

# Creation of Helical and Worm Gears & Utilization of Design Tables on Standard Gears

## Overview

Initial object of this project was to generate a library of multiple gear types, including Standard, Helical, and Worm gears, utilizing Design Tables in the CATIA cad program. However with the complications inherent in gear design, the scope of the project was limited to the production of a tutorial for building Helical and Worm gears in CATIA, supplemented with the creation of Design Tables on the simplest of standard gears.

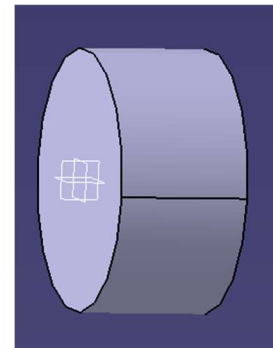
## Creation of Helical Gears

- 1) Open **Part Design Workbench** and set units of length to inches  
**Tools>Options>General>Parameters and Measure**

- 2) Create a sketch on the **yz plane**.  
-Sketch a circle with a **1" diameter** about the origin.  
-Exit the sketch.



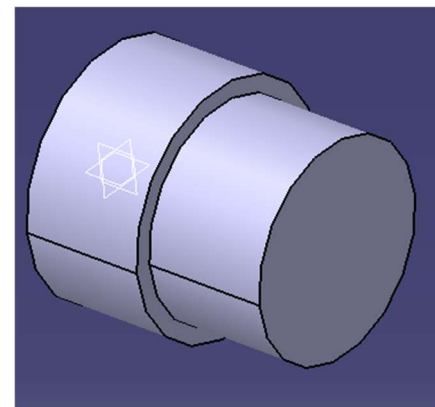
- 3) Select the **pad** icon  
-Extrude the sketch to a **length of 0.5"**  
-Click "Reverse Direction" to set the origin on the front of the face.  
-Click "OK"  
-This is the **face** of the gear



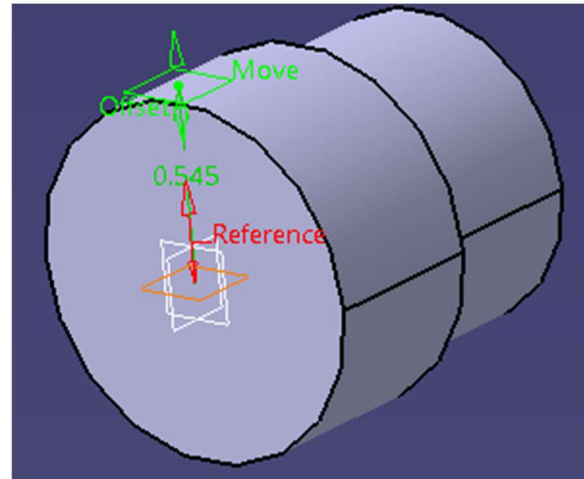
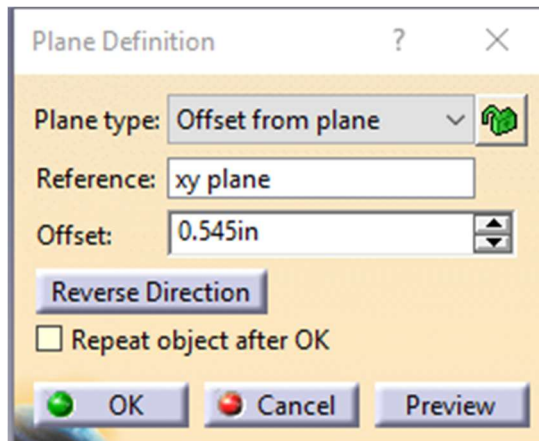
- 4) Create a sketch on the backside of the face (surface opposite of the origin)  
-Sketch a circle with a **diameter of 0.86"**  
-Exit the sketch.



- 5) Select the **pad** icon  
-Extrude this circle to a **length of 0.5"**  
-Click "OK"  
-This is the **hub** of the gear.




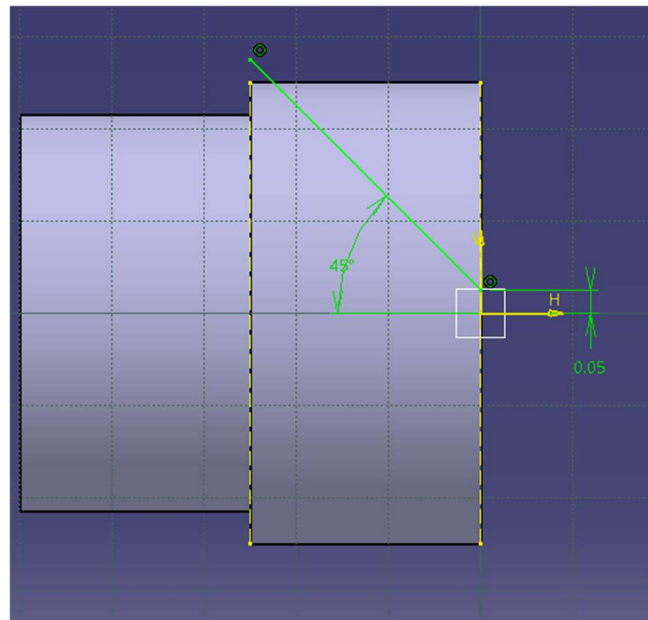
- 6) Select the plane icon  
-Offset from the **xy plane** at a **distance of**



**0.545"**  
-Click "OK"

- 7) Begin a sketch on the plane created in step 6.

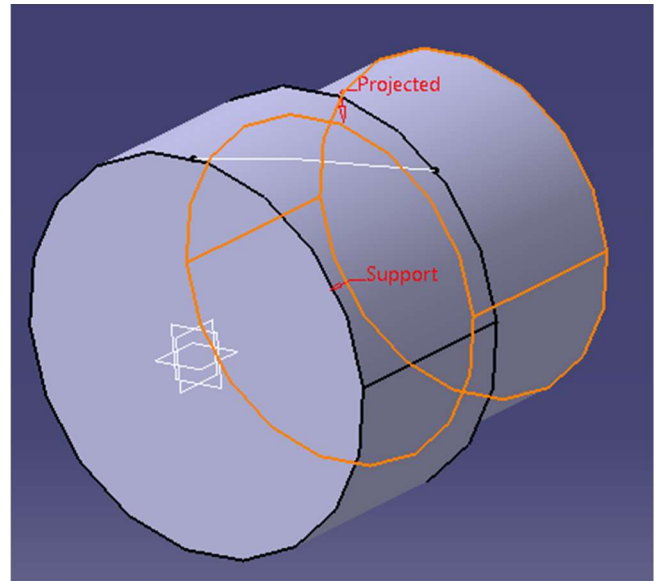
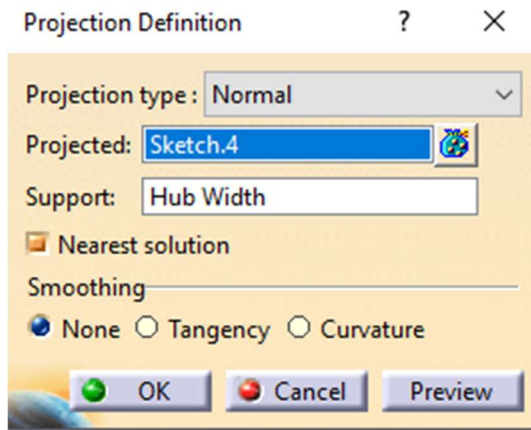
- Select **Project 3D Elements** 
- Click the right edge of the face. Click "OK".
- Do the same for the left edge of the face.
- Double click the right yellow projection line, and check the "Construction element" box.
- Do the same for the left yellow projected line.
- Draw a line on the right projected line, **starting 0.05" above the H-axis**. Drag the line to the left projection line, with the endpoint of the line higher than the start point of the line.
- Create an **angle of 45°** between the inside of the line and the H-axis as shown.




8) Enter the **Generative Shape Design** workbench. **Start>Shape>Generative Shape Design**

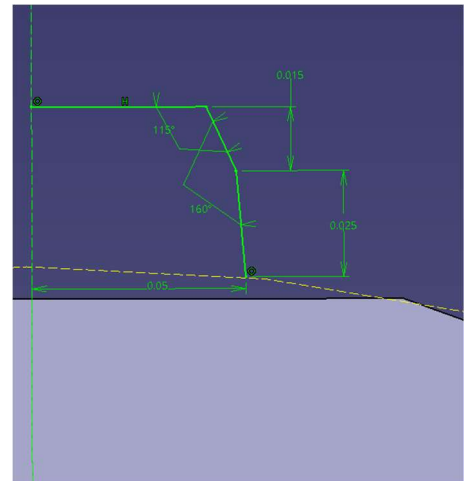
Select the **Projection** icon 

- Projected box: select the sketch of the line created in step 7.
- Support box: select the pad of the hub created in step 5.
- Click "OK"



9) Enter the **Part Design** Workbench  
Begin a sketch on the **yz plane**.

- Select **Project 3D Elements**  and select the outer edge of the face. Turn this projected line into a construction element (see step 7 for instructions on how to do this).
- Create a vertical axis construction line on the V-axis (set end length at an arbitrary distance above yellow construction line).



-Begin the sketch of the tooth with straight lines of the dimensions shown.

**\*\*Note:** Be sure that the nearly vertical line connected to the yellow projected construction line does **not** have a vertical line constraint

-Select the **Corner** icon



-Create a corner between the two upright lines with a **radius of .016"**

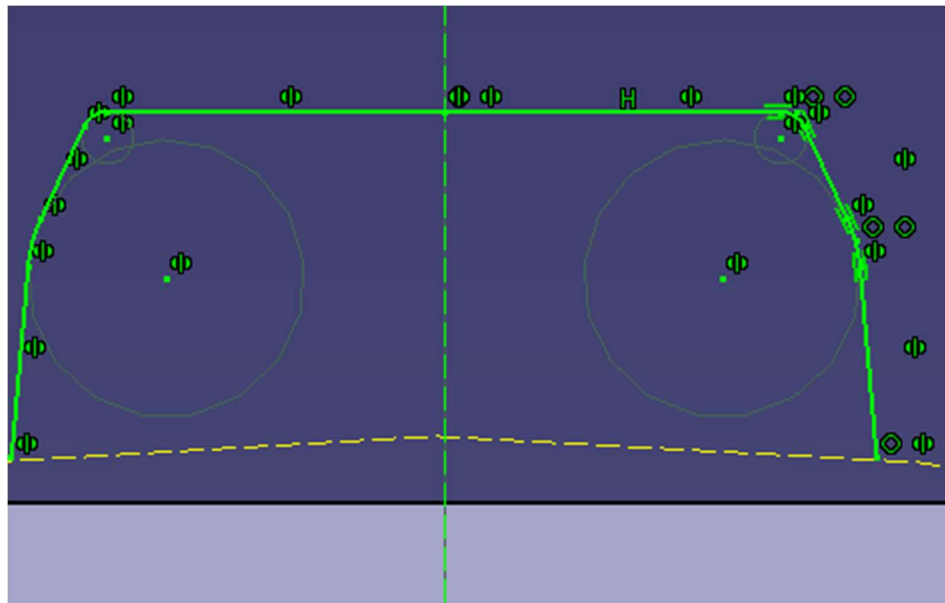
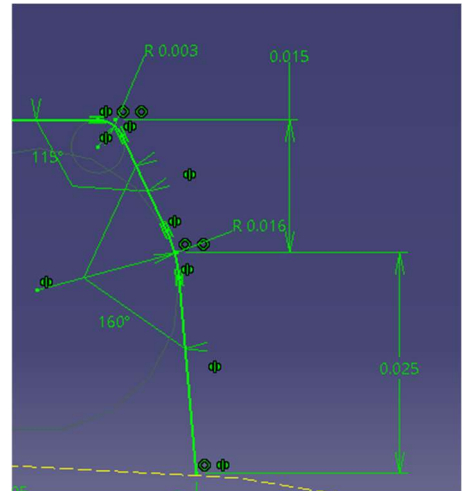
-Create a corner between the horizontal and its connected line with a **radius of 0.003"**

-Select the **Mirror** icon



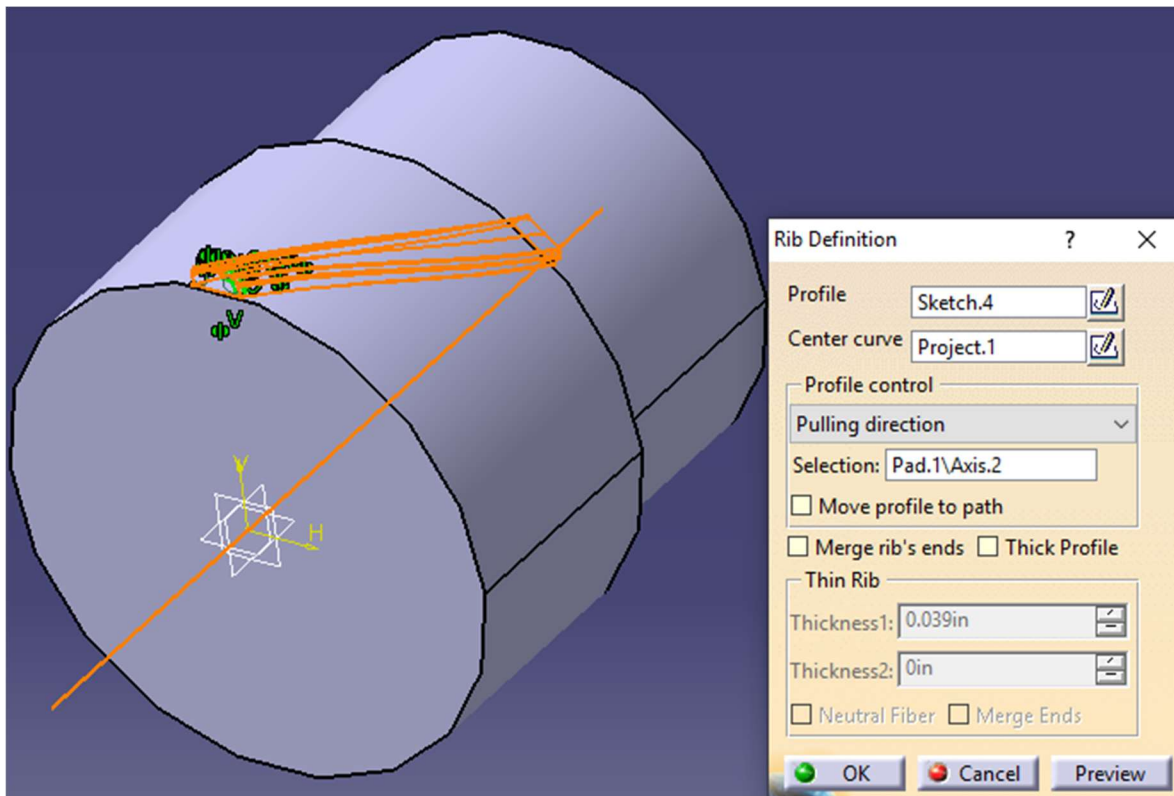
-Mirror the **three lines and two corners** about the vertical construction axis to complete the sketch of the tooth.

-Exit the sketch




10) Select the **Rib** icon 

- Profile** box: select the sketch of the tooth created in step 9.
- Center curve** box: select the projected line created in step 8.
- Select **Pulling Direction** in the drop down box under Profile control. Select the center axis that appears when the gear is hovered over.
- Click "OK"



-If loft/blend errors arise, they may be ignored in this application.

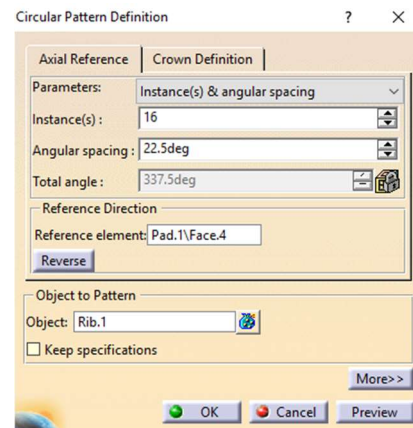
11) Select the **Circular Pattern** icon . It is located in the drop down box of the rectangular pattern icon.

-**Instances** box: 16 (number of teeth on the gear)

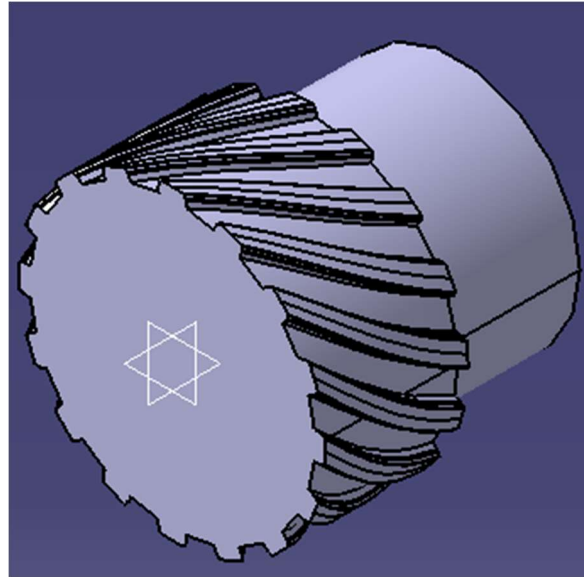
-**Reference element**: Select the circular curved surface of the face that the tooth is lying on.

-**Object to Pattern**: Select the rib created in step 10.

-Click "OK"



This is how the helical gear should look through step 11.




**12) Begin a sketch on the yz plane**

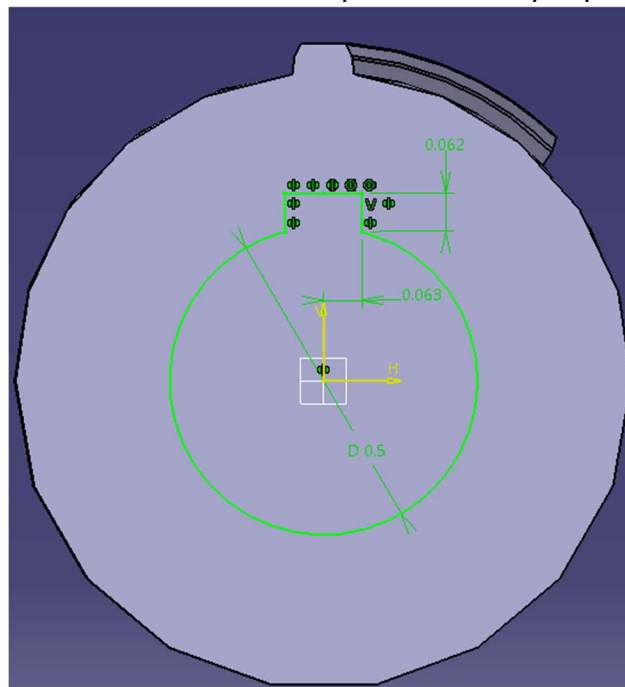
-Sketch the keyway of the gear as shown

-Create a circle with a **diameter of .5"** about the origin.

-Sketch the lines for the keyway width and depth with dimensions as shown.

-Select the **Quick Trim** icon . It is located in the drop down box of the Trim icon within the sketch toolbar.

-Trim away the portion of the circle directly below the keyway as shown.



-Exit the sketch

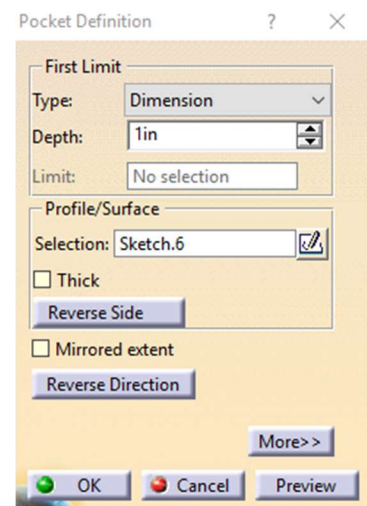
**13) Select the Pocket icon**



-Select the sketch created in step 12.

-The pocket runs up to the next surface, or a depth of 1”.

-Select “OK”

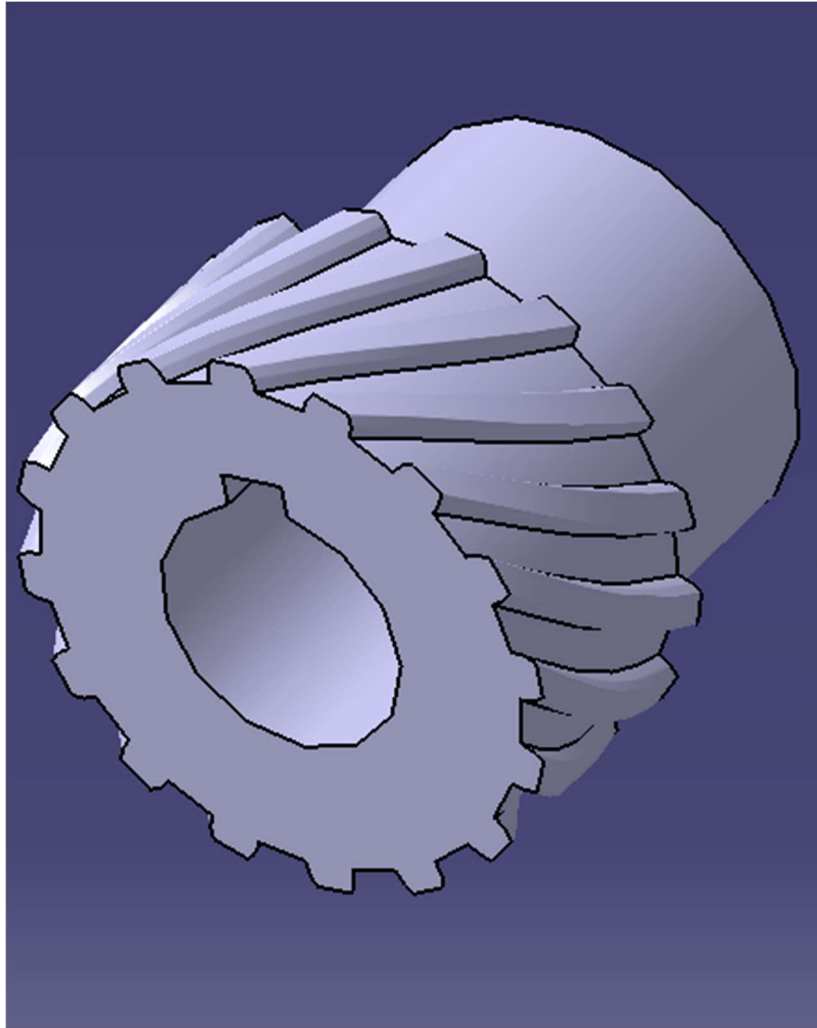


**14) Select the Shading with Edges without Smooth edges**



Finished Helical Gear with 45° Helix Angle

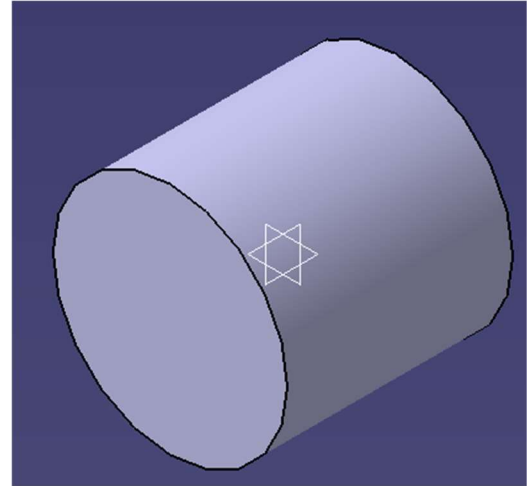





## Creation of Worm Gears


- 1) Open **Part Design Workbench** and set units of length to inches  
**Tools>Options>General>Parameters and Measure**

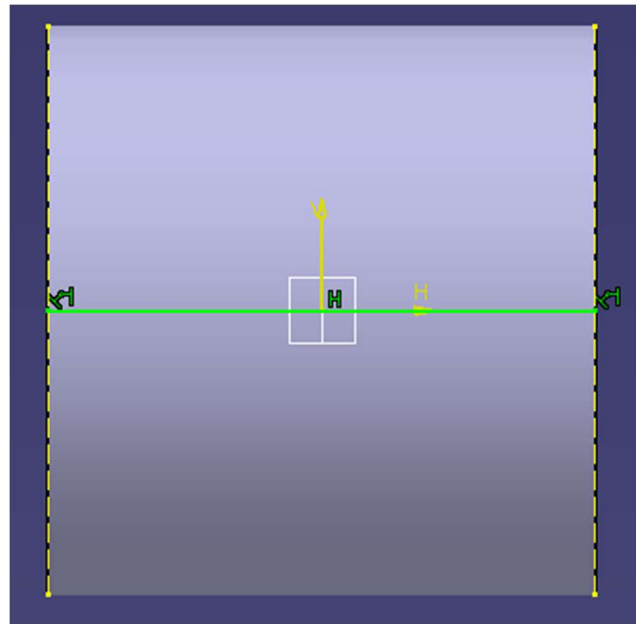
- 2) Create a sketch on the **yz plane**.
  - Sketch a circle with a **1.17" diameter** about the origin.
  - Exit the sketch



- 3) Select the **pad** icon 
  - Extrude the sketch to a **length of 0.563"**
  - Check the "Mirrored Extent" box to set the origin in the center of the gear.
  - Click "OK"

- 4) Create a sketch on the **zx plane**.

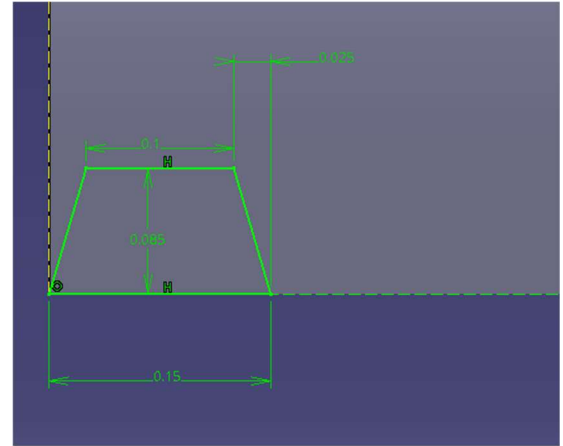
- Select **Project 3D Elements** 
- Click the right edge of the cylinder. Click "OK".
- Do the same for the left edge of the face.
- Double click the right yellow projection line, and check the "Construction element" box.
- Do the same for the left yellow projected line.
- Sketch a horizontal line from one construction line to the other, lying on the H-axis.
- The total length of this line should be **1.126"** (.563" on each side of the origin)
- Exit the sketch



5) Create a sketch on the **zx plane**

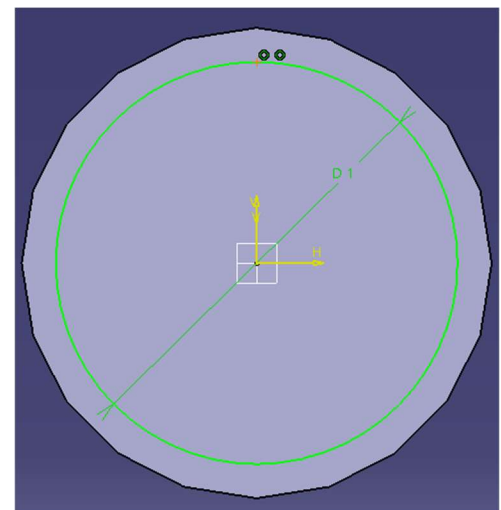


- Select **Project 3D Elements**
- Follow the same procedure as step 4 on creating the projected construction lines on each edge of the cylinder.
- Sketch a horizontal line coincident with each construction line at a **distance of .585" below the H axis.**
- Double click this line and check the "construction element" box.
- Sketch the tooth as shown in the bottom left corner of the cylinder.



6) Create a sketch on the leftmost face of the cylinder

- Sketch a circle about the origin with a **diameter of 1"**
- Create a point coincident with the circle directly above the V-axis.
- Exit the sketch

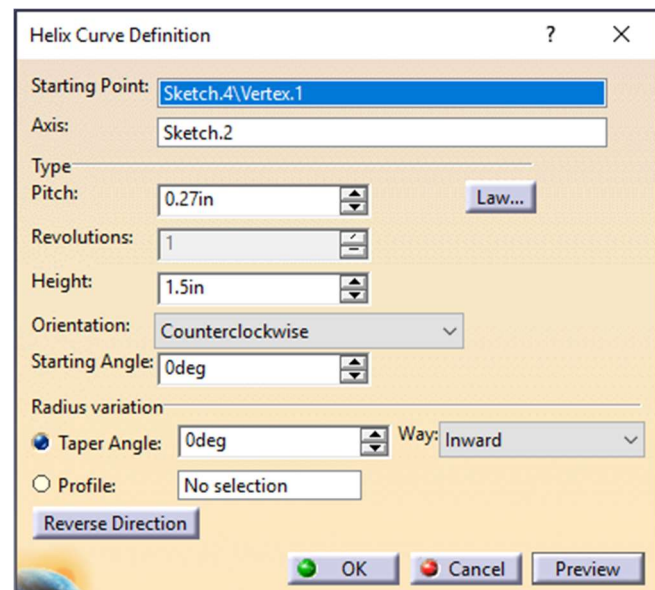
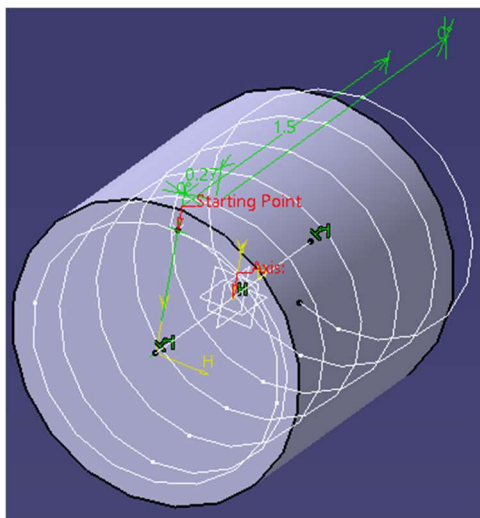


7) Enter the **Generative Shape Design** workbench. **Start>Shape>Generative Shape Design**



- Select the **Helix** icon. It is located in the drop-down box of the spline icon in the toolbar.
- Starting Point** box: Select the point created in step 6.
- Axis** box: select the horizontal line created in step 4.

**Pitch**  
box:



**0.27in**

**-Height box: 1.5in**

**-Click "OK"**

**8) Enter the Part Design Workbench**



**-Select the Slot icon**

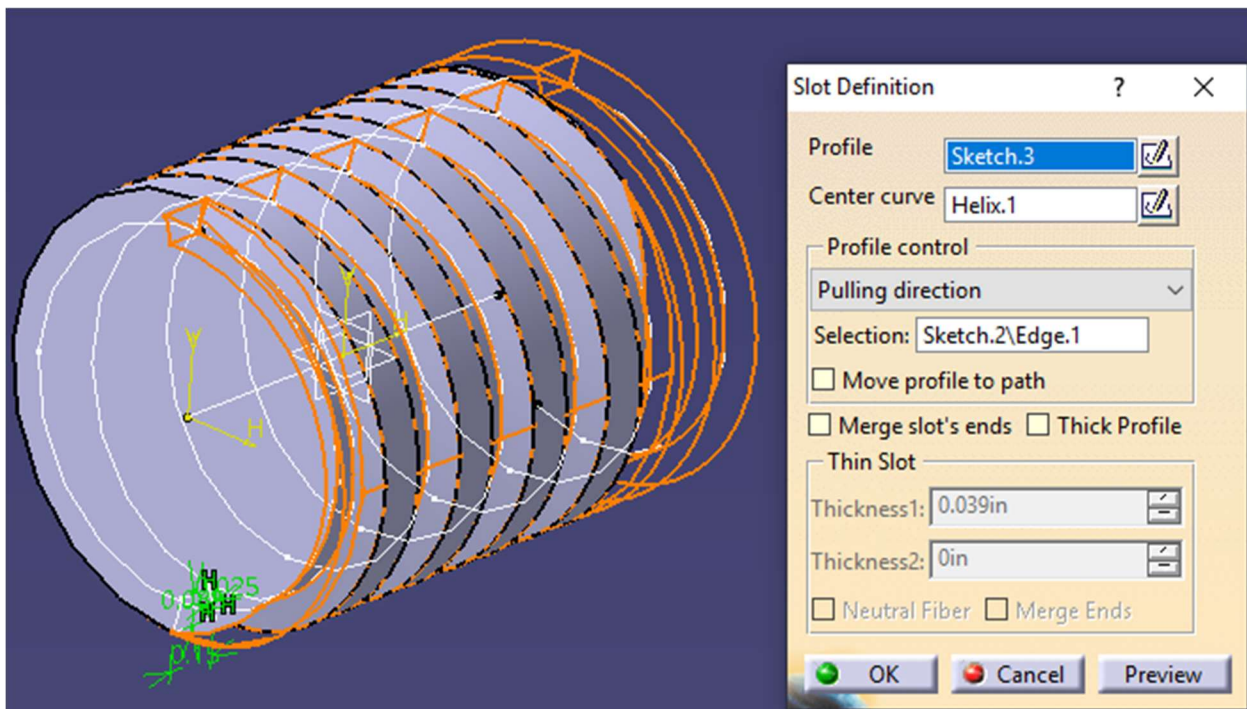
**-Profile box:** Select the tooth sketch created in step 5.

**-Center curve box:** Select the helix created in step 7.

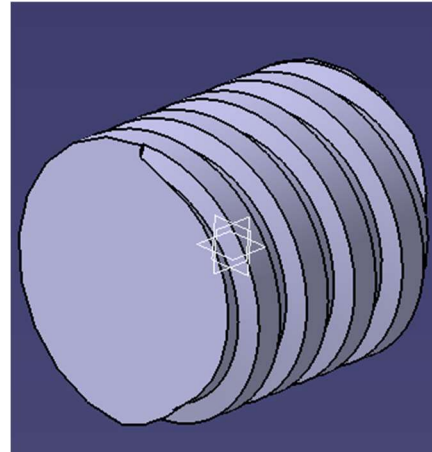
**-Profile control:** Select **Pulling Direction** in the drop down box

**-Selection box:** Select the line sketch created in step 4.

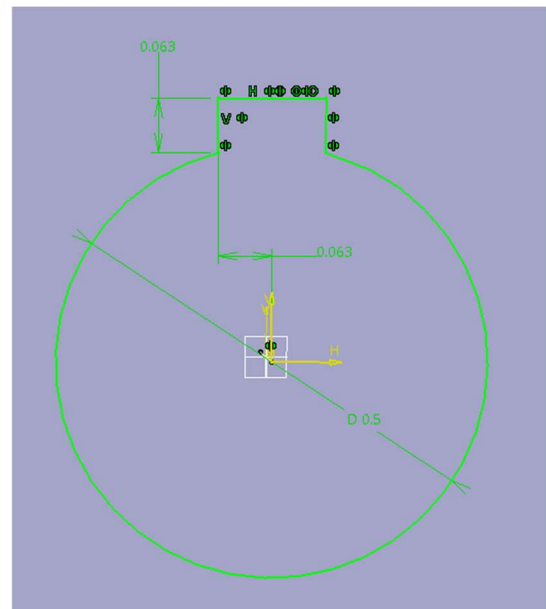
**-Click "OK"**



This is how the gear should look through step 8:



- 9) Begin a sketch on the leftmost face of the gear  
-Create the keyway with dimensions as shown  
-Exit the sketch

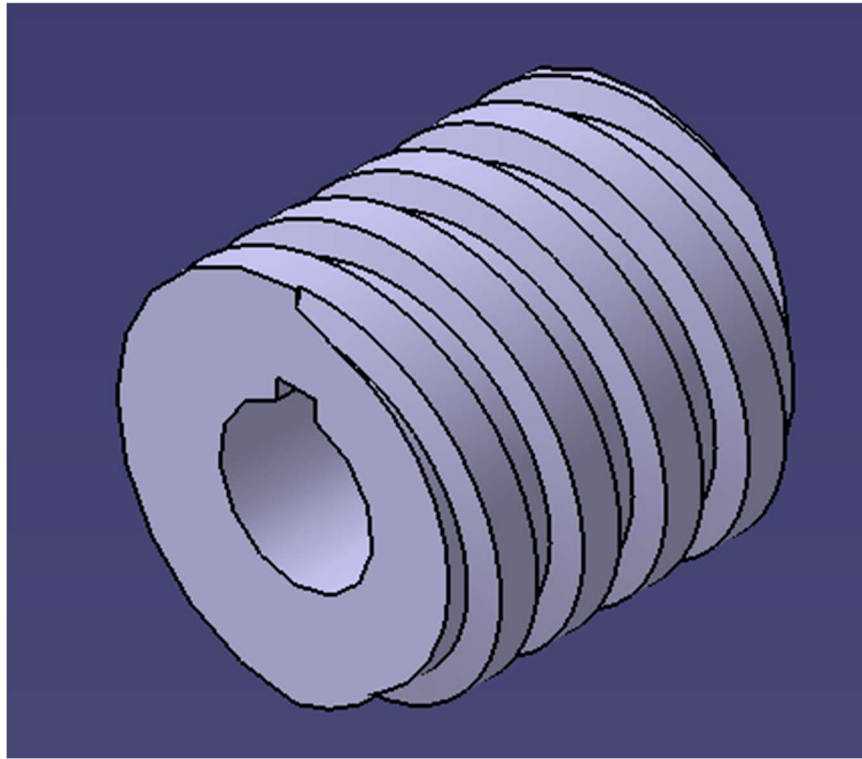


- 10) Select the **Pocket** icon



- Type** box: Up to next (through the gear to the rightmost face)
- Select the sketch created in step 9 as the feature to be pocketed.
- Click "OK"

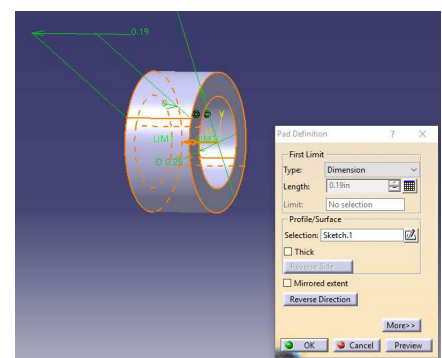
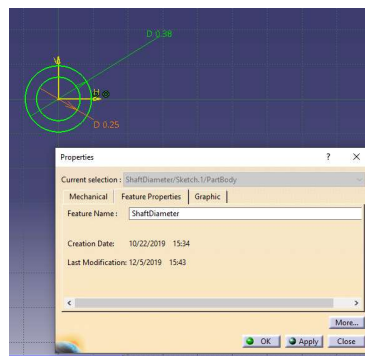
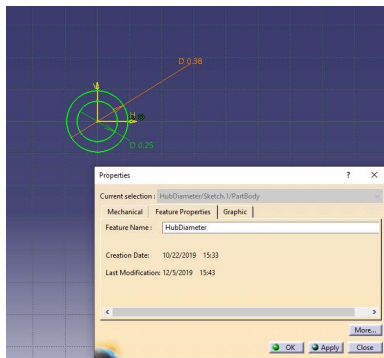
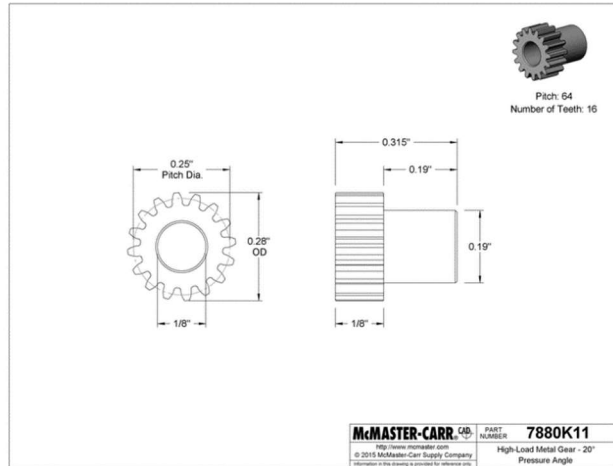
Finished Worm Gear



## Creation of Standard Gear Design Table

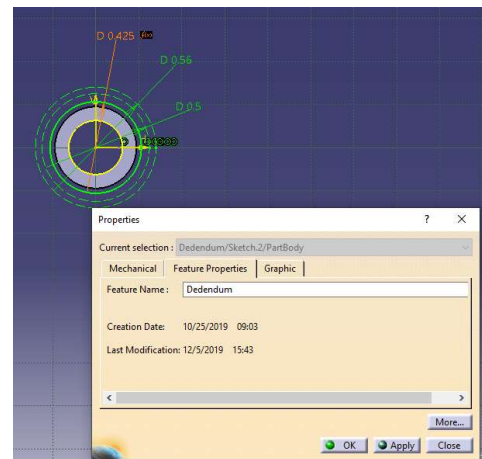
The creation of the standard gear design tables starts with designing a base part to make associations with in the design table. To start, obtain the dimensions of a gear from McMaster-Carr and begin emulating the part in CATIA. I began with the part 7880K11 as it was the first on McMaster-Carr's gear list.

Start the part by creating the hub body and shaft hole as a single sketch. Using the properties window of each dimension be sure to give them a distinctive names to easily associate to the design table later on. (HubDiameter, ShaftDiameter)



Extrude this drawing to the hub length dimension provided by McMaster-Carr. Update Pad properties to rename Pad to distinctive name to easily associate to the design table later on. (HubPad)

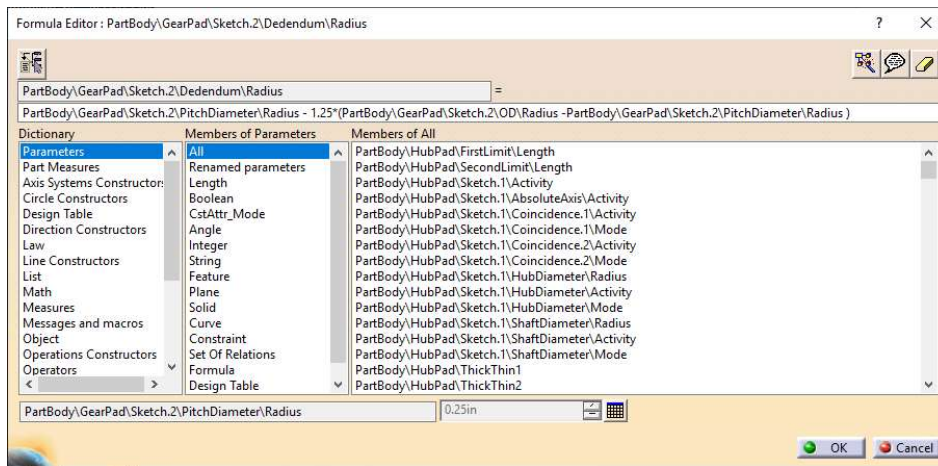
Next create a sketch for the gear pad. This sketch is where we will start to use some formulas to calculate specific gear aspects to simplify the creation of involute gears. Using construction tools create the Pitch Diameter circle, and the OD Circle, be sure to give them a distinctive names to easily associate to the formula editor and the design table later on. (PitchDiameter, OD)



Project the shaft diameter to the current sketch. Then create the final circle shown called the Dedendum. This will be the bottom of the gear teeth and will give a base off of which to build the teeth. It is controlled by a formula and the formula editor window can be utilized to parametrically link to the other defined dimensions.

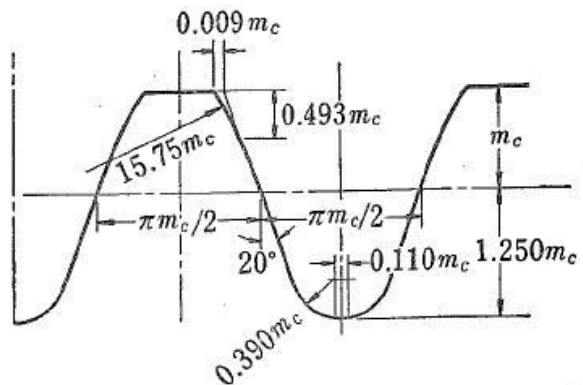
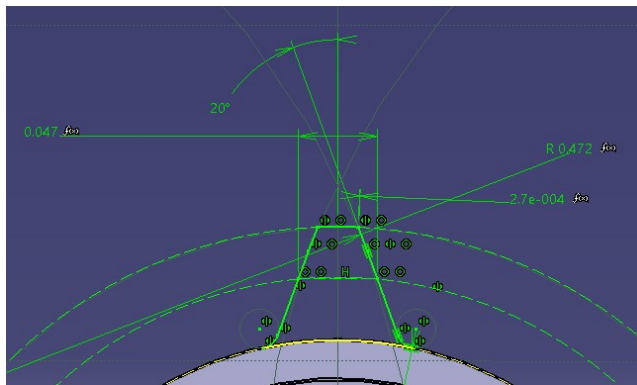
$$\text{Module } (m) = \text{Addendum } (ha) = \text{OD} - \text{PD}$$

$$\text{Dedendum } (hf) = 1.25 * m = 1.25 (\text{OD} - \text{PD})$$



Extrude this sketch to create the GearPad. This will allow for the tooth to be individually designed interstecting the solid at the dedendum diameter.

Create another sketch on the same plane that the GearPad sketch was drawn on. This sketch will dimension out the teeth themselves. Using the module found above and the figure shown here we can create a formula driven sketch of the teeth that will scale when used with the design table.





Extrude this sketch as ToothPad with the type "Up to plane" and select the opposite GearPad face. There for creating one solid tooth to the appropriate length that can then be patterned by means of the Circular Pattern tool.

For the angular spacing callout again utilize the formula editor and define as such. (Be sure to include deg)

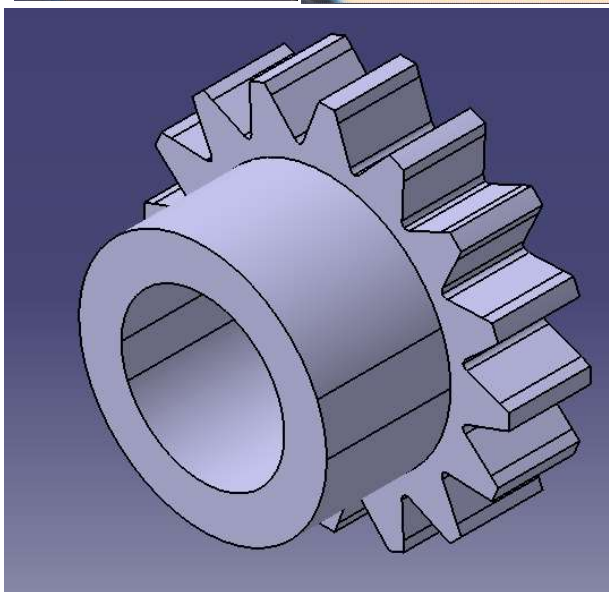
$$\text{AngularSpacing} = 360 \text{ deg} / \text{Angular Number}$$

This concludes the initial construction of the gear, the result should look similar to shown. Next is the creating and association of the design table to the created part and its features.

Start by opening Excel and McMaster-Carr to appropriate page, in this case "Metal Gears and Gear Racks—20° Pressure Angle" Select the table from McMaster-Carr and copy into Excel. Include title row and all columns.

In excel set the column titles to match Catia Appropriate Titles meaning no spaces. For example Pitch is fine but Number of Teeth can be reduced to NumberOfTeeth without a space.

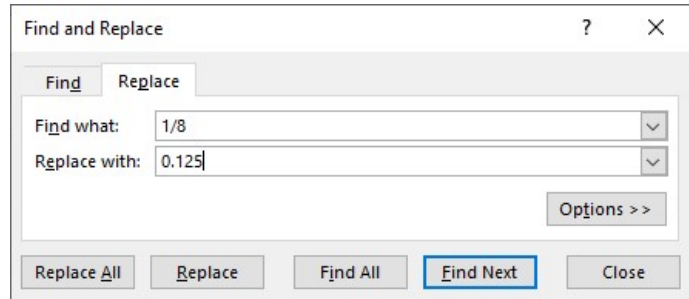
Number of Teeth	Module	Pressure Angle	Overall Width	or Shaft	Material	Teeth Treatment	Hub Dia.	Face Wd.	Set Screw Thread Size	Part No.	Price
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K11	\$12.09	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K12	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K13	15.53	
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16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K67	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K68	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K69	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K70	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K71	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K72	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K73	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K74	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K75	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K76	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K77	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K78	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K79	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K80	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K81	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K82	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K83	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K84	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K85	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K86	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K87	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K88	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K89	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K90	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K91	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K92	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K93	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K94	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K95	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K96	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K97	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K98	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K99	15.99	
16	1.0	20	0.515	0.6	Brass		0.515	0.19	7880K100	15.99	



Pitch	Number of Teeth	Pitch	OD	Face Wd.	Overall Wd.	For Shaft Dia.	Material	Teeth Treatment	Hub Dia.	Wd.	Set Screw Thread Size	Each
16	16	1.0	0.515	0.19	0.515	0.19	Brass		0.515	0.19	7880K11	\$12.09
16	16	1.0	0.515	0.19	0.515	0.19	Brass		0.515	0.19	7880K12	15.99
16	16	1.0	0.515	0.19	0.515	0.19	Brass		0.515	0.19	7880K13	15.53
16	16	1.0	0.515	0.19	0.515	0.19	Brass		0.515	0.19	7880K14	15.99
16	16	1.0	0.515	0.19	0.515	0.19	Brass		0.515	0.19	7880K15	15.99
16	16	1.0	0.515	0.19	0.515	0.19	Brass		0.515	0.19	7880K16	15.99
16	16	1.0	0.515	0.19	0.515	0.19	Brass		0.515	0.19	7880K17	15.99
16	16	1.0	0.515	0.19	0.515	0.19	Brass		0.515	0.19	7880K18	15.99
16	16	1.0	0.515	0.19	0.515	0.19	Brass		0.515	0.19	7880K19	15.99
16	16	1.0	0.515	0.19	0.515							

(PitchDiameter, OD, FaceWidth, OverallWidth, ShaftDiameter, HubDiameter, HubWidth, PartNumber)

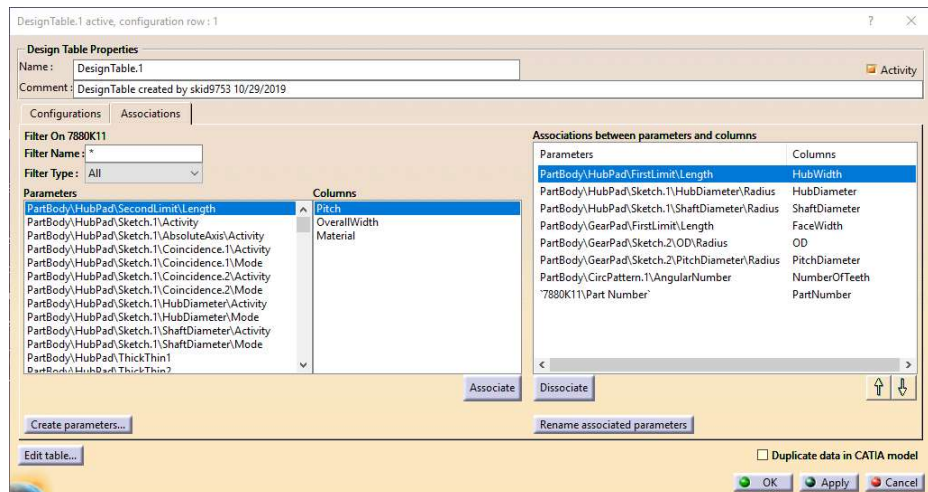
Important, Catia can only recognize three decimal places in its design tables. An important aspect of utilizing this excel sheet is to convert the fractions to decimals. While tempting to remove the quotation marks and have Excel convert them for you, this will most likely result in it returning a date value instead of a decimal. It was also discovered that Catia requires an indication of dimension therefore leave the quotation marks alone and instead manually Find/Replace fractions with their decimal equivalent not exceeding three decimal places. At this point remove columns and rows that are not required. Final formatting should look as such.



	Pitch	NumberOfTeeth	PitchDiameter	OD	FaceWidth	OverallWidth	ShaftDiameter	Material	HubDiameter	HubWidth	PartNumber
1	64	16	0.25"	0.28"	0.125"	0.315"	0.125"	Brass	0.19"	0.19"	7880K11
2	64	24	0.375"	0.41"	0.125"	0.315"	0.125"	Brass	0.28"	0.19"	7880K13
4	64	28	0.438"	0.47"	0.125"	0.315"	0.125"	Brass	0.34"	0.19"	7880K14
5	48	12	0.25"	0.29"	0.125"	0.315"	0.125"	Brass	0.18"	0.19"	7880K17
6	48	15	0.312"	0.35"	0.125"	0.315"	0.125"	Brass	0.22"	0.19"	7880K18

Returning to Catia utilizing the desing table tool select the excel list you have created and begin associateing the named dimions in your part to the columns of the table. The resulting associations should resmbel the following.

From the feature tree you should now be able to select Relations --> DesignTable.1 --> Configuration, right click at the very bottom select Definition. You may have to do this twice for the appropriate selection screen to pop up. Then you should be able to select the configuration you desire and select ok, and the part including part number will update automatically.



DesignTable.1 active, configuration row : 1

**Design Table Properties**  
Name: DesignTable.1  
Comment: DesignTable created by skid9753 10/29/2019

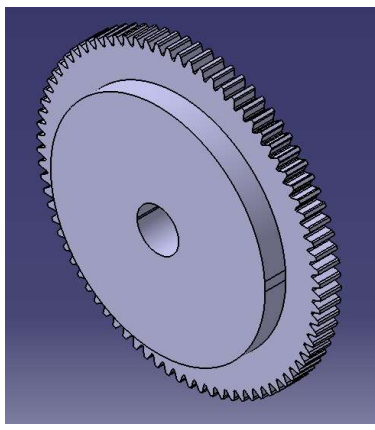
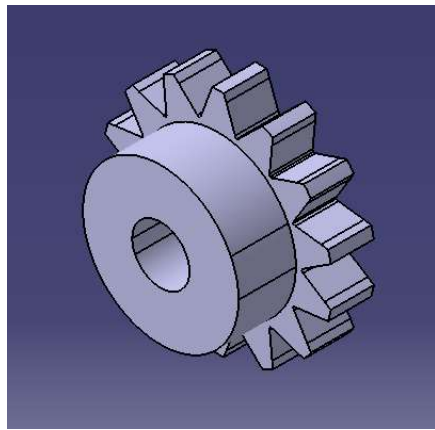
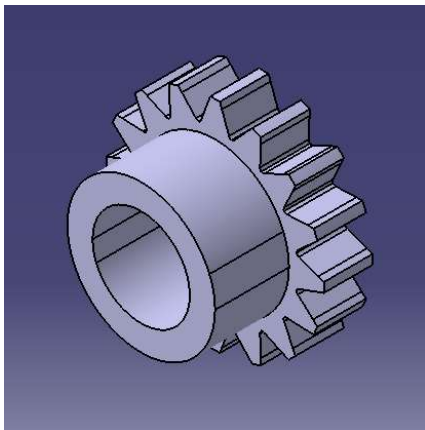
Configurations | Associations

Filter:  Edit...

Line	PartBody\HubPad\FirstLimit\Length	PartBody\HubPad\Sketch.1\HubDiameter\Radius	PartBody\HubPad\Sketch.1\ShaftDiameter\Radius	PartBody\GearPad\FirstLimit\Len
<1>	0.19in	0.19in	0.125in	0.125in
2	0.19in	0.28in	0.125in	0.125in
3	0.19in	0.34in	0.125in	0.125in
4	0.19in	0.18in	0.125in	0.125in
5	0.19in	0.22in	0.125in	0.125in
6	0.19in	0.28in	0.125in	0.125in
7	0.25in	0.38in	0.188in	0.125in
8	0.25in	0.5in	0.188in	0.125in
9	0.25in	0.63in	0.25in	0.125in
10	0.25in	0.28in	0.125in	0.188in
11	0.25in	0.34in	0.125in	0.188in
12	0.25in	0.4in	0.188in	0.188in
13	0.25in	0.47in	0.188in	0.188in
14	0.25in	0.53in	0.188in	0.188in

Edit table...  Duplicate data in CATIA model

OK Apply Cancel



## Lessons Learned

Kirk

Initially, this project was supposed to include complex gears with the use of design tables. After many failed attempts, it was learned that this feat could not be accomplished for certain gears. If it could be accomplished, it is outside of the scope for this course.

- 1) The main issue I encountered with the helical gear was that when attempting to change the diameter of the face, CATIA would respond with an error message saying that there were elements that needed to be converted into construction elements. This face happens to be where the helical teeth lie. To create the teeth, there needed to be a center curve that the sketch could follow when using the rib feature. (The projected line in step 8 of the helical gear tutorial). This was the element that was not a construction element that the error message was referring to. An attempt was made to convert this projected center curve into a construction element. However, a rib cannot be created following a construction element, making this attempt impossible.
- 2) This same issue arose when creating the worm gear. This time, the center curve for the slot feature being the helix curve created in step 7 of the worm gear tutorial.
- 3) I did learn how to create these gears using the helix feature for the worm gear as well as the projected line at an angle for the helical gear.

On a positive note, I did learn how to create these complex gears in a relatively simple manner. This project was engaging and challenging. Creating these tutorials solidified my knowledge on how to make them, as I just recreated them for the tutorials purpose. There's likely other ways to go about creating these types of gears. After Google searching and Youtube videos on how to try and create them another way, I concluded that this was the best and most straightforward way without having to bang your head against a wall.

Will

I was interested in creating a design table gear that followed the involute gear requirements however to create an accurate involute gear requires so many curve objects as to be prohibitive in this scope. I settled for a common spacing and tooth design as found on khkgears.net. This provided me with functions that relate to the module of a gear which can be found from the difference of McMaster-Carr OD and Pitch Diameter. With these dimensions I was able to produce a passable gear tooth design that would scale with the design table.

I learned that CATIA has specific requirements for the dimensions used in the design table, firstly requiring the dimension units, in this case maintaining the quotation marks, in order for the result to be non-nonsensical.

After placing units back into the design table I had columns that would no longer associate with the project. Kirk was very helpful and noted that some of my fraction

conversions to decimal were 4 decimal places, however, CATIA only recognizes 3 decimal places for design tables. I updated the design table and the associations functioned perfectly.