Sarge's Siege: A Solidworks Suitability Investigation

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16th October 2017

Executive Summary

Solidworks is an Computer Aided Design Program with Visual Basic for Application Libraries and an extensive Application Programming Interface. An investigation was undertaken to investigate the suitability of implementing Solidworks within the workplace of Sarge's Siege.

Solidworks is Intuitive and easy to learn. Dassault Systems installed comprehensive tutorials and has set up an online community, enabling fast learning and adoption of Solidworks.

The modelling of an axial, a component of a catapult, was used to investigate the suitability of the product. The component of an axial was split into five components; Wheel Rim Scalable, Tire, Swivel Mechanism, Wheel Fastener and Drive Axial. Each part was constructed using the functionalities of Solidworks. A combination of sketches, extrudes, revolves and patterns were used to model each part. Each component was assembled using standard, advanced or mechanical 'mating' relationships in an assembly. Solidworks can produce drawing to convey information, demonstrated by the Swivel Mechanism and Axial drawings.

Solidworks API and VBA libraries allow model parameters to be changed by assigning pointers to the Solidworks application from the excel workbook. Parameters are modified through replacing values or strings. The global variables manipulated where; the radius of the wheels linked to the wheel size, the length of the axial, linked to the catapult size and the material of the wooden components.

Solidworks was a great learning experience with smart, intuitive features and comprehensive guidance.

Solidworks is intuitive and comprehensive. The functionalities of the program are easy to use. Model parameters are easy to manipulate external from an excel workbook using subroutines. It is recommended Sarge's Siege adopt Solidworks.

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1 Introduction

The market for Computer Aided Design (CAD) products is saturated. There are many alternative products capable of producing three dimensional models, sketching and performing analysis. Solidworks is a CAD program specifically targeted towards engineers. Solidworks has an extensive list of Visual Basic for Application (VBA) libraries and an extensive Application Programming Interface (API). Sarge's Siege have used Solidworks to develop a model and manipulate the model in excel based on the users input, where the API is used to access the VBA programming libraries. The end goal was to use excel to manipulate the Solidworks model's parameters. Solidworks was the focus for our suitability investigation to reach a conclusion on whether the CAD package would meet our scopes of use.

2 Assessment

2.1 Ease of Use

CAD program must be intuitive and easy to use to aid firm wide implementation. Solidworks meets this classification. The application is easy to navigate and design with. Solidworks has a series of in-built tutorials, introducing new users to the functionalities of Solidworks. Tutorials include; basics and advanced techniques, productivity tools, design evaluation and more. See Figure 46 in 6.11. The gradient of the learning curve and time invested in learning the package is significantly reduced, excellent for educating draftsman quickly and a fast adoption. The tools in sketching are smart and easy to use. The 'Smart Dimension' tool is the most intuitive. You can dimension any sketch in relation to another by creating a driving, named dimension by two or more simple clicks, dragging, and then placing the dimension. You can create three-dimensional object from two-dimensional sketches through extrudes, revolves, lofts, sweeps and other functions. Assembling different components is easy through 'mating' different parts using standard, advanced or mechanical mates. The drawing wizard within Solidworks is powerful and intuitive. You can create drawings with multiple views, magnified sections and dimensioned. These drawings are useful to convey information to suppliers and inhouse compartments. Solidwork's graphical interface makes modelling, assembly and drawing very easy. There is also an extensive online community with support and learning opportunities (Dassault Systems, 2017).

2.2 Model Creation

A model had to be created to assess the programs suitability. We focused on the front axial component of the catapult due to the time constraints of this report. The findings from the investigation may be apply to the full product. The complexity of our axial led us to build the product in a series of discrete steps. The axial was split into 2 two sets of five components: Drive Axial, Swivel Mechanism, Tire, Wheel Fastener and Wheel Rim Scalable. The dimensions of each part were driven off the two global variables, radius and length. 'Radius' relates to radius of the wheel and 'Length' the width of the axial. The wooden material property is changed in the components as well.

2.2.1 Wheel Rim Scalable

This part was the most complicated to model (Figure 1). We sketched the rims cross section, with all dimensions derived from the radius. The sketch was revolved 360˚ to form the framework of the rim. A second cut extrude made slots on the edge of the rim. A revolve features created the hub cap from a sketch incorporating splines. A cut extrude features made slots in the circular hub cap. Mirroring the previous cut extrude, and using a circular pattern, helped formed the resemblance of a hub cap. The centre of the wheel rim was made from a series of circular cut extrudes and another central revolve around the central axis. The edges of the rim were filleted to increase realism. The dimensions were derived from ratios to scale accordingly. The design of the wheel rim was inspired by one produced by CAD CAM Tutorial (CAD CAM Tutorial, 2016).See Figure 11 through to Figure 16 in 6.1.

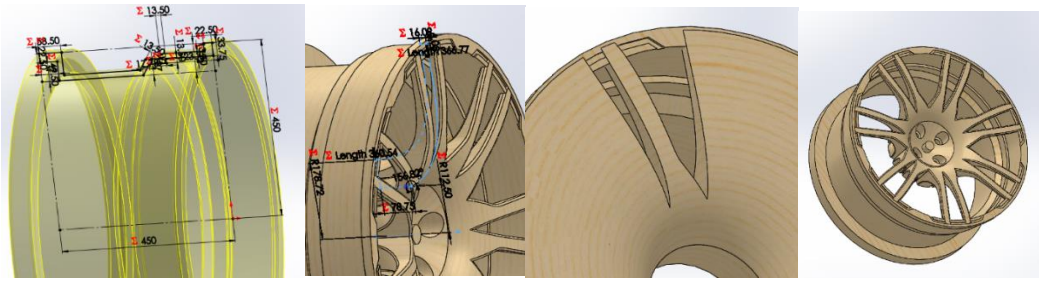


Figure 1: Wheel Rim Scalable Development

2.2.2 Tire

The tire part (Figure 2) was simple relative to the Wheel Rim Scalable. The sketch of the tire's cross section mirrors the Wheel Rim Scalable parts cross section to aid mating in assembly later. Tolerances were not accounted for as unnecessary for determining the suitability of Solidworks. However, tolerances in this part and all parts in the assembly will need to be accounted for use in machining. The sketch was revolved around the central axis. The edges of the tires were filleted to make the tire more realistic. After establishing the base of the tire, the tread was added by sketching the tread on a plane tangential to the tire's edge, using a boss extrude up until the surface of the tire and using a circular pattern to repeat the extrude around the edge of the tire. See Figure 17 through to Figure 20 in 6.2 for an in-depth walkthrough of the tire's creation.

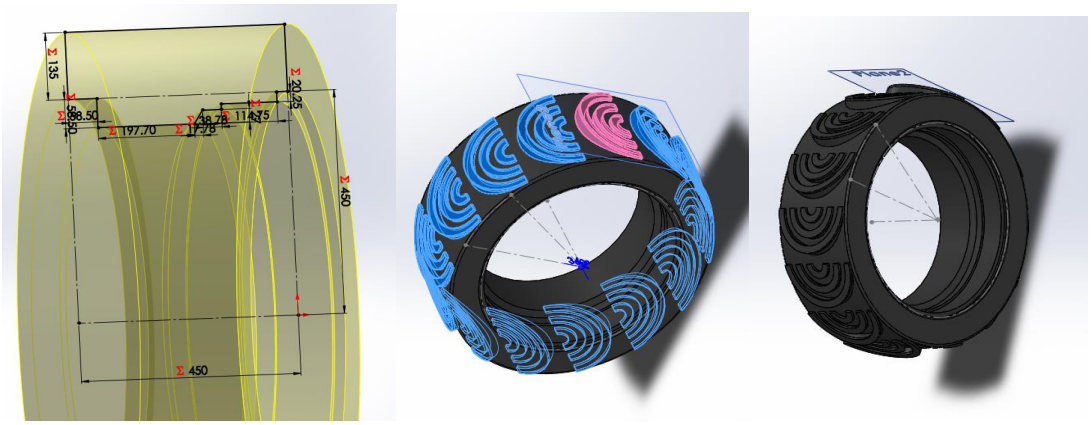


Figure 2: Tire Development

2.2.3 Drive Axial

The next part created was the drive axial, another simple part (Figure 3). First, the basic dimensions were sketched. Instead of using a standard extrude, we chose to use a series of loft extrudes where surfaces are formed between two parallel sketches. Four additional planes were made, each a quarter of the length apart. Circular and spline incorporating sketches were made on the planes with four separate loft extrudes connecting the sketches. To complete the part, two equal-distance circular cut extrudes were made on the axial. An walkthrough of the drive axial's modelling is shown in 6.3, Figure 21 through to Figure 23.

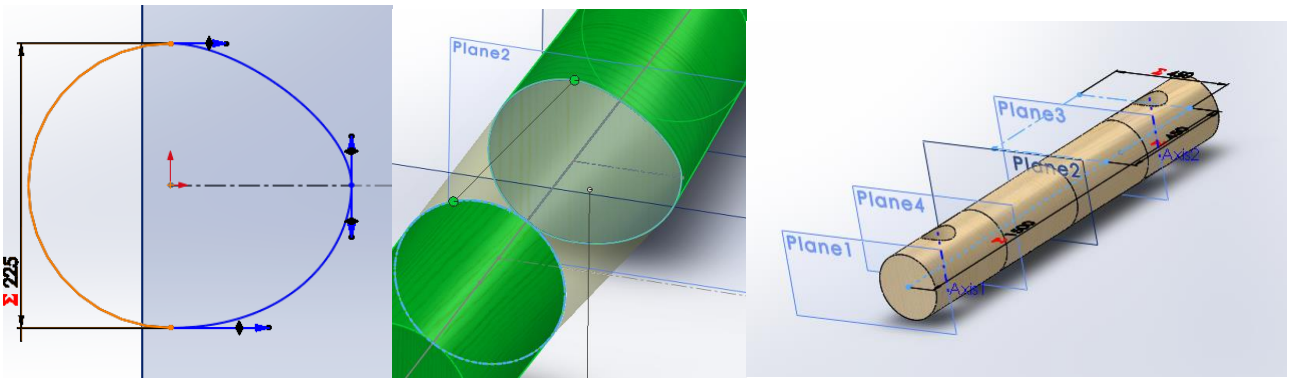


Figure 3: Drive Axial Development

2.2.4 Wheel Fastener

The wheel fastener part connects the wheel rims to the subsequent swivel mechanism (Figure 4). Like the swivel mechanism, this part is a series of simple extrudes. First, a circular extrude is made with four, equally dimensioned, equidistance circular extrudes protruding from on face. On the adjacent, parallel face to the four circular extrudes, another circular extrude is made, hollowed out by an internalised cut extrude. This part connects with the swivel mechanism and turns 360˚ in theory. The model is for demonstration purposes, and, due to time constraints, bearings

which enable the turning mechanism were not designed. The bearings and there housing will sit in the cut extrude if this part was to be machined. See 6.4, Figure 24 though to Figure 26.

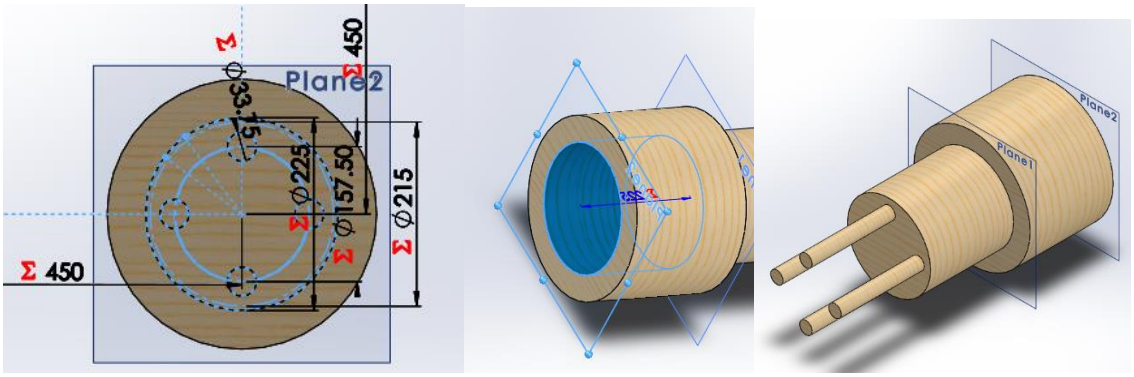


Figure 4: Wheel Fastener Development

2.2.5 Swivel Mechanism

The swivel mechanism connects the drive axial to the wheel fastener. This part was fabricated computationally using several complex functions (Figure 5). First, two circular extrudes with different radii were made off each other. A pin like component was to be incorporated into one of these extrudes. A square cut extrude was made to make room for this pin. After room was made, a circular extrude with two directions was put in place, extruding up to until the surface. The swivel mechanism has an arc shaped extrude, cut, with another pin adjacent to the first. This is a complex feature and was difficult to do at first and define with a set of equations. Two additional planes were made to enable a swept boss. A square sketch is swept along a spline to create the arc. The spline was difficult to define via an equation at first. We initially used one quarter of an ellipse's parameter. Determining the equations for the perimeters of an ellipse is very difficult as is calculated by approximations or two sets of infinite series (Maths is Fun, 2017). To circumnavigate this complexity, a simple spline equation was used. To complete the part, the Swept blend was cut out and the second pin added. See Figure 27Figure 29 in 6.5 from an in depth walkthrough.

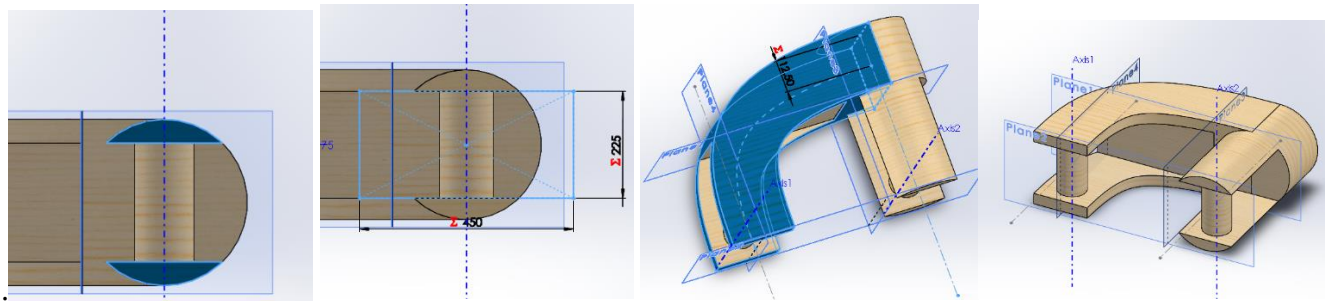
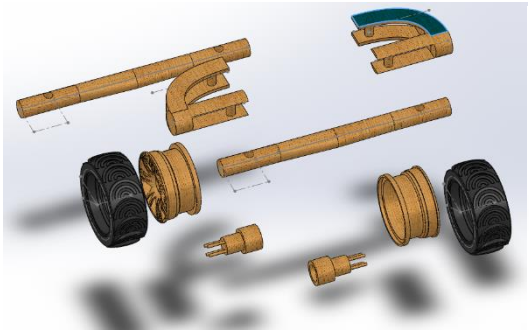


Figure 5: Swivel Mechanism Development

2.2.6 Assembly

After finalising the model components, they can each be individually placed in an assembly (Figure 6). Solidworks allows you to mate individual components together. There are standard, advanced and mechanical mates. The wheel rims, tires and wheels fasteners are mated using standard co-incident and concentric mates. Mechanical hinge mates were used to mate the wheel fasteners and swivel mechanisms. The hinge constraint combined the concentric, co-incident and rotational constraints to enable the wheel fastener and swivel mechanism to rotate with one degree of freedom. If the axial was to be machined and assembled, a slot mate would be used. The swivel mechanism and drive axials were mated using a combination of hinge and concentric constraints. The final assembly is under defined due to the rotational permissions from the hinge mates, allowing the wheels to rotate 360˚ and swivel 40˚ (Figure 7). See Figure 30 through to Figure 34 in 6.6 for the final images of all individual parts and Figure 35 through to Figure 39 in 6.7 for pre assembly, multiple assemblies and a mating pair.



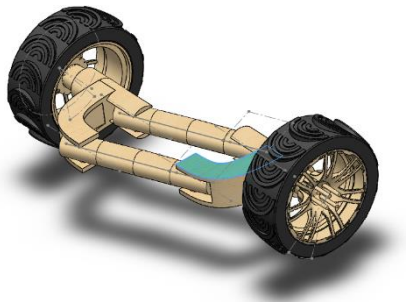


Figure 6:Individual Components placed in Assembly (Left)

Figure 7: Completed Assembly (Right)

2.2.7 Drawings

Once the assembly is complete, Solidworks has the capability of producing drawings of our parts or assemblies. Drawings are an excellent way to convey the part or assembly information. Different views of the part may be displayed on the drawing. We have displayed a top, right, front and isometric view of the Axial assembly and the Swivel Mechanism part. Dimensions may be added to individual views or the entire model. You can also produce section and detailed view to drawings. Solidworks allows you to add reference and machining information to the drawings files, enabling the accurate filing and revision of drawings. Assemblies have many dimensions from aggregate parts, cluttering the sheet. It is better to display the assembly without drawings and dimension the individual parts. See Figure 40 for the axial drawings and Figure 41 for the swivel mechanism drawings in 6.8.

2.2.8 Additional Features

Solidworks is a complex program with many other features. There are; productivity tools, animation, simulation, costing, flow analysis, dimension expert tools, mould designs and more (Figure 46,6.11). After the modelling of your product, you can test its suitability though simulation. Solidworks also has rendering capabilities to present your model in a high presentation quality in Dassault System's premium product. In addition to rendering, background images may be set in the assembly planes (Figure 45, 6.11). Solidworks has the ability to install add ins to use new features, properties and increase the functionality of the product.

2.3 Implementation within the Excel Workbook

After the assembly was finalised and the dimensions were appropriately referenced global variables, the next step was to integrate the excel workbook with Solidworks and the model. To begin with, the Visual Basic Solidworks API was investigated, learning the capabilities. Excel is able to send, receive and manipulate information to, from and in the powerful API. The Solidworks Application is accessed as a VBA object, using pointers in sub routines to point to, activate and manipulate objects in the API. This functionality allows parameters within Solidworks to be manipulated. The relevant file in Solidworks must be open to make changes, enabling the use of 'pointers'. The exert of code to establish pointers to the Solidworks application, assembly files, part files and the equation manager is in Figure 8.

```
'Set Assembly Name'
assembly = "Axial.SLDASM"
'Set pointer to the Solidworks Application'
Set swApp = CreateObject("Sldworks.Application")
'Make the assembly the active document model'
swApp.ActivateDoc (assembly)
'Set pointer to make changes to the assembly'
Set objFile = swApp.ActivateDoc(assembly)
'Set pointer to Equation manager tool'
Set swEqnMgr = objFile.GetEquationMgr
'Change the Radius and Length Global variables'
```

Figure 8: VBA pointers to Solidworks Application, Parts, Assembly and Equation Manager

After the code assigns the assembly and equation manager as the current objects, adjustments are made to the radius and length global variables saved in the assembly. The modifications to the global variables are based on the user's input in the excel worksheet. Excel VLOOKUP functions create a Solidworks manipulation table, depicted in Figure 9. The table resides in the "ReferenceData" sheet in the excel workbook. After selection, the user can press the Solidworks button to the right of the table to manipulate the model. The use of pre-determined sizes and materials matches the functionality of our previous excel workbook.

Solidworks Dimensions and Parameters (mm)						
	Size	Wheel Size	Material		Solidworks Parameters	
Small	2000	300	Oak	Oak	Size	3000
Medium	2500	450	Mahogany	Mahogany	WheelSize	300
Large	3000	600	Birch	Pine	Material	Mahogany
			Willow	Teak		
			Ash	Cedar		

Figure 9: Solidworks Modification Table

The methodology of updating the model in Solidworks is very simple. The global variables 'Radius' and 'Length' can be manipulated in the equation manager (Figure 42, 6.90. The global variables are stored as strings. The values stored can be converted to string, split removing the old value and replacing the old value in the global variable newly converted string. The index position of each global parameter is known. The snippet of code is shown in Figure 10.

```
'Convert Values in the excel spreadsheet into strings'
length = CStr(Worksheets("ReferenceData").Range("SolidworksWheelSize"))
radius = CStr(Worksheets("ReferenceData").Range("SolidworksSize"))
'Split the global variable storage equation strings at the equals sign'
strSplit1 = Split(swEqnMgr.Equation(0), "=")
strSplit2 = Split(swEqnMgr.Equation(1), "=")
'Append the length and radius value onto the end of the strings'
swEqnMgr.Equation(0) = strSplit1(0) & "= " & length
swEqnMgr.Equation(1) = strSplit2(0) & "= " & radius
```

Figure 10:VBA Updating Global Variables

The global variables 'Radius' and 'Length' are linked to the 'Radius' and 'Length' global variables in each part. Every dimension, excluding angles, is referenced to these variables. 'Length' adjusts the length of the axial, attributable to the size of catapult. 'Radius' adjust the radius of the wheels, attributable to the wheel size. The dimensions scale according to the global variables, retaining assembly mates and definitions. The material parameters are updated on each part in the assembly by resetting the material name in the material properties manager (See 6.9, Figure 43), one at a time. Solidworks had oak and mahogany as material options but did not have Birch, Willow and Ash. I used Cedar, Teak and Pine as placeholders to import later. Once all updates have been made, the assembly is rebuilt with the user able to view their custom-built axial related to their catapult design.

The entirety of the code was input into the "SolidworksIntegration" module as one subroutine, "SolidworksManipulator". A copy of this subroutine may be found in Figure 44, 6.10. When the user reaches the "Individual Invoice" sheet and confirms their catapult design, a create more catapults userform will show. If the user chooses no, the "SolidworksManipulator" subroutine will be called. The user will be presented with a prompt, asking the user to have all relevant Solidworks files open before continuing. This is to guide the user to open all files and avoid a confusing error message. The user follows the design process in the excel workbook before confirming the catapult. The user will only see updates to their model if all files are open and they selected yes in the prompt. If the user selects no, the subroutine to update the model will not run.

3 Learning Experiences

Overall, this investigation proved useful and informative. The Solidworks tutorials are informative, requiring less external exploratory based learning experienced with other CAD packages. The creator functions in Solidworks are intuitive and straightforward to use. Part creation, the assembly window and the drawing environment were simple to learn. The drag and drop functionality for all tools was fun and engaging. Solidworks was helpful with error messages, guiding you to the right methods such as fully dimensioning drawings and defining assemblies. The use of equations to drive dimensions is brilliant. I had some difficult with fully dimensioning drawings and accessing the correct indices and value of global variables in the equation manager. All other errors were quick fixes, either adjusting syntaxes or changing dimensions to reference equations. The VBA was intuitive, apart from the concept of global variables stored as strings. Using VBA to access the API of Solidworks was an enlightening experience. I look forward to investigating and experimenting with accessing other software packages using VBA. The online forum set up by Dassault Systems, the creator of Solidworks was insightful on the rare occasion the Solidworks user interface could not help. Solidworks is a great product. I planned out my parts in the beginning, performing sketches of the axial before modelling. I was set on what I wanted to design in the beginning. Next time, I will spend more time at the beginning iterating concepts to find the best possible solution. I used my knowledge from PTC CREO, assuming Solidworks had the same functionality. This worked well however I will look to investigate other features in Solidworks first to ensure I don't miss using a more appropriate tool. The way I integrated excel and VBA was very simple. I will look into thoroughly completing the VBA and API tutorials for future products to gain a better understanding of the interface, whilst, expanding my capability on what I can do with VBA and the Solidworks product. I have loved my experience with Solidworks and cannot wait to use it in other academic and personal projects.

4 Conclusions

- Solidworks is an intuitive and engaging product, straightforward to learn. The adoption of Solidworks would not improve productivity immediately. However, Solidworks is quick to learn, improving efficiency in the long term.
- Modelling parts, assembly and producing drawings is an easy and engaging process. Creativity and engineering specifications are your only constraints to what you can create, exhibited through the creation of the catapult axial.
- Using VBA to update dimensions from an Excel subroutine is easy, as well as other Visual Basic for Application functions.
- The subroutine required to update dimensions and material properties fits seamlessly with the current version of *Sarge's Siege Catapult Workbook.*
- It is recommended our company adopt Solidworks to aid design.

5 References

CAD CAM Tutorial. (2016). Solidworks Tutorial | Sketch Wheel Rim in Solidworks. Retrieved from https://www.youtube.com/watch?v=_C8gy-OI1GU

Dassault Systems. (2017). SOLIDWORKS Community. Retrieved from http://www.solidworks.com/sw/community.htm

Maths is Fun. (2017). Perimeter of an Ellipse. Retrieved from https://www.mathsisfun.com/geometry/ellipse-perimeter.html

6 Appendices

6.1 Appendix 1: Wheel Rim Scalable Development

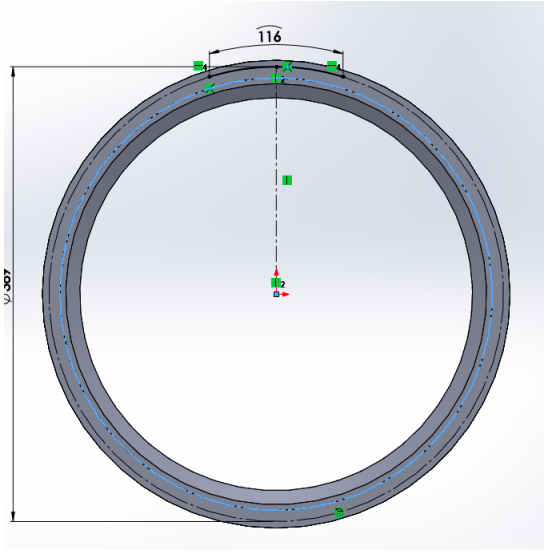


Figure 11: Initial Revolve

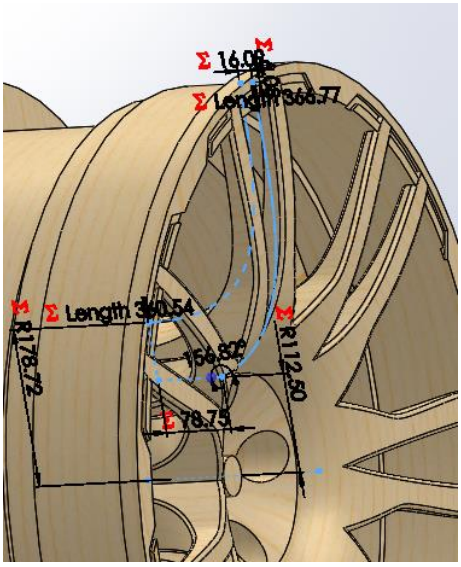


Figure 12: Sketch for Hub Cap

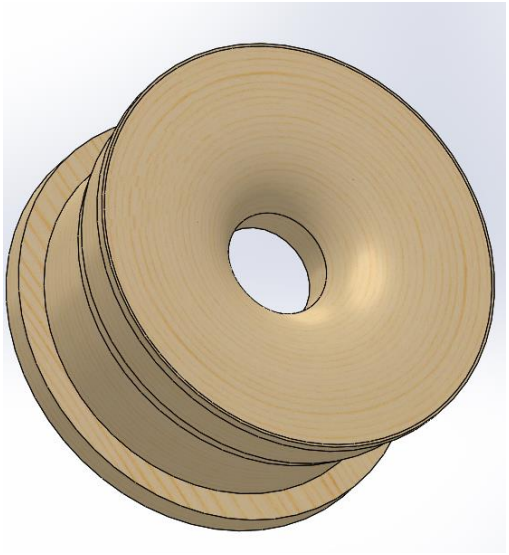


Figure 13: Revolve of Complex Hub Cap Sketch

Figure 14: Wheel Rim Hub Cap Cut

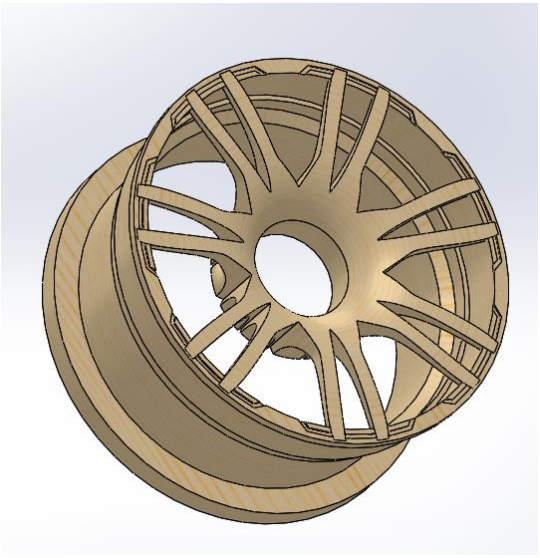


Figure 15: Using the Circular Pattern Tool on the Hub Cap Cut Extrude

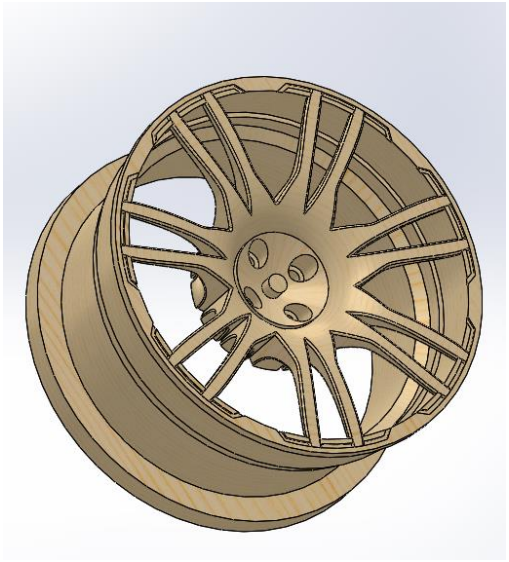


Figure 16: Finishing the Wheel Rim with Circular Extrudes

6.2 Appendix 2: Tire Development

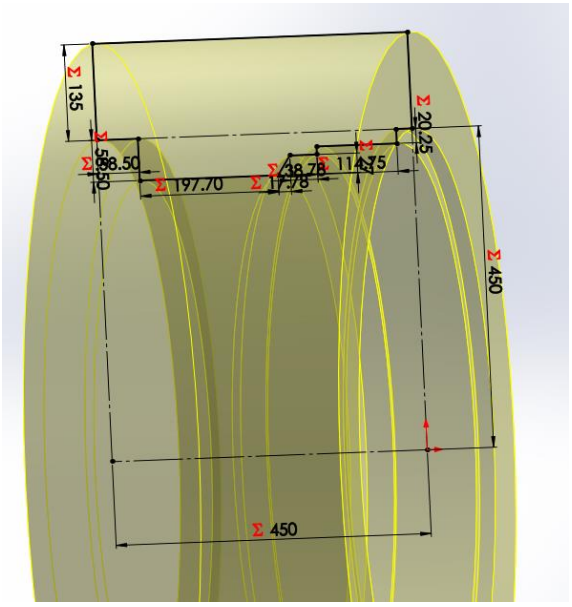


Figure 17: Sketch of Tire Thickness mirroring the Wheel Rim Thickness

Figure 18: Using the Revolve feature around a Circular Axis

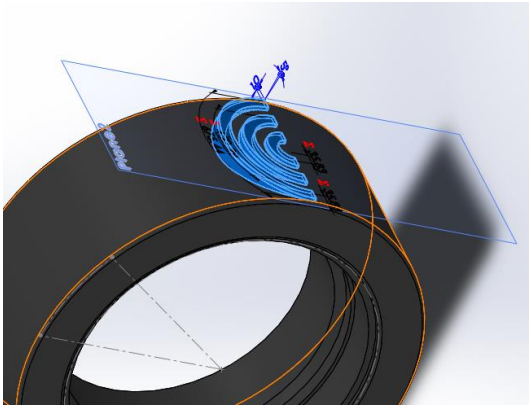


Figure 19: Creating the Tire Tread on a New Plane

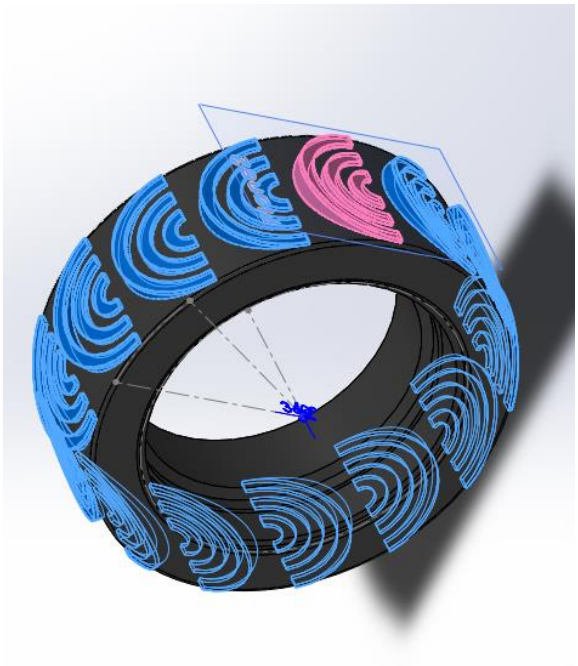


Figure 20: Using a Circular Pattern to repeat the Tire Pattern

6.3 Appendix 3: Drive Axial Development

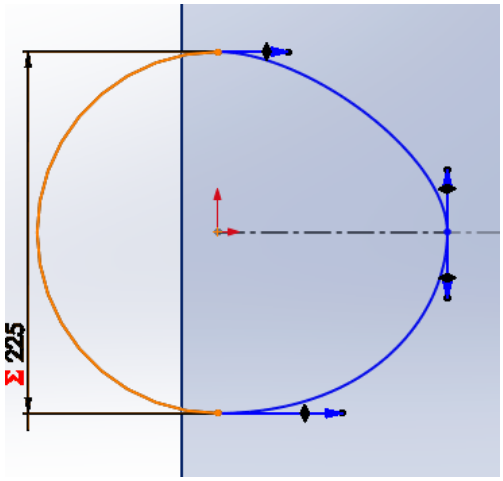


Figure 21: Sketch for Loft Mid-Section

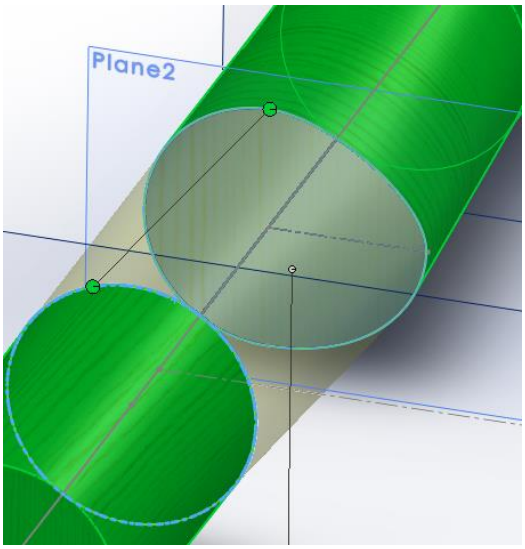


Figure 22: Using the Loft Feature to create an Extrude

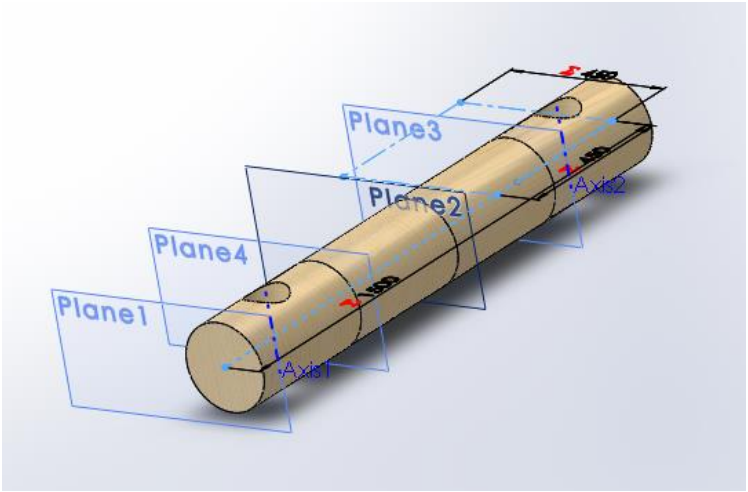


Figure 23: Creating Multiple Planes to create the Loft

6.4 Appendix 4: Wheel Fastener Development

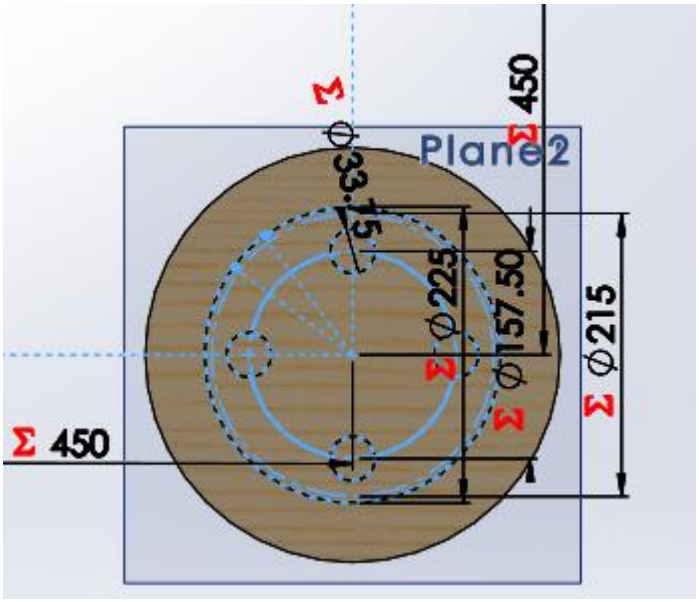


Figure 24: Sketch for first extrude with referenced dimensions

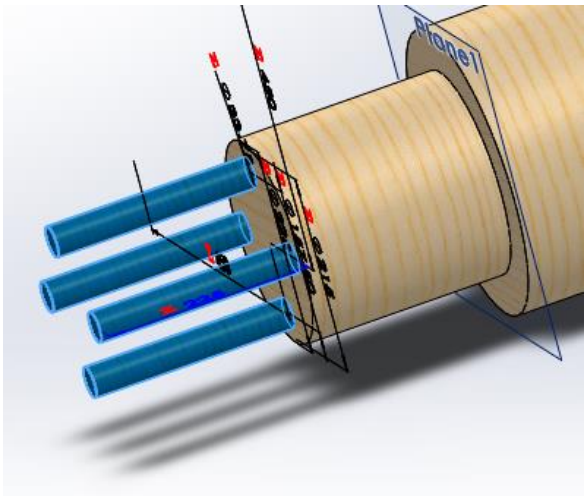


Figure 25: Four Circular extrudes to connect with the wheel

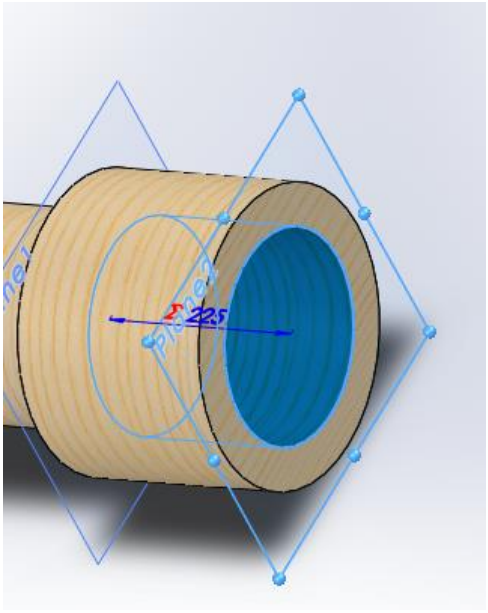


Figure 26: Hole features created using a circular cut extrude

6.5 Appendix 5: Swivel Mechanism Development

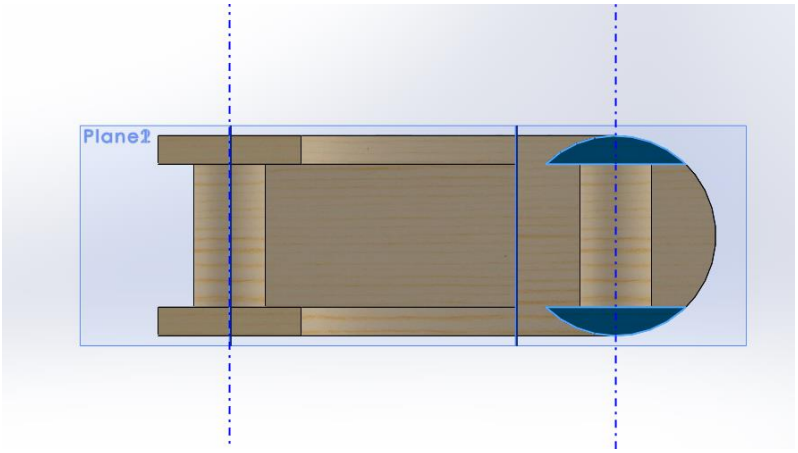


Figure 27: Front View of the First Circular Extrude

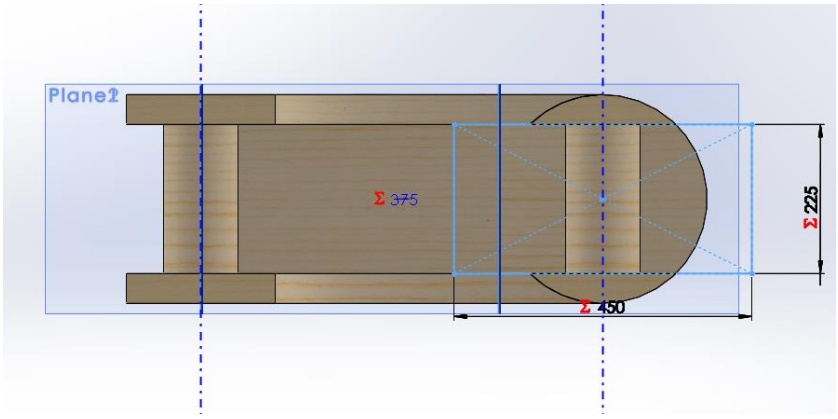


Figure 28: Using a Cut Extrude to hollow out the face

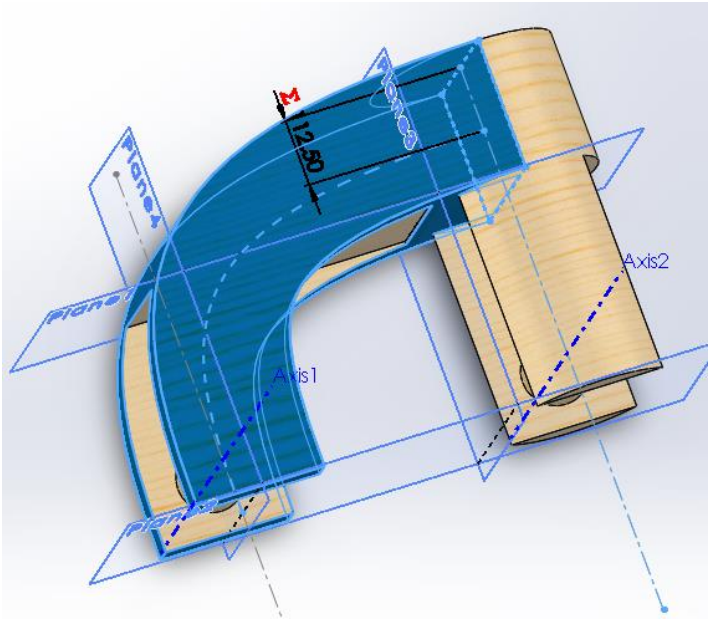


Figure 29:Using a Swept Blend to make a complex feature

6.6 Appendix 6: Completed Parts

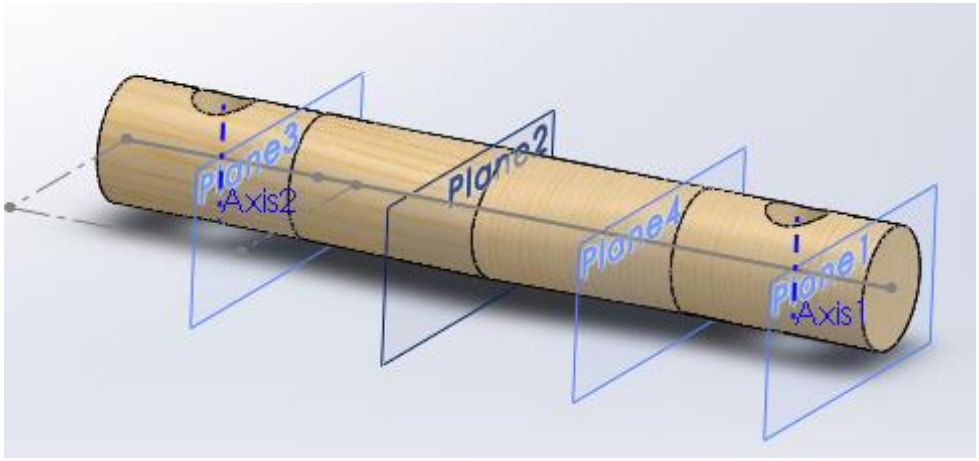


Figure 30: Drive Axial Final

Figure 31: Tire Final

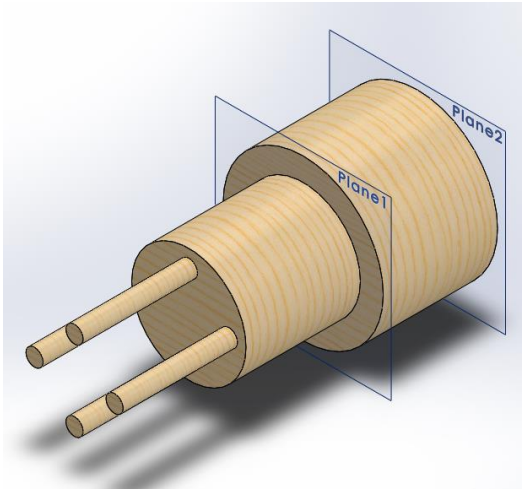


Figure 32: Wheel Fastener Final

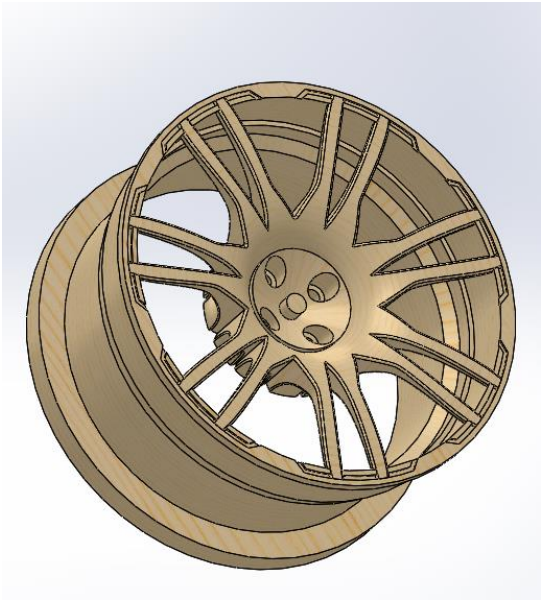


Figure 33: Wheel Rim Scalable Final

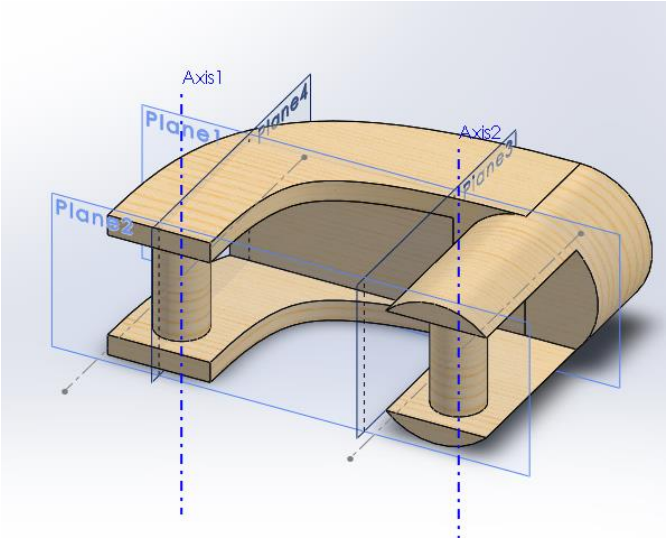


Figure 34: Swivel Mechanism Final

6.7 Appendix 7: Final Assembly

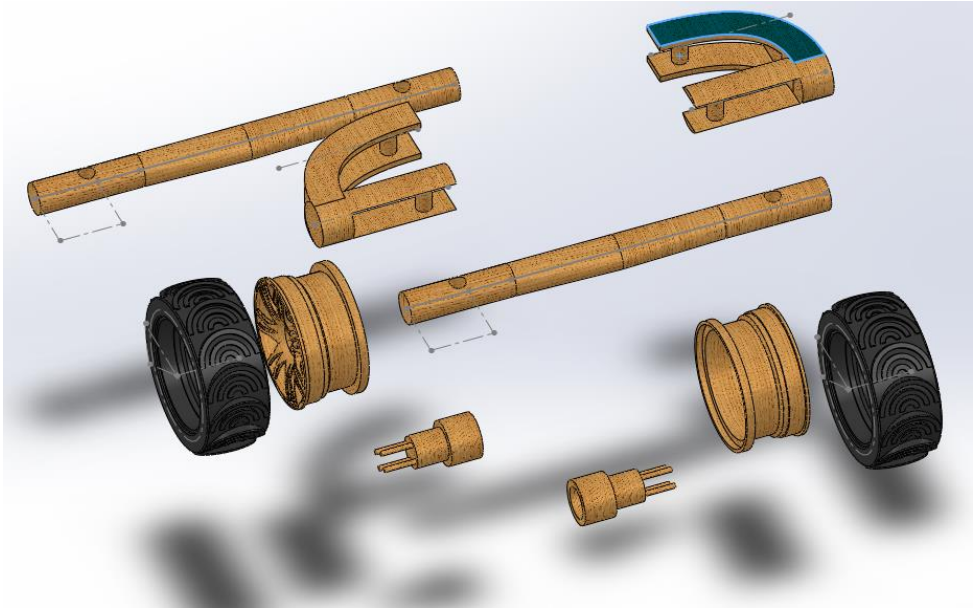


Figure 35: Pre- Assembly with Parts Individually Placed

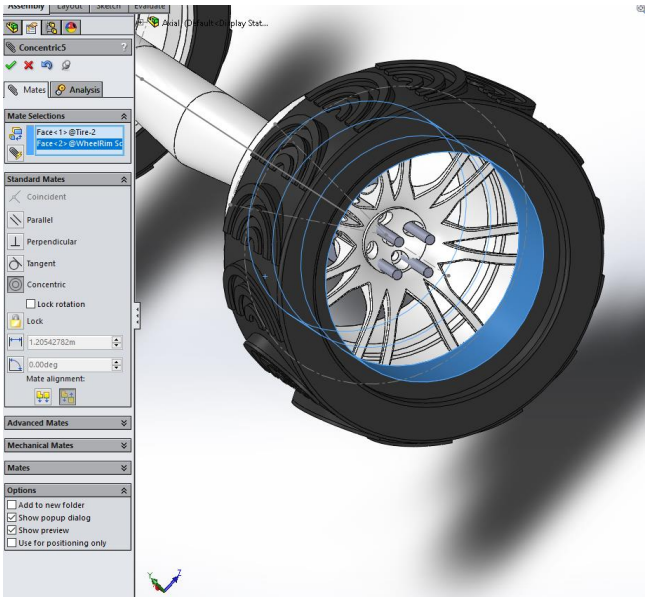


Figure 36: Using a concentric mate to place the wheel

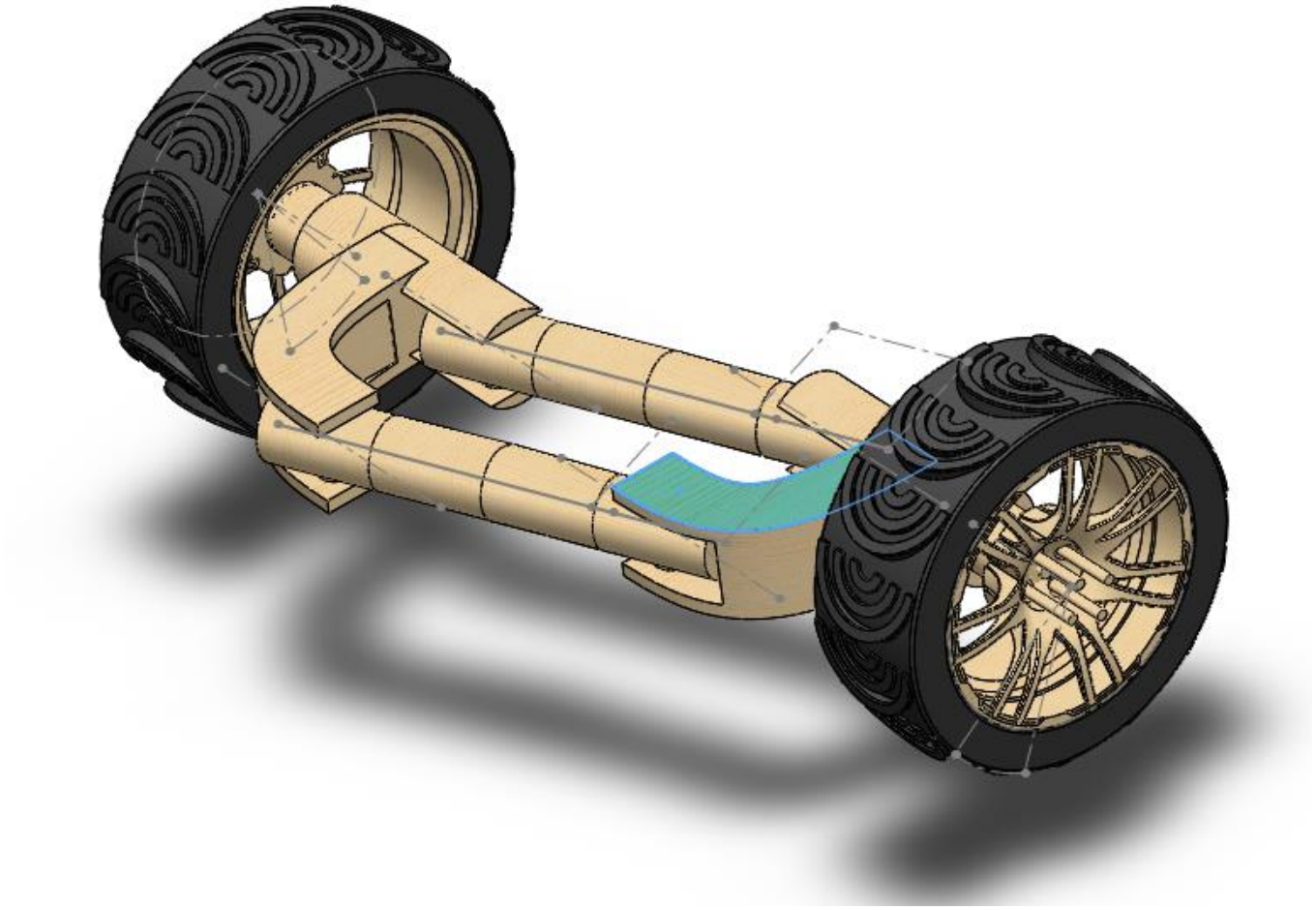


Figure 37: Final Assembly with spinning wheels and swivelling wheels

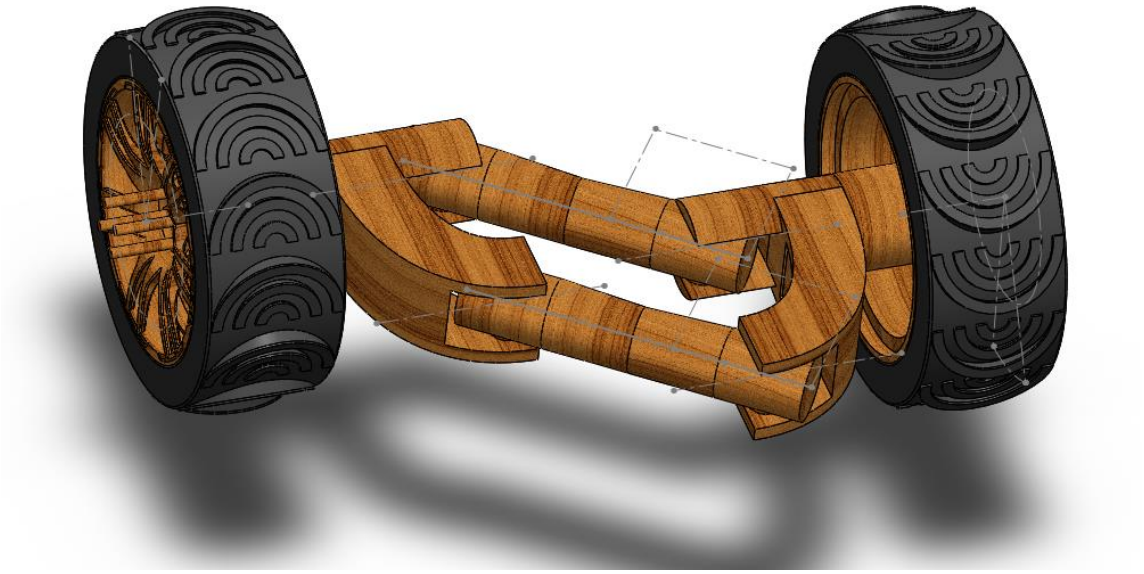


Figure 38: Axial Assembly with mahogany material, large wheel size and small catapult size

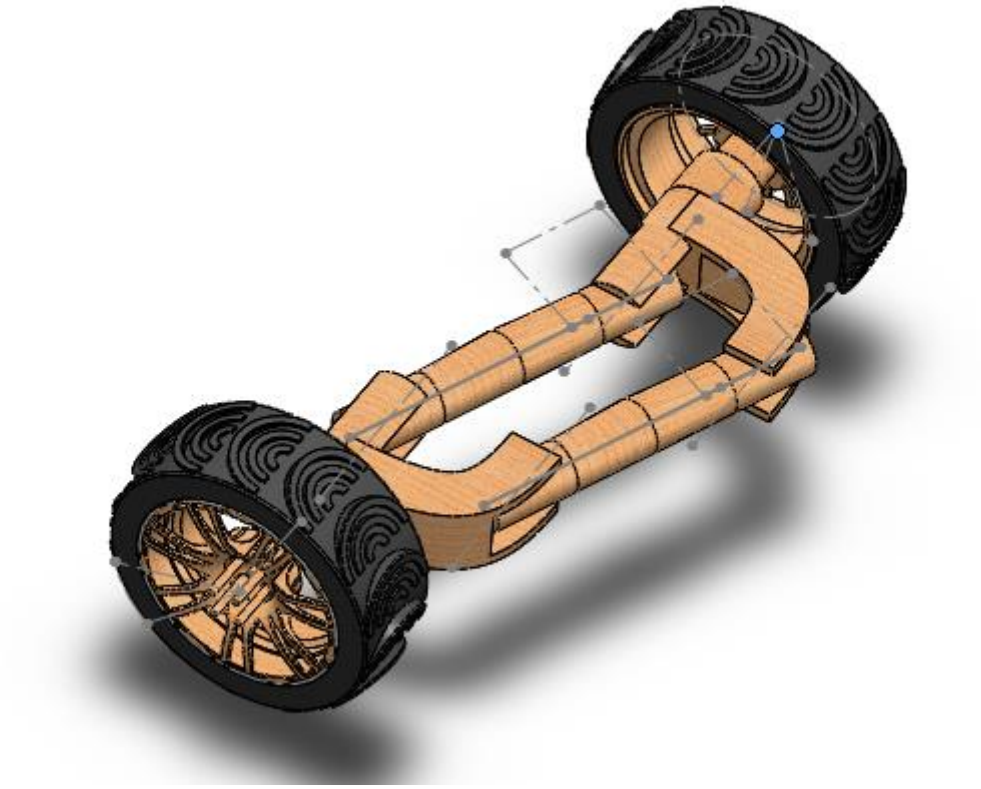


Figure 39: Axial Assembly with a cedar material

6.8 Appendix 8: Drawings

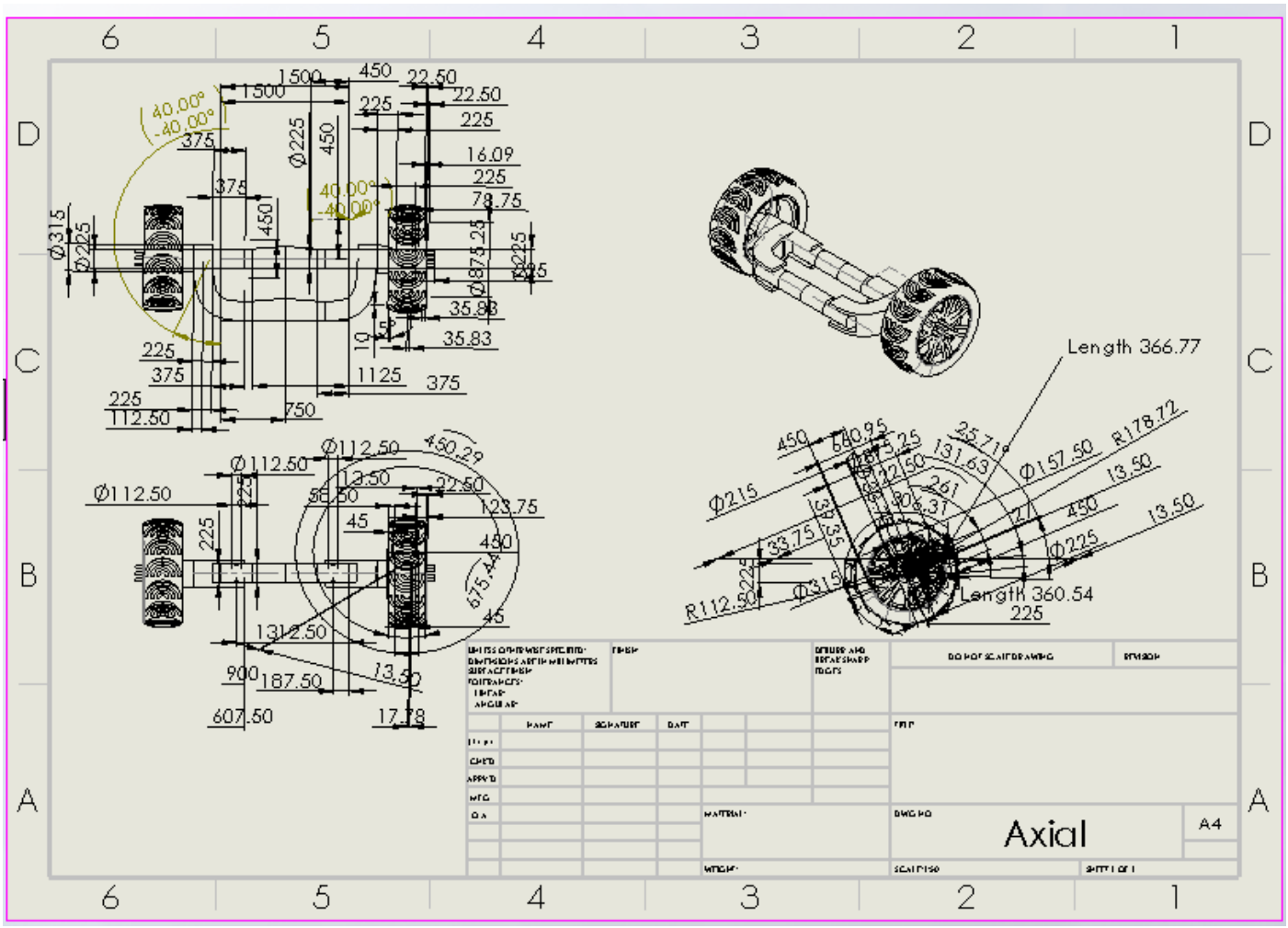


Figure 40: Axial Assembly Drawing

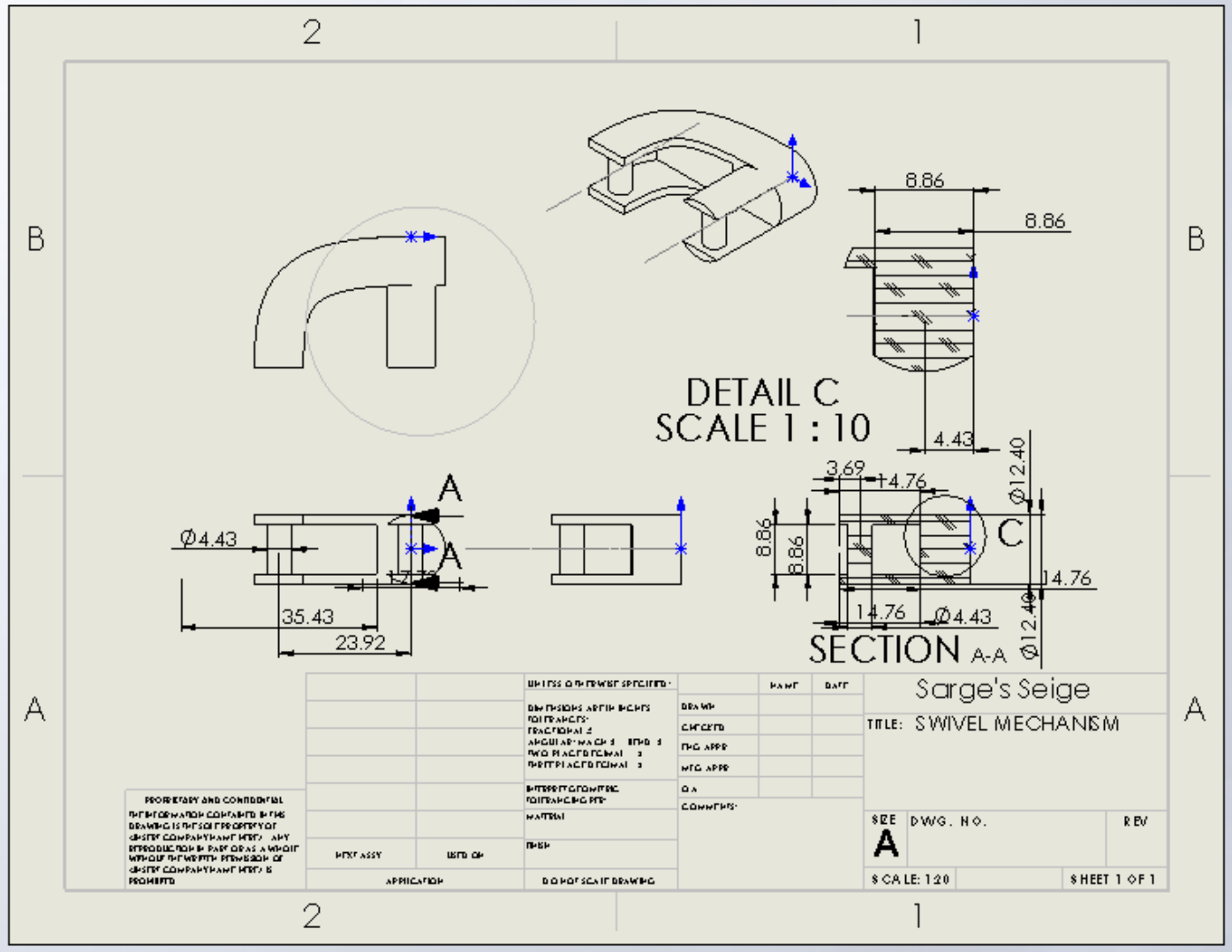


Figure 41: Swivel Mechanism Drawings with a top, front, right, isometric, segment and detailed views

6.9 Appendix 9: Equation Manager, Material Properties and Referenced Equations

Equations, Global Variables, and Dimensions

Filter All Fields

Name	Value / Equation	Evaluates to	Comments
Global Variables			
"Radius"	= "Radius@Axial.Assembly"	450	
"Length"	= "Length@Axial.Assembly"	1500	
Add global variable			
Features			
Add feature suppression			
Equations			
"D1@Sketch1"	= "Radius" * 2 / (200 / 70)	315mm	
"D1@Boss-Extrude1"	= "Radius" / 2	225mm	
"D1@Plane1"	= "Radius" / 2	225mm	
"D1@Sketch2"	= "Radius" * 2 / (200 / 70)	315mm	
"D1@Boss-Extrude2"	= "Length" / 4	375mm	
"D1@Plane2"	= "Length" / 4	375mm	
"D1@Sketch6"	= "Radius" / 2	225mm	
"D2@Sketch6"	= "Radius"	450mm	
"D1@Cut-Extrude2"	= "Length" / 4	375mm	
"D1@Sketch7"	= "Radius" / 4	112.5mm	
"D1@Plane4"	= "Radius"	450mm	
"D1@Sketch17"	= "Radius" + "Radius" / (200 / 70)	607.5mm	
"D2@Sketch17"	= "Radius" / 2	225mm	
"D3@Sketch17"	= "Radius" * 2	900mm	
"D1@Sketch18"	= "Radius" / 4	112.5mm	
"D1@Sketch19"	= "Radius" / 4	112.5mm	
"D2@Sketch19"	= "Length" / 16	93.75mm	
"D2@Sketch7"	= "Length" / 16	93.75mm	
Add equation			

OK | Cancel | Import... | Export... | Help

Automatically rebuild | Angular equation units: Degrees | Automatic solve order

Link to external file:

Figure 42: Equation Manager with Reference Equations

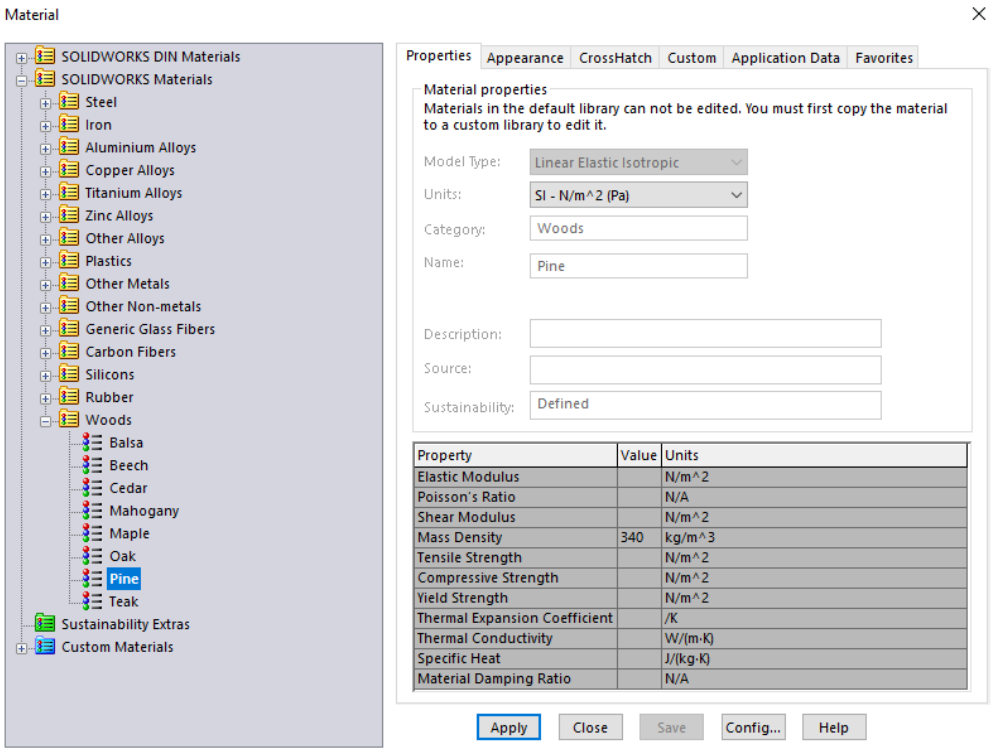


Figure 43:Material Property Manager

6.10 Appendix 10: Solidworks Manipulator Sub Routine

```
Option Explicit
Sub SolidworksManipulator()
    'Connor McDowall'
    'This sub changes the parameters of the Axial components based on the users input'
    'The drive axial, Wheel Fastener, Tire, WheelRim Scalable and Swivel Mechanism
    'Radius, Length and Material Parameters are Changed'

    'Dimension Variables'
    Dim swApp As Object 'Solidworks Application Pointer'
    Dim objFile As Object 'Solidworks Part or Assembly Pointer'
    Dim assembly As String 'Name of the Assembly you are changing'
    Dim part As String 'Name of the part changing'
    Dim response As Integer 'Response from message box'
    Dim swEqnMgr As Object 'Equation Manager for part or assembly'
    Dim i As Integer 'Counting Variable'
    Dim strSplit1 As Variant 'Splitting Variable '
    Dim strSplit2 As Variant 'Splitting Variable '
    Dim material As Variant 'Material variable'
    Dim length As String 'length string value'
    Dim radius As String 'Radius String Value'

    'Make sure the user does the right thing'
    response = MsgBox("Select yes if wish to modify model and have all relevant files open. Select no otherwise", vbYesNo)
    If response = vbNo Then
        Exit Sub
    End If

    'Set Assembly Name'
    assembly = "Axial.SLDASM"
    'Set pointer to the Solidworks Application'
    Set swApp = CreateObject("Sldworks.Application")
    'Make the assembly the active document model'
    swApp.ActivateDoc (assembly)
    'Set pointer to make changes to the assembly'
    Set objFile = swApp.ActivateDoc(assembly)
    'Set pointer to Equation manager tool'
    Set swEqnMgr = objFile.GetEquationMgr
    'Change the Radius and Length Global variables'

    'Convert Values in the excel spreadsheet into strings'
    length = CStr(Worksheets("ReferenceData").Range("SolidworksWheelSize"))
    radius = CStr(Worksheets("ReferenceData").Range("SolidworksSize"))
    'Split the global variable storage equation strings at the equals sign'
    strSplit1 = Split(swEqnMgr.Equation(0), "=")
    strSplit2 = Split(swEqnMgr.Equation(1), "=")
    'Append the length and radius value onto the end of the strings'
    swEqnMgr.Equation(0) = strSplit1(0) & "= " & length
    swEqnMgr.Equation(1) = strSplit2(0) & "= " & radius
```

```
    'Change the material of each wooden part
    'Drive Axial'
    part = "Drive Axial.SLDPRT"
    'Make the part the active document model'
    swApp.ActivateDoc (part)
    'Set the pointer to the object'
    Set objFile = swApp.ActivateDoc(part)
    'Change the material'
    material = objFile.SetMaterialPropertyName("SOLIDWORKS Materials.sldmat", Worksheets("ReferenceData").Range("SolidworksMaterial"))

    'WheelRim Scalable'
    part = "WheelRim Scalable.SLDPRT"
    'Make the part the active document model'
    swApp.ActivateDoc (part)
    'Set the pointer to the object'
    Set objFile = swApp.ActivateDoc(part)
    'Change the material'
    material = objFile.SetMaterialPropertyName("SOLIDWORKS Materials.sldmat", Worksheets("ReferenceData").Range("SolidworksMaterial"))

    'Swivel Mechanism'
    part = "Swivel Mechanism.SLDPRT"
    'Make the part the active document model'
    swApp.ActivateDoc (part)
    'Set the pointer to the object'
    Set objFile = swApp.ActivateDoc(part)
    'Change the material'
    material = objFile.SetMaterialPropertyName("SOLIDWORKS Materials.sldmat", Worksheets("ReferenceData").Range("SolidworksMaterial"))

    'Wheel Fastener'
    part = "Wheel Fastener.SLDPRT"
    'Make the part the active document model'
    swApp.ActivateDoc (part)
    'Set the pointer to the object'
    Set objFile = swApp.ActivateDoc(part)
    'Change the material'
    material = objFile.SetMaterialPropertyName("SOLIDWORKS Materials.sldmat", Worksheets("ReferenceData").Range("SolidworksMaterial"))

    'Rebuild the assembly'
    Set objFile = swApp.ActivateDoc(assembly)
    objFile.EditRebuild
End Sub
```

Figure 44: SolidworksManipulator Sub Routine

6.11 Appendix 11: Additional Tools, Tutorials and Features

Figure 45: Using Different Enviroments as Backgrounds

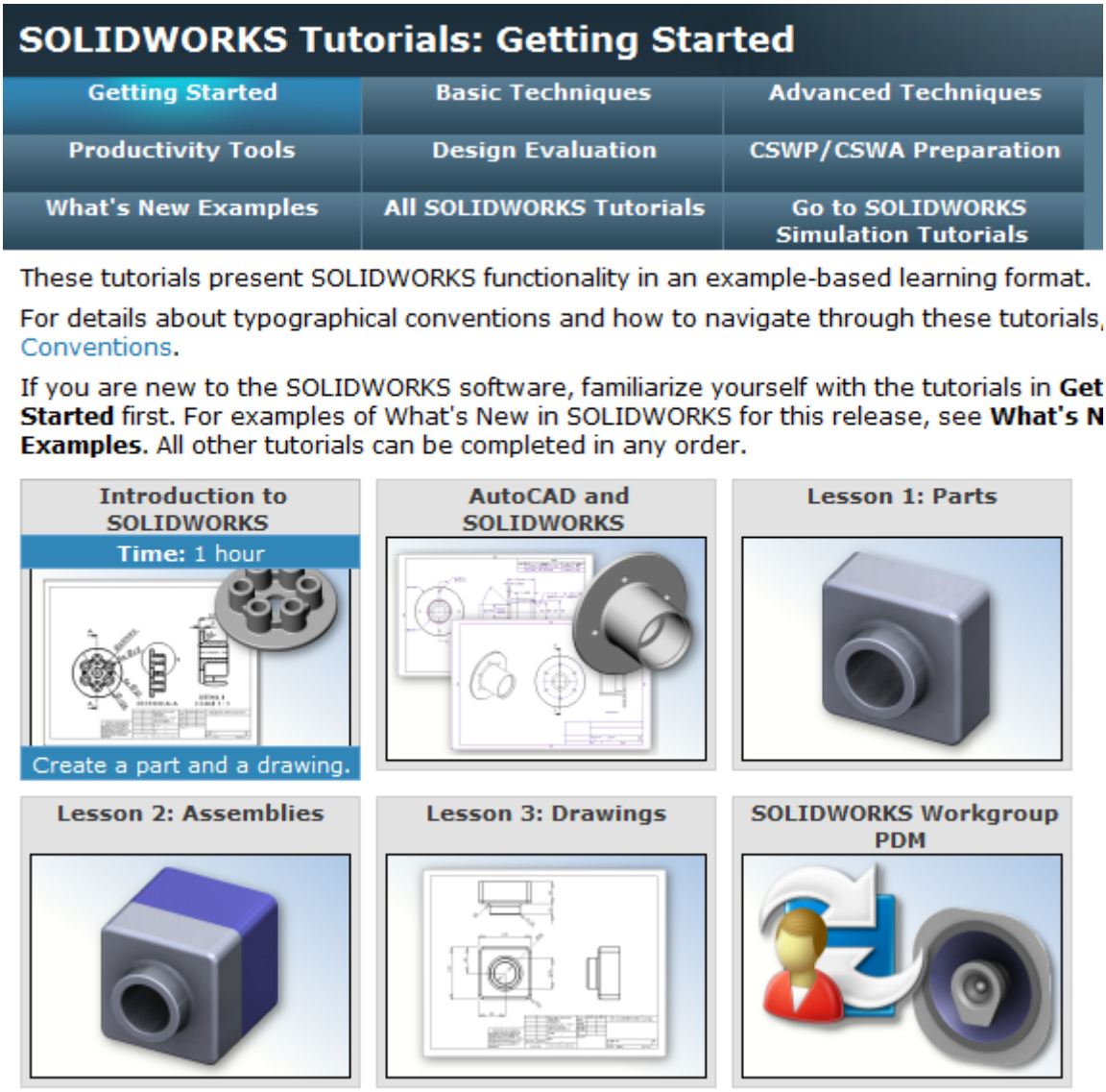


Figure 46: Solidworks Tutorials