

Weekly Design Evaluation Name _____
- Spur Gear Design Project - Class Design Section (Partial) - 20 Points

Instructions for modeling a pair of spur gears in SolidWorks 2016.

This assignment utilizes typical spur gear parameters, formulas and sketching methods to design a simple involute spur gear pair for use in an assembly.

Requirements:

- **Precision: 4 significant digits – no more – no less**
- Material: a metal of your choice
- To understand the terminology and methods of gear design read and understand the information provided through the links on the Resources Page of your website and through other relevant resources.

Formulas needed for this assignment:

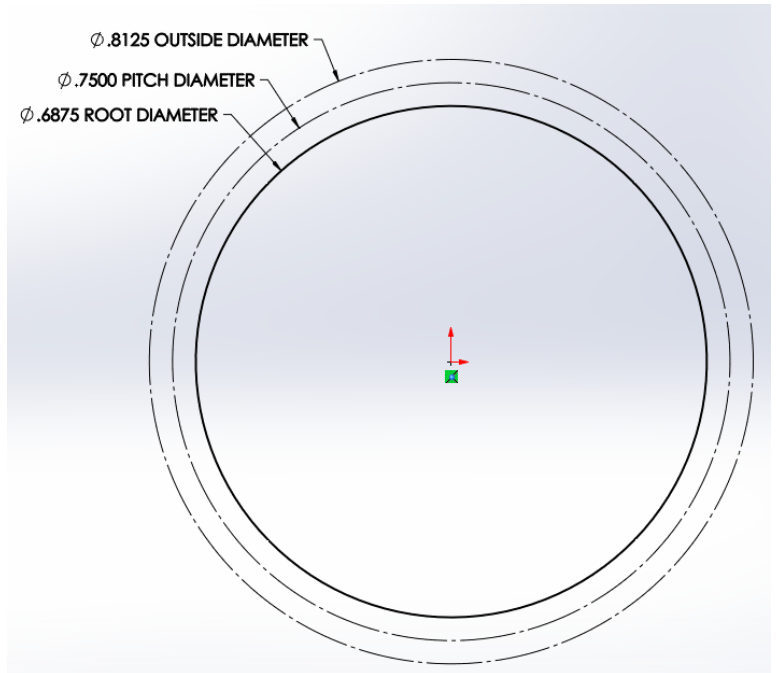
Gear Ratio	Gr	$Gr=N2/N1$	Given	Divide the teeth on the Gear Wheel by the teeth on the Pinion Gear
Diametral Pitch	Pd	$Pd=Nx/Dp$	Given	Ratio of the number of teeth per arc inch of Pitch Diameter (a common value between both gears)
Teeth, Pinion Gear	N1	$N1=Dp1*Pd$	Given	
Teeth, Gear Wheel	N2	$N2=Dp2*Pd$	Determine	<u>Use Gear Ratio</u> for this assignment
Outside Diameter	Do	$Dox=(Nx+2)/Pd$	Determine	Maximum extent of gear tooth
Pitch Diameter	Dp	$Dpx=Nx/Pd$	Determine	Imaginary circle approximately in the center of the teeth, both gear pitch diameters will contact each other tangentially
Root Diameter	Dr	$Drx=(Nx-2)/Pd$	Determine	Maximum extent of tooth cut
Pitch Point Diameter	Dpp	$Dppx=1/4*Dpx$	Determine	Used to model the involute edge on the top portion of each tooth
Base Diameter	Db	$(Dbx=Dpx \cos \emptyset)$	Use sketch geometry	One of two elements that defines a Pitch Point which is the center of a Pitch Point Diameter circle
Angular Circular Pitch	Ta	$Tax=(360/Nx)$	Determine	Angular distance on the Pitch Diameter between one tooth and the same geometric reference on the next tooth in degrees
Quarter Angular Circular Pitch	Ta4	$Ta4x=Tax/4$	Determine	Quarter of the Angular Circular Pitch which is half the width of a tooth. Used for mirroring partial tooth sketch profiles.
Center to Center Distance	Xc	$Xc=(Dp1+Dp2)/2$	Determine	Distance between centers of both meshed gears used for the mounting plate part in the assembly.

Notes: The x in the notations above, such as Nx or Dpx refer to either the Pinion Gear (1) or the Gear Wheel (2). For example in the equation for Pitch Diameter $Dpx = Nx/Pd$, if you are determining the Pitch Diameter for the Gear Wheel then the formula will be $Dp2 = N2/Pd$

Student Number		Spur Gear Design		
		Given		
		Gear Ratio	Diametral Pitch	Pinion Gear Teeth
1	0757	3	27	18
2	5816	1.5	40	26
3	2117	2	32	18
4	9298	3	28	17
5	2851	2.75	16	16
6	6561	2.25	20	20
7	3242	3	30	19
8	4614	1.75	30	16
9	5442	2.25	24	16
10	9848	2.75	18	20
11	4605	3.25	26	16
12	9255	2	25	16
13	5412	2.5	32	20
14	8908	3.5	22	14
15	9414	3	12	20
16	4174	3.5	18	16
17	2325	2	27	17
18	4491	2.5	20	14
19	2257	3	22	16
20	2169	2	31	21
21	8326	2.5	24	16

Procedure:

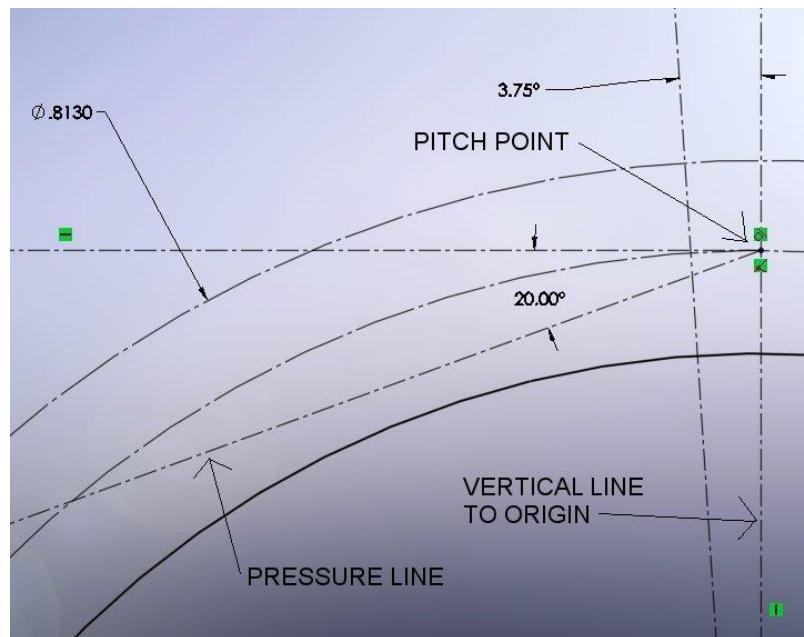
- Using the given Gear Ratio, Diametral Pitch and the number of teeth on the Pinion gear determine the values shown in the table above (except for the Base Diameter (we will solve this using to be established sketch geometry)). For the example demonstrated below we will use the following parameters: Gear Ratio (Gr) of 2.5, Diametral Pitch (Pd) of 32 and the Pinion Gear having 24 teeth (N1).



- Open SolidWorks, open your IPS part template (change the precision to 4 digits after the decimal).

- We will design the Pinion Gear first. Using the values given and derived from the formulas in the table above, sketch three concentric circles from the Origin on the Front Plane: Outside Diameter (Do) circle, Root Diameter (Dr) circle and Pitch Diameter (Dp) circle. The Root Diameter circle will be a Model Arc Sketch Entity which will form the edge of gear body with the two other circles being Construction Lines used for sketch geometry in the forming of the gear teeth. Your sketch should look similar to the nearby image.

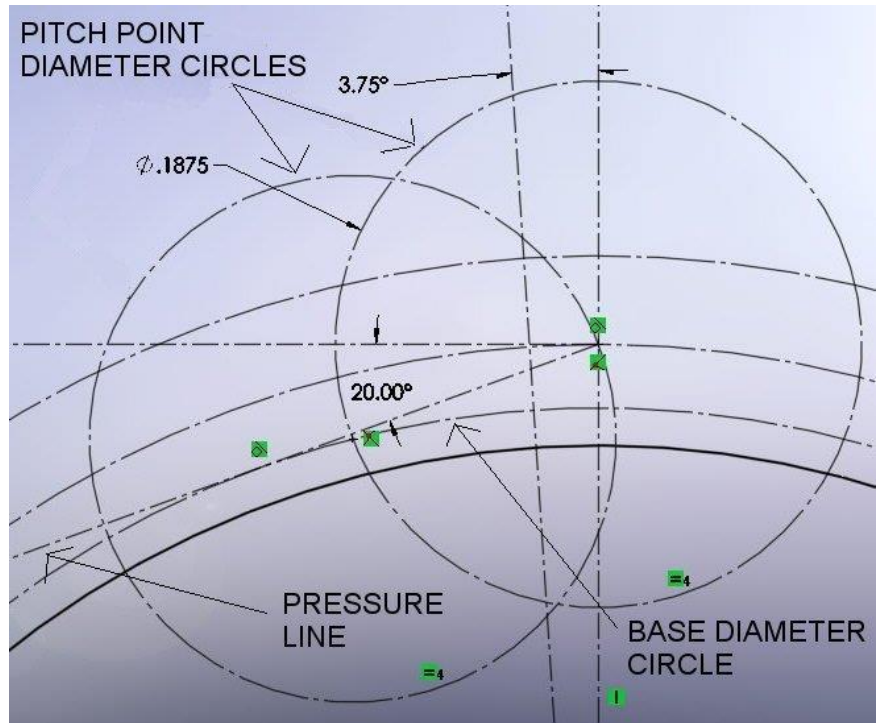
- Sketch a vertical Construction Line from the Origin beyond the top of the Outside Diameter circle. Determine the Quarter Angular Circular Pitch (Ta4) (half of the width of a gear tooth). Sketch a second Construction Line, from the origin, but vertical, slightly to the left of the first and place, beyond the top of the Outside Diameter circle, and place an angular Smart Dimension between these two lines using the value derived for Ta4. The second angular Construction Line will serve as the center line to mirror a partial tooth sketch profile to the other side of the tooth.



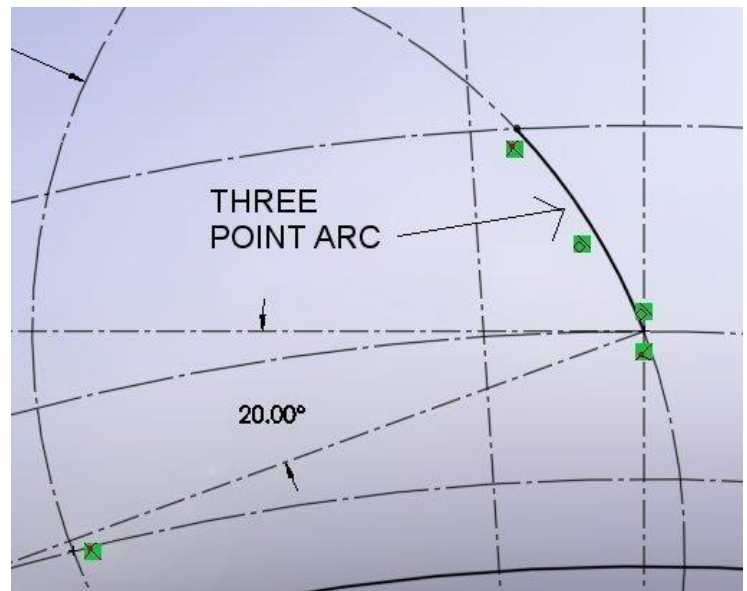
- Sketch a horizontal Construction Line to the left from the intersection of both the Pitch Diameter circle and the vertical Construction Line sketched in the previous step. The point at this intersection is named the Pitch Point. Sketch a Construction Line from the Pitch Point 20° below the horizontal line. This is the Pressure Line and represents the line of pressure - from the powered gear tooth to the non-powered tooth of the opposing gear - and will be approximately perpendicular to the tooth surface at this point. In our gear assignment the more common 20° pressure angle will be used (Designs using other angles, such as 14.5° are less common). Your sketch should look similar to the image below.

- Now sketch the Base Diameter (D_b) circle as opposed to using a formula. This circle has its center at the Origin and is tangent to the Pressure line.

- Sketch the first Pitch Point Diameter (D_{pp}) circle with the center at the Pitch Point. Sketch the second Pitch Point Circle with the center at the intersection of the first Pitch Point Circle and the Base Diameter. A portion of this circle will represent the upper arc or the involute portion of the gear tooth.



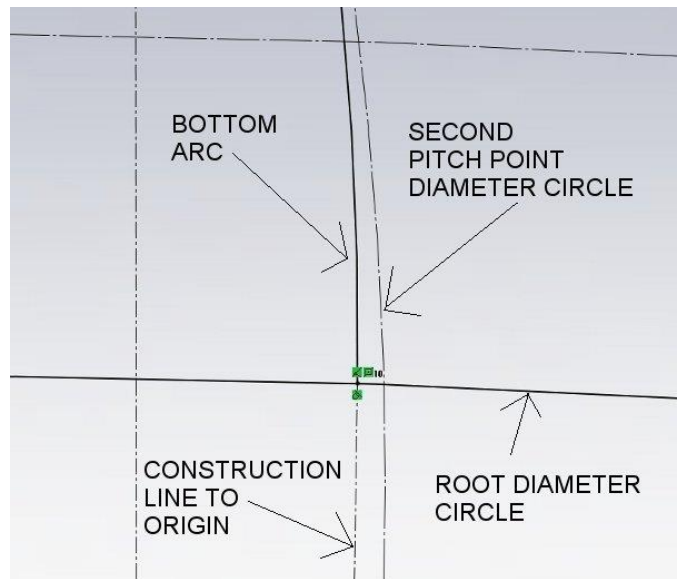
- Now sketch a 3 Point Arc named the Top Arc. This arc will represent the top or involute surface of one of the gear teeth. Place the first point of the arc at the intersection of the Outside Diameter circle and the second Pitch Point Diameter circle, the second point at the intersection of the Pitch Diameter circle and the second Pitch Point Diameter circle with the third point somewhere on the second Pitch Point Diameter circle. When placing the third point, a Tangent Sketch Relation is added automatically by SolidWorks. If the last step provides an error or is difficult, then snap the third point of the arc in an arbitrary, non-constrained location near the circle. Then establish a co-radial Sketch Relation between the arc and the second Pitch Point Diameter circle. Refer to the image above.



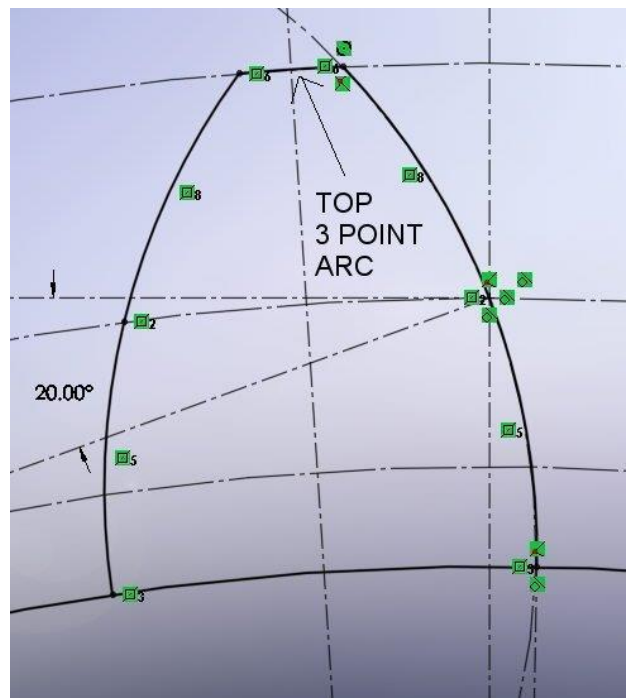
- Now sketch a Tangent Arc named the Bottom Arc. This arc will represent the bottom surface of one of the gear teeth. Start this arc at the bottom end point of the 3 Point Arc - sketched in the previous step – with the second endpoint coincident to the Root Diameter circle. This arc needs to be perpendicular to the Root Diameter circle (which will be covered in the next step). Refer to the image nearby.



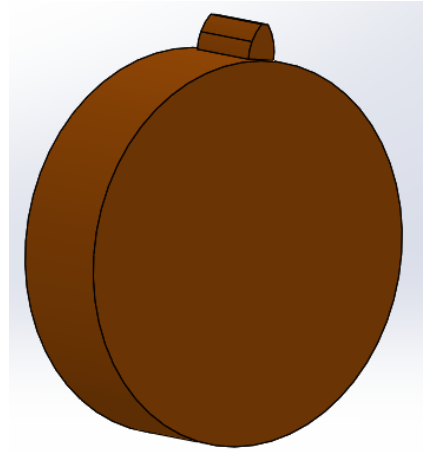
- To make the Bottom Arc perpendicular to the Root Diameter, sketch a Construction Line from the endpoint of the arc - precisely sketched – and then make it coincident with the Origin. Then provide a Tangent Sketch Relation between this Construction Line and the Bottom Arc as shown in the image to the right. You may have to move the intersection of the Construction Line and arc to one side of the second Pitch Point Diameter circle - or the other - in order to get this tangency to work. This arc will not be co-radial to the second Pitch Point Diameter circle!



- Now mirror the two arcs (the Top Arc and Bottom Arc) - sketched in the steps above - using the Construction Line representing Quarter Angular Circular Pitch as a mirror sketch reference. Sketch a 3 Point Arc connecting the two top endpoints of the mirrored arcs with the third point snapped to the Outside Diameter circle. If the last step provides an error or is difficult, then snap the third point of the arc in an arbitrary, non-constrained location near the circle. Then provide a co-radial Sketch Relation between that arc and the Outside Diameter circle. You should now have 2 areas of enclosed geometry, one for the gear tooth and the other the gear body. You are now ready for some solid features.

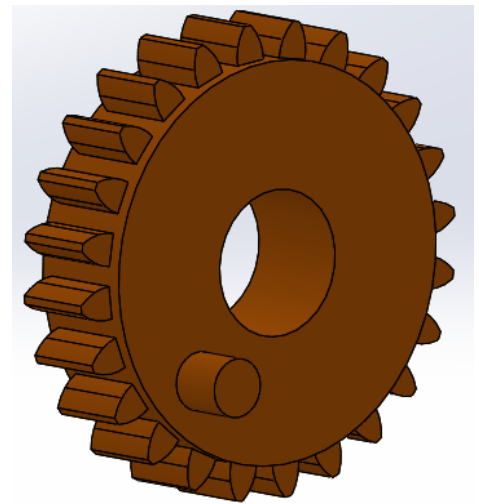


12. On the Features tab on the Command Manager – select the “Extrude Boss/Base” and extrude using the Mid Plane direction option, to a distance of your choice, selecting the center region of the gear in the “Selected Contours” section of the Properties Manager. Name this feature “Base Extrude”. Borrow the sketch from the first feature and using the “Extrude Boss/Base” feature again extrude the region for the gear tooth a distance less than the first extrude. Name this feature “Gear Tooth”.



13. From the Hide/Show tool on the Heads Up toolbar select “Temporary Axes” to “Show” and using the “Circular Feature Pattern” - pattern the “Gear Tooth” around the gear the required number of times.

14. Add additional features such as a center hole or axle for mounting to the Mounting Plate in your assembly. You may want to add insets and/or spokes in the gear body that would increase the strength of your gear while reducing the material needed. You may want to add fillets between the tooth and gear base as you would expect to find. Add these fillets before the Circular Feature Pattern and make them small enough so they do not impact movement in the assembly. You may also want to add features that will enhance the motion and interaction of the gears. These may include knobs and arms on the front gear face. Be prepared to “suppress” all of these extra features if you are having trouble with motion.



15. Design the Gear Wheel in a similar manner as the Pinion Gear as described above along with a Mounting Plate. Take these three components and create an assembly.

