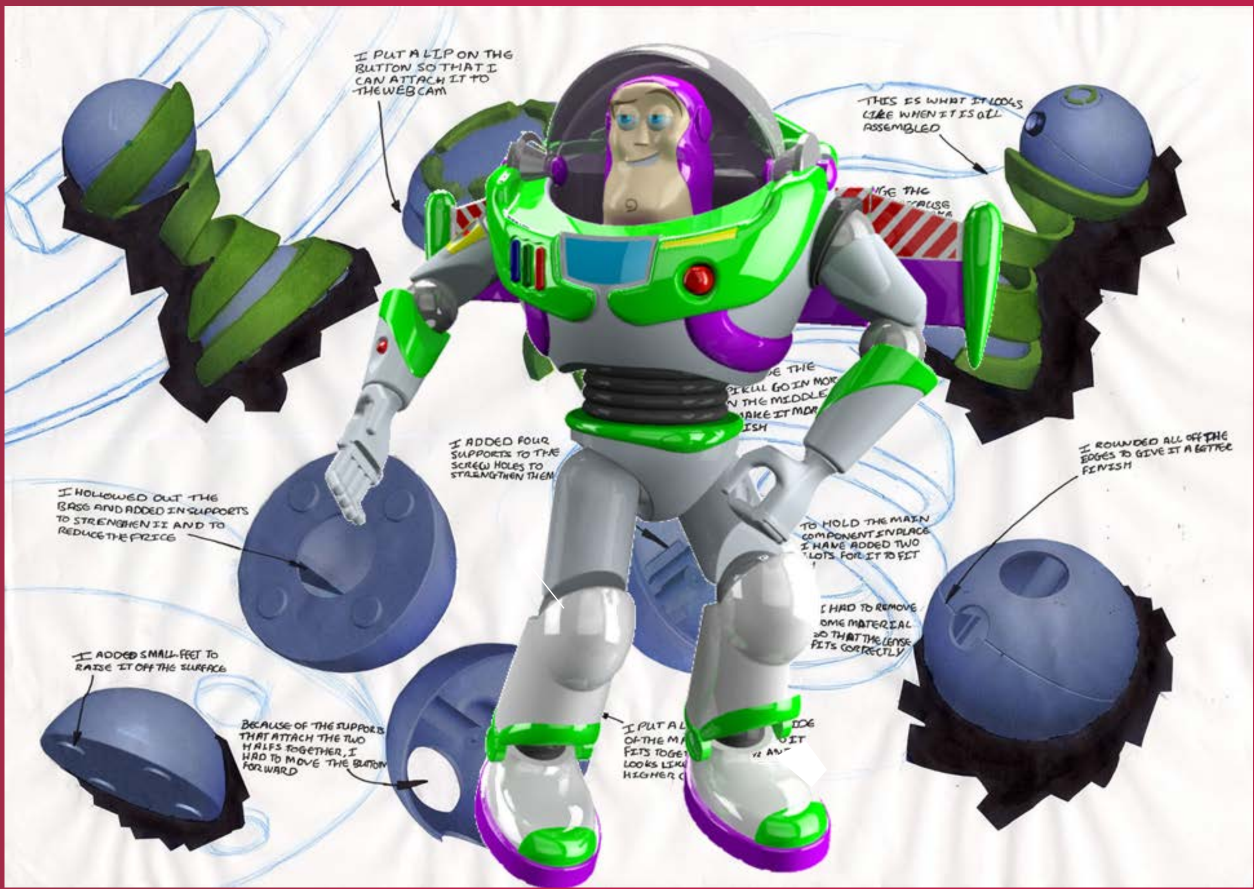
Section

1

HOW TO MODEL ALMOST

ANYTHING



Get started guide

PRODUCT DEVELOPMENT MANUAL

How to Model Almost ANYTHING

WRITTEN BY JORDAN COX

NOTICE OR RIGHTS

TRADEMARKS

ACKNOWLEDGEMENTS

Scott Morris, Mark Fischer, Adam Haas, Ayora Berry

Section

1

GETTING STARTED

Solid Modeling in Industry and the Classroom

All the employees at PTC Inc. are proud and happy to introduce to you the exciting world of product modeling. We at PTC are committed to providing our 27,000 customers worldwide with software and services that give them a significant advantage in the competitive markets of today.

This manual has been created to introduce students to the same world of product development as our customers. The purpose of this manual is to provide a guide for workshops and in-class curriculum which prepares students to plan and create effective 3D solid models. Each year volunteers from PTC and their customers help provide hands-on workshops using this manual to students all over the world. We hope that the instruction given in this manual will help make your experience in product modeling successful.

Case Study

Industry

WHIRLPOOL

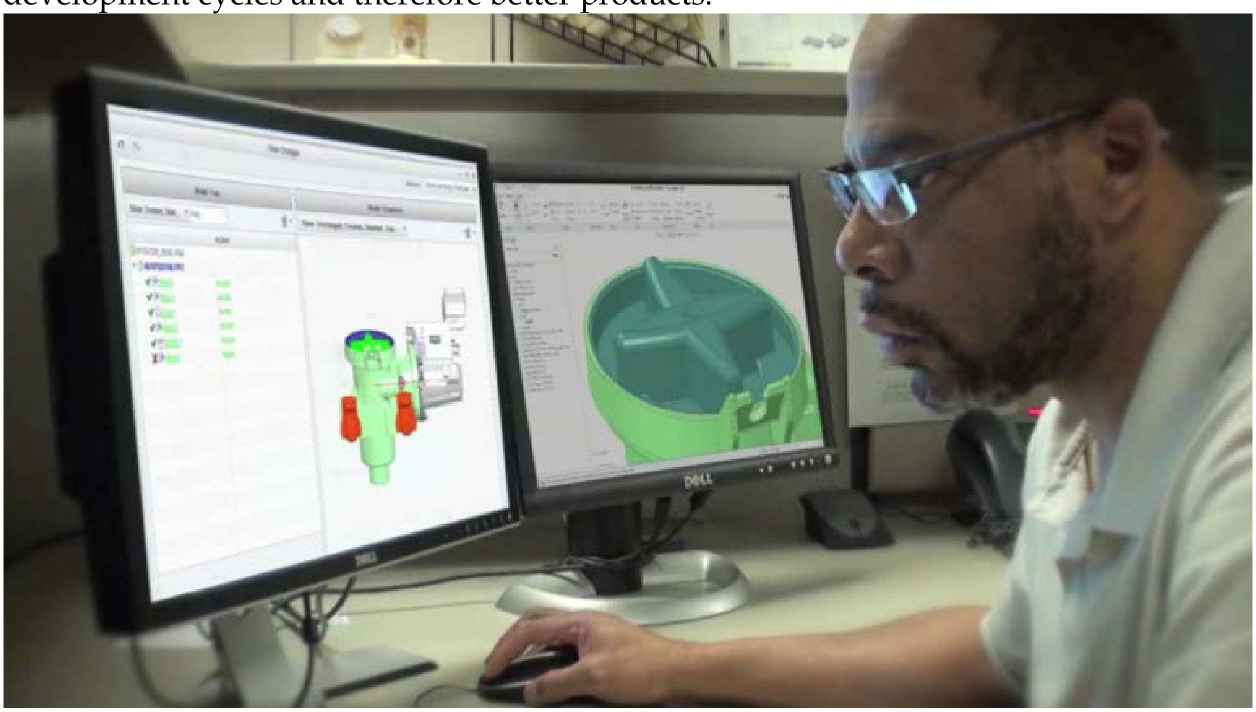
SOLID MODELING TRANSFORMED THE DESIGN OF NEW PRODUCTS

Whirlpool is considered the world's #1 major appliance company. Its annual revenue is close to $20 billion, and it employs around 70,000 employees worldwide. Whirlpool creates its products with global design teams and provides products and services in over 170 countries. Whirlpool faces stiff competition and holds itself to high standards in producing and offering products to its customers across the globe.

Whirlpool has been a PTC customer since 1986. It implemented the new solid modeling technology that PTC had developed and offered in its Pro/Engineer CAD software

(Now PTC Creo). Whirlpool's ability to design new products and deliver them to market was dramatically improved. Today, Whirlpool has standardized on the complete array of PTC product and service advantage software to support their entire range of products and services. Through the Whirlpool Constellation program all of their product data is managed throughout the product lifecycle.

Whirlpool engineers use solid modeling to create concepts, detailed product models, and production and assembly plans. Solid modeling has led to shorter product development cycles and therefore better products.



Case Study

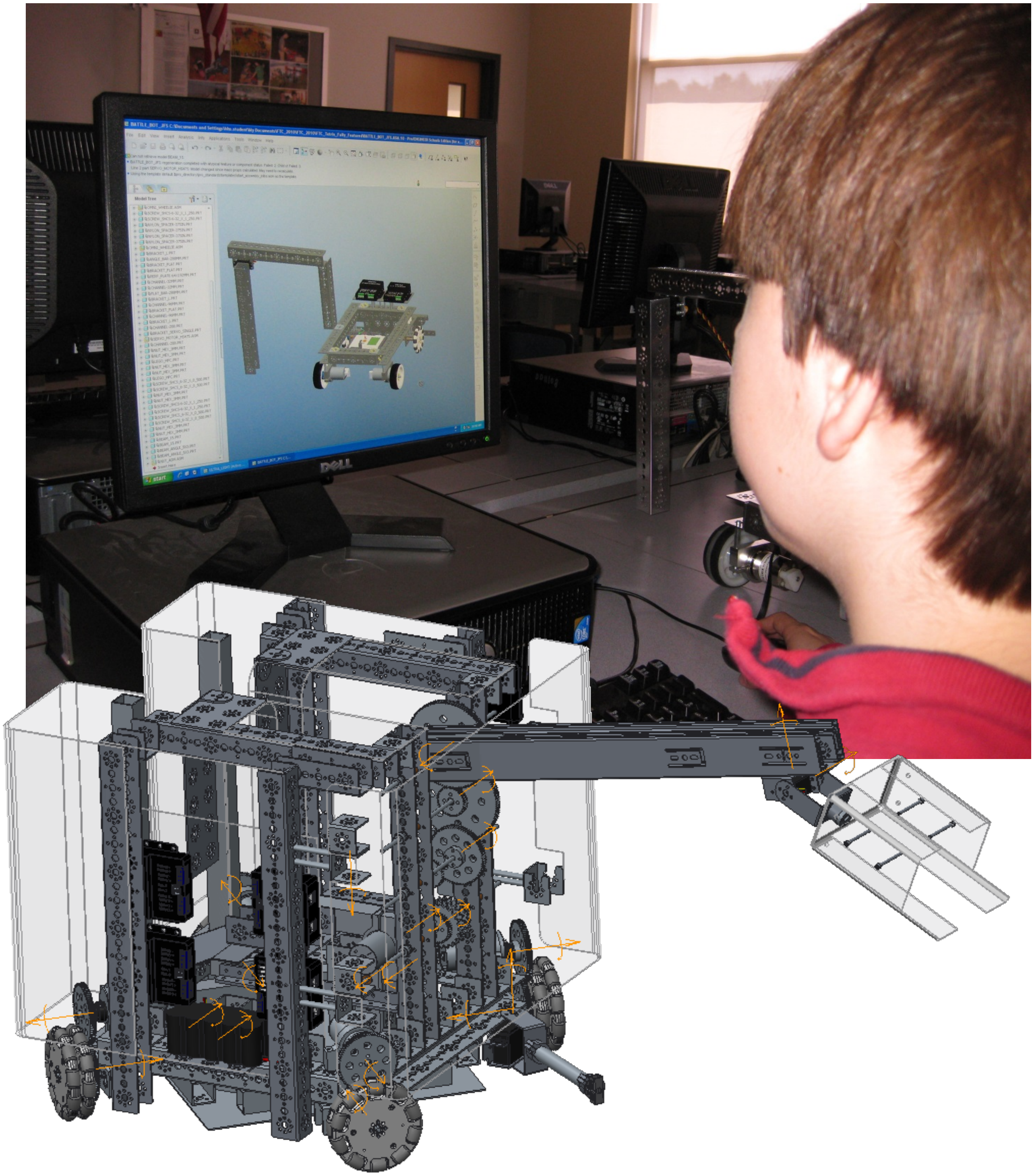
Academe

Hudson High School

TRANSFORMING VIDEO GAME SKILLS INTO PRODUCT DESIGN SKILLS

Students today are immersed in a 3D world of video games and computer animation that transforms their ability to think and visualize in the real world. Ryan Daley, a teacher at Hudson High School has embraced this strength by introducing solid modeling into the classroom. He introduces design challenges and robotics which require that the students learn and become proficient in PTC Creo.

High School students begin to use their 3D visualization skills to conceptualize and design new robots that solve specific challenges.



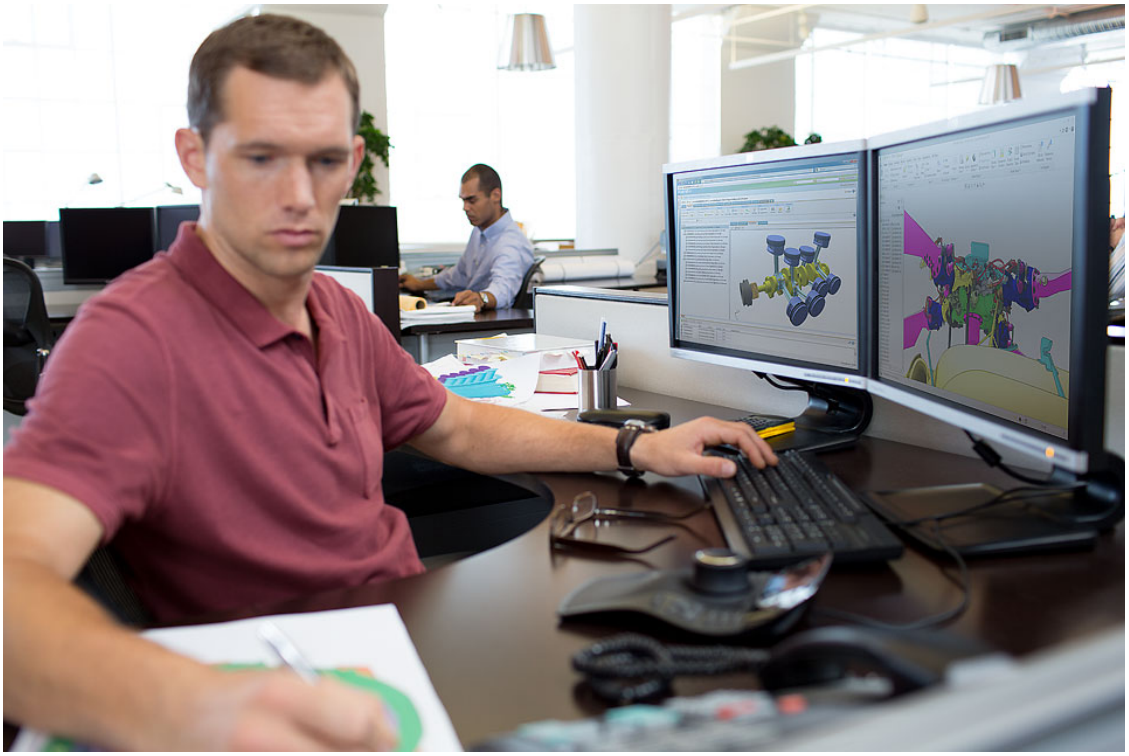
Section

1

SOLID MODELING

UNDERSTANDING HOW TO CREATE MODELS OF PARTS AND PRODUCTS

One of the greatest developments evolving out of the computer age is the development of 3-dimensional modeling of parts and products. Computer technologies have enabled the representation of 3D parts and facilitated the viewing and rendering of these models so that it is easier to create and present product concepts and bring those concepts to market. In order to fully understand this exciting field it is important to understand some important fundamentals.



3D solid models are mathematical models of real or virtual parts & products.

3D solid models are mathematical models of real or virtual parts and products. In this chapter, we will look at the anatomy or elements of solid models, how solid models are organized, and we will also introduce you to the solid modeling tool "PTC Creo".

Effective 3D solid models require careful planning. Good practices translate into models that can be used again and again to launch successful products.

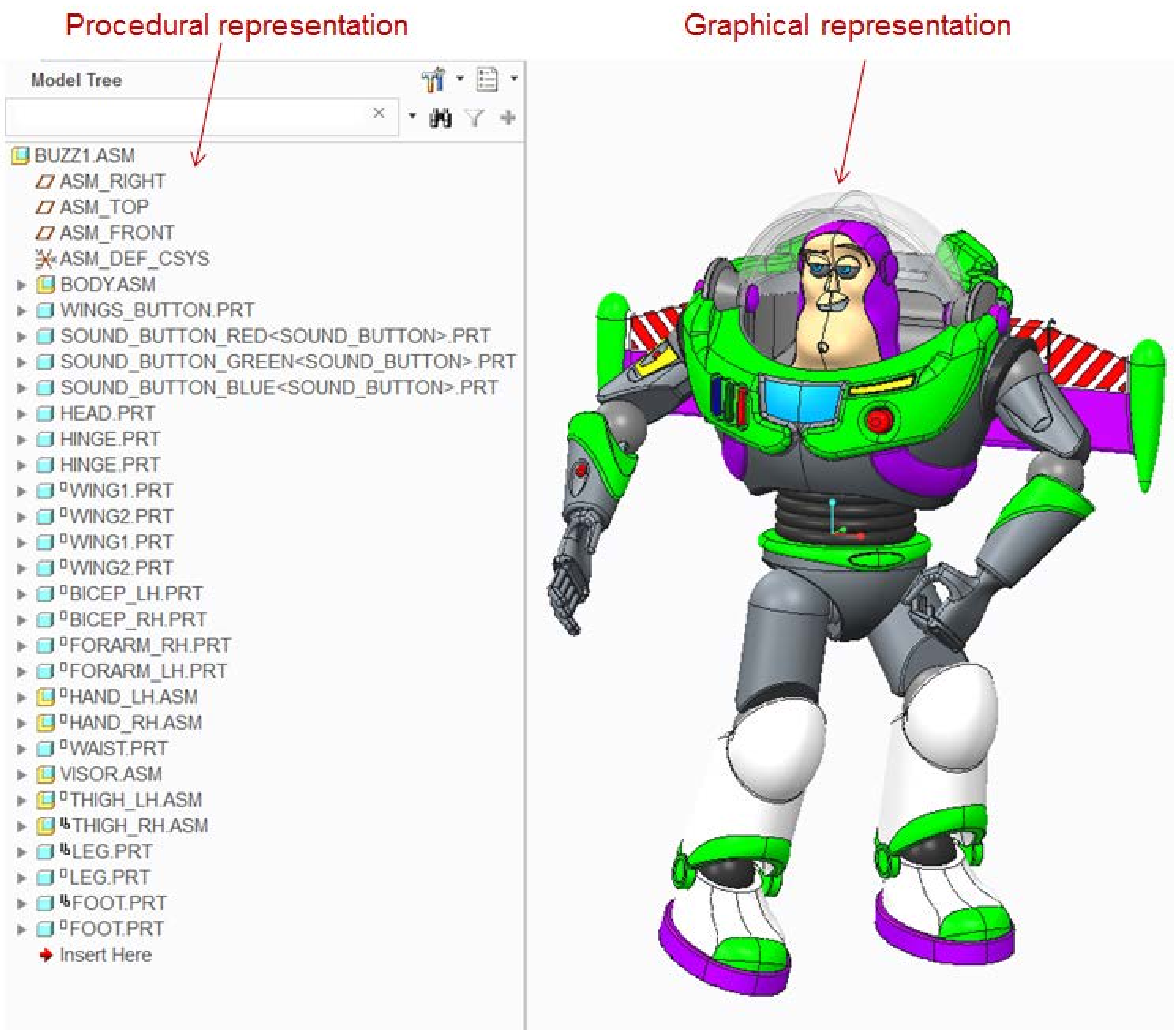
Before you begin to develop a model plan, it is important to understand what 3D solid models consist of and how they are constructed.

Characteristics of a Solid Model

3D solid models are constructed by adding and subtracting geometric shapes to create the final shape of a part or product. This use of "sums" of geometry has become a language for representing ideas for new parts and products. So let's begin by exploring the anatomy or elements of a solid model.

MODEL ANATOMY

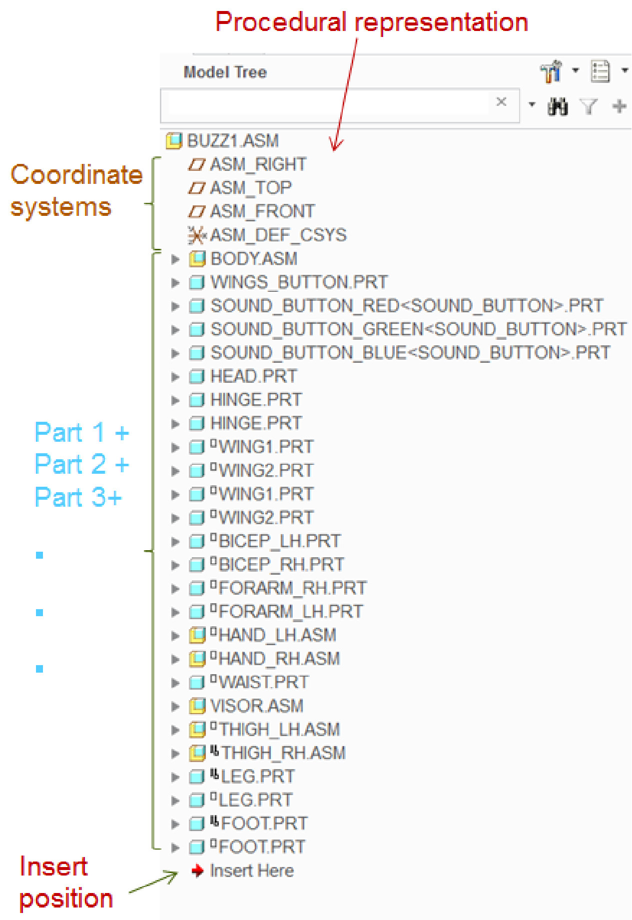
A solid model is really a sophisticated mathematical set of equations represented in software; however, you hardly ever have to work with those equations. You are able to create and modify solid models simply by working with two representations of the model; a procedural representation and a graphical representation.



The procedural representation is like a list of instructions which when followed will produce the solid model. It is called a “Model Tree”. It doesn’t look much like a tree in this view because it is just a list. However, we can look at it in a different way that makes it look very much like a tree.

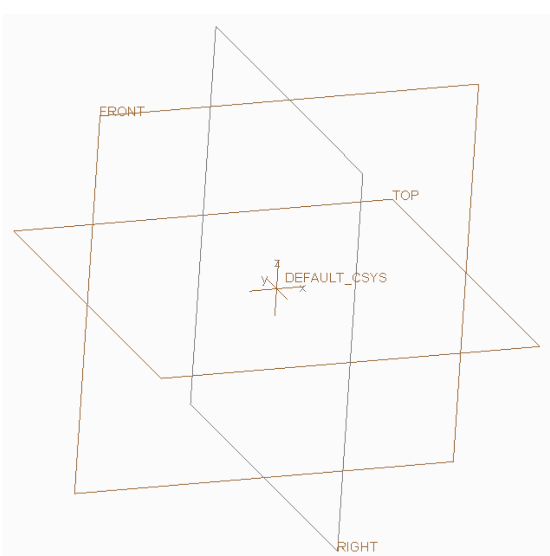
Notice that there are three sections to the model tree. The tree begins at the top with coordinate systems that are important to the model. Next is the list of all the parts in the model. It is sort of like a big sum of all the parts. Finally there is the insert location which is the place where a new part can be added. It turns out that the order in which things are added is important.

The graphical representation is what the solid model looks like. That is easy to understand. You will use both of these representations when building or modifying solid models.



COORDINATE SYSTEM

All geometry is built in reference to a global coordinate system. It is usually referred as the "origin". This is simply a point in space that is recognized as the starting point or (0,0,0) location. This point is usually represented by three crossing lines as shown. Product models may have several origins, one for each part in the assembly.



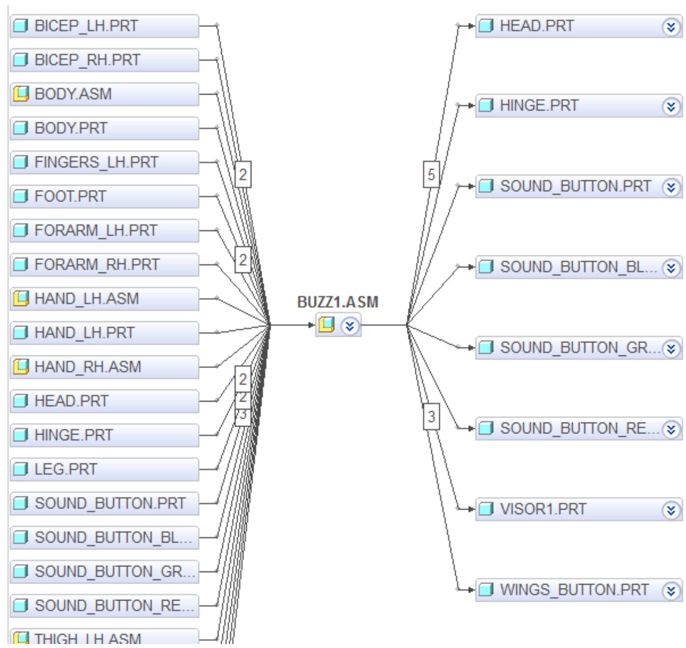
There are also other types of references that are used in solid models such as planes, axes, and points. All of these references are called "Datums" and are typical referred to specifically as "datum planes, points, and axes".

A default set of datums includes the origin coordinate system, and the three coordinate planes (FRONT: X-Z, RIGHT: Y-Z, TOP: X-Y). Axes can be defined at the intersection of any of two of these planes.

Carefully selecting what datums you use as references for creating geometry will make sure that your model is robust (doesn't break or fail) and helps insure that you will be able to create all the different aspects of your model as easily as possible. We'll show you how to plan these datums and references later and also show you how to create them in PTC Creo.

SUM OF PARTS

Most of the model tree consists of a list of parts. These parts can be single parts or sub-assemblies of parts (sort of a model inside of a model). This means that there are relationships between the parts in the list. All of the parts in a model are the children of that model. The top-level model is the parent. These relationships are referred to as Parent-Child relationships and just represent which models own what parts.

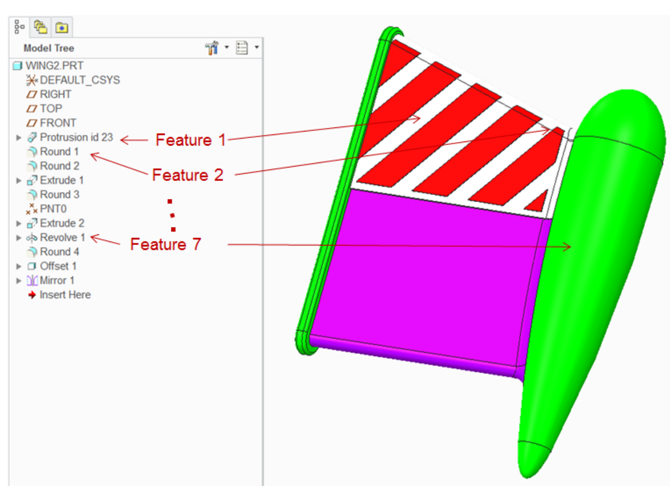


It is these types of relationships that turn the model tree from a list into a tree. Look what happens when we draw a hierarchical representation of the parent-child relationships in our model. These relationships will become important to you as you build your model and we will explain how to create them later on.

SUM OF FEATURES

So far we have talked about how assembly models are made up from a collection of parts. What about the model of a single part? It isn't a collection of parts; instead it is a collection of geometric features.

Geometric features are "chunks" of geometry that make up the model of the part. If we look at the model tree of a part model we will see these features.



Part models are different from assemblies or models of products (collections of parts). Part models can have positive and negative features. The features or chunks of geometry can add solid material or can subtract solid geometry.

Review

So let's review, there are two types of solid models; an **assembly model** which is a collection of part models for modeling products, and **part models** which are collections of positive and negative chunks of geometry called **features**.

There are two types of representations for these models; **procedural** (**model tree**) and **graphical**. We will use both in creating and modifying these models.

Finally, there are references used in the creation of the models called **datums**. These consist of **coordinate systems**, **datum points**, **datum planes**, and **datum axes**.

Exercise 1:

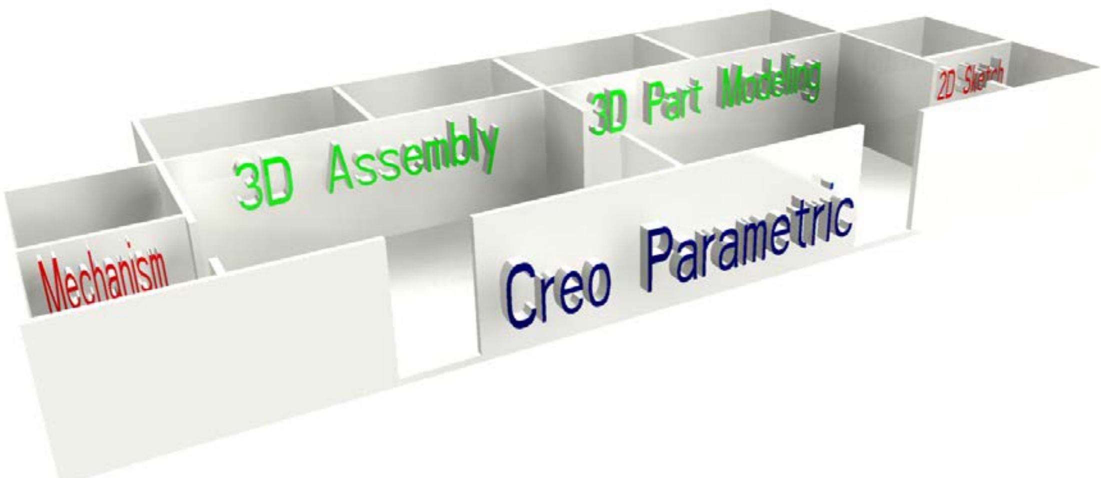
In this exercise we are going to open PTC Creo and explore the different elements of a solid model. Let's start by exploring a relatively simple model of a next generation bike.

First of all let's find each of the parts in the Model tree for the assembly and then we'll open a part and look at the features in the model tree of the part.

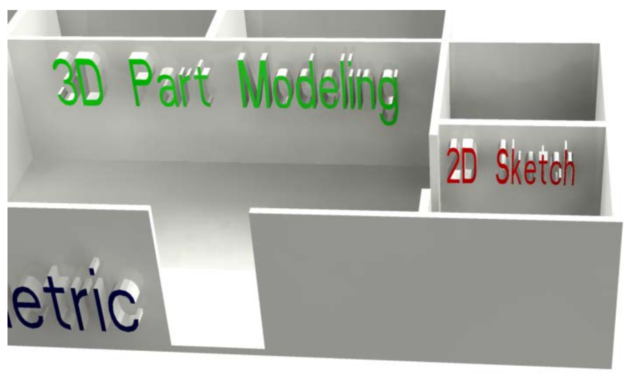
*Find in the Appendix "**Exercise 1: The Model Tree**" and follow the instructions to complete the exercise.*

THE ART OF SOLID MODELING

PTC Creo is a software tool that helps you build part models and assembly models. It is like a virtual art studio where you can build 3D solid models. There are many different modes where different functions can be completed. Within each of the modes the tool sets are different. Moving from one mode to another is important part of PTC Creo.



When creating part and assembly models, we will use the 3D Part Modeling mode and the 3D Assembly mode. But we will also use a 2D Sketch mode in order to create the sketches needed in building the conceptual model. It can be helpful to think of PTC Creo as a virtual studio made up of different rooms as shown.



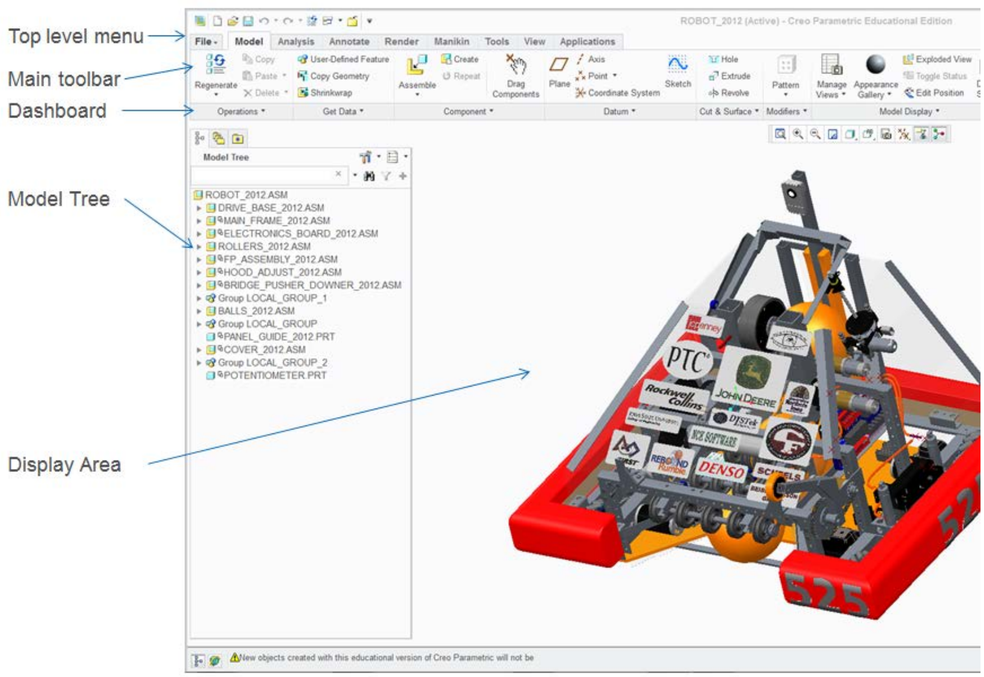
For creating parts and assemblies we will stay primarily in the 3D Part Modeling room and in the 2D Sketch room. Later on we will explore some of the other rooms.

The other unique aspect of PTC Creo is the amount of user interaction that is required in creating geometry. When you think about how you would use your hands, fingers, and all types of tools if you were creating or sculpting in an art studio, you can understand why it requires so much user interaction in creating geometry.

Because PTC Creo is software, the only means of user interaction is through the buttons and motion of the mouse and keyboard. You will find that you will use more buttons and button combinations in interacting with PTC Creo than any other program you have used.

Tour of the Creo Parametric Screen

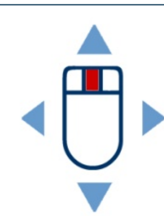
Before we start working in **PTC Creo** we should take some time to understand the user interface and methods to navigate and view models.



The **Creo Parametric** user interface is the now familiar Windows Ribbon environment. The Windows Ribbon makes it easy to navigate functions arranged in toolbars and menus. Take some time to familiarize yourself with the various menus as these will be used throughout this modeling activity.

Orientating the View in Creo Parametric

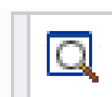
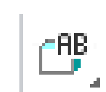
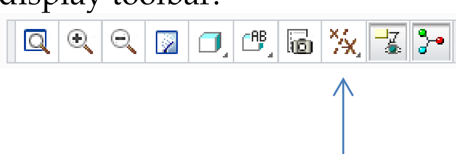
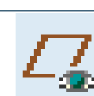
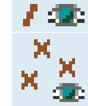
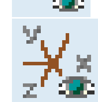
Creo Parametric offers easy-to-manipulate model views so that engineers can view their designs from different perspectives.

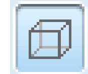
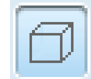
Rotate	**Hold** down the middle mouse button and move the mouse.	**Middle-Hold + Drag**	

Zoom in/out	Use the middle mouse button to scroll forward or backward. To zoom into a specific location of your model, move your mouse to that location before scrolling the mouse wheel.	**Middle-Scroll:** **Forward = Out** **Backward = In**	
Pan	**Hold Shift, then press and hold** the middle mouse button. Moving the mouse from left to right will then pan the view.	**Shift + Middle-Hold + Drag**	

DISPLAY OPTIONS IN CREO PARAMETRIC

Creo Parametric also has a number of predefined display settings and views to help you visualize your design.

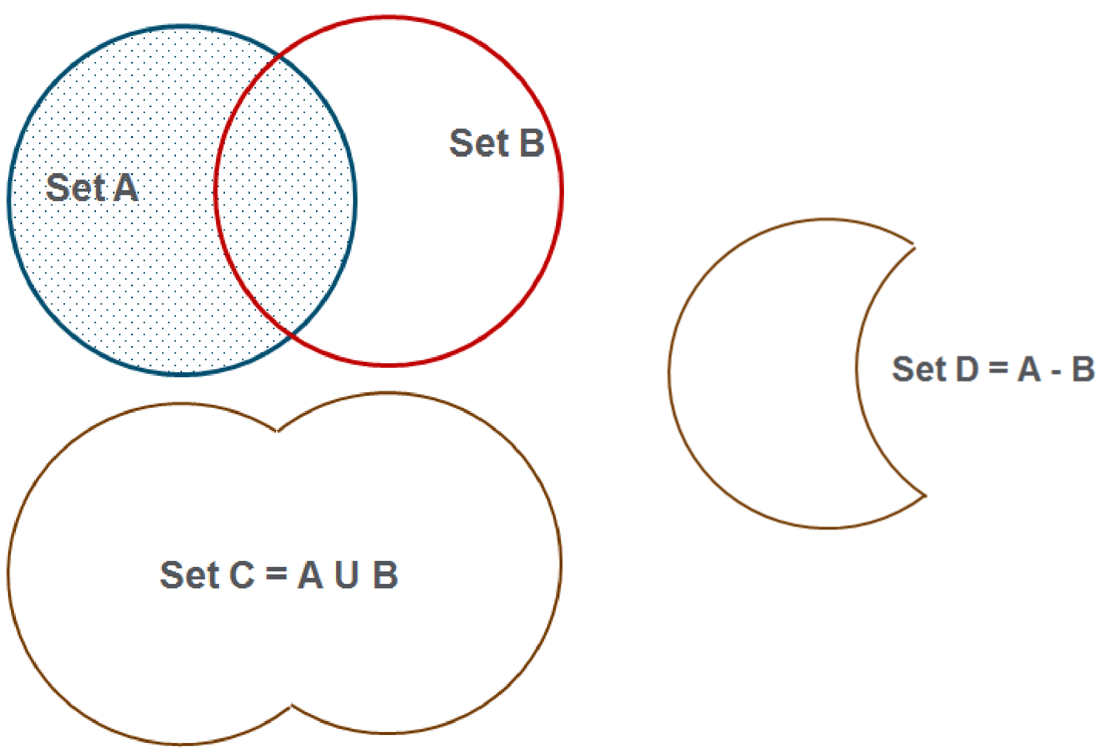
Refit Window	Click to refit your model in the main graphics area.	
Named Views	Creo Parametric has a number of predefined saved views which enable you to view your model in different orientations such as BACK, BOTTOM, FRONT, and so on.	
Datum Display	The display of datum features can be toggled on and off from the datum display toolbar. 	**Planes**  **Axes**  **Points**  **Coordinate Systems** 

Model Display	The display of your model can be quickly set from the model display toolbar. 	**Wireframe**  **Hidden Line**  **No Hidden**  **Shade**  **Real Time Render** 

This was a brief introduction to the PTC Creo user Interface. As you progress through this activity you will explore and use many more functions within PTC Creo.

SOLID MODELING OPERATIONS

As discussed previously, part models are collections of positive and negative features. Let's talk about features and how they are constructed. Features are "chunks" of geometry that can be positive or negative. They are often referred to as volumes. When constructing the model of a part, volumes are added or subtracted to create the final shape. This is much like using Venn diagrams to create set theoretic sums.



The "sets" in our solid modeling context are collections of points within a volume. Each volume represents a different set of points and these volumes can be unioned, differenced, or intersected. But first we must construct the volumes and then we can apply the set theoretic operations of union and difference.

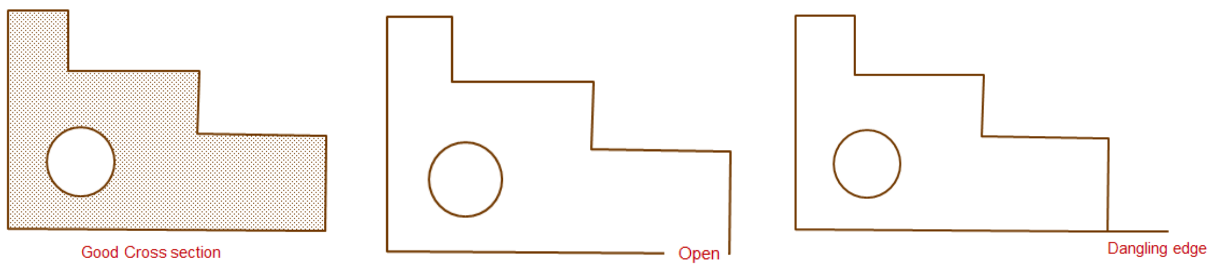
Constructing Volumes or Features

You are probably familiar with primitive geometric shapes, such as spheres, cubes, and pyramids. Since these are 3D shapes they have a defined volume. Analyzing these 3D shapes you will notice they are constructed from extending 2D sketches into 3D. This is the process we will use when creating features in PTC Creo, which will result in forming our models. There are four basic operations for doing this. Let's look at each one.

So remember first of all we have to do is draw a defined 2D sketch. However, there are special requirements for creating a robust 2D sketch which we need to follow.

- 2D sketches must be on the same plane
- 2D sketches must be closed, meaning that there are no openings in the sketch
- 2D sketches should not have any dangling edges or free floating geometry.

Let's look at each of these cases.

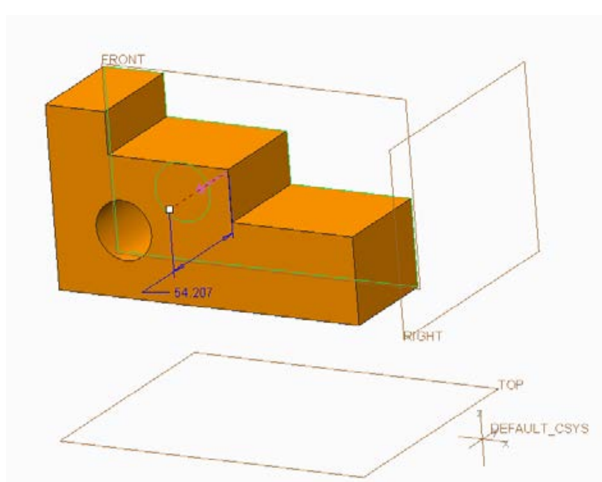


Once we have a good 2D cross section, we can do four different operations to extend them into 3D.

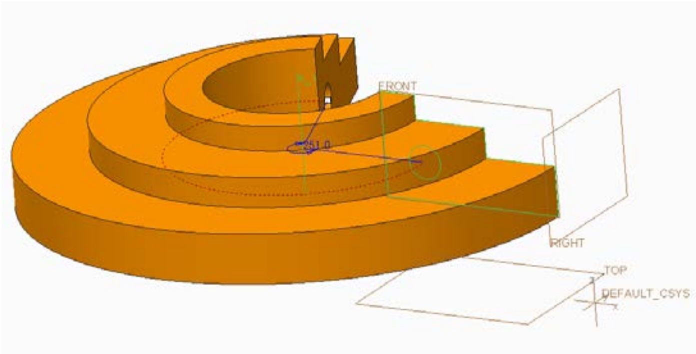
Four Operations

There are four basic operations that are used to create volumes or features.

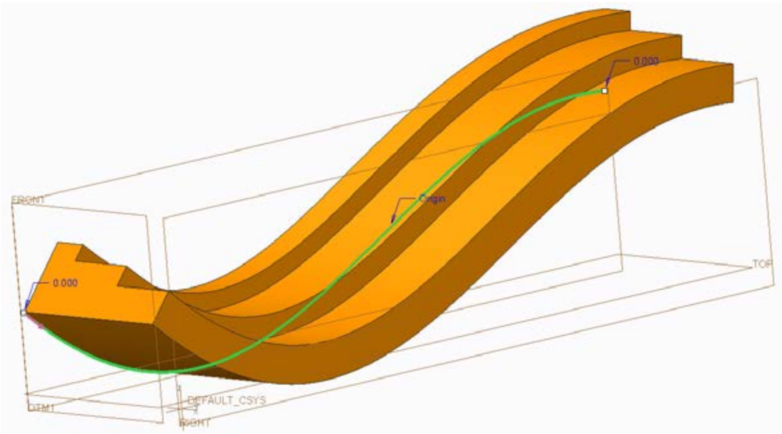
1. **Extrude** is taking a cross-section and extending it in a straight line into 3D as shown.



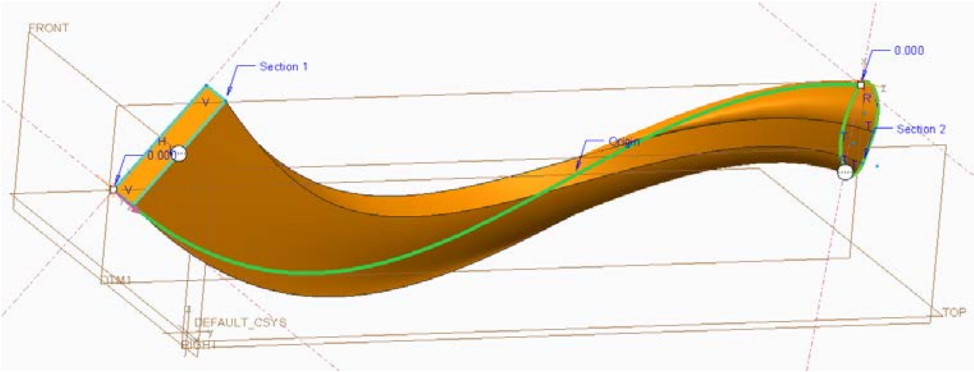
2. **Revolve** is taking the cross-section and revolving it about an axis as shown.



3. **Sweep** is taking the cross-section and sweeping it along a curve as shown.



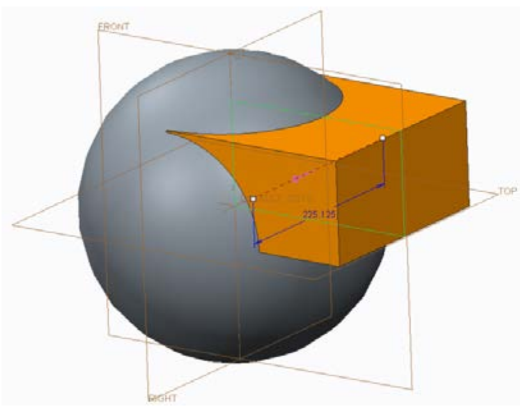
4. **Blend** is taking two or more cross-sections and blending between them along a curve.

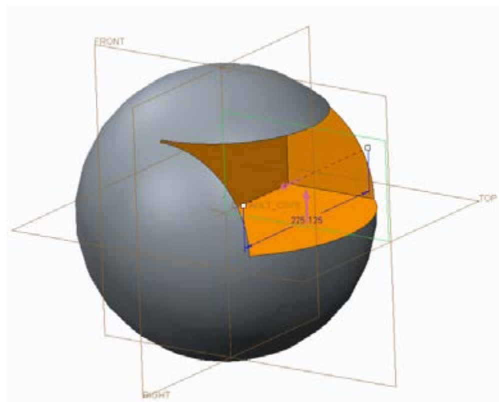


Using these four operations it is possible to construct a significant number of 3D models. There are other additional operations, but since these four can represent almost all types of 3D parts, we will work with these.

BOOLEAN ADDITION AND SUBTRACTION

Once you have created a volume, you can then create a second volume and add or subtract it from the first one to create a new shape. This can continue volume after volume until you have created the final shape of the part you desire. For example, here is a sphere and a rectangular cube. In the first situation they are added together. In the second the rectangular cube is subtracted from the sphere.





Therefore, using the four operations for creating volumes and then adding or subtracting them allows for a great deal of diversity in the 3D models.

Now let's explore a new assembly model and discover how each of the parts in the model were made using the different solid modeling operations we have learned about.

This assembly model is of a robot and is made of three parts (the arms are one model used twice).

Find in the Appendix **"Exercise 2: Solid Modeling"** *and follow the instructions to complete the exercise.*

Creating Product Models

Now that we know about part models, let's learn about assembly models. Assembly models as we discussed before are collections of part models and are used to represent products.

Assembly Operations

When constructing a 3D assembly of parts, we use a different set of operations to construct the product model. For example, we don't use Boolean adds or subtracts since we don't want to subtract parts in the assembly. Instead we use assembly constraints that determine how parts relate one to another. There are many different types of constraints, we will present a few.

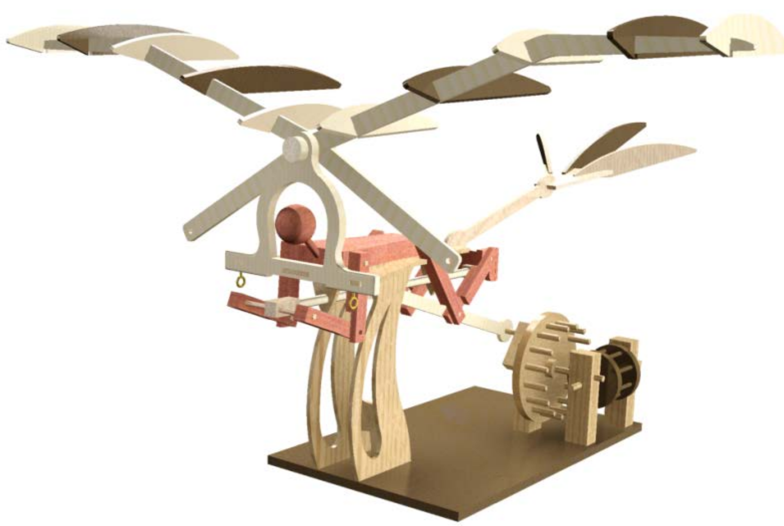
When creating an assembly the first part must be grounded or placed in reference to the default coordinate system. This is usually done with a **default** constraint.

When the next part is brought into the assembly, it must be placed in reference to the first part. In this example the neck is placed within the neck hole in the body part using a **coincident** constraint to align the axes and then another **coincident** constraint to identify where along the axis the neck must rest.

Each additional part must be assembled using constraints that locate it with respect to the existing parts until the entire assembly is finished.

There are other types of constraints that allow parts to move with respect to each other. We call these types of constraints "**Kinematic**" constraints. In order for the arms and legs of this wooden man to move with respect to each other, we must use kinematic constraints.

Once we have our product model assembled, we can analyze the characteristics of the model. For example, we can calculate its total weight, volume, size, etc. We can also cause the parts to move and expose them to virtual forces.



To finish this chapter, let's explore an assembly model and find all the different elements that we have talked about. This model is of Davinci's ornithopter design.

*Find in the Appendix "***Exercise 3: Model Analysis***" and follow the instructions to complete the Exercise.*

Section

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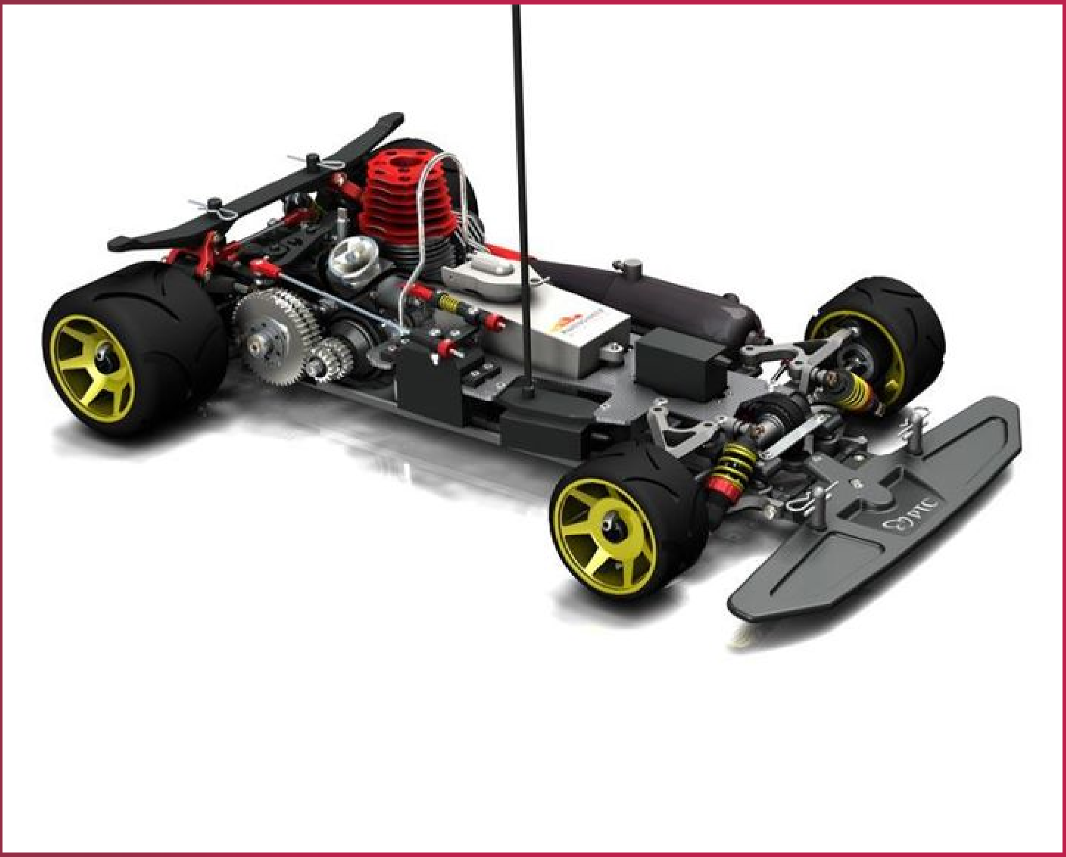
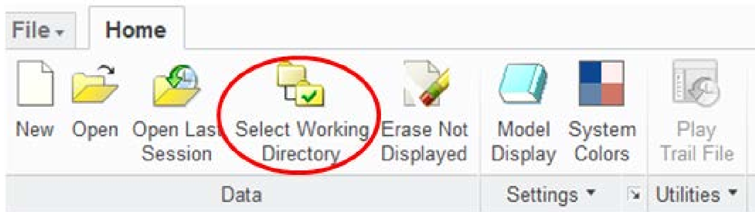
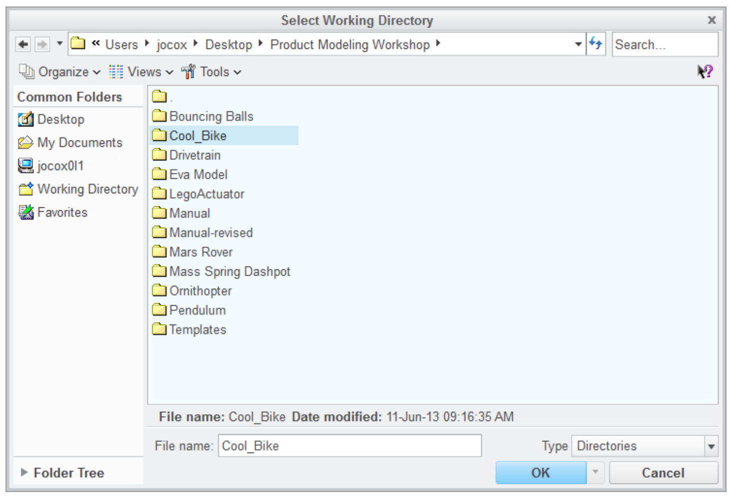
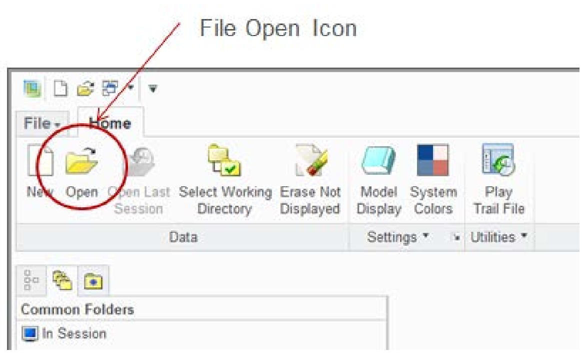
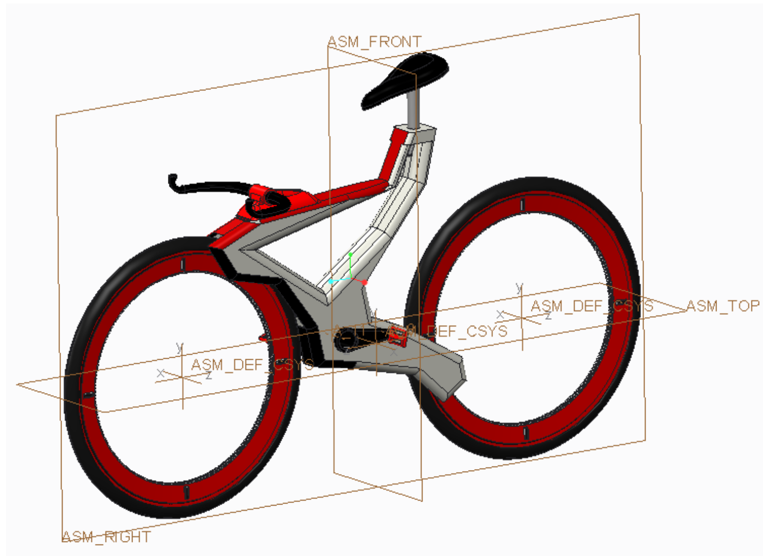


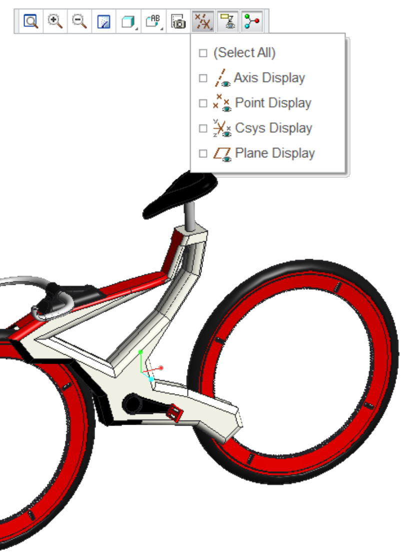
APPENDIX

EXERCISE 1: THE MODEL TREE

Open **Creo Parametric 2.0** and click the icon called **Select Working Directory** to set the folder in which you will be working.	
Navigate to the folder called "Cool_Bike" and click **OK**.	
Now click on the **Open** icon to open the file: **"cool_bike_top_assembly.asm"**. Double click on the file or select it and click **Open**.	

<table>
<tr>
<td>When the file opens you will see the cool bike model and its associated datum planes.</td>
<td>
</td>
</tr>
<tr>
<td>Notice the reference planes displayed on the model. These planes are called reference datums and are used during part and assembly modeling.

Find all 9 of the datum planes and turn the display off. Clicking on the Datum Display Filters icon, uncheck the box beside 'Select All'.

We can easily toggle the display of the datum during modeling.</td>
<td>
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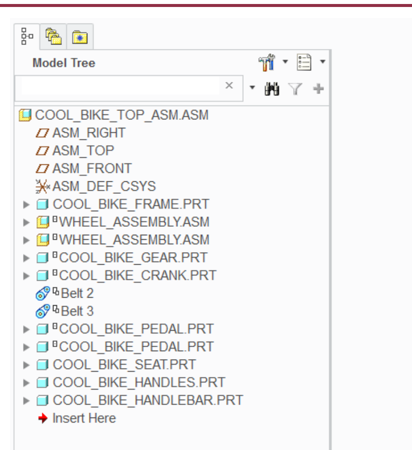
<tr>
<td>Now let's examine the model tree.

Look at the left side of the screen and locate the Model Tree.</td>
<td>
</td>
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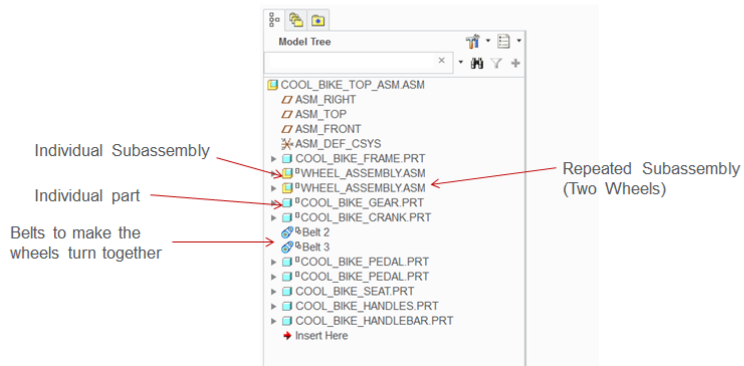
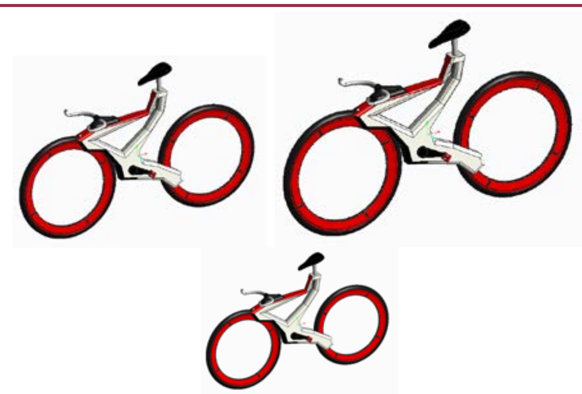
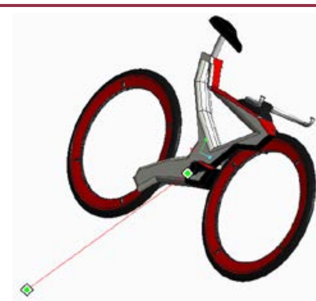
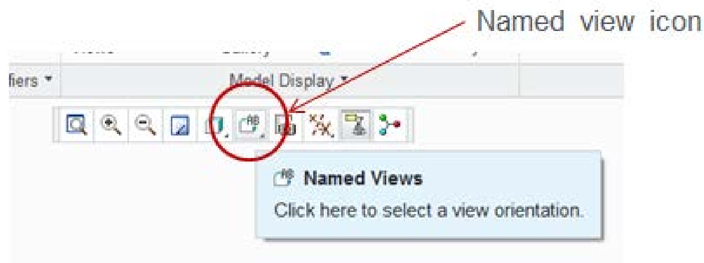
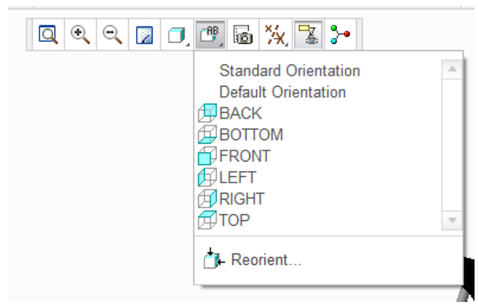
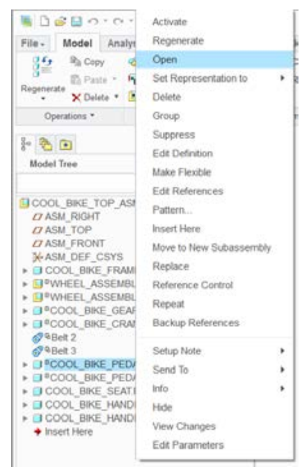
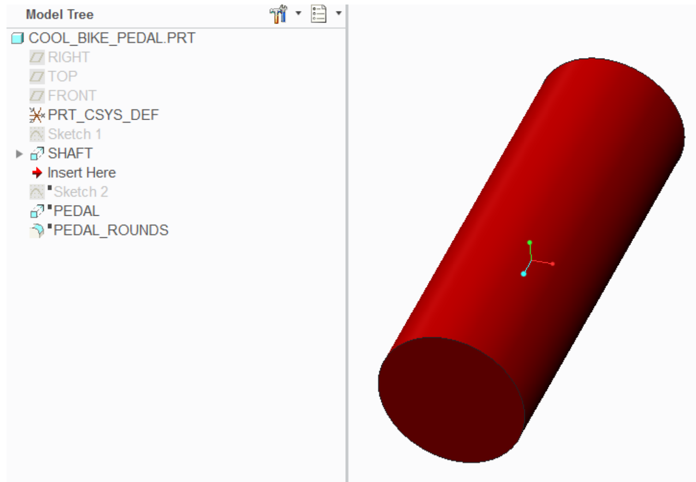
</table>

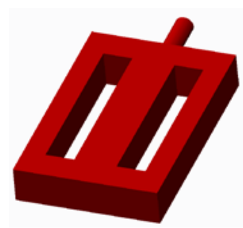
How many total parts are there? How many parts are repeated?	
Now let's try zooming, rotating and panning the model. Start by scrolling the middle wheel of the mouse. It will cause your bike model to zoom in and out. 	
Next try rotating or spinning your model by pushing and holding the middle mouse button down while moving your mouse. 	
Panning is done by holding down both the **Shift** key and the middle mouse button while moving your mouse. 	

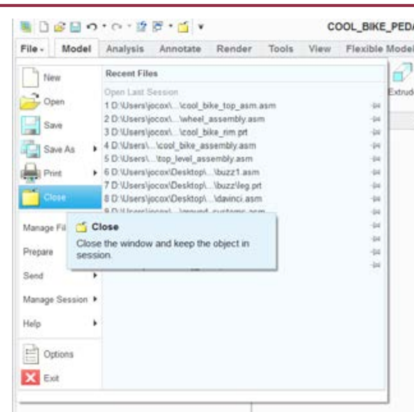
There are some pre-defined views that you can use during modeling. To access them simply click the **Named Views** icon.	
You can select any of these predefined views and **Creo** will orient the model accordingly.	
Now let's explore one of the parts in this assembly model. Find the **COOL_BIKE_PEDAL** in the **Model Tree**. Right-click on it and select **Open**.	
Notice that you are now looking at the pedal model.	

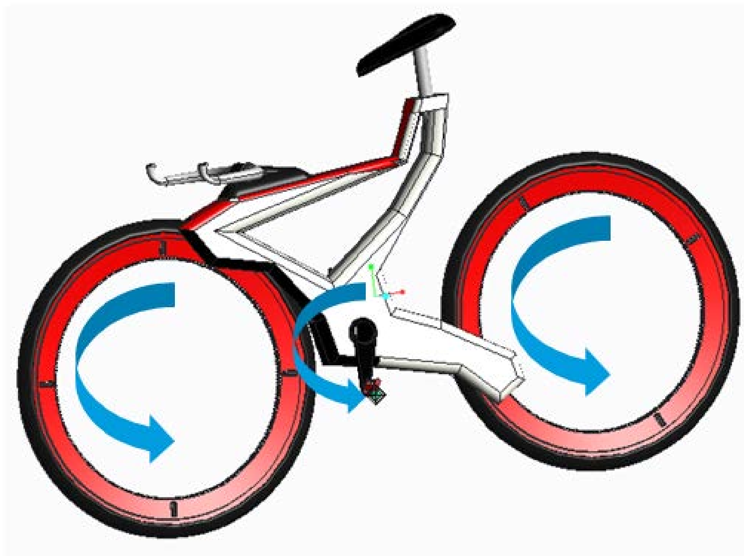
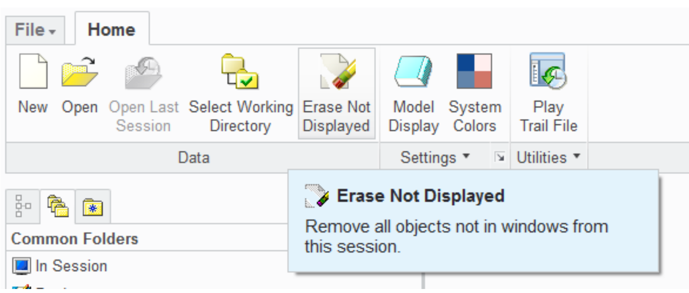
<table>
<tr>
<td>Let’s explore the actual features used to build this model one by one.

Left-click and hold the red arrow next to “Insert Here” in the Model Tree and drag it up under SHAFT.

The first feature in this model is the shaft.</td>
<td>
</td>
</tr>
<tr>
<td>Next, move the red arrow below the next feature “PEDAL”.

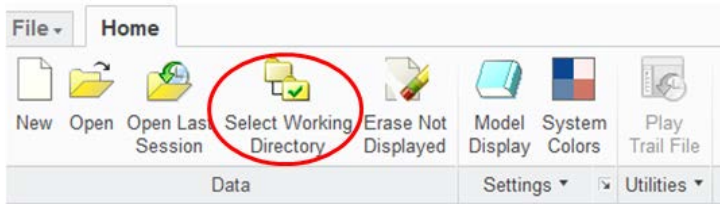
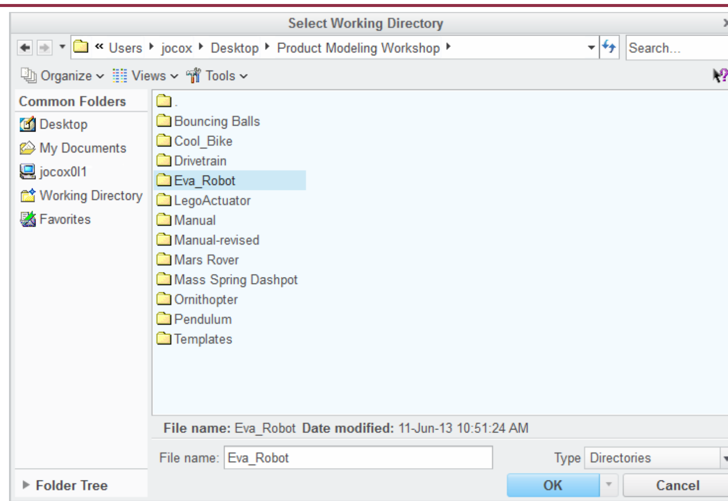
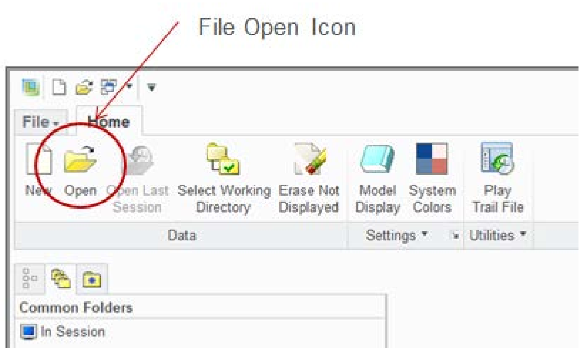
You can see that the next feature is a positive volume made from an extrude feature.</td>
<td></td>
</tr>
<tr>
<td>Finally move the red arrow below “PEDAL_ROUNDS” which is a special operation used to round the edges of parts.

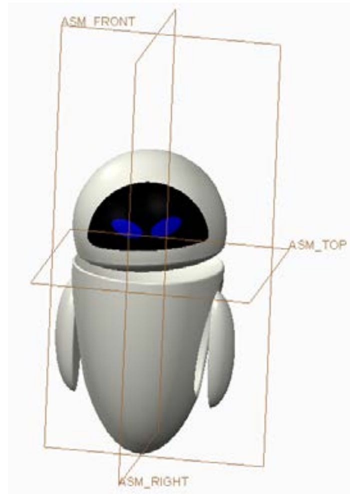
Notice how the model plan is captured by the model tree.</td>
<td></td>
</tr>
<tr>
<td>Close this model by clicking on the File tab in the top left of the screen and selecting Close.</td>
<td>
</td>
</tr>
</table>

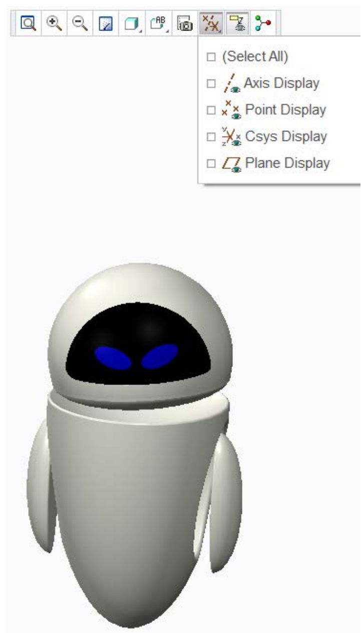
We’re back to the assembly model of the bike. Depending on how the parts were assembled, if kinematic constraints had been used, parts can move with respect to each other. To make them move, we use a special combination of keys. Hold down the **CTRL-ALT** button on the keyboard and then left-click and hold the pedal in the graphics screen. Dragging your mouse, you will notice that the wheels turn.	
Now to finish this exercise, close the model and erase it from the memory of your computer. Click on the **File** tab in the top-left of the screen and select **Close**. Now click on the **Erase Not Displayed** tab and then click **OK** to erase the files from memory. (Don’t worry the files are still on your disk)	

Congratulations You have finished this exercise

EXERCISE 2: SOLID MODELING

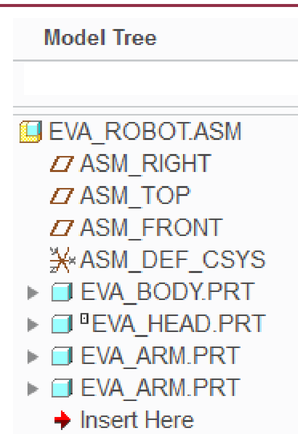
Open **Creo Parametric 2.0** and click the icon called **Select Working Directory** to set the folder in which you will be working.	
Navigate to the folder called "**Eva_Model**" and click **OK**.	
Now click on the **Open** icon to open the file: "**eva_robot.asm".** Double click on the file or select it and click **Open**.	

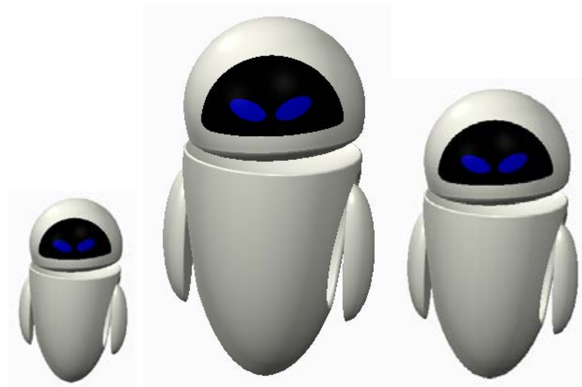
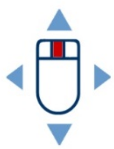
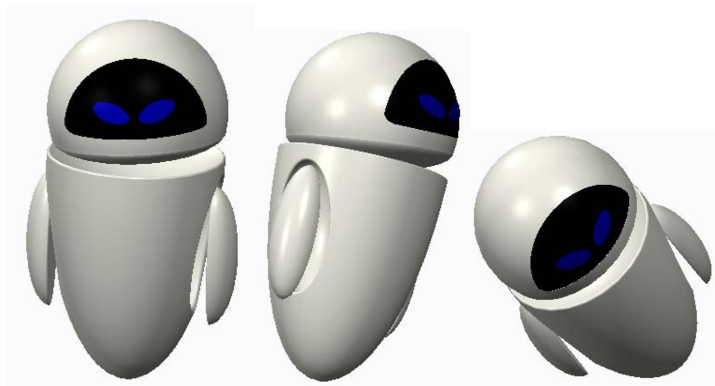
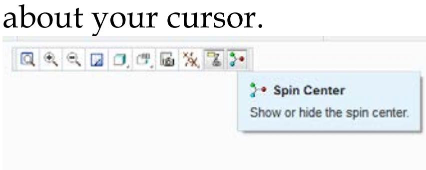
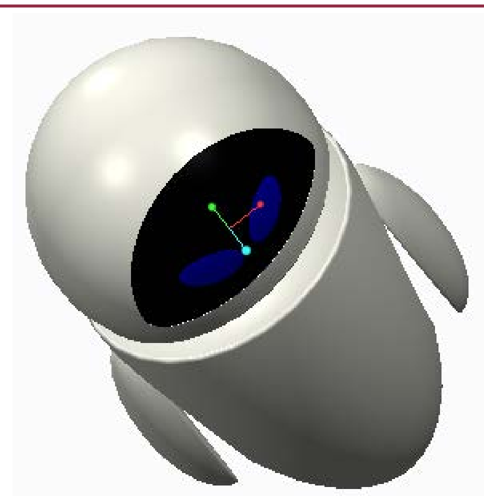
<table>
<tr>
<td>Notice the reference planes displayed on the model. These planes are called reference datums and are used during part and assembly modeling.</td>
<td>
</td>
</tr>
<tr>
<td>Turn the datums off by clicking on the Datum Display Filters icon and uncheck the box beside 'Select All'.

We can easily toggle the display of the datum during modeling.</td>
<td>
</td>
</tr>
<tr>
<td>Now let's examine the model tree.

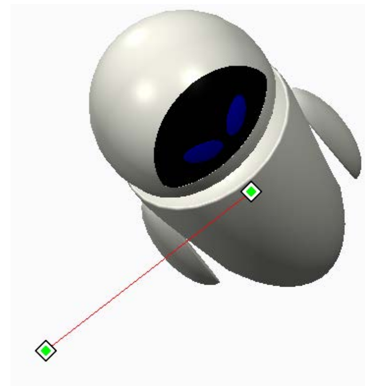
Look at the left side of the screen and locate the Model Tree.

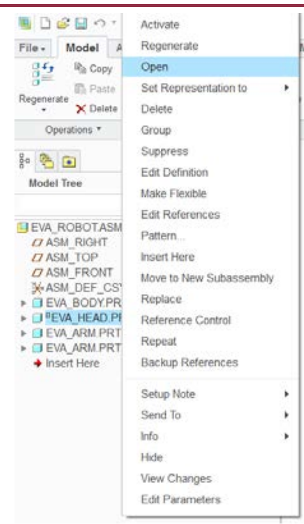
How many total parts are there?

How many parts are repeated?</td>
<td>
</td>
</tr>
</table>

Now let's review zooming, rotating and panning the model. Start by scrolling the middle wheel. It will cause your model to zoom in and out. 	
Next try rotating or spinning your model by pushing and holding the middle mouse button down while moving your mouse. 	
In the center of the graphics screen is a small coordinate system called the "**Spin Center**". It defines the center of rotation. You can toggle it off by clicking the Spin Center icon in the view menu and then the model rotates about your cursor. 	

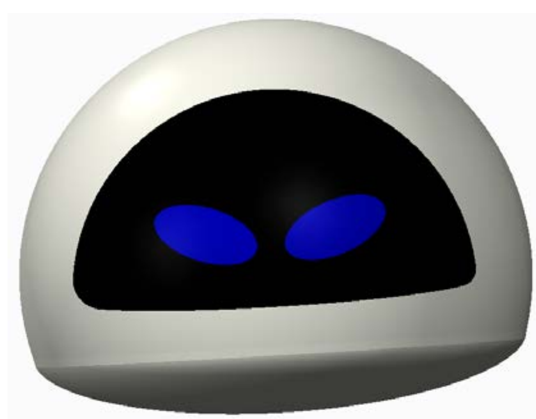
<table>
<tr>
<td>Panning is done by holding down both the Shift key and the middle mouse button while moving your mouse.

</td>
<td></td>
</tr>
<tr>
<td>Now let’s explore each of the parts of this model.

Start by locating the head in the Model Tree. Right-click on “EVA_HEAD.PRT” and select Open.</td>
<td>

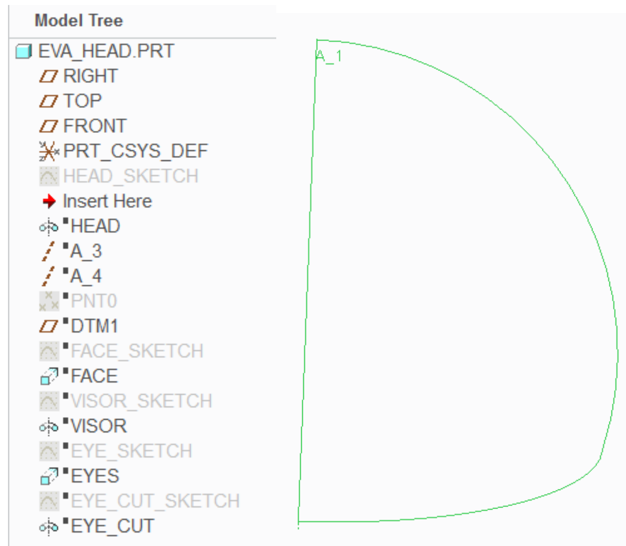
</td>
</tr>
<tr>
<td>Eva is staring right at you!

Let’s explore how the part was made.

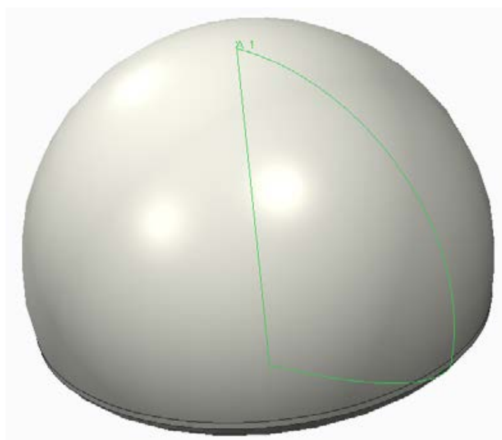
In the Model Tree, drag the red arrow next to “Insert Here” below HEAD_SKETCH.</td>
<td></td>
</tr>
</table>

Since HEAD_SKETCH is hidden, the graphics screen appears blank. Clicking on it in the model tree will display the cross-section sketch of the head.

Move the red arrow below the **HEAD** revolve feature and you will see the result of the revolve operation.



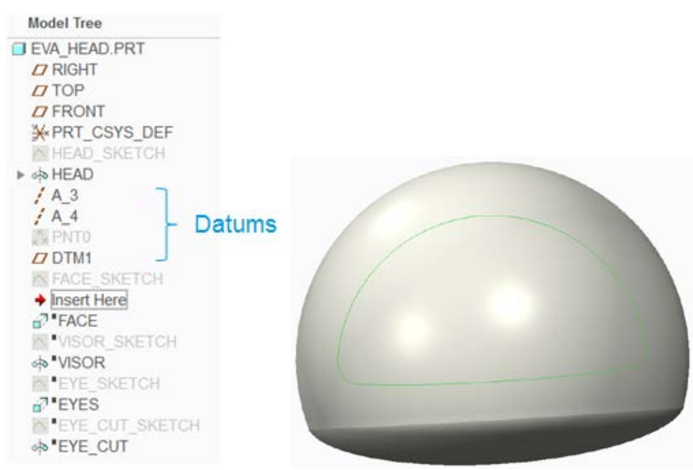
This is a positive solid or volume created by rotating or revolving the cross section around the axis **A_1**.

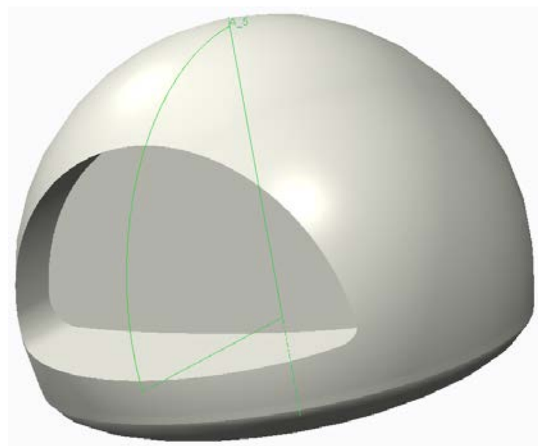
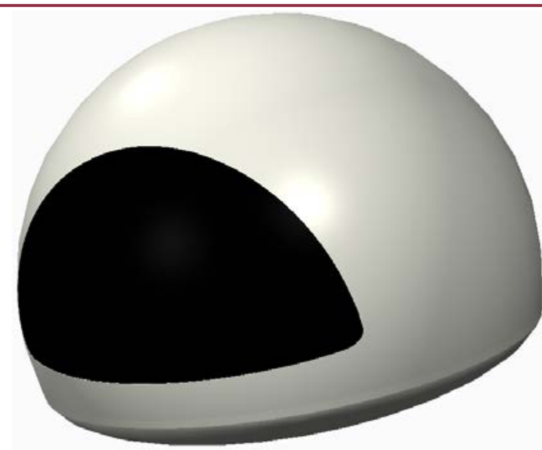


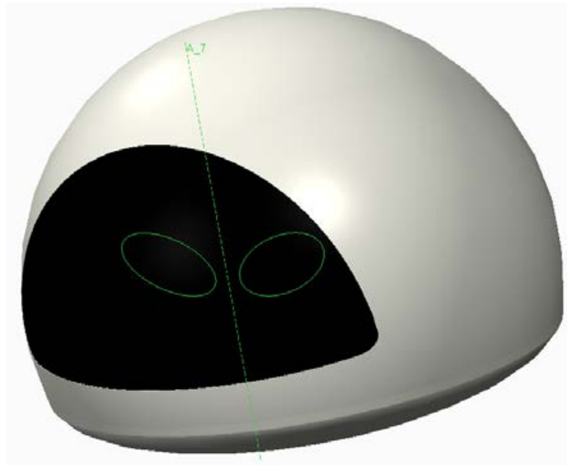
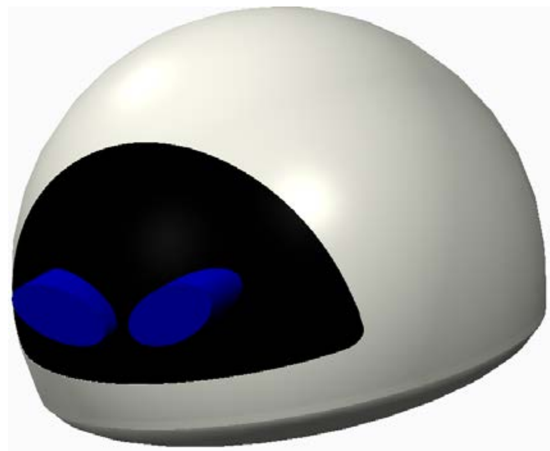
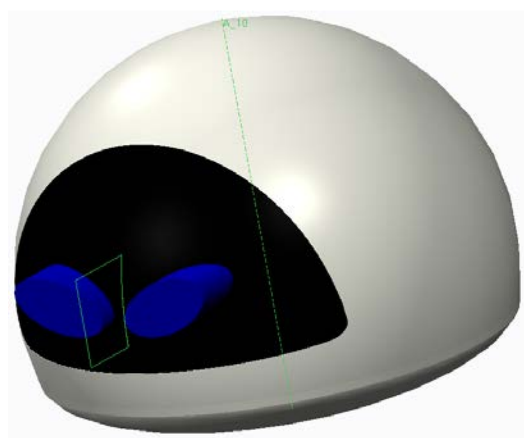
Now move the red arrow below **FACE_SKETCH**. Then click on **FACE_SKETCH** to see the cross section sketch.

In the model tree, you will notice several hidden datums. These features are hidden to keep the model display clean. They can be unhidden at any time.

This is typical once a model is completed.

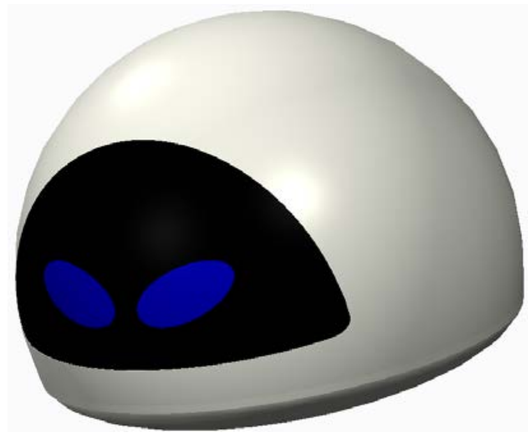


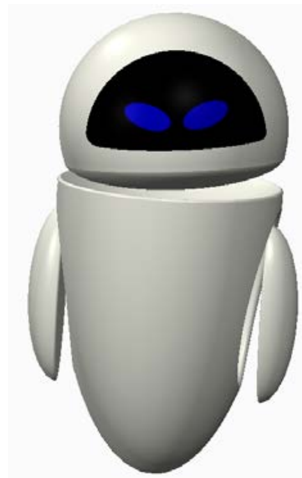
Next, move the red arrow below “**FACE**”. You can see that the next feature is a negative volume made from an extrude feature. You may need to rotate your model a little to see the cut out.	
Now move the red arrow below “**VISOR_SKETCH**” and then click on **VISOR_SKETCH**. Once again notice that it is a cross section with a defined axis. A revolve is coming… Move the red arrow below **VISOR**.	
This revolve created Eva’s visor. Now move the red arrow below **EYE_SKETCH** and then click on **EYE_SKETCH**.	

In this case, you can see the cross sections for Eva's eyes. While there is an axis created within the sketch, we will use an extrude feature to create the eyes. Move the red arrow below **EYES**.	
We now have Eva's eyes. The last operation must trim the eyes flush with the visor. Move the red arrow below **EYE_CUT_SKETCH** and click on **EYE_CUT_SKETCH**.	
Notice that there is a cross section and an axis. The cross section will be revolved just enough to trim the eyes. Move the red arrow below **EYE_CUT**.	

<table>
<tr>
<td>Eva’s head is now complete.

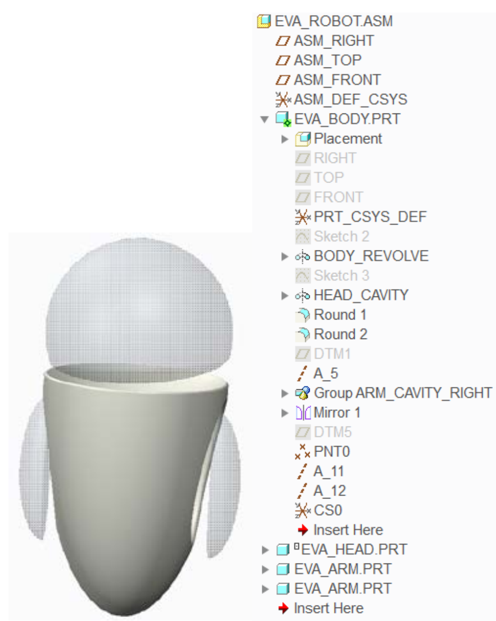
Let’s close this model and go back to the assembly model.

Click on the File tab and choose Close.</td>
<td></td>
</tr>
<tr>
<td>Now let’s explore how the body was made, but this time we will do it without leaving the assembly model.

Right click on “EVE_BODY.PRT” and click Activate.</td>
<td></td>
</tr>
<tr>
<td>Notice that all the other parts are grayed out. The body is now the active part.

Let’s explore how it was created.

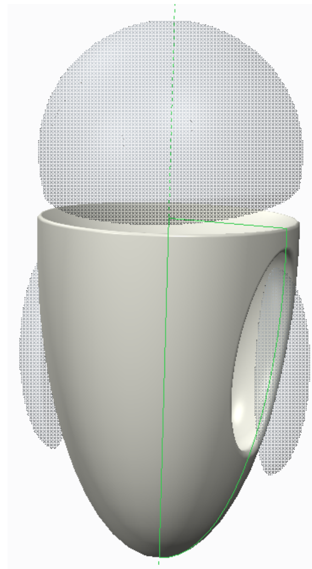
Start by expanding the model tree under EVA_BODY.PRT by clicking on the triangle just left of it.

Notice that all of the features are displayed in the model tree.</td>
<td>
</td>
</tr>
</table>

Note: The red arrow will not move in this mode. It is only accessible in part mode.

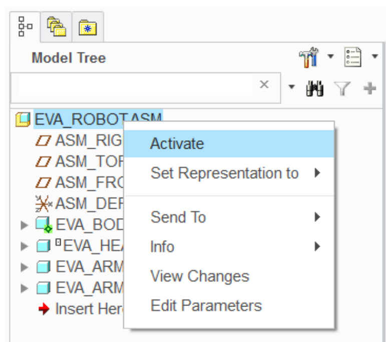
However, you can still click on the sketches to see each one.

Step down through the model by clicking on the sketches in sequence to see how the body was made.



Double click the **EVA_BODY.PRT** in the model tree to collapse the part feature tree.

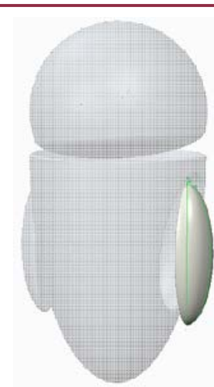
Right-click on the top level assembly, **EVA_ROBOT.ASM** and select **Activate** to reactivate the entire assembly again.

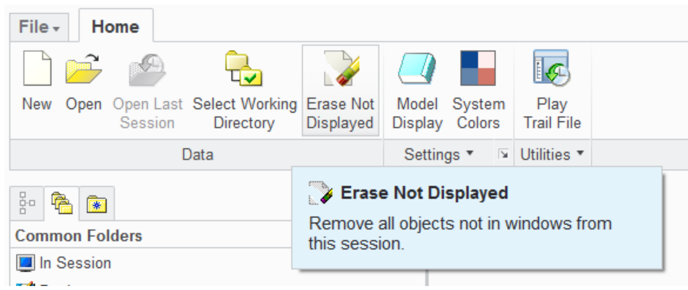


Now explore the arms by either opening up the part model or activating the part model.

Review the features.

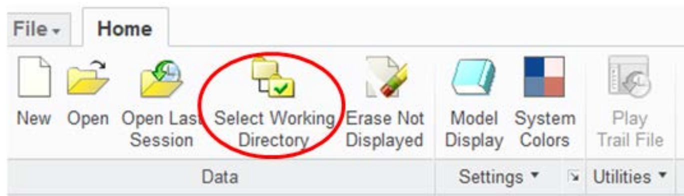
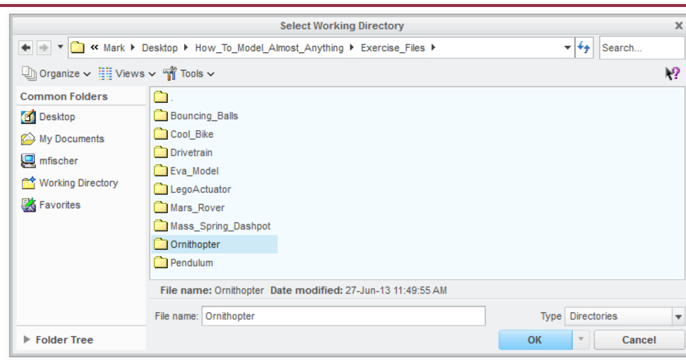
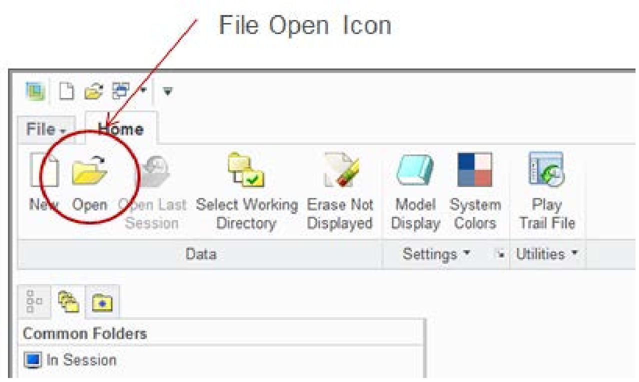
How was the part created?



Now to finish this exercise, close the model and erase it from the memory of your computer. Click on the **File** tab in the top-left of the screen and select **Close**. Now click on the **Erase Not Displayed** tab and then click **OK** to erase the files from memory. (Don't worry the files are still on your disk)	

Congratulations You have finished this exercise

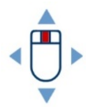
EXERCISE 3: MODEL ANALYSIS

<table>
<tr>
<td>Open Creo Parametric 2.0 and click the icon called Select Working Directory to set the folder in which you will be working.</td>
<td>
</td>
</tr>
<tr>
<td>Navigate to the folder called “Ornithopter” and click OK.</td>
<td>
</td>
</tr>
<tr>
<td>Now click on the Open icon to open the file: “top_level_assembly.asm”. Double click on the file or select it and click Open.</td>
<td>
</td>
</tr>
</table>

Now that the assembly is open, you will notice that all of the datums in this model are hidden.

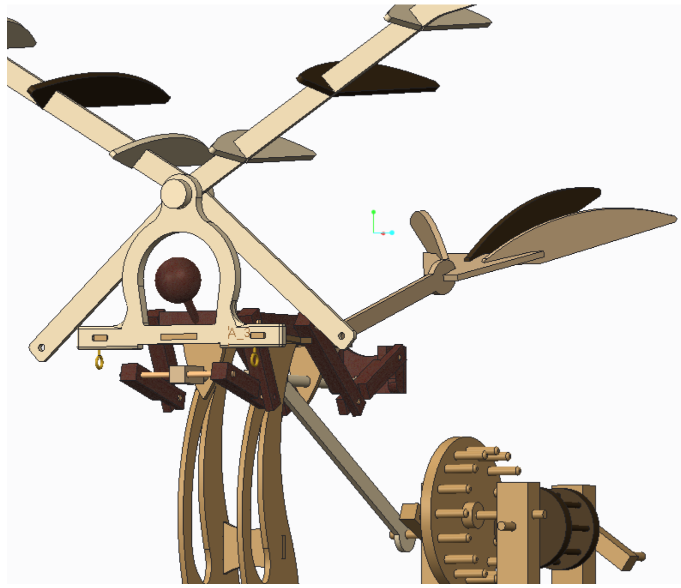
Explore the ornithopter by zooming, rotating and panning.

Zoom



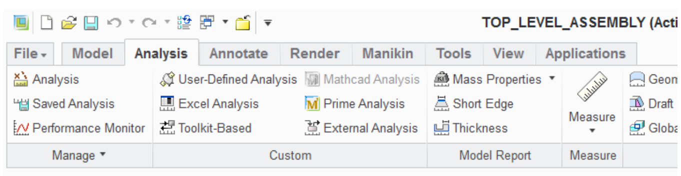
Rotate

Pan



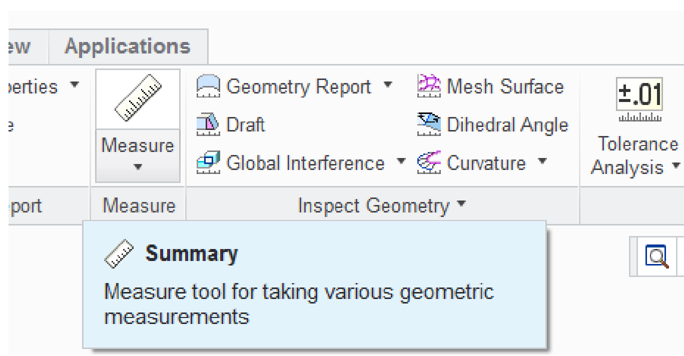
Once a model has been completed, we can analyze it in many ways.

Let's start by clicking on the **Analysis** tab in the top menu.



Next, let's do some measuring. Click on the **Measure** icon in the Analysis ribbon.

A dialog box will appear that we can use to measure different aspects of your model.

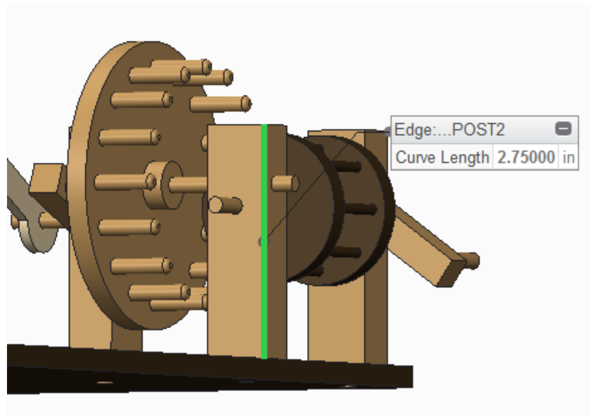


Notice that we can measure length, distance, angle, diameter, surface area, and volume.

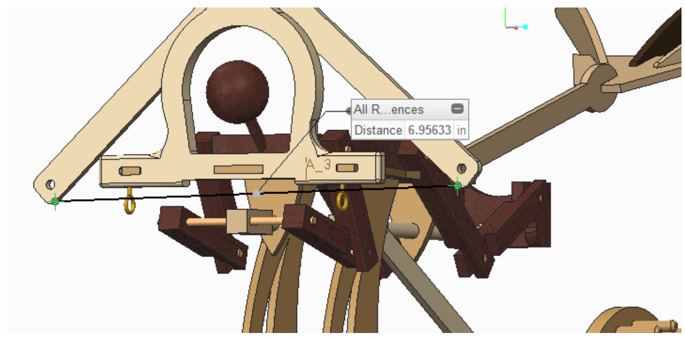
Let's try measuring some of these quantities.

Click on the length measure tool and then click on the edge of one of the posts to find out its length.

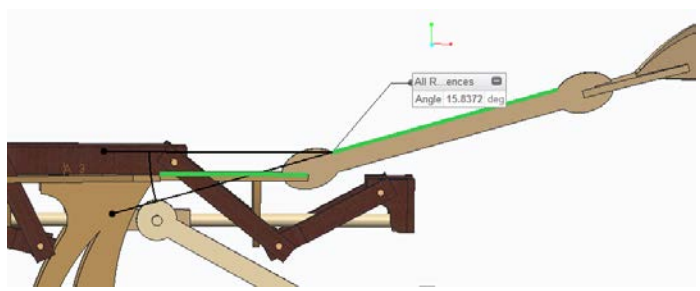
Note: Draging your mouse over your model will pre-highlight different elements to allow you to select the entity you want.



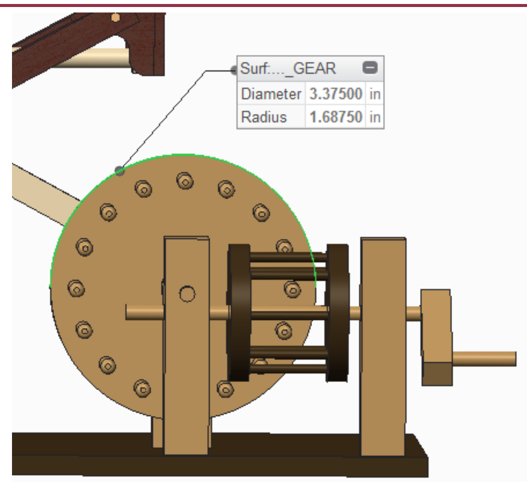
Now click on the distance tool and select two points by clicking the first point and then holding the **CTRL** button down on the keyboard and selecting the second point. This will show the distance between the two points.

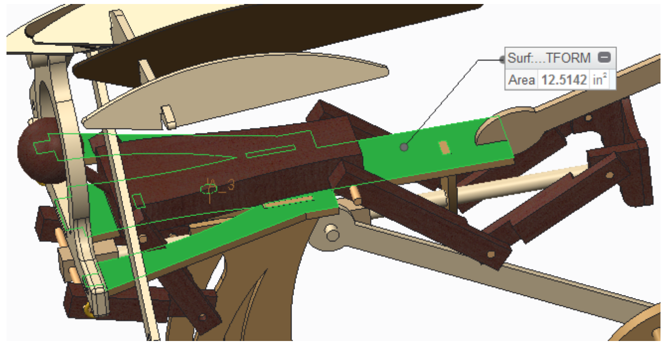
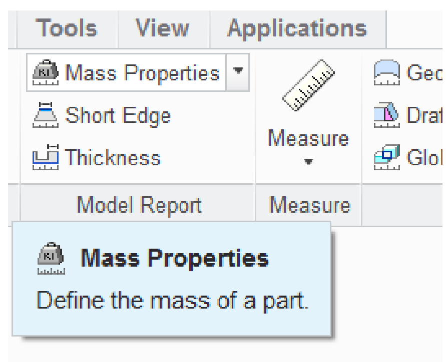


Try the angle tool next and then select two edges once again by using the **CTRL** key on the keyboard.

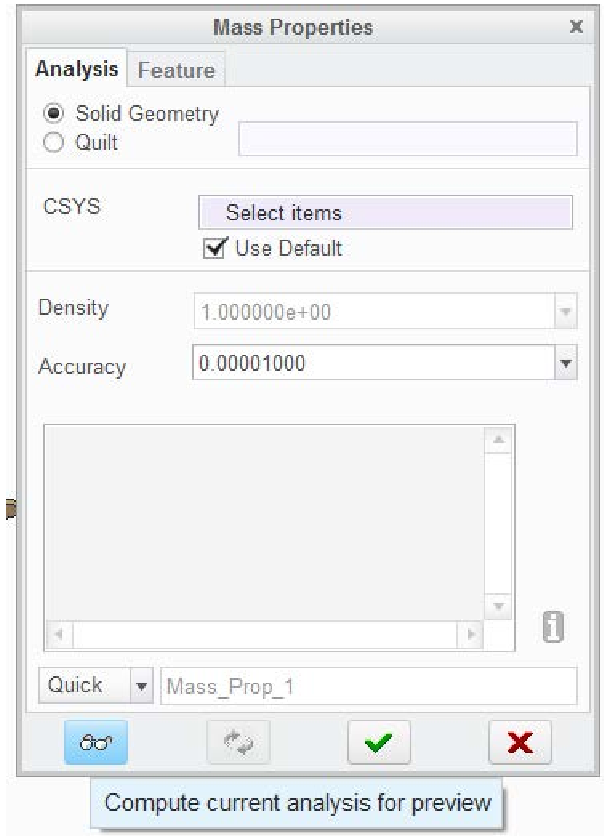


Next let's measure the diameter of one of the gears. Click on the diameter tool and then click on the edge of one of the gears.

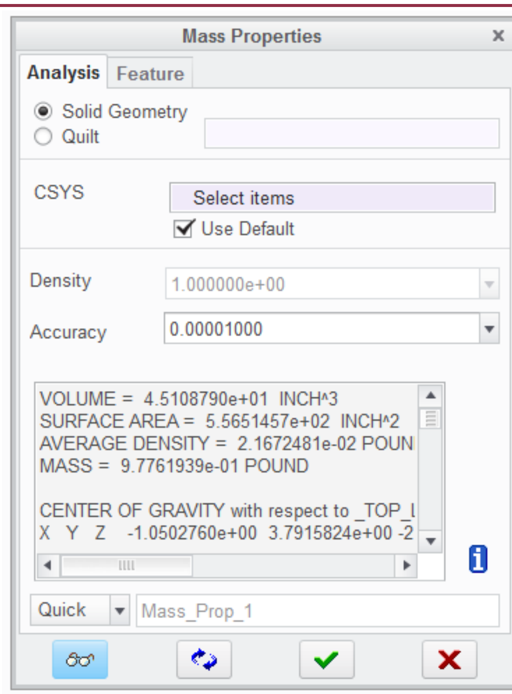


Now measure the surface area of the top platform by selecting the surface area tool  and selecting the face of the top platform.	
Let's finish by measuring the volume of one of the wing fins. Select the volume tool  and then select one of the wing fins.	
The measurement tools are powerful and can be used for a variety of purposes. Let's try measuring the "**Mass Properties**" of this model which are properties of the entire model. Close the measure dialog box and click the **Mass Properties** icon in the Analysis ribbon.	

Once again a dialog box will open. Find and select the eye glasses at the bottom of the dialog box to compute the mass properties for this model.

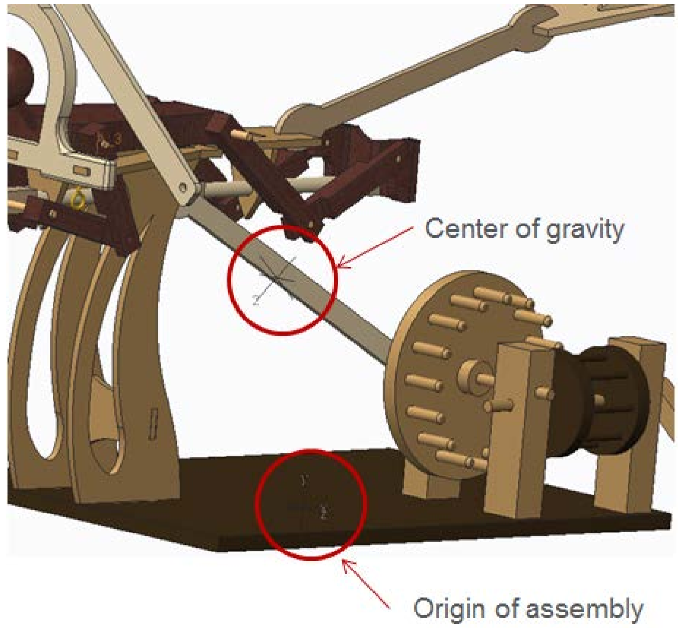


Notice that the mass of the entire assembly is just less than a pound. You can scroll through the rest of the information using the scroll bar on the right.



Another important calculation that is made when you click on the eye glasses is the center of gravity for the entire product.

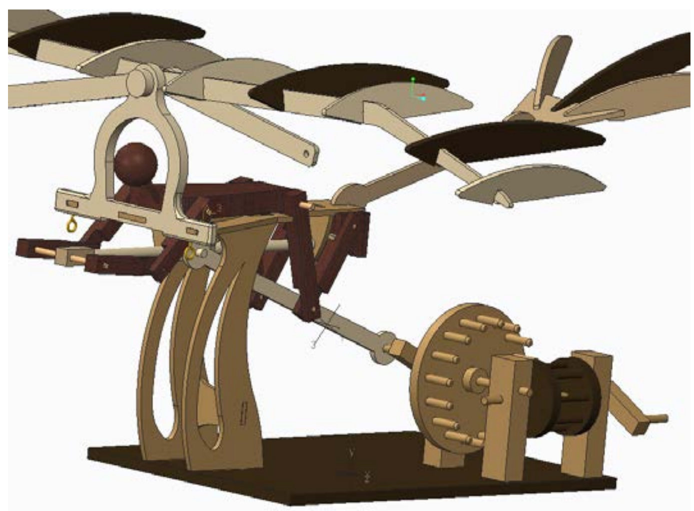
Rotate and zoom the ornithopter to find the coordinate frame labeled with 1, 2, & 3. This is the center of gravity. The coordinate values are displayed in the dialogue box.



Left-click on the crank handle while pressing **CTRL-ALT** on the keyboard and drag your mouse around to turn the crank until the wings are at their highest point and then click on the eye glasses again.

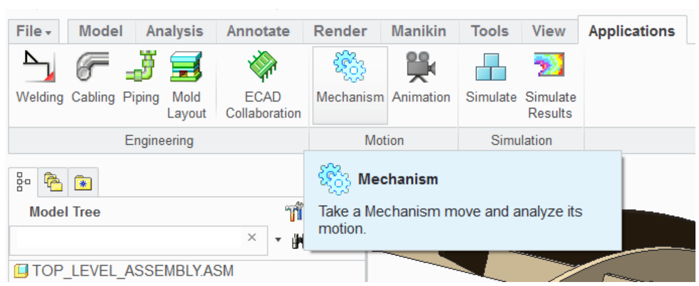
Does the center of gravity change? If so, why did it move?

Close the dialogue box.



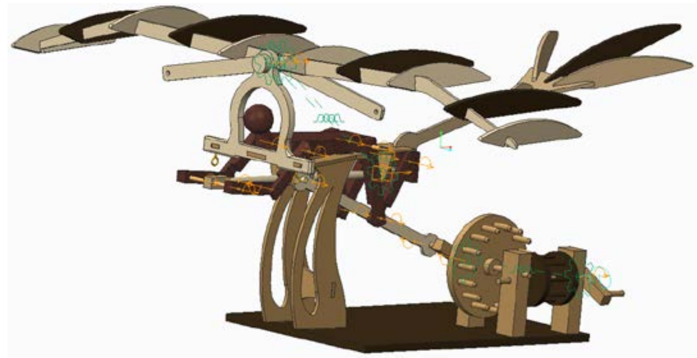
Now find the **Applications** tab in the upper menu and click on it. Then click on **Mechanism** icon.

This will open a new area of **Creo** that is used for mechanism simulation.



Notice that there are a number of orange symbols shown in the model. These are mechanism connections created during the assembly process to define a components motion.

Notice that there are also green symbols that define gear pair connections between the crank and gears. This defines that those components are connected.

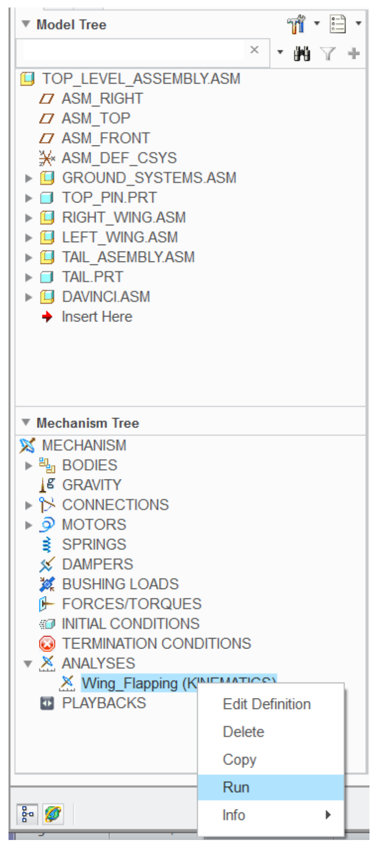


You will notice that there is a new section in the model tree called the mechanism tree.

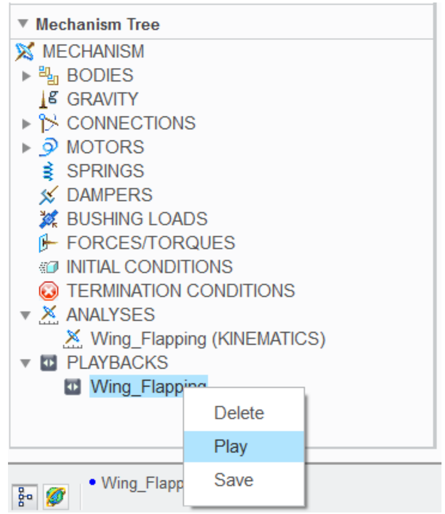
This tree has information about all of the kinematic connections, gear pairs and so forth.

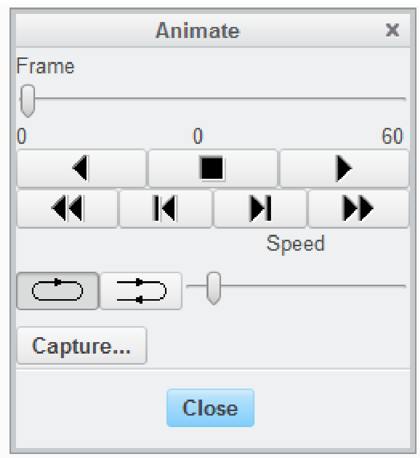
Expand the tab called **ANALYSES** by clicking on the triangle next to it. Then right-click on the analysis called “**Wing_Flapping (KINEMATICS)”** and click Run.

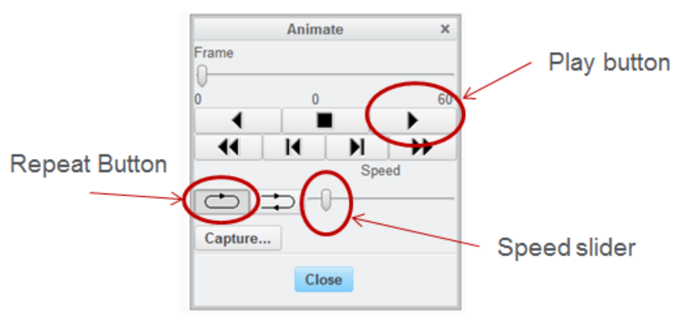
This turns on a virtual motor and runs the entire ornithopter.



<table>
<tr>
<td>The ornithopter ran through one complete revolution. This was the definition of the analysis that you just ran.

Now replay the results by expanding the PLAYBACKS tab and then right clicking on “Wing_Flapping” and select Play.</td>
<td>
</td>
</tr>
<tr>
<td>A dialog box appears and allows you to play back the entire analysis over and over again. It allows you to change the speed and capture it as a video. Let’s try each of these activities.

Start by clicking the Play button.</td>
<td>
</td>
</tr>
<tr>
<td>You can change the speed of the playback by sliding the speed slider back and forth while it is running.

Note: You can rotate and zoom the model while it is playing back.</td>
<td>
</td>
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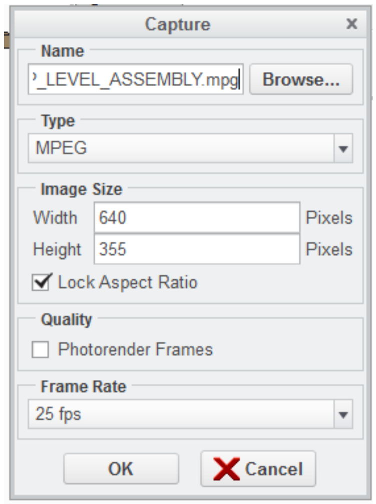
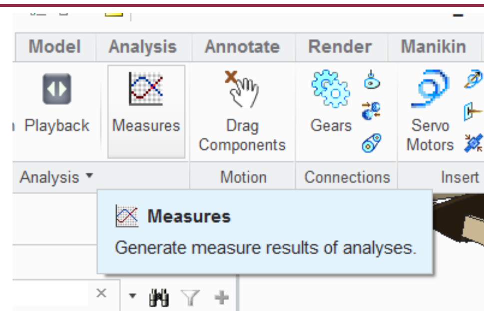
</table>

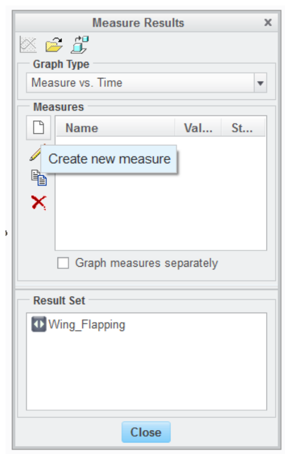
<table>
<tr>
<td>Now click on the Capture button.

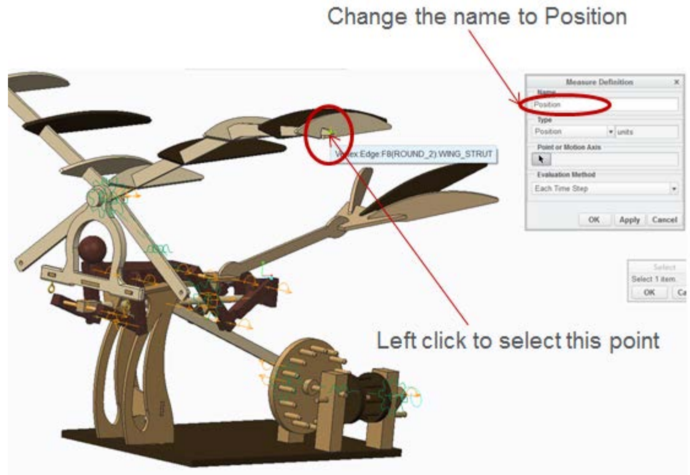
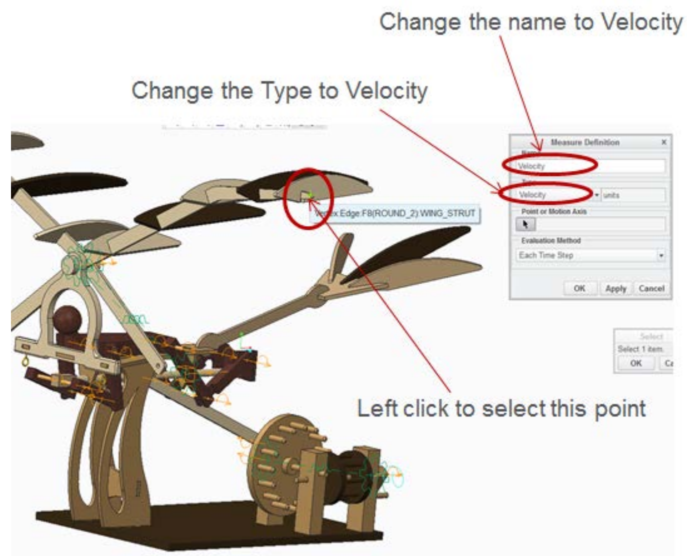
The default output is an MPEG type video file. You can specify a different file name, output format, output screen size and frame rate. This output file will allow you to play the video outside of Creo.

Close this dialog box by clicking on the Cancel button.

Close the playback dialog box as well.</td>
<td></td>
</tr>
<tr>
<td>After the analysis has been run, you can also perform analysis measures to interrogate the simulation.

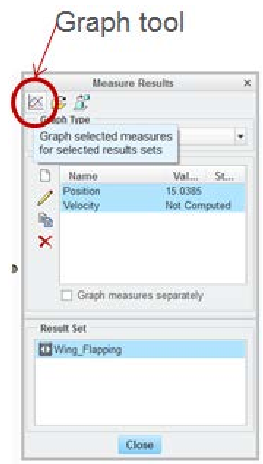
Find and click the Measures tab in the upper menu.</td>
<td></td>
</tr>
<tr>
<td>A new dialog box appears.

Click on the Create New Measure icon .</td>
<td></td>
</tr>
</table>

Change the name of the measure to **Position.** On the model, left-click on the tip of the wing to select a point shown by a small green crosshairs. Click **OK**. Now create another measure by clicking on the **Create New Measure** icon again.	
Change the name of the measure to **Velocity**. Change the type to **Velocity** and then click the same point you selected in the previous step. Now click **OK**	

Now select both of the measures you created using the **CTRL** key and left clicking. Then click on the **Result Set: Wing_Flapping**.

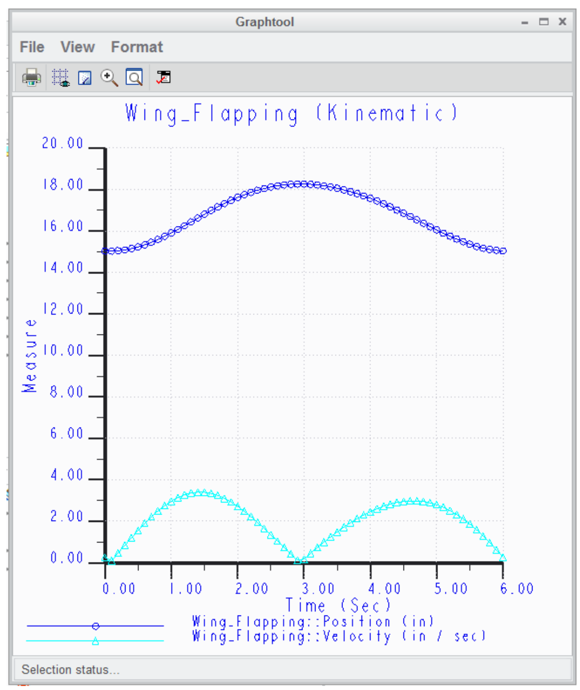
Once selected, click the **Graph Tool** icon in the upper left corner of the dialog box to generate a graph of the measures.



The two measures are graphed for the duration of the analysis.

Now move the graph to the right of the screen by clicking on the top bar of the graph dialog and dragging it to the right.

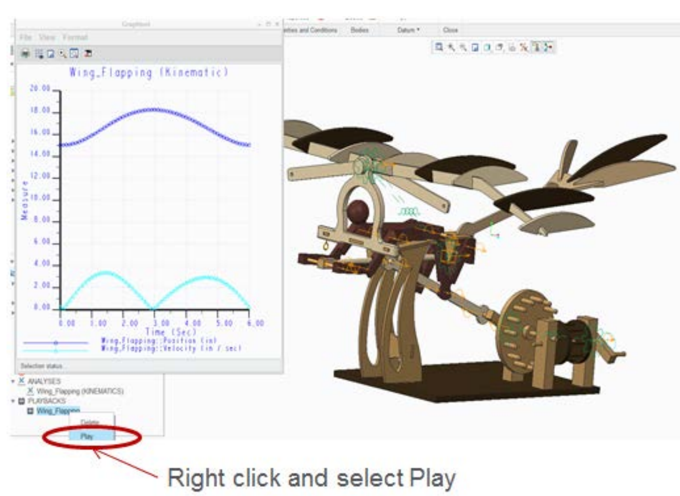
Next, close the **Measure Results** dialog box by clicking on the **Close** button at the bottom of the dialog box.



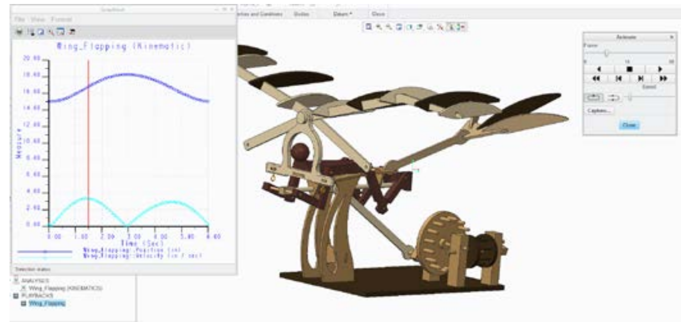
Right-click on the playback in the mechanism tree again and select **Play**.

Click the play button to start the play back sequence.

You will notice that a red bar appears on the graph to show where in the analysis you are.



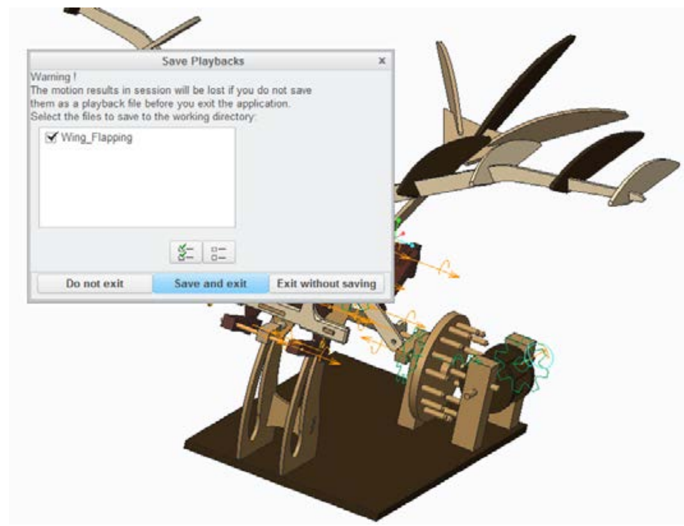
This allows you to review the graph while the analysis is playing so that you can rotate and zoom to observe how your mechanism is operating.



Once you have finished observing the analysis playback, close the playback dialog box by clicking **Close**. Also, close the graph by clicking on the x in the upper right hand corner.

We are done with this exercise so you can select the **File** tab in the upper left of the screen and select **Close**.

Creo will ask you if you want to save the playbacks but just **Exit without saving**.



Congratulations, you have finished this exercise!

This exercise is based on a model of Leonardo da Vinci's Ornithopter by Pathfinders Design from his Codex Atlanticus notebook in about 1488. Leonardo drew many designs of flying machines and this model is based on both those drawings, and his drawing of a bevel gear in one of his other notebooks around the same time. Many of Leonardo's machine drawings were designed to convert rotational movement into reciprocating motion, and in modern industry and manufacturing we continue to do the same!