GearTrax 2007

COMPONENTS FOR YOUR DRIVE

Camnetics
INCORPORATED

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Made in the USA

Document date: 11/13/2005

Introduction: GearTrax/SolidWorks Model

GearTrax interacts directly with the SolidWorks API to create solid models.

Spur gears are an extrusion; a circle that is the same diameter as the addendum circle and another circle that is the bore. The 1st tooth is an extrude-cut feature on the base feature. All other teeth, if so chosen, are a pattern of this first tooth. The tooth form is created with a true involute profile. The tooth profile is a series of points that define a spline or a series of arcs.

Helical gears are made in the same manner as the spur gears with one exception. The tooth is swept along a helix. The user has the choice of either right or left-hand helix.

Revolving the outside profile of the gear creates a bevel gear. A plane is constructed tangent to the back cone surface. A sketch is then created on this plane for the tooth profile. The tooth profile is then drawn on the sketch. A second sketch is created that intersects with the center of the gear. A point is placed on this sketch at the Pitch Apex. These two sketches are then loft cut through the revolved base feature to produce the first tooth cut. All other teeth, if so chosen, are a pattern of this first tooth cut.

A sprocket is created by revolving the outside profile of the sprocket. If multi-strands are selected, then the sketch includes that geometry. A tooth is then drawn on a sketch and is extrude-cut through the base feature. All other teeth are a pattern of this first tooth cut.

Gear belt (timing belt) pulleys are created revolving a profile of the gear including flanges. A tooth sketch is then created through the center of the gear. A tooth profile is drawn and extrude-cut though the gear or up to the flanges. All other teeth are a pattern of this first tooth cut.

Belt pulleys are created by revolving a shape of the profile of the gear. If “multi grooves” is selected, they are also revolved at that time.

Splines may be created individually or as a pair. Similar to spur gears, the internal spline is created with a pair of circles, one defining the internal diameter and the other defining the outside diameter. The pair of circles is mid-plane extruded to form the base feature. A tooth space is drawn and mid-plane cut-extruded. All other tooth spaces are patterned from this cut. External splines are created with a single circle defining the outside diameter of the spline then mid-plane extruded to form the base feature. A tooth space is drawn and mid-plane cut-extruded. All other tooth spaces are patterned from this cut.

Mounting is a revolved feature added to the base feature. Bores with a keyway, set screws or bores for tapered bushing may also be selected.

Starting GearTrax

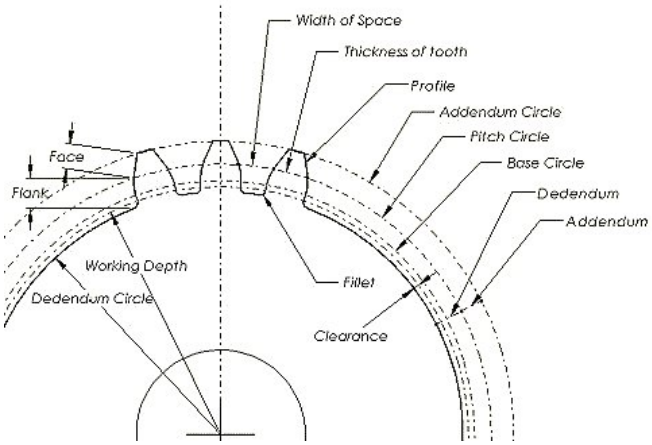
There are two ways to start GearTrax in SolidWorks:

1. Click on the 'Start' button, move to Program Files, Camnetics, and select GearTrax.

2. Starting with **SolidWorks 2001 Plus GearTrax*** will run in-process therefore GearTrax will show up as an Add-In. There are a few steps required to make GearTrax available to SolidWorks as an Add-In.

    - Start SolidWorks.
    - From the SolidWorks file menu select Open.
    - From the "Files of Type:" drop down box select "Add-Ins (*.dll)"
    - Browse the install folder for GearTrax, most likely "C:\Program Folders\GearTrax2005"
    - Select GearTrax2005.dll
    - Then click the Open button.
    - GearTrax will now be available in the SolidWorks menu tree. Simply click on Camnetics then GearTrax to activate GearTrax.

Basic gear geometry and nomenclature



Diametral Pitch verses Module

Diametral pitch is always in inches.

Pd = N/D = number of teeth to inches of pitch diameter

Module pitch is always in millimeters.

Module m = D/N = mm of pitch diameter per tooth

1 Diametral pitch = 25.400 Module.

Example:

A gear with 10 teeth and 1.000 inch pitch diameter:

10 diametral pitch = 10 / 1.000in

The same gear in metric:

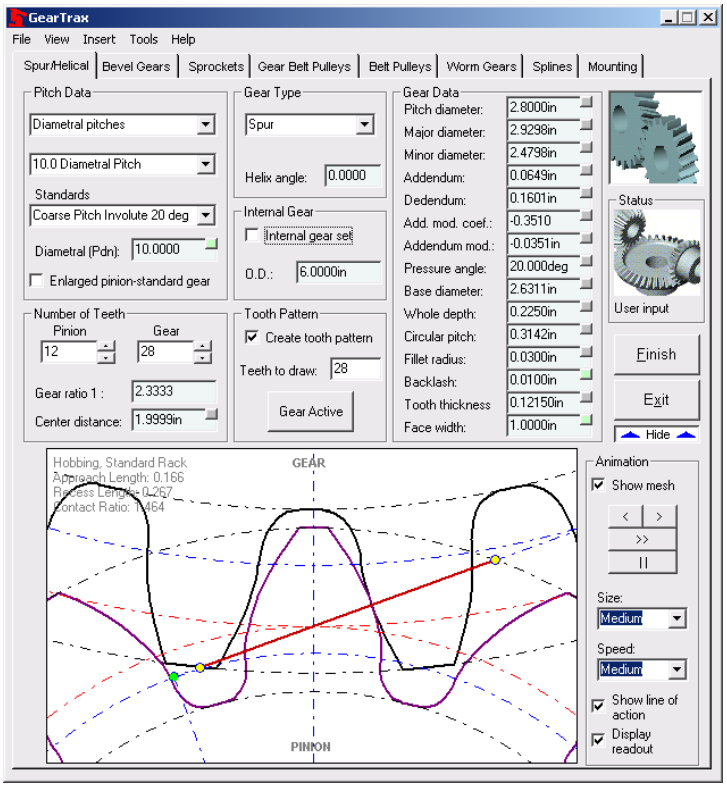
2.54 module = 25.4mm / 10

m = 25.4/Pd

Pd = 25.4/m

Creating a Spur Gear

To create a spur gear:



- Select the appropriate standard from the **Standards** drop down box.
- Select the desired pitch from the **Diametral Pitch** drop down box.
- A diametral pitch or module value may be entered in the dialog box below the Standards drop down box.
- In the **Gear Type** box select the **Spur** option button if not already selected.
- Enter a value for the face width in the **Face width** box.
- In the **Number of Teeth** box enter or use the spin box to change the number of teeth for the gear and the pinion.
- Enter a value for the tooth **Backlash**
- Enter a value for the **Face width**.
- Check the **Create Tooth Pattern** if you want the tooth to be patterned.
- If the **Create Tooth Pattern** is checked, the **Teeth to Draw** box becomes enabled. If it is desired to pattern the tooth with other than a full pattern then input the number of teeth to be drawn. This can be changed using SolidWorks after the part is drawn simply by editing the definition of the Tooth Pattern.
- If the **Internal gear set** check box is selected, enter a value for the outside diameter. It must be larger than the major diameter.

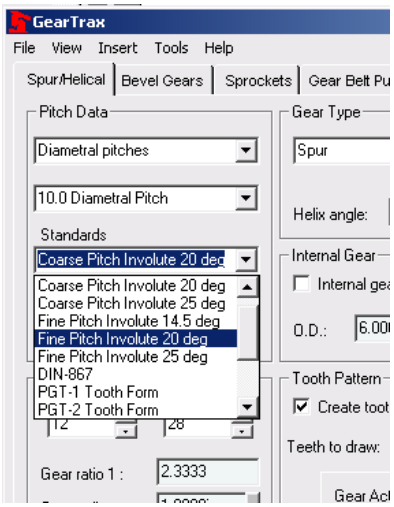
- Make any necessary changes to the hub mounting.
- Click on the **Finish** button to complete the gear.

Selecting a Gear Standard

From the Standards drop down box select one of the standards.

ANSI/AGMA Tooth Proportions

This version of GearTrax supports automatic tooth thickening or long-addendum pinions and their mating short-addendum gears. See the section on Tooth Thinning for more information.



PGT Tooth Form

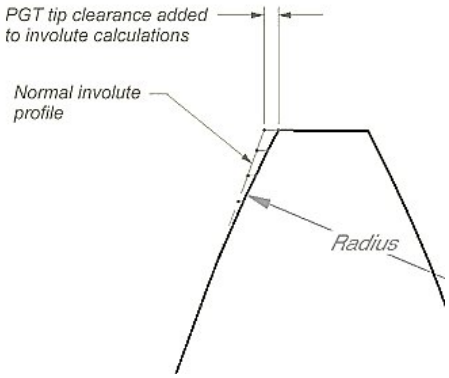
PGT has a complete and easy to understand gearing manual specifically for plastic gears.

The “Plastic Gearing” manual is available for $69.50 (1999 price) from ABA-PGT.

For more information about the PGT tooth form please contact:

ABA-PGT, INC.
1395 Tolland Turnpike
P.O. Box 8270
Manchester, CT 06040-0270

ABA (860) 649-4591
PGT (860) 643-6340
FAX (860) 643-7619
Web Site: www.abapgt.com



DIN-867 Standard

Addendum = Module x 1

Dedendum = Module x 1.25

Fillet Radius (Hob)= (Dedendum – Addendum) x 1.5

Standard pressure angle = 20 degrees

Modules between 0.3 and 75.0

JIS B 1701-1973 Standard

Addendum = Module x 1

Dedendum = Module x 1.25

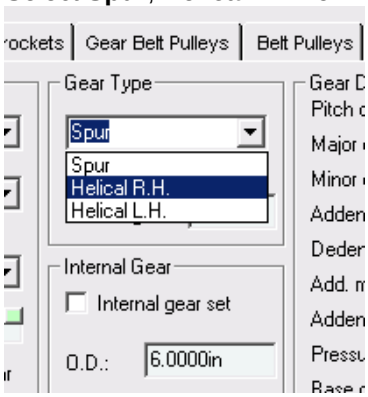
Fillet Radius (Hob)= (Dedendum – Addendum) x 1.5

Standard pressure angle = 20 degrees

Modules between 0.3 and 75.0 Series No.1 through No. 3

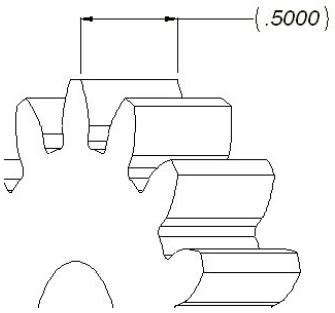
Gear Type Selection

Select **Spur**, **Helical R.H.** or **Helical L.H.** gear type.



Face Width

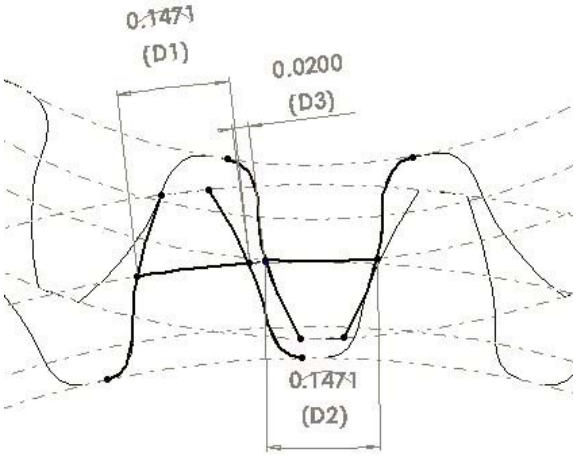
Face width is the length of the spur gear tooth.



Backlash

- The Backlash specified is the reduction in circular tooth thickness. In the case of a 10 Dp gear the tooth thickness without any back lash would be 0.1571 inches. With a backlash of 0.0100 inches the tooth thickness will be 0.1471 inches.

    - If two gears are fabricated each with 0.0100 inches of backlash then there will be 0.0200 inches backlash between the mating teeth. See the example below. Note: Backlash = circular pitch – tooth thickness, gear – tooth thickness, pinion.

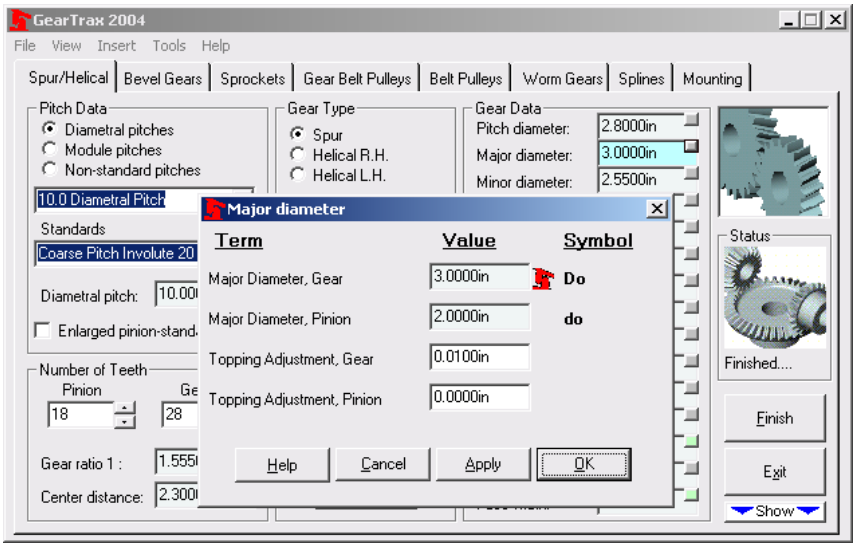


Topping Adjustment

The topping adjustment effects the outside diameter of the gear and/or the pinion.

To set the topping adjustment:

- Activate the Major Diameter dialog box.



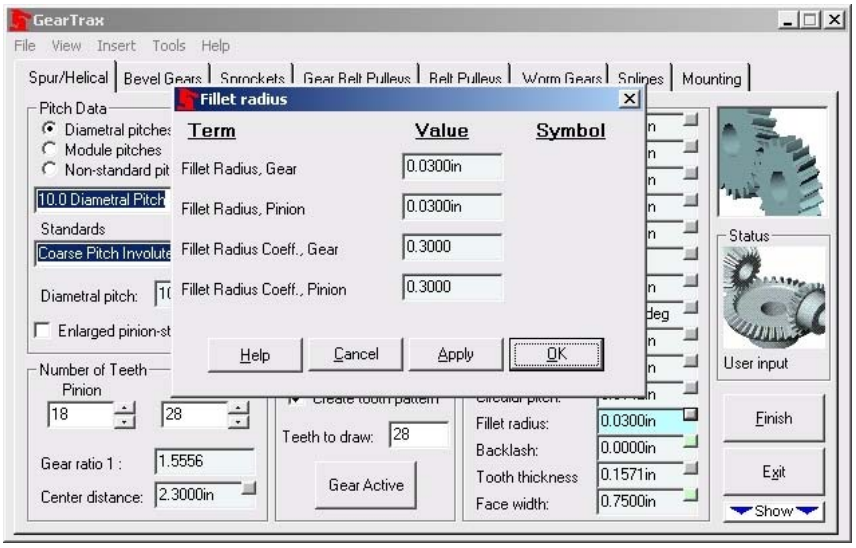
- Enter the topping adjustment value and click OK.
- The OD of the gear or pinion will be reduced by twice the value entered.

Fillet Radius, Fillet Radius Coefficient

The fillet radius and fillet radius coefficient can only be modified if Non-standard pitches is selected.

To set the fillet radius or fillet radius coefficient:

- Activate the Fillet Radius dialog box.

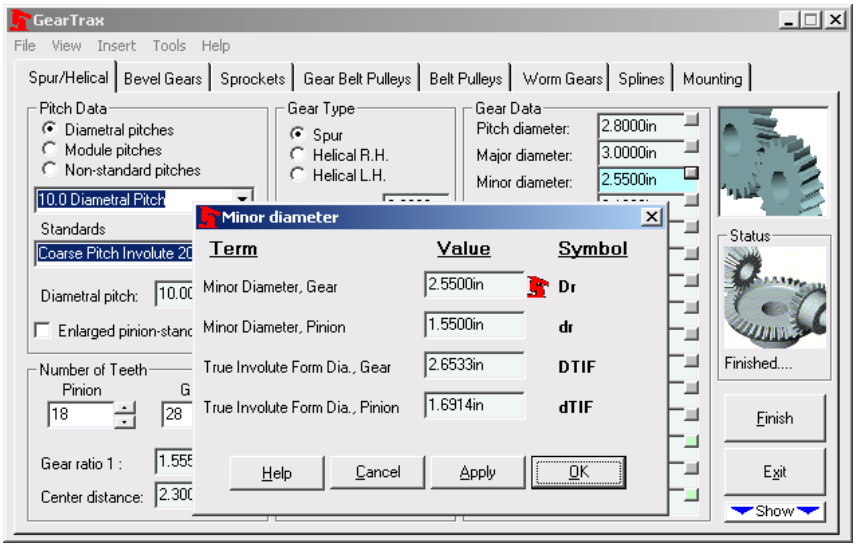


True Involute Form Diameter

The calculated value for the true involute form diameter may be viewed in the **Minor Diameter** dialog box.

To view the true involute form diameter value:

- Activate the Minor Diameter dialog box.

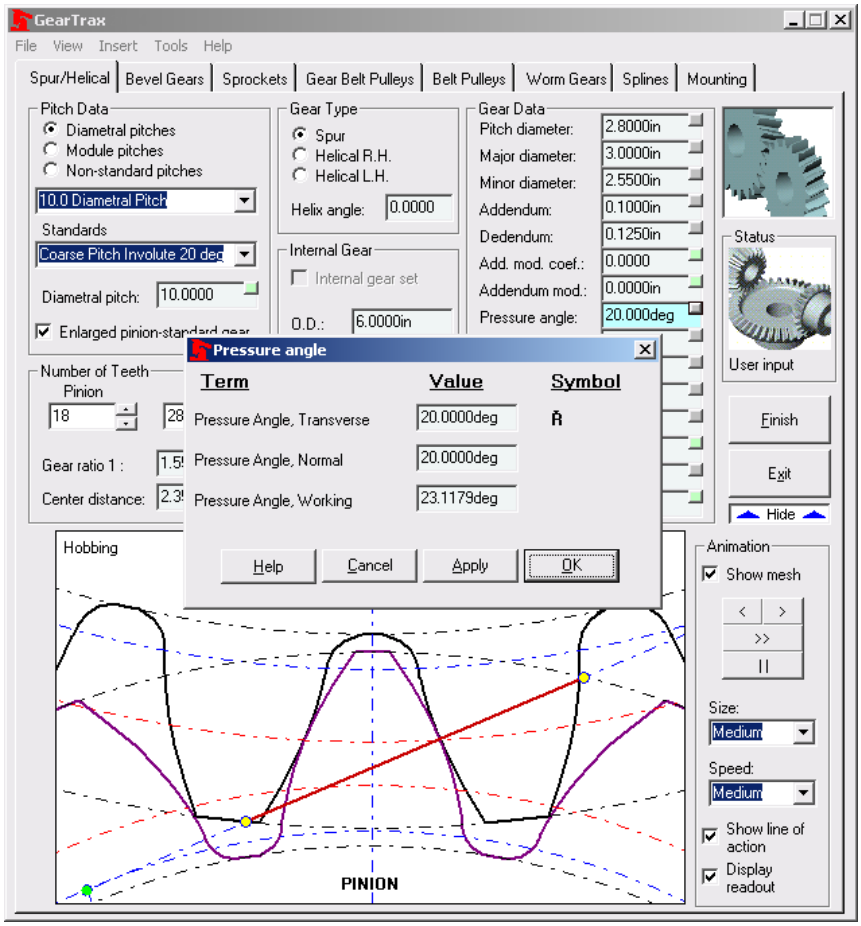


Working Pressure Angle

The calculated value for the working pressure angel may be viewed in the **Pressure Angle** dialog box.

To view the working pressure angel value:

- Activate the Pressure Angle dialog box.
- The working pressure angle may be different from the normal pressure angle if the gear set is a helical set, enlarged pinion-standard gear set or the sum of addendum modifications does not equal zero.

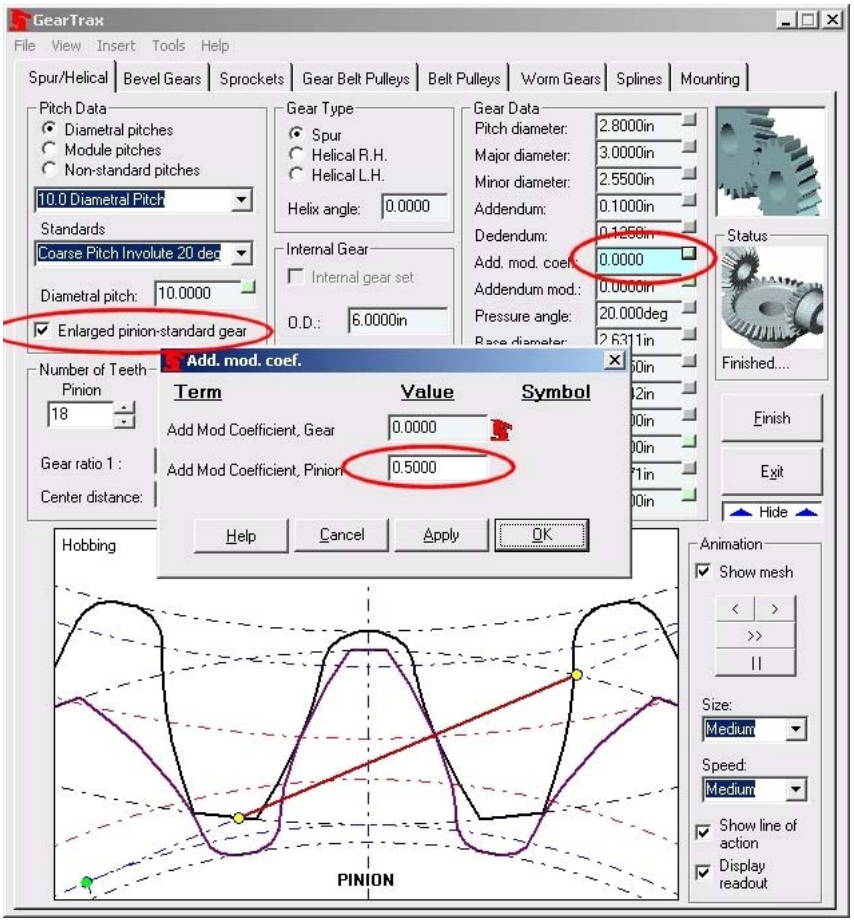


Creating an Enlarged Pinion with a Standard Gear

You can model an enlarged pinion that will interact with a standard gear.

To create an enlarged pinion:

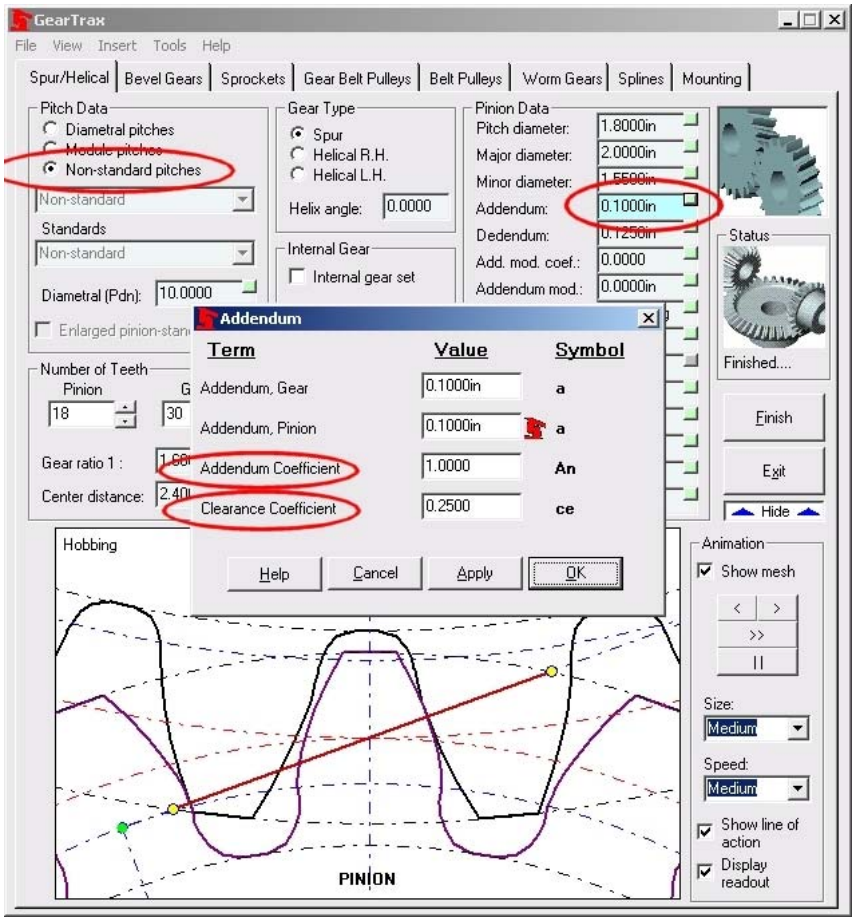
- Select the Enlarged pinion-standard gear check box.
- GearTrax calculates the addendum modification for the pinion if the center distance is manually changed. Otherwise the addendum modification or addendum modification coefficient must be provided. Click the more information box in the upper right hand corner of the Addendum modification text box. This will allow you to edit the value of the addendum modification.



Creating Non-Standard Pitch Spur Gears

Non-standard pitches give the user optimum control over all parameters of the gear construction.

- Select the “Non standard Pitches” option button.



GearTrax2005 uses the addendum coefficient, clearance coefficient and the fillet radius coefficient as the basis for the gear geometry. When switching to non-standard pitches GearTrax will display the current coefficient values. You will also be asked to select to use coefficients as the basis of the gear geometry. Selecting ‘no’ will give the most flexibility but enough data will need to be input into GearTrax for the gear to be modeled properly. Selecting ‘yes’ will force GearTrax to use coefficients. These values can be changed. The fillet radius coefficient may be changed in the fillet radius dialog box. The addendum coefficient and the clearance coefficient may be changed in the Addendum dialog box.

Many parameters of the gear are inter-related; changing one parameter may affect the values of other parts of the gear. For example, changing the pressure angle will affect the value of the base circle. Also changing the addendum will affect the major diameter (O.D), the addendum modification and the addendum modification coefficient.

The coefficient values will not change unless they are directly edited. These values are the basis of all standards.

Some values will not change unless they are directly changed. For example, the number of teeth can only be changed by directly editing the value in the box.

When creating non-standard gears you may need to specify the values for the following:

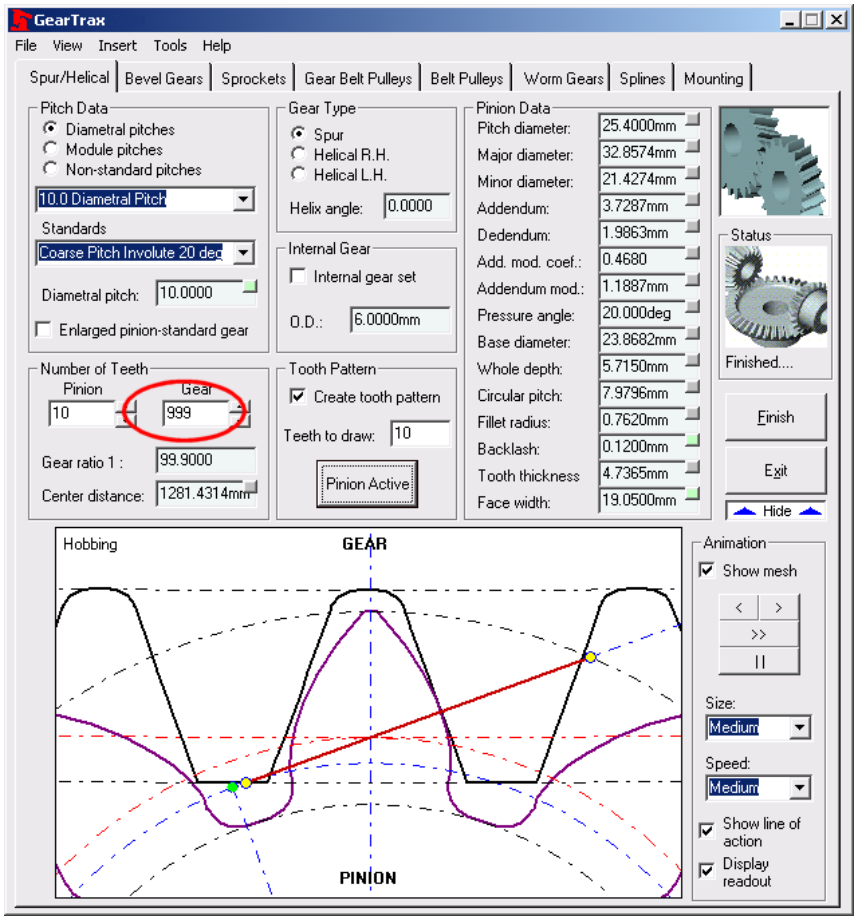
- Diametral pitch, module pitch or the pitch diameter
- Number of teeth
- Major diameter, addendum, addendum modification, or the addendum modification coefficient
- Minor diameter, dedendum, dedendum modification, or the dedendum modification coefficient
- Pressure angle or base diameter
- Fillet radius or the fillet radius coefficient
- Tooth thickness or backlash
- Face width

Creating a Spur Gear Rack

You can create a generic spur rack with 20 teeth.

To create a spur gear rack:

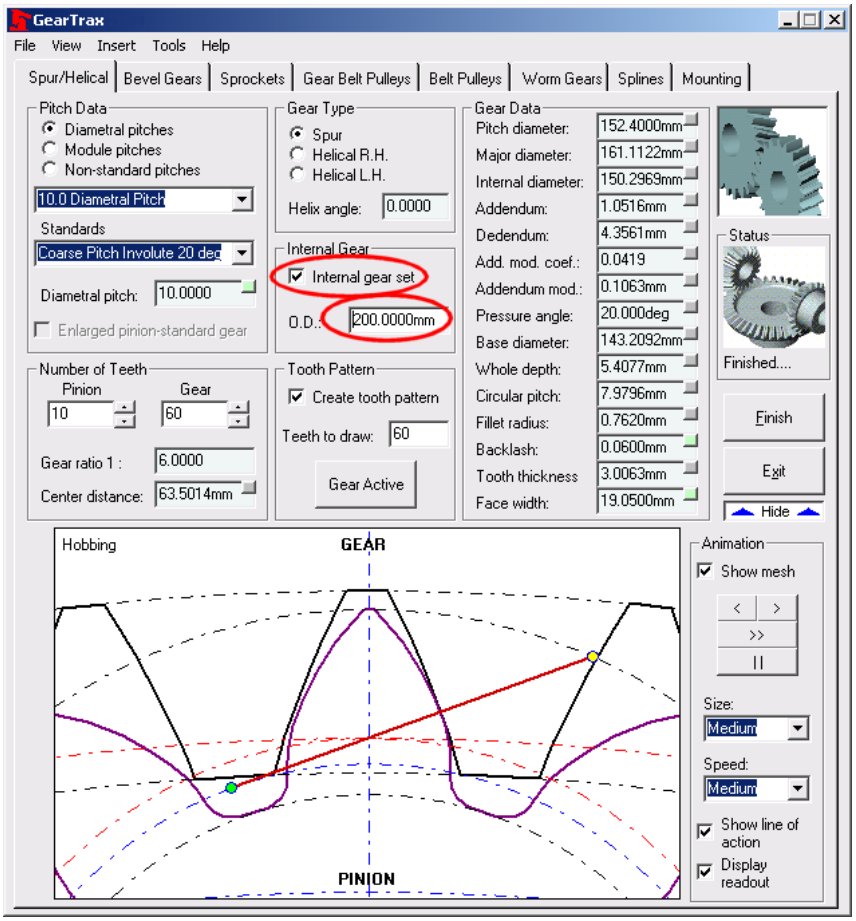
- Enter 999 into the **Number of Teeth** for the gear
- Set the number of teeth for the pinion.
- Click the **Finish** button to model the rack.
- The pitch line is at the origin.



Internal Gear Sets

To create an internal gear set:

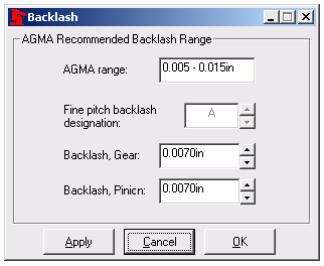
- Select the **Internal gear set** check box.
- Minimum number of gear teeth for this standard is 28.
- Enlarged pinion check box is disabled.



Selecting an AGMA Backlash Range for Spur Gears

To select a backlash range:

- Select a fine pitch standard.
- Open the **Backlash** dialog box.



- If you are using a fine pitch gear standard select a **Fine pitch backlash designation**; A through E or use the spin buttons to change the designation.
- Enter a backlash value or use the spin buttons to change the backlash value. If the value falls outside of the AGMA range the background color of the backlash input box changes to yellow or green.

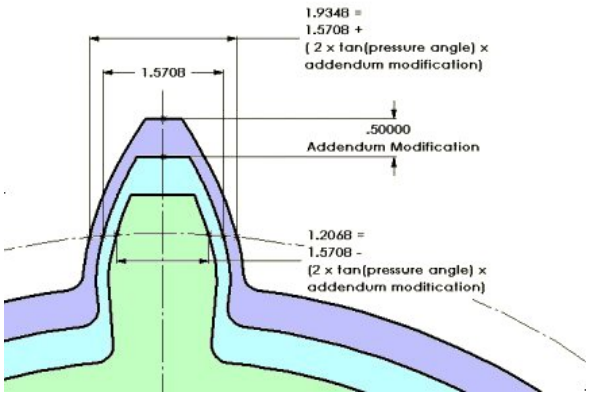
Addendum Modification

Why modify the addendum?

Addendum modification serves two main purposes.

- Addendum modification will help avoid tip interference between the gear and the pinion.
- The gear tooth is really just a short cantilevered beam. For the gear set to have a balanced strength, the teeth of the pinion need to be stronger than that of the gear. Gears made of weaker materials like plastic; this may be critical to the life and the performance of the gear set. See the PGT Tooth Form for more information.

As the addendum length changes, so does the tooth width at the pitch circle. This change in thickness is equal to twice the addendum change multiplied by the tangent of the pressure angle. Note: All width dimensions are arc lengths.

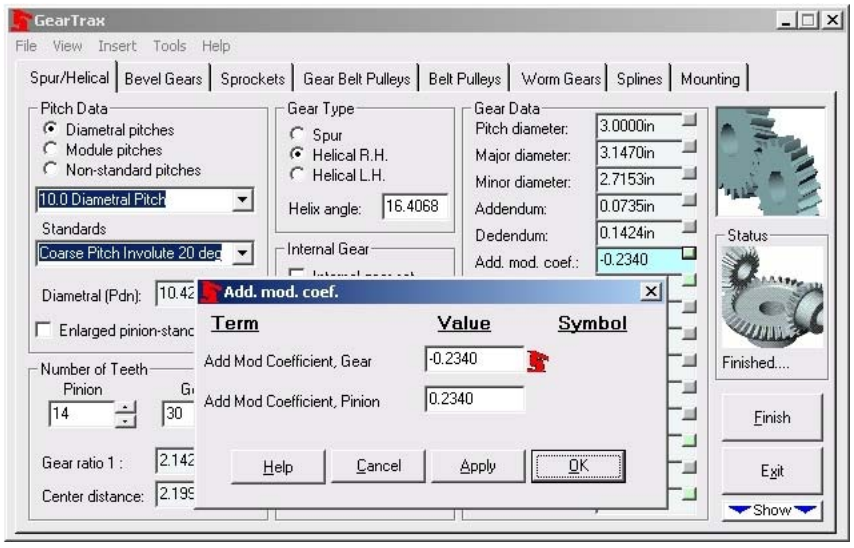


Addendum Modification Coefficients

For DIN standards you can specify the addendum modification coefficient.

To specify an Addendum modification coefficient:

- The Addendum Modification option in the Tools>Options>Spur Gears menu must be set to "Manual only".



Tooth Thinning

Tooth thinning, addendum and dedendum geometry changes to avoid under cut or interference. If the pinion or gear is modified, the mating gear will also require modification or change in the center distance.

The following tooth forms may require modification to avoid under cut or interference.

- 11 teeth or less with 25 degree pressure angle.
- 17 teeth or less with 20 degree pressure angle.
- 31 teeth or less with 14.5 degree pressure angle.

Machinery's Handbook has a section on this subject titled "Increasing Pinion Diameter to Avoid Undercut or Interference".

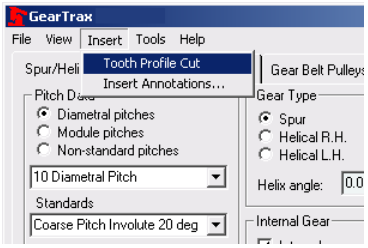
Tooth thinning may be accomplished by selecting the supported ANSI/AGMA standard. Along with tooth thinning/thickening the addendum and dedendum changes accordingly. As the tooth thickens on the pinion, the addendum increases and the dedendum decreases in size. Conversely, the gear tooth thins and the addendum decreases while the dedendum increases in size. The whole depth remains constant.

Inserting a Tooth Profile Sketch

With GearTrax you can add multiple teeth to a single part.

To insert a gear or involute spline tooth profile:

- In GearTrax configure the gear or involute spline.
- Start a new sketch In SolidWorks.
- From the GearTrax menu select **Insert** then **Tooth Profile Cut**.



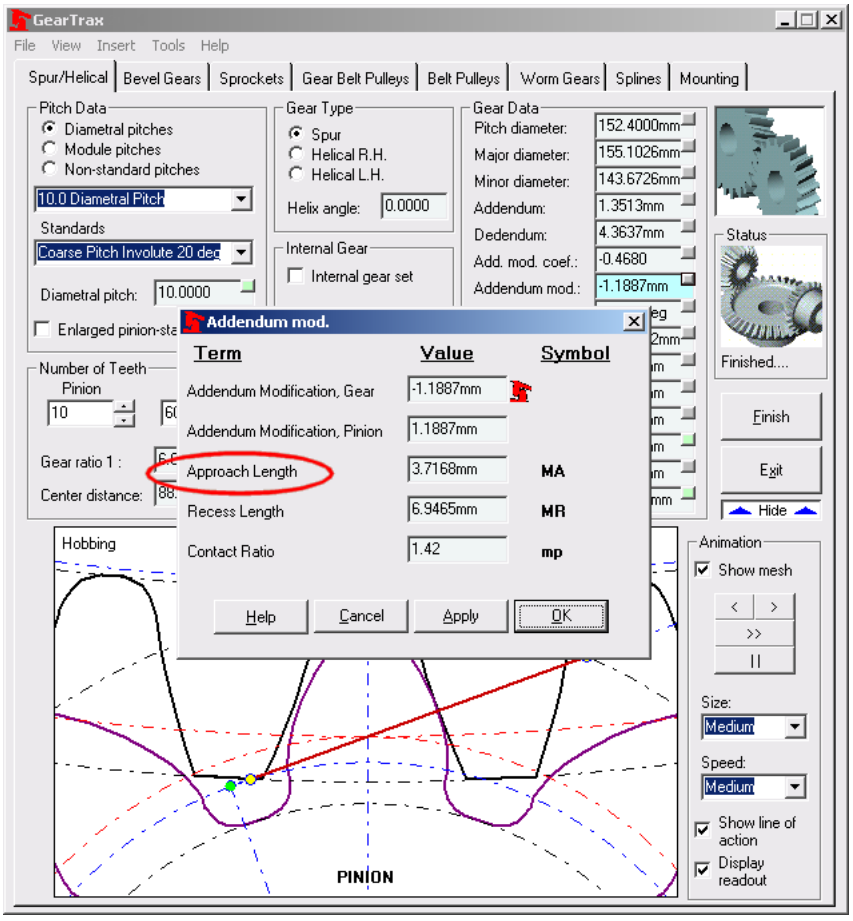
- After the tooth cut geometry has been created by GearTrax, manually extrude cut the sketch using SolidWorks.
- Pattern the tooth cut using SolidWorks.

Approach Length

With GearTrax you can view the approach length for a spur gear set.

To view the approach length:

- Open the **Addendum mod** dialog box.
- This assumes the pinion is always the driving gear and there is no tip relief.

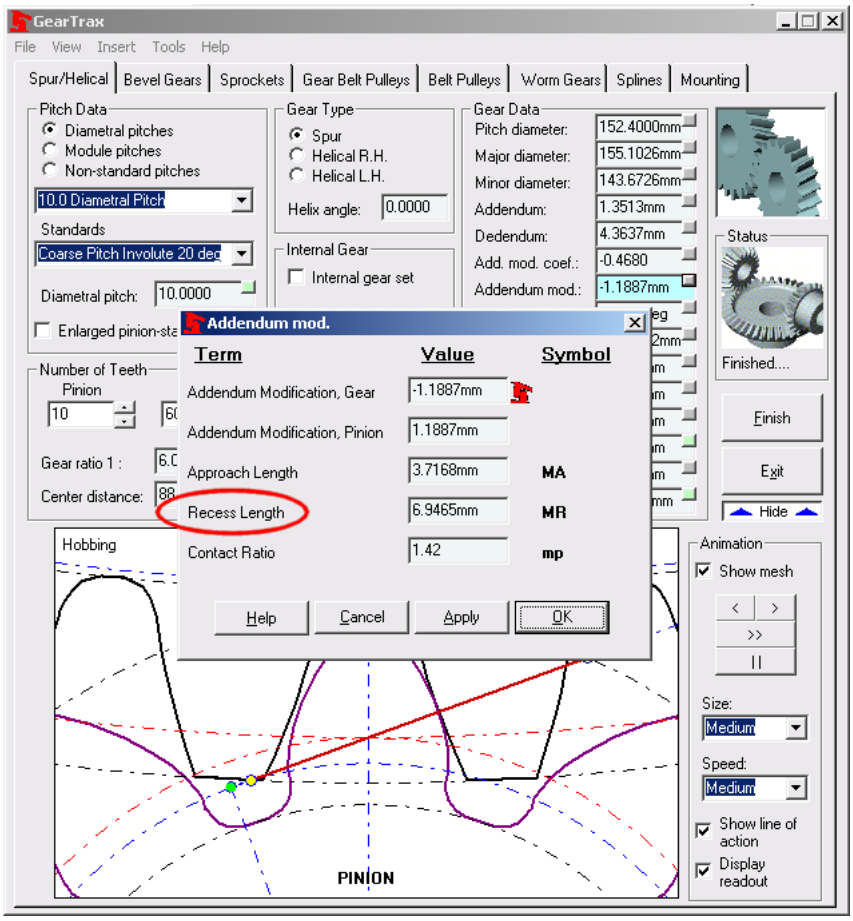


Recess Length

With GearTrax you can view the recess length for a spur gear set.

To view the recess length:

- Open the **Addendum mod** dialog box.
- This assumes the pinion is always the driving gear and there is no tip relief.

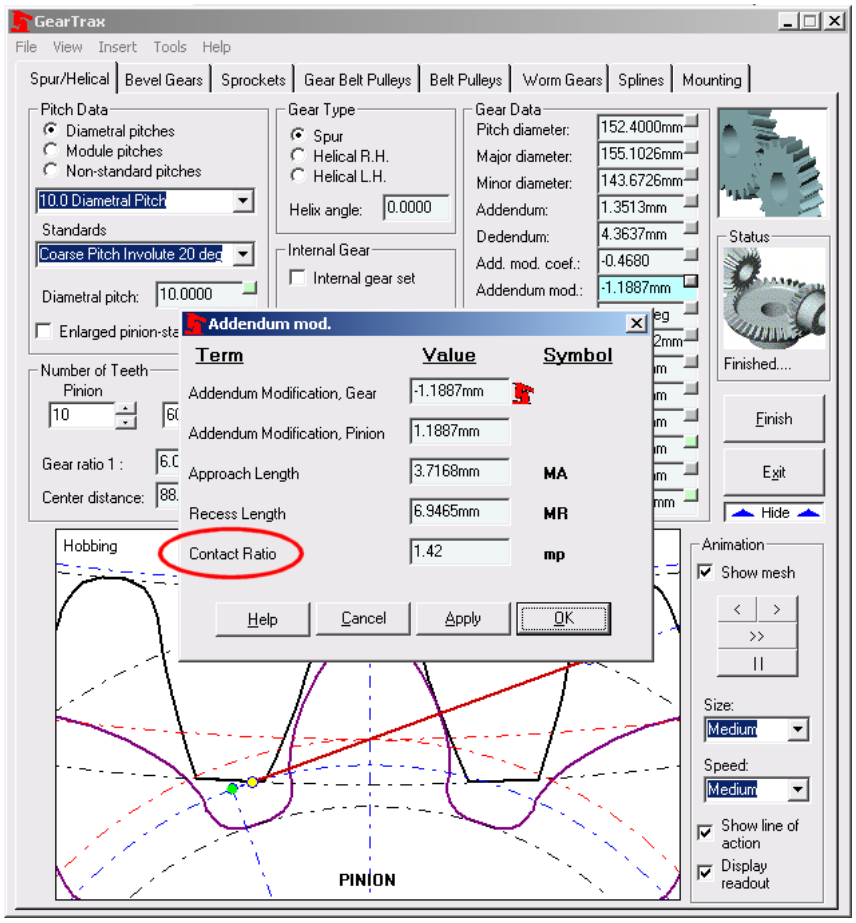


Contact Ratio

With GearTrax you can view the contact ratio for a spur gear set.

To view the contact ratio:

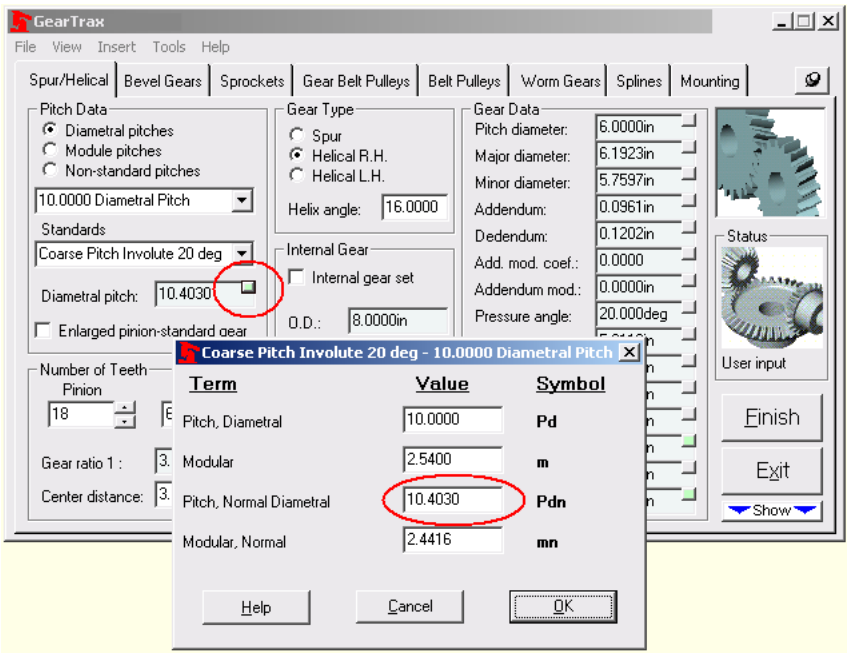
- Open the **Addendum mod** dialog box.

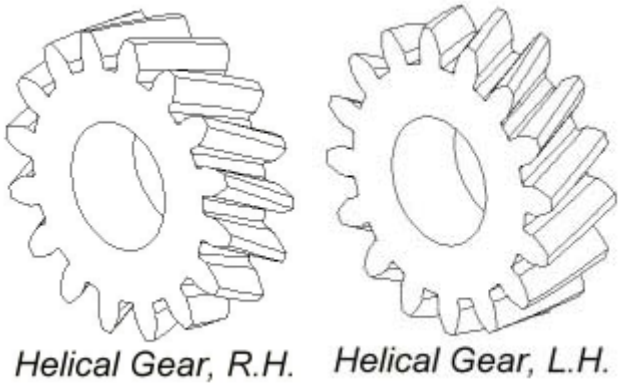


Creating a Helical Gear

To create a helical gear:

- Select the appropriate standard from the **Standards** drop down box.
- Select the desired pitch from the **Diametral Pitch** drop down box.
- In the **Gear Type** box select either the **Helical R.H.** or **Helical L.H.** from the drop down box.
- If the shafts are not parallel you will need to switch the helical direction of one of the gears. When helical is the first selected the other gear is automatically specified as the other hand.
- Selecting helix gear type enables the **Helix angle** input box. Enter the desired helix angle. This value must be greater than zero.
- To change the diametral pitch or module click on the **More information** button in the upper right hand corner of the **Diametral pitch** text box. Selecting helix gear type enables the text boxes to be edited. Enter the desired value in one of the four boxes. The other values will update accordingly. Click OK to apply the values.
- Enter a value for the face width in the **Face width** box.
- In the **Number of Teeth** box enter or use the spin box to change the number of teeth for the gear and the pinion.
- Enter a value for the tooth **Backlash** by clicking on the **More information** button in the upper right hand corner of the **Backlash** text box.
- Check the **Create Tooth Pattern** if you would like the tooth to be patterned.
- If the **Create Tooth Pattern** is checked, the **Teeth to Draw** box becomes enabled. If it is desired to pattern the tooth other than a full pattern then input the number of teeth to be drawn. This can be changed using SolidWorks after the part is drawn simply by editing the definition of the Tooth Pattern.
- If the **Internal gear set** check box is selected, enter a value for the outside diameter. This value must be larger than the dedendum circle value.
- Make any necessary changes to the hub mounting.
- Click on the **Finish** button to complete the gear.


GearTrax
File View Insert Tools Help
Spur/Helical Bevel Gears Sprockets Gear Belt Pulleys Belt Pulleys Worm Gears Splines Mounting
Pitch Data
Diametral pitches
Module pitches
Non-standard pitches
10.0000 Diametral Pitch
Standards
Coarse Pitch Involute 20 deg
Diametral pitch: 10.4030
Enlarged pinion-standard gear
Gear Type
Spur
Helical R.H.
Helical L.H.
Helix angle: 16.0000
Internal Gear
Internal gear set
O.D.: 8.0000in
Gear Data
Pitch diameter: 6.0000in
Major diameter: 6.1923in
Minor diameter: 5.7597in
Addendum: 0.0961in
Dedendum: 0.1202in
Add. mod. coef.: 0.0000
Addendum mod.: 0.0000in
Pressure angle: 20.000deg
Status
User input
Number of Teeth
Pinion
18
Gear ratio 1 :
Center distance:
Finish
Exit
Show
Coarse Pitch Involute 20 deg - 10.0000 Diametral Pitch
Term Value Symbol
Pitch, Diametral 10.0000 Pd
Modular 2.5400 m
Pitch, Normal Diametral 10.4030 Pdn
Modular, Normal 2.4416 mn
Help
Cancel
OK


Helical Gear, R.H.
Helical Gear, L.H.

Transverse diametral pitch verses normal diametral pitch

When creating helical gears:

It is necessary to understand the difference between transverse and normal diametral pitches. The normal diametral pitch is defined as normal to the cutter. The transverse pitch is on the plane of rotation.

Normal diametral pitch = Transverse diametral pitch / cos(helix angle)

If Diametral pitches are selected, the drop down box displays the transverse pitch and the **Diametral pitch** box displays the normal pitch.

If Module pitches are selected the drop down box displays the module transverse pitch and the **Module pitch** box displays the normal module.

Normal pressure angle verses transverse pressure angle

When creating helical gears:

It may not be necessary to understand the relationship between transverse and normal pressure angle but understanding how those values are displayed in GearTrax may be useful. The transverse pressure angle is measured in the plane of rotation and is always greater than the normal pressure angle. The pressure angle displayed in GearTrax is the normal pressure angle. The transverse pressure angle may be displayed by clicking on the **More information** button in the upper right hand corner of the **Pressure angle** text box.

The transverse pressure angle is effectively the real pressure angle.

The normal pressure angle is defined as normal to the cutter.

Transverse pressure angle = ATN(TAN(normal pressure angle) / COS(helix angle))

Setting the helix angle by specifying the center distance

GearTrax can set the helix angle by specifying the center distance.

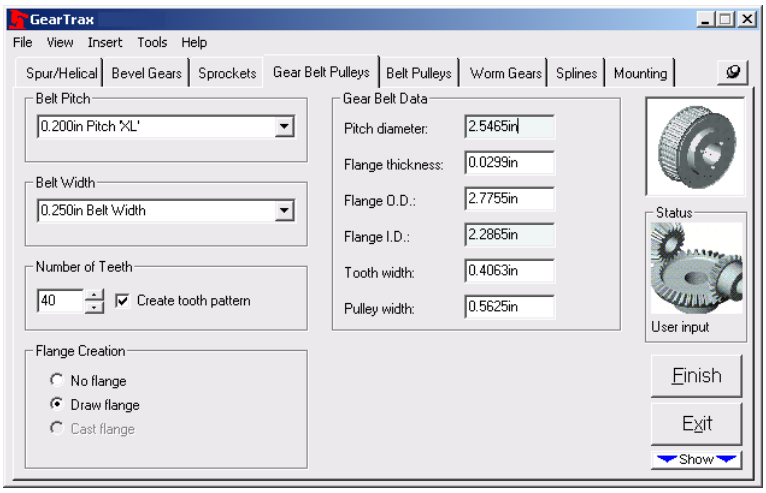
To set the helix angle:

- Enter the value in the **Center distance** text box.
- This changes the helix angle and transverse diametral pitch. The normal diametral pitch remains unchanged.

Creating a Gear Belt Pulley

To create a gear belt pulley:

- Select the desired belt pitch from the drop down box.

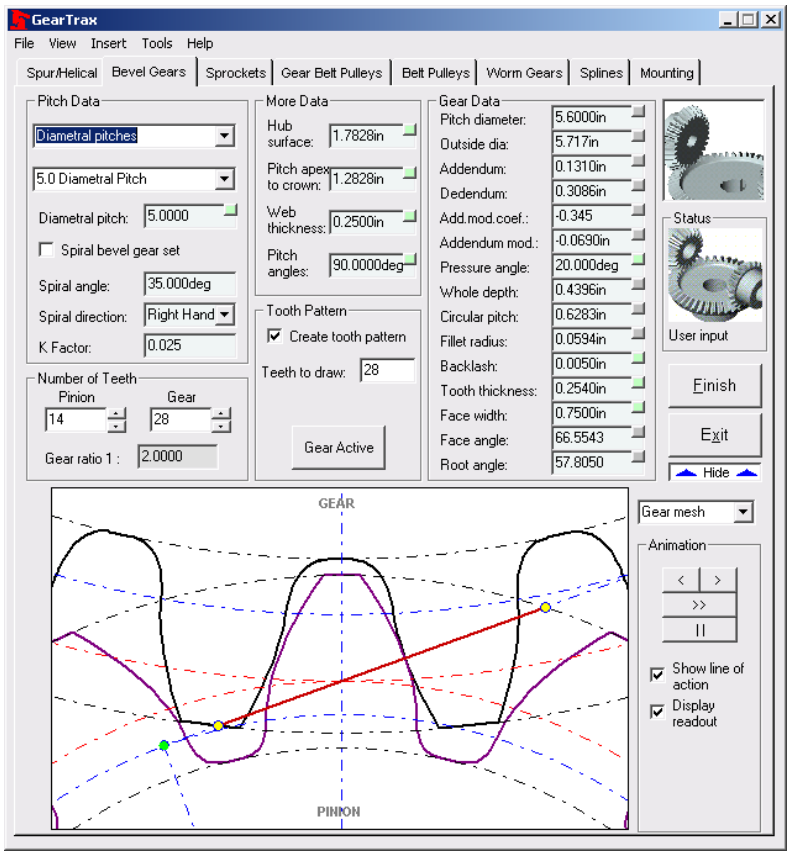


- Select the appropriate belt width.
- Enter or use the spin box to change the number of teeth. Note that the pitch diameter changes with the belt pitch and the number of teeth. The pitch diameter is a driven dimension.
- Check the **Draw Flange** if flanges are desired.
- Check the **Create Tooth Pattern** if you want the tooth to be patterned.
- Make any necessary changes to the Hub Mounting.
- Click on the **Finish** button to complete the pulley.

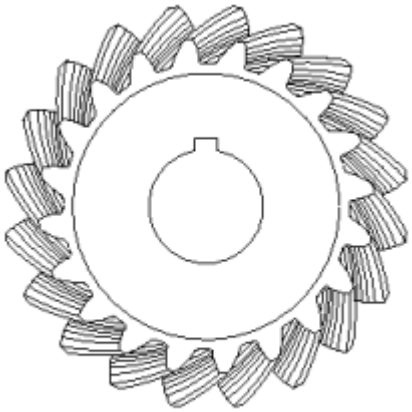
Creating a Bevel Gear

To create a bevel gear:

- Select the desired **Diametral Pitch** from the drop down box.

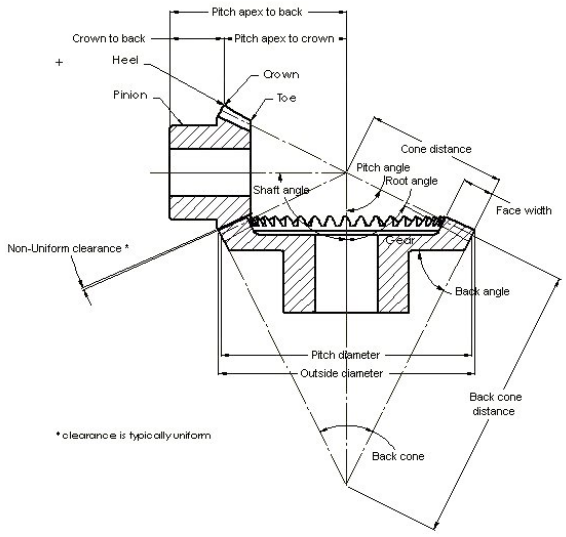


- Enter a value for the face width.
- Select the appropriate tooth proportion.
- Enter or use the spin box to change the number of teeth.
- Enter a value for the tooth clearance. See help for standard values for different diametral pitches.
- Check the **Create tooth pattern** if you want the tooth cut to be patterned.
- If the **Create tooth pattern** is checked, the **Teeth to draw** box becomes enabled. If it is desired to pattern the tooth other than a full pattern then input the number of teeth to be drawn. This can be changed after the part is drawn simply by editing the definition of the CirPattern1.
- Enter the pitch angle of the gear set (if other than 90).
- To create a spiral bevel gear place a check in the **Spiral bevel gear set** box.
- Set the **Spiral direction**. The hand of the spiral is indicated by the direction in which the teeth curve away from the axis. The mating gear is always the opposite hand.



- Enter the **K Factor** for circular tooth thickness. This value can be found in Machinery's Handbook in the 24th and earlier editions.
- Enter a value for the **Hub Mounting Surface**. This value should be between the **Min/Max Hub** Mounting values. If the mounting value is not between these values the base feature may not revolve properly.
- The pitch and outside diameters are driven dimensions.
- Make any necessary changes to the **Hub Mounting**.
- Click on the **Finish** button to complete the gear.

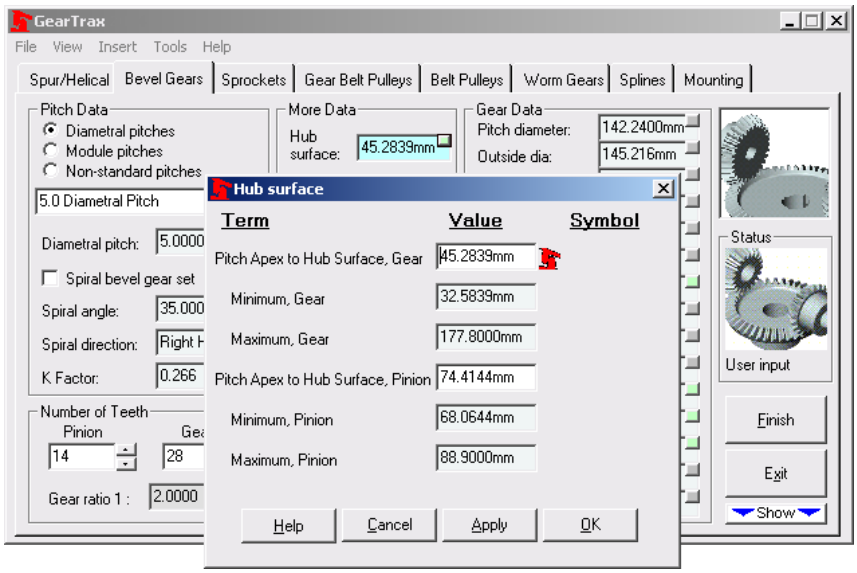
Geometry of bevel gear nomenclature.

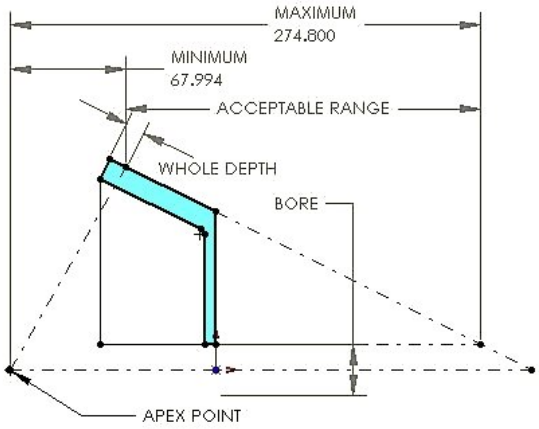


Understanding the Bevel Gear Hub Mounting Distance

To create a valid base feature for a bevel gear:

- The hub mounting distance must be between the minimum and maximum distances as shown below.
- If the hub mounting distance is less than the minimum, the base feature may revolve but the surface may be cut into by the tooth feature.
- If the hub mounting distance is greater than the maximum, the base feature will not revolve because of self-intersecting lines.





Recommended Normal Backlash for Bevel Gear Meshes

AGMA Recommended backlash range for bevel gears:

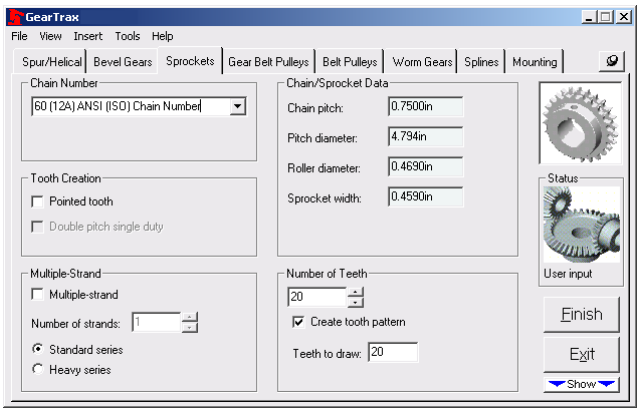
Diametral Pitch	Normal Backlash, Inch	
	Quality Numbers 7 through 13	Quality Numbers 4 through 6
1.00 to 1.25	0.020-0.030	0.045-0.065
1.25 to 1.50	0.018-0.026	0.035-0.055
1.50 to 1.75	0.016-0.022	0.025-0.045
1.75 to 2.00	0.014-0.018	0.020-0.040
2.00 to 2.50	0.012-0.016	0.020-0.030
2.50 to 3.00	0.010-0.013	0.015-0.025
3.00 to 3.50	0.008-0.011	0.012-0.022
3.50 to 4.00	0.007-0.009	0.010-0.020
4.00 to 5.00	0.006-0.008	0.008-0.016
5.00 to 6.00	0.005-0.007	0.006-0.013
6.00 to 8.00	0.004-0.006	0.005-0.010
8.00 to 10.00	0.003-0.005	0.004-0.008
10.00 to 16.00	0.002-0.004	0.003-0.005
16.00 to 20.00	0.001-0.003	0.002-0.004
20.00 to 50.00	0.000-0.002	0.000-0.002
50.00 to 80.00	0.000-0.001	0.000-0.001
80 and finer	0.000-0.0007	0.000-0.0007

This table gives the recommended normal backlash for gears assembled ready to run. Because of manufacturing tolerances and changes resulting from heat treatment, it is frequently necessary to reduce the theoretical tooth thickness by slightly more than the tabulated backlash in order to obtain the correct backlash in assembly. In case of choice, use the smaller backlash tolerances.

Creating a Chain Sprocket

To create a chain sprocket:

- Select the desired **Chain Number** from the drop down box.

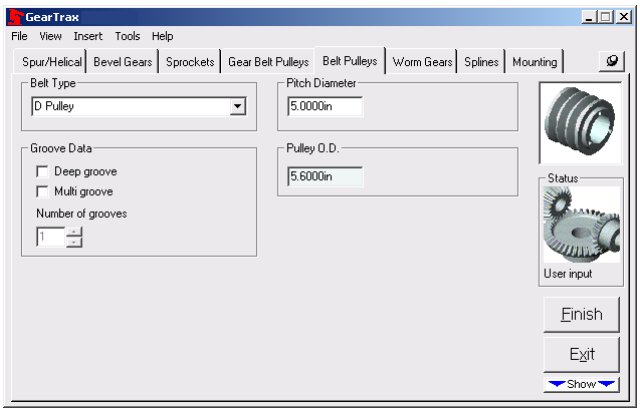


- Enter or use the spin box to change the number of teeth.
- Check the **Pointed Tooth** check box if so desired.
- Check the **Create Tooth Pattern** if the tooth is to be patterned.
- If the **Create Tooth Pattern** is checked, the **Teeth to Draw** box becomes enabled. If it is desired to pattern the tooth other than a full pattern then input the number of teeth to be drawn. This can be changed after the part is drawn simply by editing the definition of the CirPattern1.
- Select Standard Series or Heavy Series of sprocket.
- If a **Multi-strand** sprocket is to be modeled, click on the check box. This will enable the **Number of Strands** input box. Enter the number of strands to be drawn.
- The **Pitch Diameter** is a driven dimension.
- Make any necessary changes to the **Hub Mounting**.
- Click on the **Finish** button to complete the gear

Creating a Belt Pulley

To create a belt pulley:

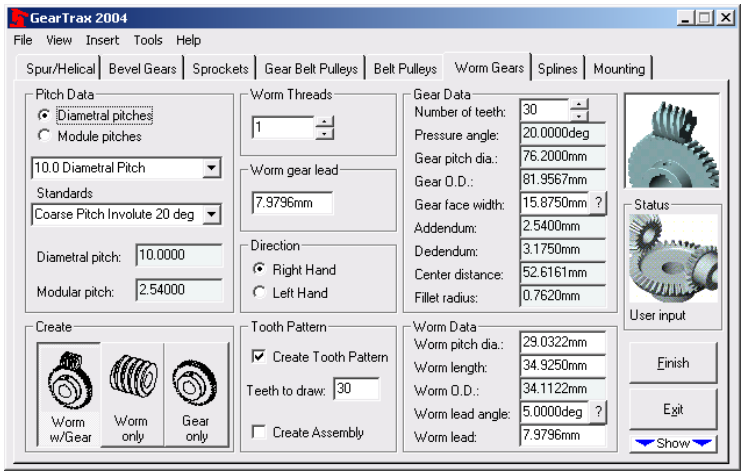
- Select the desired **Belt Type** from the drop down box.



- Check the **Deep groove** box if so desired.
- Placing a check in the **Multi groove** check box enables the **Number of grooves** input box.
- Enter a value or use the spin control to change the number of grooves to be drawn.
- Enter a value in the **Pitch Diameter** box.
- The **Pulley O.D.** value is a driven dimension.
- Make any necessary changes to the **Hub Mounting**.
- Click on the **Finish** button to complete the gear.

Creating a Worm Gear

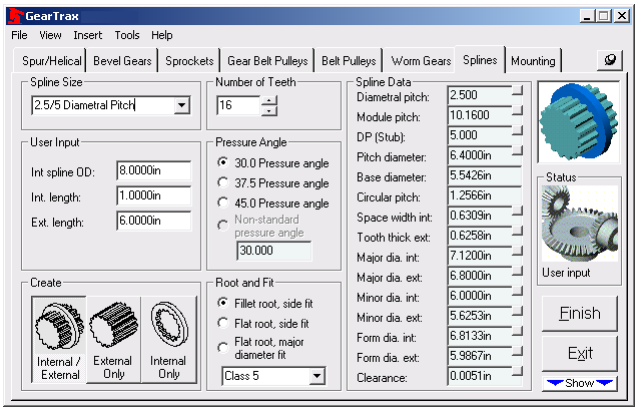
To create a worm gear:



- Select the desired **Diametral Pitch** from the drop down box.
- Select the desired **Standard** from the drop down box.
- Enter a value for the face width. To display the recommended minimum face width click on the "**?**" button.
- Select the appropriate tooth proportion.
- Enter or use the spin buttons to change the number of teeth.
- Select the desired number of threads for the worm. The gear ratio is 1: the number of gear teeth divided by the number of threads.
- Select the direction of the worm gear. Note: A right hand gear's teeth lean to the right when its face is placed on a horizontal surface.
- Check the **Create Tooth Pattern** if you want the tooth to be patterned. Note: Patterning worm gear teeth can be a time consuming process. You wish to limit the number of teeth created. You may change this value later in SolidWorks.
- If the **Create Tooth Pattern** is checked, the **Teeth to Draw** box becomes enabled. If it is desired to pattern the tooth other than a full pattern then input the number of teeth to be drawn. This can be changed in SolidWorks after the part is drawn simply by editing the definition of the **Tooth Pattern**.
- Determine the size of the worm pitch by entering a value for the **Worm Pitch Diameter** or the **Worm Helix Angle.**
- Make any necessary changes to the **Hub Mounting**.
- Select **Worm w/Gear** to model both parts, **Worm only** to model just the worm or **Gear only** to model the gear.
- Click on the **Finish button** to complete the gear.

Creating Involute Splines

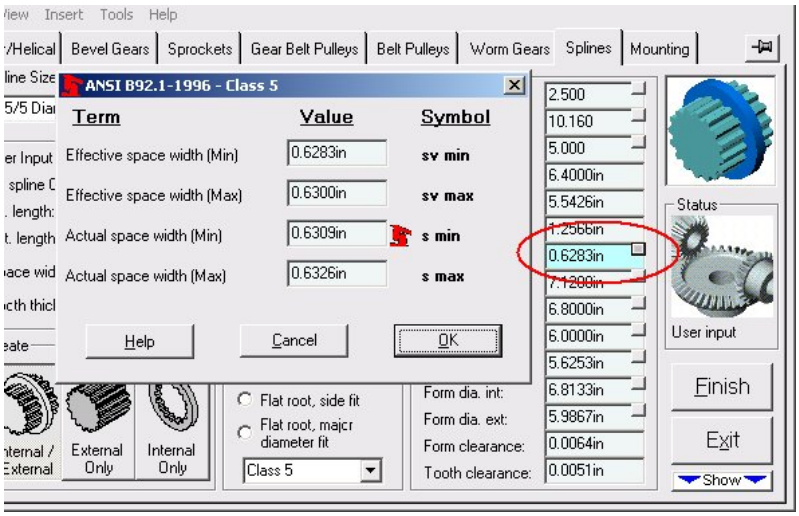
To create an involute spline:



- Select the desired spline size from the drop down box.
- Enter or use the spin box to change the number of teeth.
- Select the pressure angle.
- Select the root and fit.
- Use the drop down box to select the class of fit.
- When a class fit is select GearTrax calculates the tooth thickness and space width and prohibits the user from changing these values.
- Enter or use the spin buttons to change the space width for the internal spline.
- Enter or use the spin buttons to change the tooth width for the external spline
- To have GearTrax model both parts, select **Internal/External** button and then the **Finish** button.
- To have GearTrax model only the external spline select the **External Only** button and then the **Finish** button.
- To have GearTrax model only the internal spline select the **Internal Only** button and then the **Finish** button.

Getting more information about the Spline

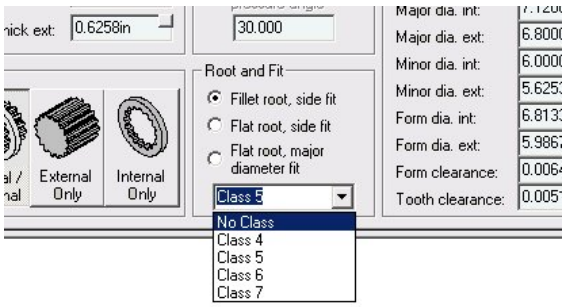
To get more information:



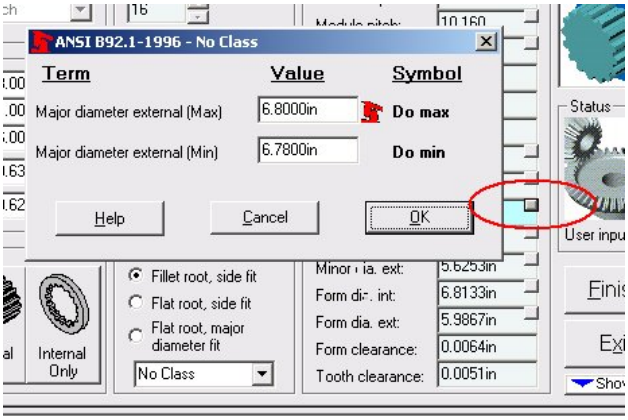
- Click on the small button in the upper right hand corner of the text box (if available). This will open a dialog box with related information for the selected item. Note: the GearTrax icon next to the text box indicates the value used for creating the geometry in SolidWorks. The values in the text boxes cannot be edited unless **No Class** is selected.

Creating a Non-Standard Spline

To create a non-standard spline:



- From the drop down box in the **Root and Fit** frame select the No Class option. This will allow the values to be edited in the dialog box. Click on the small button in the upper right hand corner of the text box (if available). This will bring up a window with related information for the selected item.



Updating a Spline

To update an involute spline:

- Automatically updating involute splines is not available in this version of GearTrax. Following the steps below can redraw the tooth profile:

- Open the tooth sketch and delete all of the sketch entities.
- In GearTrax, set the desired values for the spline.
- Click **Insert** and select **Tooth Profile** Cut.
- Close the sketch to rebuild the part.

Inserting a Spline Tooth Profile

You can create a part with multiple tooth profiles.

To add a tooth profile to a sketch:

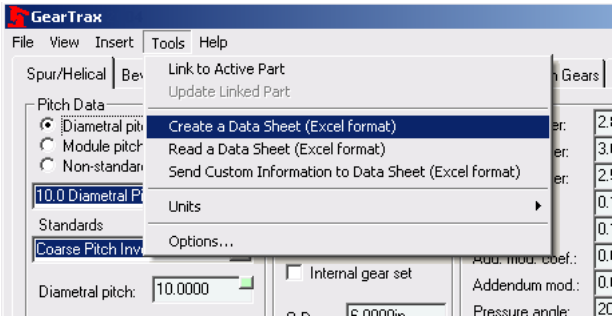
- Start a new sketch.
- In GearTrax select the **Spline** tab.
- Set the values for the spline.
- In the GearTrax menu click on **Insert.** Select **Tooth Profile Cut**.
- GearTrax will create the sketch with the center of the spline on the origin.

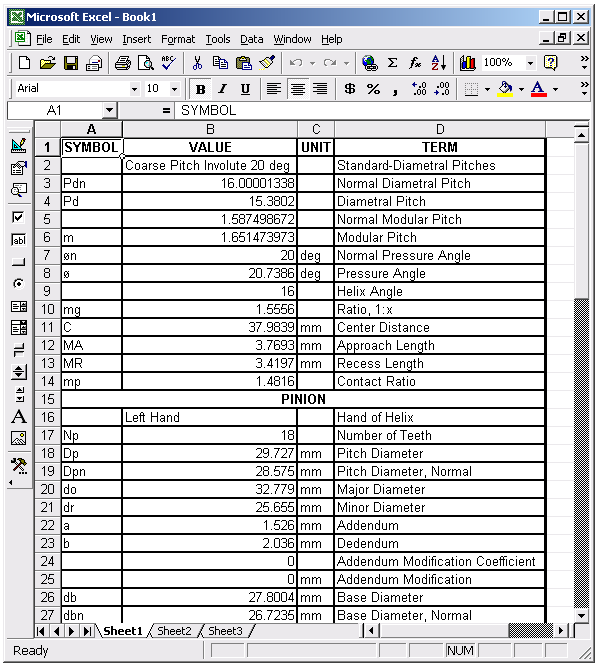
Creating an Excel Data Sheet

Allows you to create an Excel data sheet for spur/helical, bevel gears and splines.

To create an Excel data sheet:

- Select the Spur/Helical tab or the Spline tab.
- Click **Tools.** Select **Create Excel Data Sheet**.
- A new sheet will be started in Excel containing data for the current spur/helical gear or spline in GearTrax.





	A	B	C	D
1	SYMBOL	VALUE	UNIT	TERM
2		Coarse Pitch Involute 20 deg		Standard-Diametral Pitches
3	Pdn	16.00001338		Normal Diametral Pitch
4	Pd	15.3802		Diametral Pitch
5		1.587498672		Normal Modular Pitch
6	m	1.651473973		Modular Pitch
7	øn	20	deg	Normal Pressure Angle
8	ø	20.7386	deg	Pressure Angle
9		16		Helix Angle
10	mg	1.5556		Ratio, 1:x
11	C	37.9839	mm	Center Distance
12	MA	3.7693	mm	Approach Length
13	MR	3.4197	mm	Recess Length
14	mp	1.4816		Contact Ratio
15		PINION		
16		Left Hand		Hand of Helix
17	Np	18		Number of Teeth
18	Dp	29.727	mm	Pitch Diameter
19	Dpn	28.575	mm	Pitch Diameter, Normal
20	do	32.779	mm	Major Diameter
21	dr	25.655	mm	Minor Diameter
22	a	1.526	mm	Addendum
23	b	2.036	mm	Dedendum
24		0		Addendum Modification Coefficient
25		0	mm	Addendum Modification
26	db	27.8004	mm	Base Diameter
27	dbn	26.7235	mm	Base Diameter, Normal

Reading an Excel Data Sheet

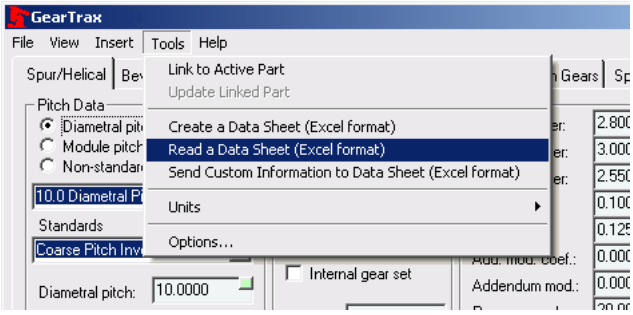
Allows you to read an Excel data sheet that was previously created with GearTrax and modified manually.

To read an Excel data sheet:

- Excel must be running with the data sheet as the current document.
  - Check for multiple instances of Excel. GearTrax will attempt to read the first instance only.

- Select the Spur/Helical tab or the Spline tab.

- Click **Tools.** Select **Read a Data Sheet (Excel format)**. GearTrax is updated with the information from the Excel data sheet.

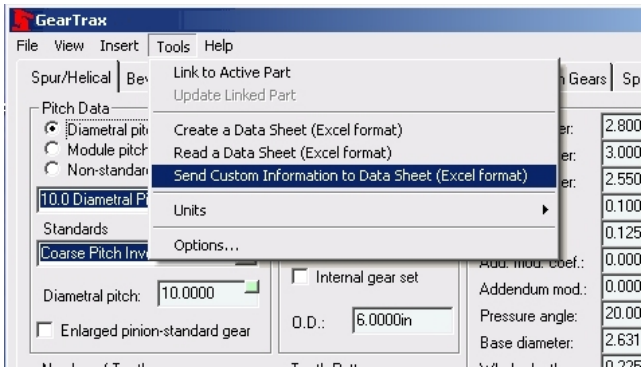


Sending the Custom Information to an Excel data sheet

Allows the user to send the Custom Information stored in any SolidWorks part file to a Excel spread sheet.

To create an Excel Spread using the Custom Information data.

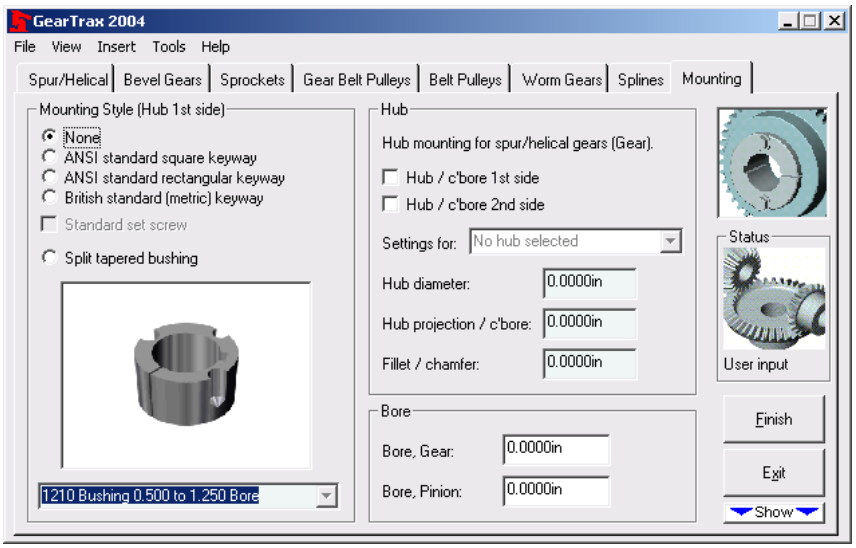
- SolidWorks must be open with a part file active.
- Excel must be available.
- Click **File>Send Custom Information to Data Sheet (Excel format)** from the GearTrax menu to dump the contents of the custom information data of the current SolidWorks part file. This does not need to be a part file created with GearTrax.



Adding a mounting style to the Gear or Pulley

To add a mounting style:

- Select a Mounting style from the **Mounting** tab.



- Selecting a keyway style will allow you to have GearTrax model a standard setscrew hole. The setscrew hole will be drawn only if a hub is being drawn. The setscrew hole will be centered on each of the hub(s).
- If a bore for a split-tapered bushing is called for, check the **Split Tapered Bushing** button. Select the appropriate **Split Tapered Bushing** from the drop down box.
- Note: The split-tapered bushing is not drawn only the bore and mounting holes. See the Camnetics web site www.camnetics.com or the CD for libraries of standard split tapered bushings.
- Select Hub 1st side and/or Hub/c'bore 2nd side.
- Set the values for the appropriate hub. Use the **Settings for:** drop down box to enter values for each of the hubs to be drawn.
- Click on the **Finish** button to complete the drive component.
- A counter bore may be drawn on the second side by entering a negative number in the **Hub projection / c'bore** input box.

Opening and Saving GearTrax files

Allows the user to save and open data they have entered for the current tab.

To save a GearTrax file:

- Click on the **File>Save** menu option.
- Enter a file name.
- Click on **Save** to invoke the file browser.
- Browse to the folder you would like the GearTrax file saved to.
- Enter a file name for the GearTrax file.
- Click on the **Save** button to save the GearTrax file.
- Or click **Cancel** to cancel the save option.

To open a GearTrax file:

- Click on **File>Open** to invoke the file browser.
- Browse to the folder where the GearTrax file is located.
- Click on the file name.
- Click on the **Open** button to open the GearTrax file.
- Or double click on the file name to open the GearTrax file.
- Or click **Cancel** to cancel the open option.

Capturing the GearTrax Screen

Allows the user to capture the GearTrax screen

To Capture the GearTrax screen to a file.

- To capture the current GearTrax screen to a bit map file click **File>Capture GearTrax Screen to File.** The file will be saved at <Program file location>/Capture.bmp

To Capture the GearTrax screen to the clipboard.

- To capture the current GearTrax screen to the clip board click **File>Capture GearTrax Screen to the Clipboard.** The user may paste the image from the clipboard to most paint programs.

To Capture the GearTrax screen to the MS Paint.

- To capture the current GearTrax screen to MS Paint click **File>Capture GearTrax Screen to MS Paint.** The user may then save the image to different locations and file formats using MS Paint.

To Empty the Clip Board.

- To Empty the Clipboard click **File>Empty Clipboard.** This will free up any memory being used by the clipboard.

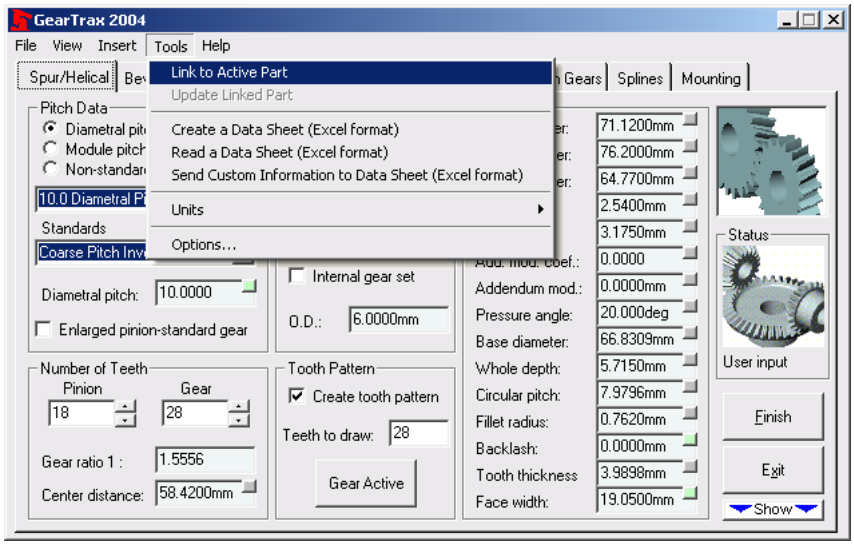
Modification of Parts

Spur/helical and straight bevel gears created in GearTrax can be modified.

For example, you have a spur gear in an assembly and it was decided to change the tooth profile from a full involute 20-degree pressure angle with a 20-diametral pitch to stub involute with an 18-diametral pitch.

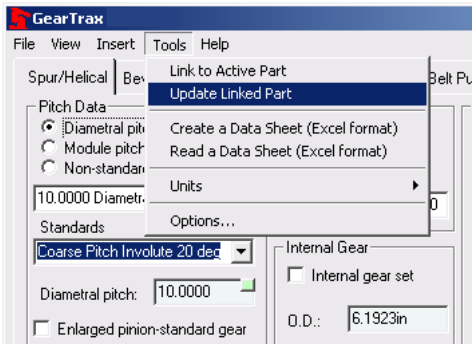
GearTrax can modify the tooth profile, face width and the number of teeth.

Because of the complicated nature of the various involute forms, the tooth will be reconstructed rather than parametrically changed while other features of the gear will be parametrically changed.



To modify an existing part:

- Make the part (gear, sprocket, etc) the active document in SolidWorks.
- From the main GearTrax menu select the **Tools** option.
- Select the **Link to Active Part** option.



- The design information of the part is loaded into GearTrax.
- Make the necessary changes to the GearTrax forms.
- Modify the part by selecting **Update Active Part** in the GearTrax tools menu.
- The part may also be recreated as a new part by selecting the **Finish** Button.
- Changes made to the part in this fashion will require the least amount of editing to the assembly mates.
- If manual changes have been made to the part prior to this, the results may be unpredictable.

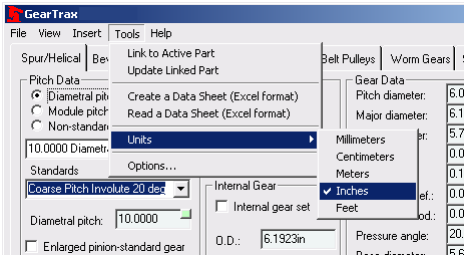
Tip: For best results, make any manual modifications to the part by adding new features and leaving the base features unaltered.

Setting the Length Units

The GearTrax length setting is determined by the current SolidWorks default length setting. The GearTrax setting may be changed without regard to the SolidWorks setting.

To set the Length Units:

- Click the **Tools** option from the pull down menu, and then select **Units**.
- Click the appropriate length unit.
- This will set the length unit of GearTrax and will change the SolidWorks part unit setting when a new part is created.

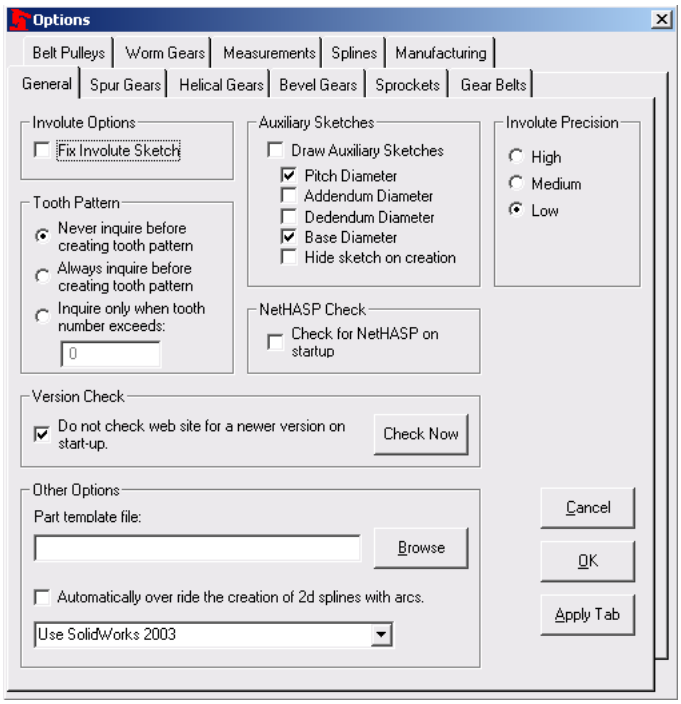


GearTrax Options

Allows you to customize GearTrax options and calculate over-pin-measurements.

To set options:

- Click **Tools** from the pull down menu, then **Options**.
- Select the appropriate tab for the settings you want to change.
- To register the new settings for only the current tab click **Apply Tab**.
- To register the new settings for all the tabs click **OK**.
- To exit the options without registering any changes click **Cancel**.

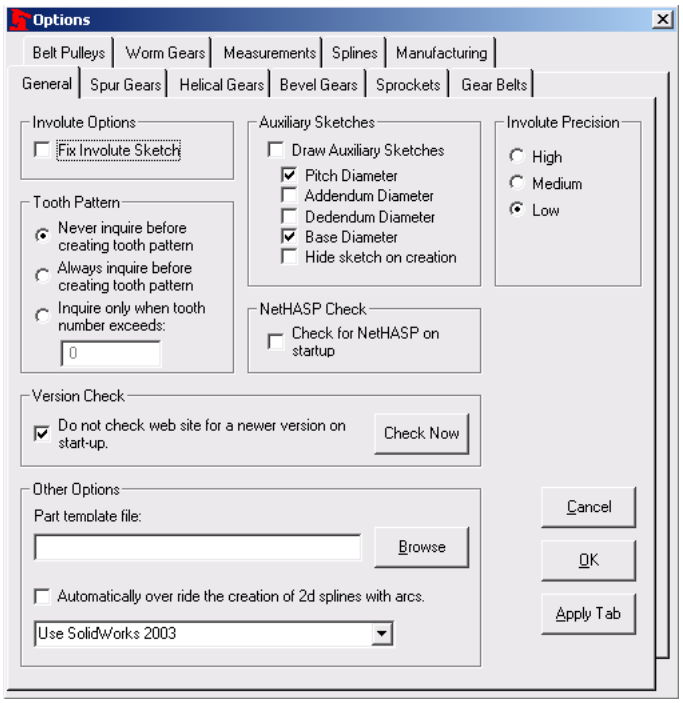


General Options

Lets you set properties for involute sketches, tooth patterns and auxiliary sketches.

To set options:

- Click **Tools** option from the pull down menu, then **Options**.
- Select the **General** tab.
- Select options from the following list and make the changes that you prefer.
- Click **OK** to accept the changes; click **Cancel** to discard the changes and exit the dialog box; click **Apply Tab** to accept the changes for this tab only.



Fix Involute Sketch

Lets you specify if GearTrax should fix the entities of the involute sketch as GearTrax creates the model.

To set options:

- Click **Tools** option from the pull down menu, then **Options**.
- Select the **General** tab.
- Select **Fix Involute Sketch** check box if you would like GearTrax to fix the involute sketch entities as the model is created. Fixing the involute sketch may prevent unintentional dragging of the sketch points. The gear will take longer for SolidWorks to generate with this option set.

Tooth Pattern

Lets you specify if and when GearTrax will inquire before the tooth pattern is created.

To set options:

- Click **Tools** option from the pull down menu, then **Options**.
- Select the **General** tab.
- Select **Never inquire before creating tooth profile** if GearTrax should always create the tooth profile without interrupting the modeling process.
- Select **Always inquire before creating a tooth pattern** if you would like GearTrax to inquire before any tooth pattern is created.
- Select **Inquire only when tooth number exceeds** if you would like GearTrax to inquire before creating a tooth profile above a certain number of teeth. Selecting this option enables the text box. Enter that value in the text box.

Auxiliary Sketches

Lets you specify the creation of an auxiliary sketch.

To set options:

- Click **Tools** option from the pull down menu, then **Options**.
- Select the **General** tab.
- Select the **Draw Auxiliary Sketches** check box for GearTrax to create a sketch as the model is created.
- Select **Pitch Diameter** if you would like the pitch diameter included on the sketch.
- Select **Addendum Diameter** if you would like the addendum diameter included on the sketch. This option will be ignored if it does not apply to the product being modeled.
- Select **Dedendum Diameter** if you would like the dedendum diameter included on the sketch. This option will be ignored if it does not apply to the product being modeled.
- Select **Base Diameter** if you would like the base diameter included on the sketch. This option will be ignored if it does not apply to the product being modeled.
- Select **Hide sketch on creation** if you would like GearTrax to hide the sketch as the model is created.

NetHASP Check

Lets you specify if GearTrax should check for a NetHASP at startup.

To set options:

- Only use this option is you have a Camnetics NetHASP on the network.
- Click **Tools** option from the pull down menu, then **Options**.
- Select the **General** tab.
- Select the **Check for NetHASP on startup** check box if you would like GearTrax to check for a NetHASP on startup.
- Selecting this option without a NetHASP present or functioning will delay the startup of GearTrax as the network is searched for the presence of the NetHASP.
- Click **OK** to accept the changes; click **Cancel** to discard the changes and exit the dialog box; click **Apply Tab** to accept the changes for this tab only.

Version Check

Lets you specify if GearTrax should check for a more recent release of GearTrax at startup.

To set options:

- Click **Tools** option from the pull down menu, then **Options**.
- Select the **General** tab.
- Select **Do not check web site for a newer version on start-up** if you do not want GearTrax to check the web site for a more recent release. If left unchecked and you have access to the internet, GearTrax will look at the web site for a more recent release and notify you if one exists. You will need to manually download the latest file and go through the installation procedure. This is not done automatically.

2d Spline Creation Over ride

Lets you specify if GearTrax should automatically over ride the creation of 2d splines with a series of arcs.

To set options:

- Click **Tools** option from the pull down menu, then **Options**.
- Select the **General** tab.
- Select the **Automatically over ride the creation of 2d splines with arcs** if you would like GearTrax to create tooth profiles with arcs rather than 2d splines.

SolidWorks Version

Lets you specify which version of SolidWorks should be used with GearTrax.

To set options:

- Click **Tools** option from the pull down menu, then **Options**.
- Select the **General** tab.
- Use the drop down box to specify which version of SolidWorks to use with GearTrax.

Part Template File

Lets you specify a template to be used when starting a new GearTrax part.

To set options:

- Click **Tools** option from the pull down menu, then **Options**.
- Select the **General** tab.
- Browse to the location of the template files and double click on the desired template.

Tooth Creation

Two methods are available for generating the tooth form, splines or arcs.

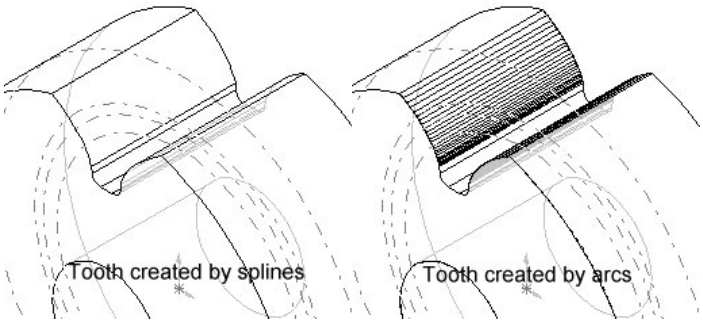
If the gear geometry will be used in conjunction with a CAM system, arcs may be the wise choice. Most CAM systems will convert splines to a series of straight lines, while the arcs will remain as arcs.

A pair of splines is defined by a series of points along the tooth form. The more complex the tooth form, the more points are used to create the tooth spline. The spline is extruded to create the tooth feature. Splines are the faster of the two options in SolidWorks.

A series of arcs are used to create the tooth form. The more complex the tooth form is, the more arcs are used to create the tooth profile.

For help in setting GearTrax to create the tooth profile in arcs or splines, see Spur Gear Options.

Also see Manufacturing Methods for Spur/Helical Gears.

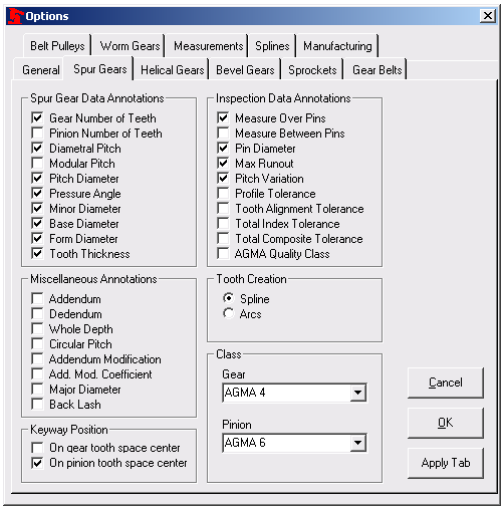


Spur Gear Options

Allows you to customize Spur Gear Annotations for drawings and how the tooth profile is created.

To set options:

- Click **Tools**, then **Options**.



- Select the **Spur Gears** tab. Selecting a **Class** option from the drop down box will "check" the appropriate boxes for that standard. Add or subtract from the check boxes for the desired annotations when adding a note on a drawing. These options are not saved with the part file and will need to be set at the time the note is created on the drawing.
- **Tooth Creation**. To create the tooth profile with a spline, select the **Spline** option or select the **Arcs** option to create the tooth profile with a series of arcs.
- Typically, creating the tooth with a spline will increase the performance of rebuilds but may make it more difficult if this geometry is use with a CAM system for fabrication.
- Creating the gear with arcs will allow the tooth profile to be mated in an assembly, but may require more time for rebuilds. Arcs may also be more appropriate for use with some CAM systems for fabrication.
- **Keyway Position**. To position the keyway on the tooth space center place a check in the appropriate box for the gear or the pinion. If left unchecked the keyway will be positioned on the tooth center. A keyway must be selected from in the mounting tab.
- **Class**. Selecting a class in this tab sets up the appropriate check boxes for a note. It does not effect the gear creation, only what is displayed in the annotation.
- **Addendum Modification**. Select either Automatic only or Manual only. If automatic only is select GearTrax, will enter the minimum modification recommended for certain standards, if available. If manual only is select then GearTrax will allow changes to the addendum modification and the addendum modification coefficient values.

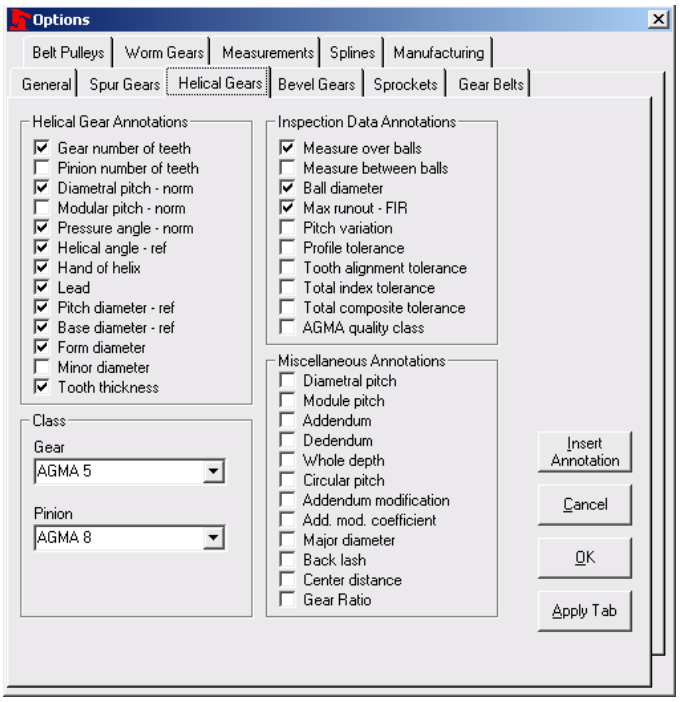
- Click **OK** to accept the changes; click **Cancel** to discard the changes and exit the dialog box; click **Apply Tab** to accept the changes for this tab only.

Helical Gear Options

Allows you to customize Helical Gear Annotations for drawings.

To set options:

- Click **Tools**, then **Options**.
- Select the **Helical Gears** tab.



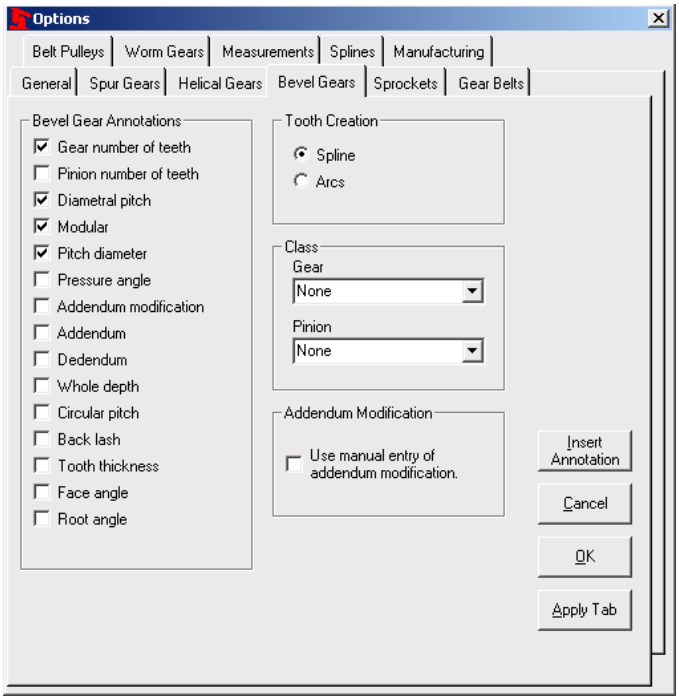
- Selecting a **Class** option from the drop down box will “check” the appropriate boxes for that standard. Add or subtract from the check boxes for the desired annotations when adding a note on a drawing. These options are not saved with the part file and will need to be set at the time the note is created on the drawing.
- Click **OK** to accept the changes; click **Cancel** to discard the changes and exit the dialog box; click **Apply Tab** to accept the changes for this tab only.

Bevel Gear Options

Allows you to customize Bevel Gear Annotations for drawings.

To set options:

- Click **Tools**, then **Options**.
- Click the **Bevel Gears** tab.



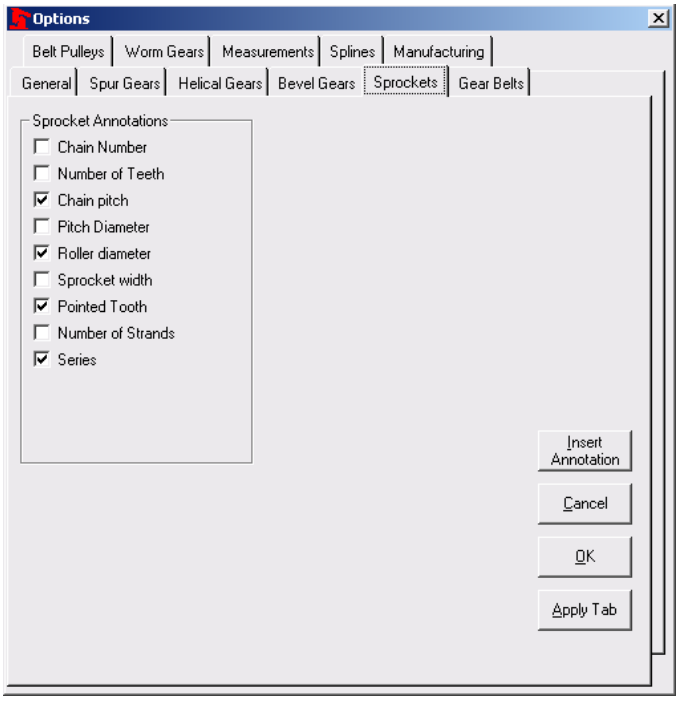
- Selecting a **Class** option from the drop down box will "check" the appropriate boxes for that standard. Add or subtract from the check boxes for the desired annotations when adding a note on a drawing. These options are not saved with the part file and will need to be set at the time the note is created on the drawing.
- To create the tooth profile with a spline, select the **Spline** option or select the **Arcs** option to create the tooth profile with a series of arcs.
- **Addendum Modification**. If this option is unchecked GearTrax will modify the addendum based on Gleason standards. If this option is check the user will need to specify the desired addendum modification.
- Click **OK** to accept the changes; click **Cancel** to discard the changes and exit the dialog box; click **Apply Tab** to accept the changes for this tab only.

Sprocket Options

Allows you to customize Sprocket Annotations for drawings.

To set options:

- Click **Tools**, then **Options**.
- Select the **Sprockets** tab.



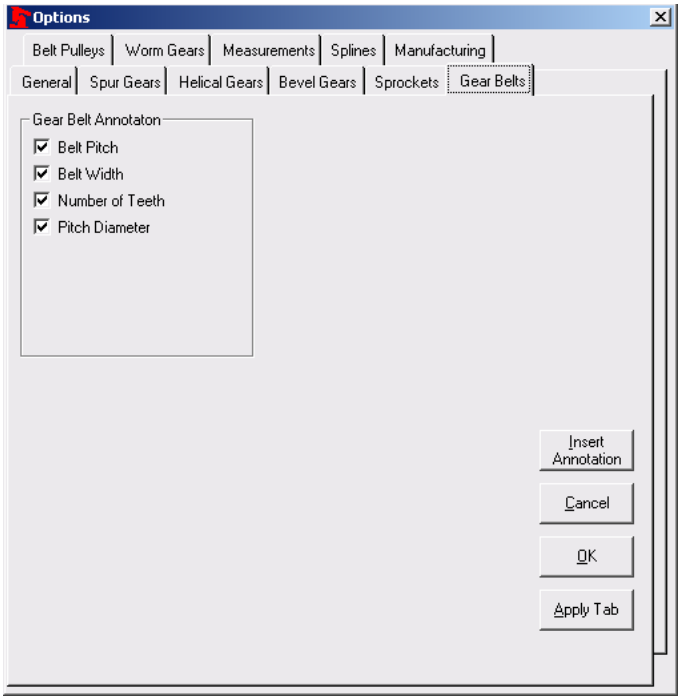
- Add or subtract from the check boxes for the desired annotations when adding a note on a drawing. These options are not saved with the part file and will need to be set at the time the note is created on the drawing.
- Click **OK** to accept the changes; click **Cancel** to discard the changes and exit the dialog box; click **Apply Tab** to accept the changes for this tab only.

Gear Belt Options

Allows you to customize Gear Belt Annotations for drawings.

To set options:

- Click **Tools**, then **Options**.
- Select the **Gear Belts** tab.



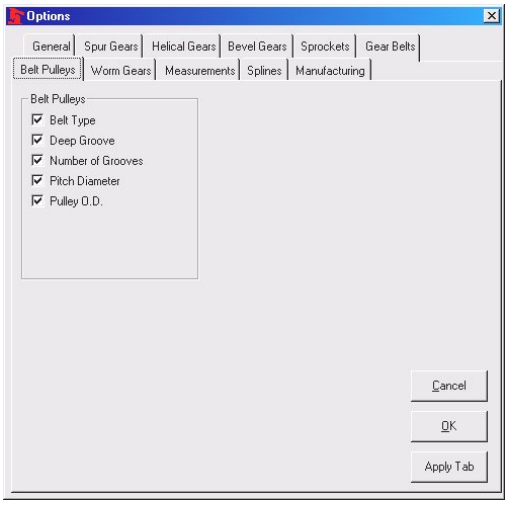
- Add or subtract from the check boxes for the desired annotations when adding a note on a drawing. These options are not saved with the part file and will need to be set at the time the note is created on the drawing.
- Click **OK** to accept the changes; click **Cancel** to discard the changes and exit the dialog box; click **Apply Tab** to accept the changes for this tab only.

Belt Pulley Options

Allows you to customize Belt Pulleys for drawings.

To set options:

- Click **Tools**, then **Options**.
- Select the **Belt Pulleys** tab.



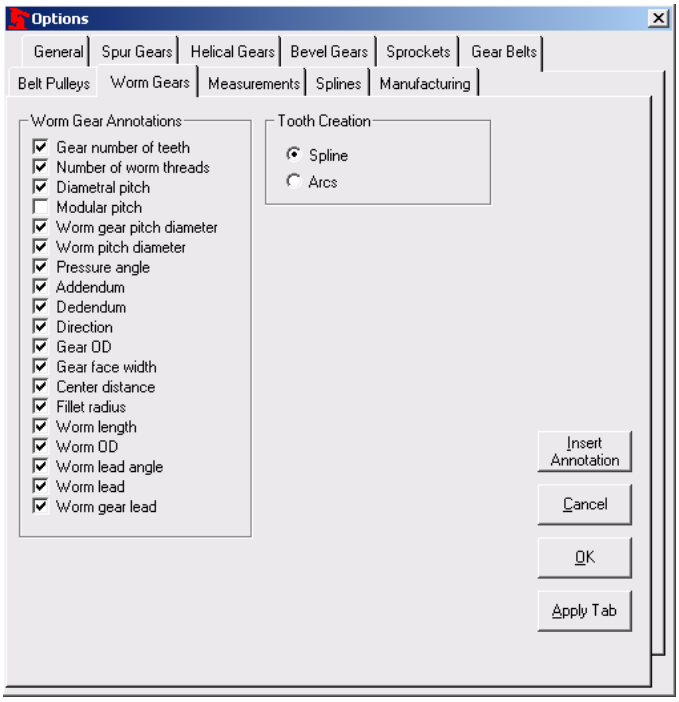
- Add or subtract from the check boxes for the desired annotations to be added to the note on a drawing. These options are not saved with the part file and will need to be set at the time the note is created on the drawing.
- Click **OK** to accept the changes; click **Cancel** to discard the changes and exit the dialog box; click **Apply Tab** to accept the changes for this tab only.

Worm Gear Options

Allows you to customize Worm Gear Annotations for drawings.

To set options:

- Click **Tools**, then **Options**.
- Select the **Worm Gears** tab.



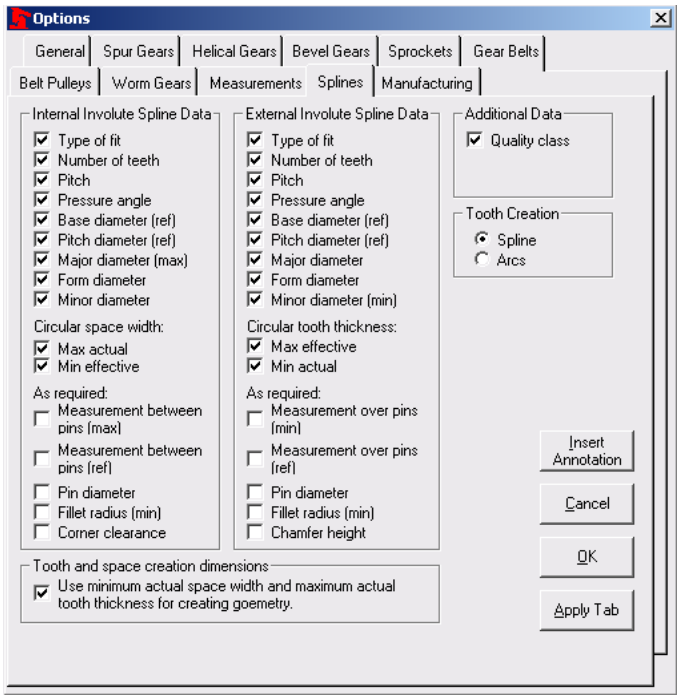
- Selecting a **Class** option from the drop down box will "check" the appropriate boxes for that standard.
- Add or subtract from the check boxes for the desired annotations when adding a note on a drawing. These options are not saved with the part file and will need to be set at the time the note is created on the drawing.
- To create the tooth profile with a spline, select the **Spline** option or select the **Arcs** option to create the tooth profile with a series of arcs.
- Click **OK** to accept the changes; click **Cancel** to discard the changes and exit the dialog box; click **Apply Tab** to accept the changes for this tab only.
-

Spline Options

Allows you to customize Spline Annotations for drawings and how the tooth profile is created.

To set options:

- Click **Tools**, then select **Options.**
- Click on the **Splines** tab.



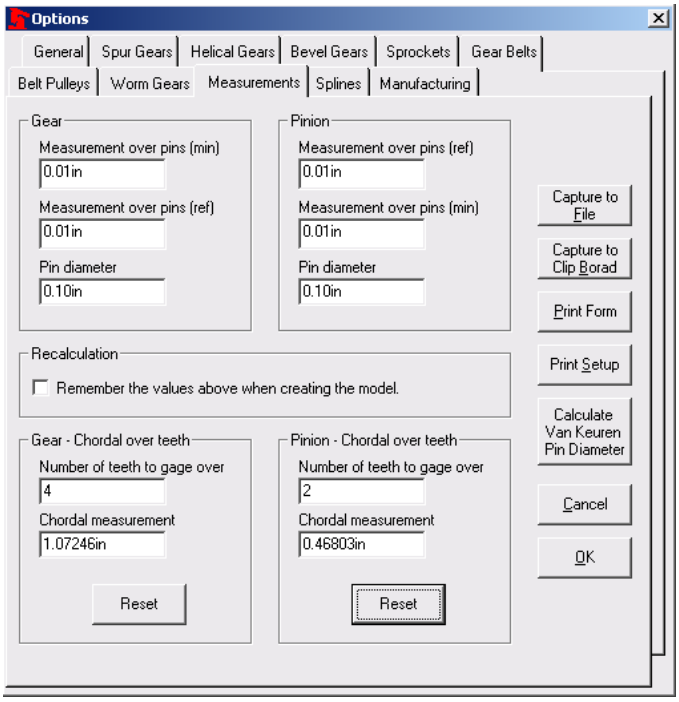
- The involute spline tooth cut may be created with splines or a series of small arcs. Select the appropriate option button.
- Add or subtract from the check boxes for the desired annotations to be added to the note on a drawing. These options are not saved with the part file and will need to be set at the time the note is created on the drawing.
- Click **OK** to accept the changes; click **Cancel** to discard the changes and exit the dialog box; click **Apply Tab** to accept the changes for this tab only.

Measurements Options

Allows you to calculate the over the pin measurements.

To set options:

- Click **Tools**, then **Options**.
- Select the **Measurements** tab.



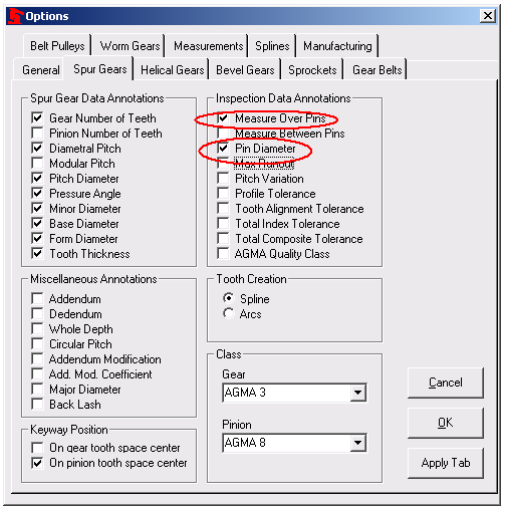
- Click the **Calculate Van Keuren Pin Diameter** button to use the industry standard for pin sizes.
- You can calculate the “Over the Pin Measurement” with a desired pin size by entering that value into the **Pin Diameter** text box. **WARNING:** GearTrax will not inform you if the pin size is usable.
- GearTrax can calculate the tooth thickness for spur, helical or spline by entering the pin diameter and then the measurement over or under pins.
- The standard **Chordal measurements** and **the Number of teeth to gage over** is displayed for external spur gears and pinions.

How does GearTrax store these values?

- GearTrax places the measurement values in the custom information properties of the part. When GearTrax creates the part model, these values will be recalculated using the Van Keuren Pin Diameters. If you want to use a specific pin diameter other than the Van Keuren value, enter the diameter and check the box **Use the value above when creating**

the model before creating the model. This will prevent GearTrax from recalculating using the Van Keuren pin diameter during the creation of the gear or spline.

- If you would like to change these values later you will need to manually change the values in SolidWorks. Click File, Properties, Custom tab. See “Summary Info” help section in SolidWorks for more information.
- These values can be added to a drawing by checking the appropriate boxes in the spur gears option dialog box. The boxes checked will be used when inserting an annotation to a drawing. See Inserting a Drawing Annotation for more information.



Calculating a Tooth Thickness

GearTrax can be used to calculate the tooth thickness when the over the pin dimension and pin diameters are known.

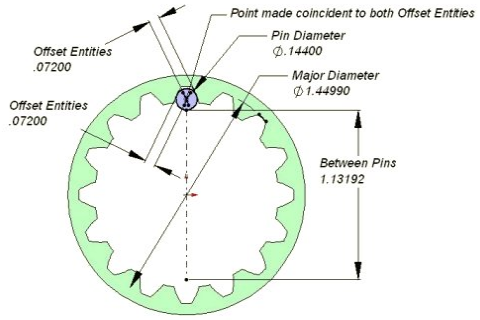
To calculate the tooth thickness:

- Select either the Spur/Helical tab or the Spline tab.
- Specify the number of teeth, diametral pitch or the Module
- Open the Measurements tab in the Options menu.
- Specify a pin diameter for the pinion, gear or the internal or external spline.
- Enter a value in one of the four over or under the pin measurement text boxes.
- GearTrax will return a message with the calculated tooth thickness or a message saying the dimension could not be calculated.

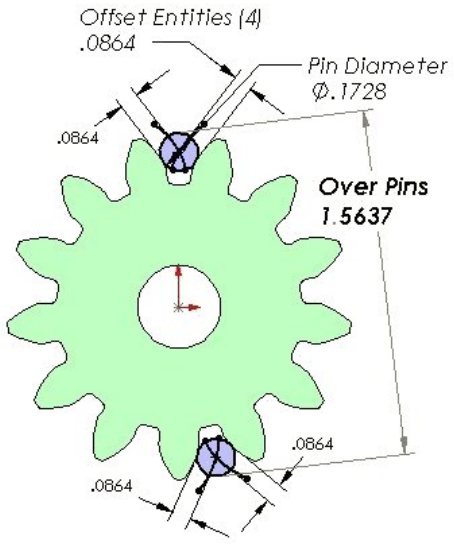
Between Pins Example

- Between pin measurements can be created manually and can be used to verify data.
- Create a new sketch on the face of the gear.

- Select a single tooth spline and create an offset entity equal to ½ of the pin diameter.
- Select the spline on the other side of the tooth space. Create another offset entity equal to ½ the pin diameter.
- Add a point on one of the new offset entities.
- Add a coincident relationship between the point and the offset entity.
- Add a circle with the center on the point. Dimension the circle equal to the pin diameter.
- Add a centerline to the sketch.
- Add a midpoint relationship between the centerline and the center of the gear.
- Add a coincident relationship between the pin outside diameter and an end point of the centerline.
- Add a coincident relationship between the pin center point and the centerline.
- Add the between the pins dimension by adding a dimension to the length of the centerline.



- Sample of a manual verification of between the pins measurements using SolidWorks.
- If the gear has an odd number of teeth you will need to create the sketch with 2 pins.



Sample of a manual verification of over the pins measurements using SolidWorks.

General Formula for Checking External and Internal Spur Gears by Measurement over Pins

M = measurement over wires for external gears or measurement between wires for internal gears.

D = pitch diameter

T = arc tooth thickness on pitch circle

W = wire diameter

N = number of gear teeth

A = pressure angle of gear

a = angle, the cosine of which is required in formulas 2 and 3

1 $$\text{inv}\, a = \text{inv}\, A \pm \frac{T}{D} \pm \frac{W}{D \cos A} \mp \frac{\pi}{N}$$

2 For even numbers of teeth, $$M = \frac{D \cos A}{\cos a} \pm W$$

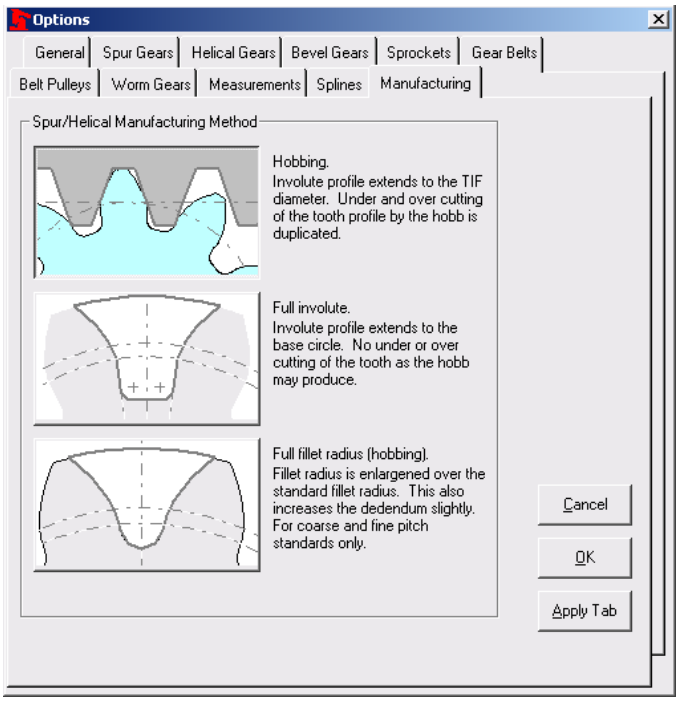
3 For odd numbers of teeth, $$M = \left(\frac{D \cos A}{\cos a}\right)\left(\cos \frac{90^\circ}{N}\right) \pm W$$

Selecting a Manufacturing Method for Spur/Helical Gears

Allows you to specify the Manufacturing method for spur/helical gears.

To set options:

- Click **Tools**, then select **Options.**
- Click on the **Manufacturing** tab.
- Select one of the three manufacturing methods by clicking on the picture:
- **Hobbing**. Involute profile extends to the TIF diameter. Under and over cutting of the tooth profile by the hob is duplicated.
- **Full involute**. Involute profile extends to the base circle. No under or over cutting of the tooth as the hob may produce.
- **Full fillet radius (hobbing).** Fillet radius is enlarged over the standard fillet radius. This also increases the dedendum slightly. For coarse and fine pitch standards only. This is a deviation from the standards.

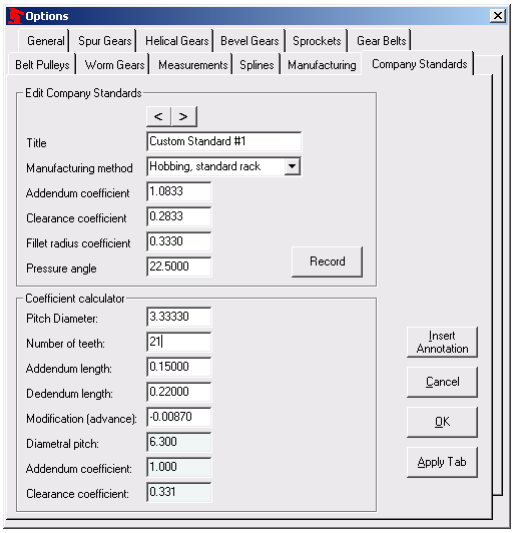


Select Company Standards for Addendum, Clearance and Fillet Radius Coefficients

Allows you to specify three company standards.

To set standards:

- Click **Tools**, then select **Options.**
- Click on the **Company Standards** tab.
- Use the arrow buttons to index through the three different standards.
- Change the title of the standard, if desired.
- Select manufacturing method to be the default for the standard
- Input the values for
    - Addendum coefficient
    - Clearance coefficient
    - Fillet radius coefficient
    - Pressure angle
- Press the record button or index to the next standard.
- These standards will be available for spur and helical gears.

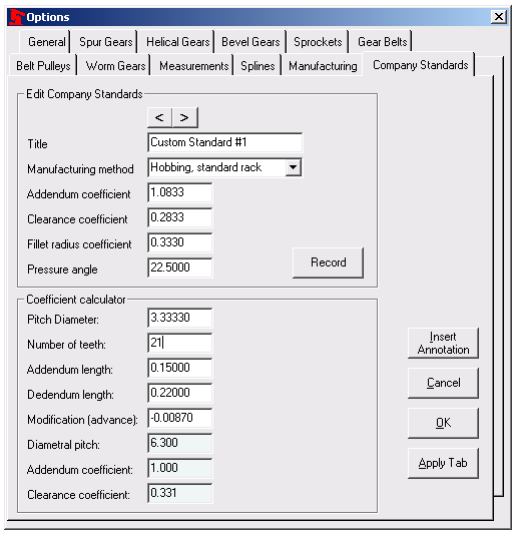


Calculating Addendum and Clearance Coefficients

This may help decipher 3rd party specifications with limited information.

To calculate the coefficients:

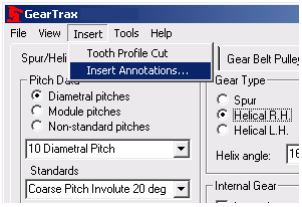
- Click **Tools**, then select **Options.**
- Click on the **Company Standards** tab.
- Enter the values for the following:
  - Pitch diameter
  - Number of teeth
  - Addendum length
    - Note: Addendum = (major diameter – pitch diameter) / 2
  - Dedendum length
    - Note: Dedendum = (pitch diameter – minor diameter) / 2
  - Modification, this is sometimes called tool advance. It's normally negative for the gear and positive for the pinion.



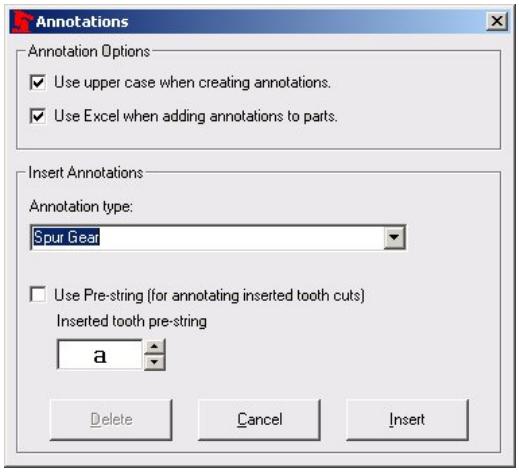
Inserting a Drawing Annotation

To insert a note on a drawing:

Define the note by setting the options for the product. See GearTrax Options



From the GearTrax menu select **Insert** and then **Insert Annotations** to display the **Annotations** dialog box.



To set Annotation options:

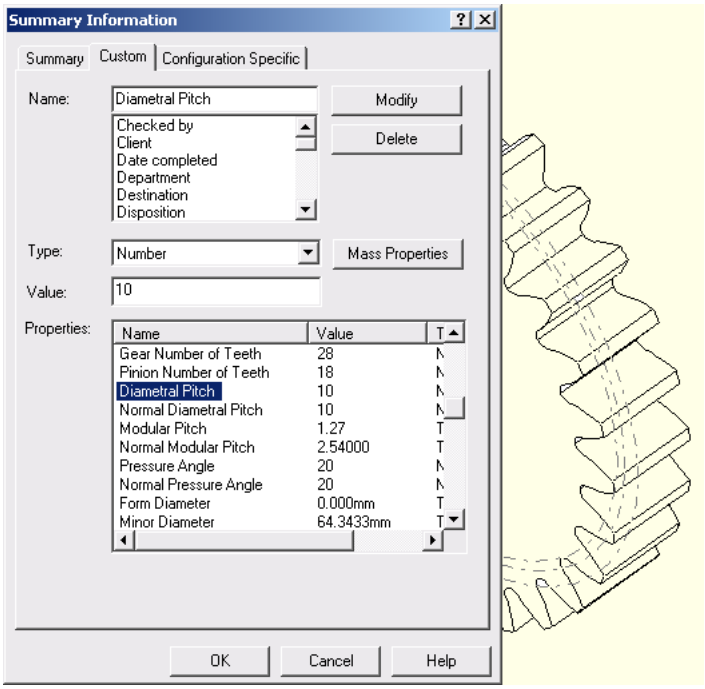
- Place a check in the appropriate box to have annotations created using all upper case letters.
- Place a check in the appropriate box to have annotations created using Excel.

To insert an Annotation to a part or a drawing:

- From the **Annotations type** drop down box select the appropriate annotation type.
- If the desired annotation is for an inserted tooth cut profile then:
- Place a check in the **Use Pre-string** check box.
- The per-string character needs to be specified. Enter the letter in the box or use the up-down arrows to change the letter. If you are unsure of which letter to use you may want to review the Custom Properties of the part in SolidWorks.

To delete the custom information for an inserted tooth cut:

- Place a check in the **Use Pre-string** check box.
- Set the per-string character for the desired set of custom properties.
- Press the **Delete** key



Sample of the Summary Information window in SolidWorks.

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