

GearTrax

COMPONENTS FOR YOUR DRIVE



 **Camnetics**
INCORPORATED

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Introduction

The first seat of GearTrax was sold in 1998. The original GearTrax was programmed using Visual Basic and is limited to 32bit compiling. The new GearTrax is programmed using Visual Studio. We have tried to retain the user friendly interface of the original GearTrax but incorporate the technologies that were developed for our GearTeq product. While many features of GearTeq are now available with the new GearTrax, some of the advanced features will not be available, such as profile modification and crowning and planetary gear sets to mention a few. We like to say the new GearTrax has received a face lift and a brain transplant!

GearTrax is an object oriented/property driven gear design program. It is not the intent of GearTrax to replace your CAD system but to augment the CAD system with a user interface that will allow the gear designer to accurately visualize the components before they are modeled in the CAD system.

GearTrax has a predetermined number of components. For example, the spur tab always has 2 gears, a pinion and a gear. For bevel gears this is important because bevels are normally defined as a set. The number of teeth in both gears defines such properties as the pitch angles. When GearTrax creates in CAD, the user can select between creating only the individual models or the models and the assembly.

GearTrax is programmed in the USA using 100% renewable energy.



Definitions:

CAD

CAD (Computer Aided Design) is the system this program was compiled to run with, either as an add-in or add-on.

Assembly

An assembly is a number of parts placed in a CAD assembly document. For gears that can be created as a set, GearTrax can create the assembly in CAD.

Component

Component is a single GearTrax gear, sprocket, pulley, etc.

Part

Part is a single CAD part document file that contains one GearTrax component.

Model

Model is a CAD part document.

Annotation

Annotation is a text or Excel note placed on the CAD part or drawing document.

Starting GearTrax

GearTrax is compiled and published as a Click-once application. To have GearTrax available on a computer, each user of that computer must install GearTrax. Use the Windows Start menu to start GearTrax by clicking on Start>All Programs>Camnetics, Inc>GearTrax*. Communication with the CAD program is done through the controls in the CAD group box on each of the tabs.

GearTrax Menus

File

Save GearTrax File

Save GearTrax File saves to the .gtx extension.

Open GearTrax File

Open a previously saved GearTrax document. GearTrax documents have .gtx as the file extension.

Capture GearTrax screen to the clipboard

Capture GearTrax screen to the clipboard captures the GearTrax screen, including borders, to the clipboard.

Convert Legacy GearTrax file

Convert Legacy GearTrax file opens a GearTrax file that was saved in the original version of GearTrax. The legacy file extension is .gtx.

Exit

Exit closes GearTrax. The user is prompted to save any unsaved data.

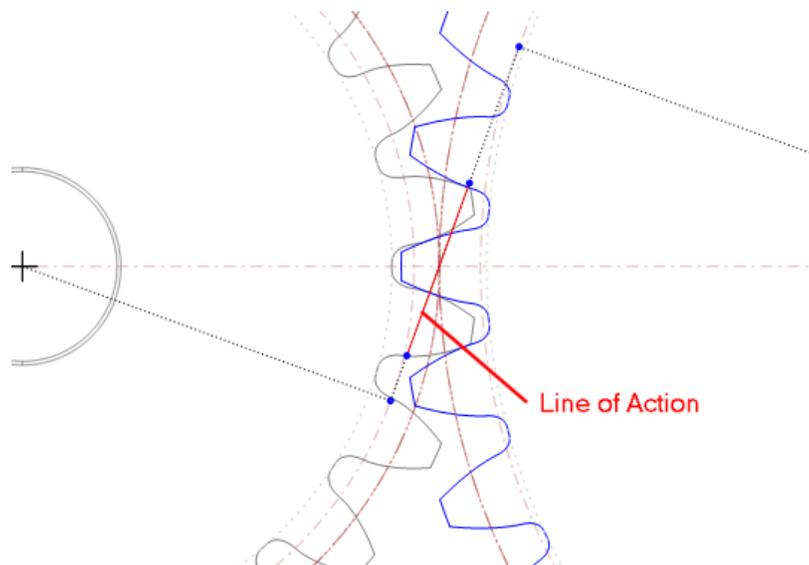
View

Redraw - fit all

Redraw – fit all redraws the assembly to fit on the screen with the front view and resets the drive position to zero.

Show Line of Action

Show Line of Action, if checked, will display the line of action on spur gear pairs with gear mates.



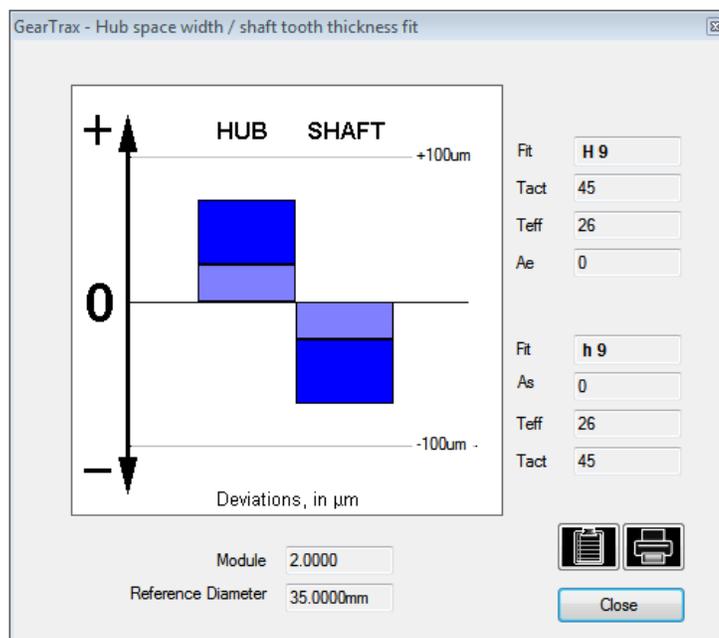
Show Over/Under Pins

Show Over/Under Pins, if checked, will draw a red circle between a pair of spur gear teeth.

Deviation Chart



Deviation Chart opens the deviation chart window. This menu item is normally disabled unless the active component is a DIN or ANSI Module spline. The short cut key is F3.



If Grade, Deviation, Fit and Tolerances values are changed in the main window the chart is updated immediately.

More Data

Spur/Helical

Class and Tolerance menu item opens a window to view and adjust the class and tolerances for the gears.

Operating Diameters menu item opens a window to view the operating diameters of the gear set.

Topping Adjustment menu item opens a window to view and adjust the topping adjustment.

TIF Diameters menu item opens a window to view the true involute form diameters.

Contact Ratio menu item opens a window to view the contact ratio for the gear set.

Splines

Major Diameters menu item opens a window to view and adjust the class and tolerances for the splines.

Minor Diameters menu item opens a window to view and adjust the class and tolerances for the splines.

Space Width menu item opens a window to view and adjust the class and tolerances for the splines.

Tooth Thickness menu item opens a window to view and adjust the class and tolerances for the splines.

Tools

Reverse Animation Direction

Clicking 'Reverse Animation Direction' reverses the direction of the simulation.

Create a Data Sheet for active component

Excel file

Clicking 'Excel file' creates a data sheet of the active component in Microsoft Excel, if available.

Text file

Clicking 'Text file' creates a data sheet of the active component in Microsoft Notepad, if available.

Comma Separated Values (CSV) file

Clicking 'Comma Separated Values file' creates a data sheet in Microsoft Notepad, if available, that can be imported into most spread sheet programs.

Spur03.txt - Notepad

Name: Spur03
Path:
Date:
Time:

| SYMBOL | VALUE | UNIT | TERM |
|--------|-----------------------------|------|-----------------------------------|
| | Coarse_Pitch_Involute_25deg | | Standard |
| Pdn | 8.000000 | | Normal Diametral Pitch |
| Pd | 8.000000 | | Diametral Pitch |
| m | 3.175000 | | Normal Modular Pitch |
| mn | 3.175000 | | Modular Pitch |
| ø | 25.0000 | deg | Normal Pressure Angle |
| ø | 25.0000 | deg | Pressure Angle |
| mp | 0.0000 | deg | Helix Angle |
| C | 0.3929 | | Ratio, 1:x |
| MA | 2.4516 | in | Center Distance |
| MR | 0.2368 | in | Approach Length |
| mp | 0.2436 | in | Recess Length |
| Np | 1.3577 | | Contact Ratio |
| Dp | 11 | | Number of Teeth |
| Dpn | 1.3750 | in | Pitch Diameter |
| do | 1.3750 | in | Pitch Diameter, Normal |
| dr | 1.6536 | in | Major Diameter |
| a | 1.0911 | in | Minor Diameter |
| b | 0.1393 | in | Addendum |
| b | 0.1420 | in | Dedendum |
| | 0.11428 | | Addendum Modification Coefficient |
| db | 0.0143 | in | Addendum Modification |
| dbn | 1.2462 | in | Base Diameter |
| ht | 1.2462 | in | Base Diameter, Normal |
| p | 0.2813 | in | Whole Depth |
| pn | 0.3927 | in | Circular Pitch |
| B | 0.3927 | in | Circular Pitch, Normal |
| t | 0.0375 | in | Fillet Radius |
| tn | 0.0000 | in | Backlash |
| F | 0.20967 | in | Tooth Thickness |
| F | 0.20967 | in | Tooth Thickness, Normal |
| dw | 0.7500 | in | Face Width |
| M | 0.2160 | in | Size Between Pins |
| M | 1.6794 | in | Pin Diameter |
| M | 1.6710 | in | Measurement over Pins |
| | | | Measurement over Pins-Minimum |
| | | | Chordal over Teeth |
| | 0 | | Number of Teeth to gage over |
| | 0.0000 | in | Chordal Measurement |
| | 7 | | AGMA Quality class |
| | 0.0032 | in | Max Runout |
| | 0.00099 | in | Pitch Variation |
| | 0.0013 | in | Profile Tolerance |
| | 0 | in | Tooth Alignment Tolerance |
| | 0.0026 | in | Total Index Tolerance |
| | 0.0062 | in | Total Composite Tolerance |
| | 0.00500 | in | Tooth Thickness Tolerance |

Sample of a data sheet created as text file

Balance Addendum Modification...



The Balance Addendum Modification menu option opens a dialog window that can adjust the addendum modification for the active gear and its mate. The mate must be a spur gear with a gear mate.

The change in addendum modification coefficient can be automatically estimated. The addendum modification coefficients for the current component and its mating component can be balanced by the following formula:

$$x1 = 1 / 3 * (1 - 1 / u) + (x1 + x2) / 1 + u$$

In this formula, u is the gear ratio while x1 and x2 are the addendum modification coefficients of the pinion and the gear, respectively. The pinion is the gear with the lesser number of teeth.



The user may manually enter values for the sum and individual addendum modification coefficients. The screen will be updated to reflect these changes.

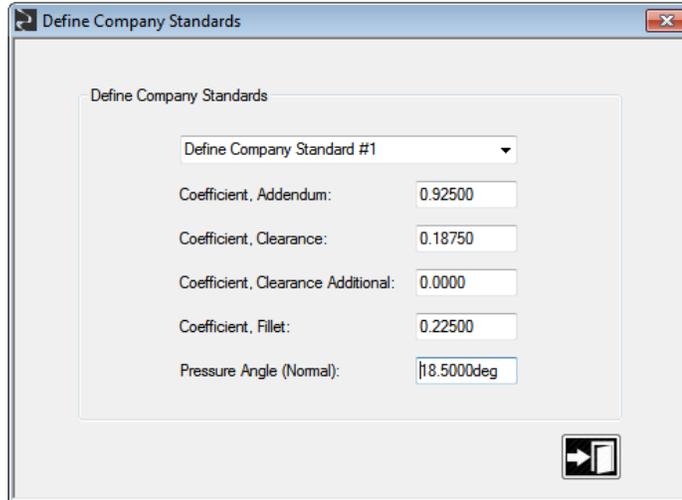
The slider bar may also be used to change the distribution of the modification.

Click the Cancel button to restore the original values and close the window.

Click the Accept button to use the values and close the window.

Company Standards

Company Standards allows the user to configure up to three different company standards.



Define Company Standards

Select one of the three company standards to load it into the property grid. Make any changes to the standard and then use the component selector to switch back to a component.

Coefficient Addendum

Coefficient Addendum defines the constant that is used to calculate the length of the addendum.

addendum = coefficient addendum / diametral pitch (inches)

addendum = coefficient addendum * module (millimeters)

Valid values are greater than 0 and less than 2.

Coefficient Clearance

Coefficient Clearance defines the constant that is used to calculate the length of the addendum.

dedendum = (coefficient addendum + coefficient clearance) / diametral pitch (inches)

dedendum = (coefficient addendum + coefficient clearance) * module (millimeters)

Valid values are greater than or equal to 0 and less than or equal to 1.

Coefficient Clearance Additional

Coefficient Clearance Additional is a linear dimension that defines the amount of additional length to be added to the

dedendum. This value does not take into consideration the units (inches or millimeters) of the component.

Coefficient Fillet

Coefficient Fillet defines the constant that is used to calculate the length of the addendum.

The hob tip fillet radius = coefficient fillet / diametral pitch (inches)

The hob tip fillet radius = coefficient fillet * module (millimeters)

Valid values are greater than 0 and less than or equal to 1.

Pressure Angle Normal

Pressure Angle Normal defines the pressure angle for the standard.

Backup All Company Standards

Backup All Company Standards allows the user to backup the company standards in a location that will not be destroyed when removing or installing a new version of GearTrax. This can also be used to store the standards so that other users can have access to them.

Restore or Import Company Standards

Restore or Import Company Standards allows the user to restore company standards after a new installation of GearTrax or to import the company standards from another location.

Open install folder using Explorer

GearTrax is compiled and published as a "Click-once" application. Therefore, the installation folder is not in the program folder as it is for most programs. The installation folder can be opened with this command.

Help

GearTrax Help Topics

Clicking 'GearTrax Help Topics' opens the GearTrax help topics.

GearTrax Manual

Clicking 'GearTrax Manual' opens a PDF of the manual. Adobe Acrobat must be installed on the computer.

Registration...

Clicking 'Registration...' opens the Registration window.

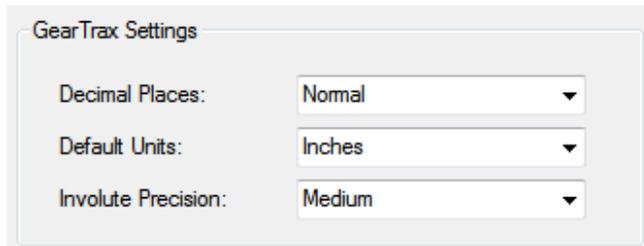
About GearTrax...

Clicking 'About GearTrax...' opens the About window. Camnetics addresses are listed along with a link to the web site. The version of GearTrax that is installed is displayed.

Options

The Options settings are available on the GearTrax Options tab.

GearTrax Settings



The screenshot shows a dialog box titled "GearTrax Settings". It contains three rows of settings, each with a label and a dropdown menu:

- Decimal Places: Normal
- Default Units: Inches
- Involute Precision: Medium

Decimal Places

Decimal Places determines the number of decimal places from normal. The normal setting is x.xxxx for inches and x.xxx for millimeters. This option will not change the standard or add one or two decimal places to most of the values displayed or presented in any data sheets. This may not affect some settings.

Default Units

Default Units defines the units (inches or millimeters) to be used when creating a new GearTrax component. This value is saved as a default for the next time GearTrax is started.

Involute Precision

Involute Precision allows the user to have some control over the number of points used to create the spline that defines the involute portion of the tooth. The number of spline points will be affected by the number of teeth in the gear and any addendum modification. The maximum number of points used to create the involute per side is:

- Low: 10
- Medium: 20
- High: 40
- Very High: 80

Bevel Gears

Bevel Gears

Tooth Cut Loft Factor Inside: 1.20

Tooth Cut Loft Factor Outside: 1.06

Tooth Cut Loft Factor, Inside

Tooth Cut Loft Factor, Inside defines how much longer the loft should be to facilitate the complete lofting of the bevel tooth cut toward the apex point. Increasing this value for wide face widths may help assure that the tooth cut is long enough.

Tooth Cut Loft Factor, Outside

Tooth Cut Loft Factor, Outside defines how much longer the loft should be to facilitate the complete lofting of the bevel tooth cut outward from the pitch diameter. Increase this value if changes to the base revolve might increase the blank diameter of the gear.

CAD Settings

CAD Settings

CAD Version: Last_Session

Constrain Involute Sketch

Assembly Template File:

Part Template File:

Tooth Creation: With_Splines

CAD Version

CAD Version defines which version of CAD to use with GearTrax when it is running as an Add-On. This option is not available when GearTrax is running as an Add-In. This value is saved as a default for the next time GearTrax is started.

Constrain Involute Sketch

The Constrain Involute Sketch check box determines if the involute sketch will be created with an anchor (or fixed) constraint. Setting this to checked will prevent unintended dragging of the sketch entities with the mouse. This value is saved as a default for the next time GearTrax is started.

Template File for Assembly

Template File for Assembly displays the template file to be used when creating a new assembly document in CAD. This is a read-

only value. Set the location and name of the template file in the GearTrax menu item CAD>Assembly Template File... This value is saved as a default for the next time GearTrax is started.

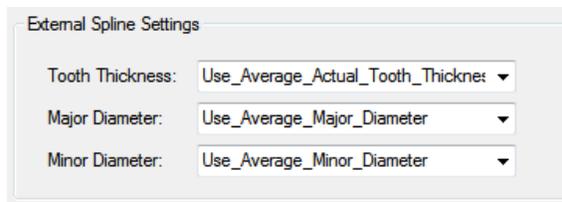
Template File for Component

Template File for Component displays the template file to be used when creating a new part document in CAD. This is a read-only value. Set the location and name of the template file in the GearTrax menu item CAD>Component Template File... This value is saved as a default for the next time GearTrax is started.

Tooth Creation

The Tooth Creation option allows the user to select between splines and arcs when creating the tooth sketch geometry. This value is saved as a default for the next time GearTrax is started.

External Spline Settings



Tooth Thickness

The Tooth Thickness combo box gives the user a method to determine how to create the tooth thickness of external splines when creating a CAD model. This option is saved as a default for the next time GearTrax is started. The options are:

- Use Average Actual Tooth Thickness
- Use Maximum Actual Tooth Thickness
- Use Minimum Actual Tooth Thickness

Major Diameter

The Major Diameter combo box gives the user a method to determine how to create the major diameter of external splines when creating a CAD model. This option is saved as a default for the next time GearTrax is started. The options are:

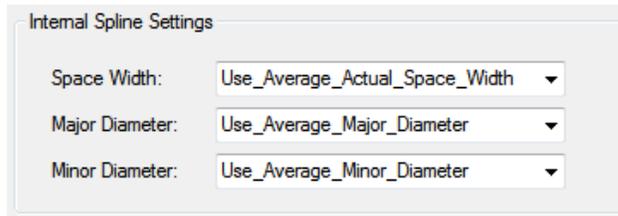
- Use Average Major Diameter
- Use Maximum Major Diameter
- Use Minimum Major Diameter

Minor Diameter

The Minor Diameter combo box gives the user a method to determine how to create the minor diameter of external splines when creating a CAD model. This option is saved as a default for the next time GearTrax is started. The options are:

- Use Average Minor Diameter
- Use Maximum Minor Diameter
- Use Minimum Minor Diameter

Internal Spline Settings



Internal Spline Settings

| | | |
|-----------------|--------------------------------|---|
| Space Width: | Use_Average_Actual_Space_Width | ▼ |
| Major Diameter: | Use_Average_Major_Diameter | ▼ |
| Minor Diameter: | Use_Average_Minor_Diameter | ▼ |

Space Width

The Space Width combo box gives the user a method to determine how to create the tooth thickness of external splines when creating a CAD model. This option is saved as a default for the next time GearTrax is started. The options are:

- Use Average Actual Space Width
- Use Maximum Actual Space Width
- Use Minimum Actual Space Width

Major Diameter

The Major Diameter combo box gives the user a method to determine how to create the major diameter of internal splines when creating a CAD model. This option is saved as a default for the next time GearTrax is started. The options are:

- Use Average Major Diameter
- Use Maximum Major Diameter
- Use Minimum Major Diameter

Minor Diameter

The Minor Diameter combo box gives the user a method to determine how to create the minor diameter of internal splines when creating a CAD model. This option is saved as a default for the next time GearTrax is started. The options are:

- Use Average Minor Diameter
- Use Maximum Minor Diameter
- Use Minimum Minor Diameter

Spur/Helical

Addendum

Addendum is the length of the tooth from the pitch diameter to the major diameter. This value cannot be changed directly unless GearTrax is in the “Free Form” mode.

Addendum Modification

Addendum Modification is the amount of addendum change of a modified tooth. This value can be positive or negative. If the sum of change to a pair of gears equals zero then there is no change to the center distance.

A positive value will increase the addendum length and a negative value will decrease the addendum length.

If GearTrax is in the “Free Form” mode, the value will be unused.

Addendum Modification Coefficient

Addendum Modification Coefficient is the ratio of change of a modified tooth. This value can be positive or negative. If the sum of change to a pair of gears equals zero then there is no change to the center distance.

A positive value will increase the addendum length and a negative value will decrease the addendum length.

If GearTrax is in the “Free Form” mode, the value will be unused.

AGMA Class

AGMA Class sets the AGMA Class for this component.

Backlash

Backlash is the thinning (or thickening, if a negative value) of the tooth profile after any modification to the tooth form. This backlash is achieved by rack shift and does not affect the diameters. See Addendum Modification for tooth thinning or thickening that also affects the diameters.

Blank O.D.

Blank O.D. is the diameter of the blank used to create the internal gear. The diameter must be large enough for the tooth cut in CAD.

Circular Pitch

Circular Pitch is the arc length of a single pitch at the pitch diameter.

Coefficient, Addendum

Addendum Coefficient is a constant that is divided by the diametral pitch to determine the length of the addendum before any modification. The addendum length of a 10 diametral-pitch gear with an addendum coefficient of 0.8 is 0.080in.

The whole depth is equal to the sum of twice the addendum coefficient and the clearance coefficient divided by the diametral pitch.

Coefficient, Clearance

Clearance Coefficient is a constant that is divided by the diametral pitch to determine the clearance in the root of the gear tooth.

The whole depth is equal to the sum of the clearance coefficient and twice the addendum coefficient divided by the diametral pitch.

Coefficient, Fillet

Fillet Coefficient is a constant that is divided by the diametral pitch to determine the radius of the hob tip that creates gear tooth root.

Contact Length, Approach

Contact Length, Approach and the recess length equal the total contact for a pair of gear teeth. To view these read-only values, click on the small button next to the Addendum Mod text boxes.

Contact Length, Recess

Contact Length, Recess and the approach length equal the total contact for a pair of gear teeth. To view these read-only values, click on the small button next to the Addendum Mod text boxes

Contact Ratio

Contact Ratio is the ratio of the arc of action to the circular pitch. This value should be over 1.4 to assure a smooth transfer of load from one pair of teeth to the next pair of teeth. To view these read-only values, click on the small button next to the Addendum Mod text boxes

Crowning

Crowning is not available in GearTrax but it is available in our GearTeq product. Crowning is the alteration to the tooth thickness along the length of the face width. This can also be referred to as lead or longitudinal crowning.

Dedendum

Dedendum is the radial length of the tooth between the pitch diameter and the minor diameter.

Base Diameter

Base Diameter is a diameter that is tangent to the pressure angle. The involute curve cannot be within this diameter.

Major Diameter

Major Diameter is the outside diameter of a gear.

Minor Diameter

Minor Diameter is the root diameter of a gear.

Pitch Diameter

Pitch Diameter is the theoretical diameter of the gear. On a face gear, this value defines the inner diameter. The outside diameter of a face gear is the pitch diameter plus twice the face width.

Pitch Diameter, Operating

Pitch Diameter, Operating is a theoretical diameter at which a set of gears meshes. It normally equals the pitch diameter except when either of the gears is modified or the center distance has been modified. To view these read-only values, click on the small button next to the Pitch Diameter text boxes.

True Involute Form Diameter

The true involute form (TIF) diameter is the smallest diameter of the involute curve. To view these read-only values, click on the small button next to the Minor Diameter text boxes.

Gear Type

Gear Type defines the component as an external gear, internal gear or rack. A face gear option is available in our GearTeq product.

Face Gears

Face gears are not available in GearTrax but is available in our GearTeq product. Face gears are more like a circular rack than an internal or external spur gear.

- The shaft angle is always at 90 degrees.
- The pinion and the face gear axis are always coincident.
- The pitch diameter of a face gear is at the internal diameter of the teeth.
- The outside diameter of a face gear is equal to the pitch diameter plus twice the face width.
- Backlash is always 0.000 and cannot be changed.
- The addendum modification is always 0.000 and cannot be changed.
- The Pitch Depth of Rack property controls the distance from the pitch line to the back of the face gear.

- Lead or longitude crowning is not available for face gears. Add any crowning to the pinion.
- Face gears are sometimes call crown gears because they look like a king's crown. But this should not be confused with gears that have their teeth "crowned".

Face Width

Face Width is the length of the tooth parallel to the shaft.

Fellows Stub Denominator

Fellows Stub standard is not available in GearTrax but is available in our GearTeq product.

The Fellows Stub Denominator sets the denominator for the Fellows Stub standard. The Fellows Stub standard must be selected to change this value. This value must be equal to or less than the diametral pitch

The user may specify any combination of nominator/denominator for the Follows Stub standard as long as the denominator is a value greater than the nominator. The standard ratios established by the Fellows Gear Shaper Co. are 4/5, 5/7, 6/8, 7/9, 8/10, 9/11, 10/12 and 12/14.

Fillet Radius

Fillet Radius defines the radius on the tip of the cutter, which forms a trochoidal curve tangent to the tooth root.

Gear Standard

Gear Standard sets the AGMA, DIN, JIS, PGT or other standards for the component. Click the small button to the right of the drop down box to change tolerance specifications.

Helix Angle

Helix Angle is the angle of the tooth from the shaft. A zero angle would be parallel to the shaft and would define a spur gear. An angle other than zero would define a helical gear.

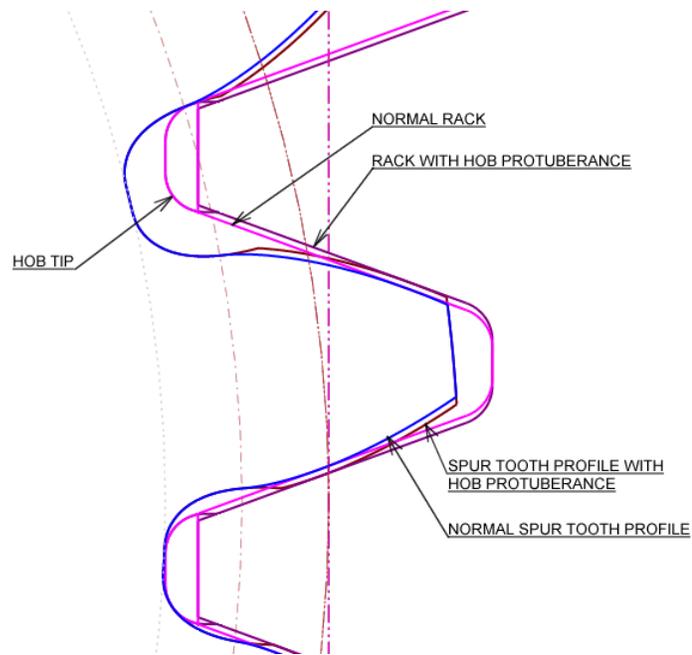
Helix Direction

Helix Direction defines the direction of the angle. The teeth on a left hand helical gear on a horizontal surface lean to the left.

Hob Protuberance

Hob Protuberance is not available in GearTrax but is available in our GearTeq product.

Hob Protuberance is the amount of undercutting of the involute surface created by the hob. The fillet remains unchanged except for a slight increase in length.



HPSTC (Highest Point of Single Tooth Contact)

Highest Point of Single Tooth Contact is the largest diameter on a spur gear at which a single tooth is in contact with the mating gear. To view these read-only values, click on the small button next to the Pitch Diameter text boxes.

LPSTC (Lowest Point of Single Tooth Contact)

Lowest Point of Single Tooth Contact is the smallest diameter on a spur gear at which a single tooth is in contact with the mating gear. To view these read-only values, click on the small button next to the Pitch Diameter text boxes.

Full Fillet Radius

Some standards allow for a full fillet radius option. If Full Fillet Radius is unchecked then the standard hobbing manufacturing method will be used to generate the tooth form. Standards that do not allow this option will ignore this check box.

Modification Profile

Modification Profile is not available in GearTrax but is available in our GearTeq product.

Module

Module (transverse) is used in metric system gears. Module equals the normal module divided by the cosine of the helical angle.

Module Normal

Module Normal is used in metric system gears and is normal to the cutter.
Module = 25.4 / diametral pitch.

Number of Teeth

Number of Teeth defines the number of teeth for each component.

Pin Diameter

Pin Diameter defines the diameter of the pins or wires used with Measurement of Pins. A user defined pin diameter may be entered. To reset the value to the standard value click the Reset button in the Spur Inspection window.

Pitch Depth of Rack

Pitch Depth of Rack is the distance from the pitch line to the back of the rack. This value is also used to define the depth of a face gear from the pitch line to the back of the gear.

Diametral Pitch

Diametral Pitch is used in imperial system gears. This defines the diametral pitch in the transverse plane.

Diametral Pitch, Normal

Diametral Pitch, Normal is used in imperial system gears and is normal to the cutter.

Pressure Angle

Pressure Angle is the angle of the tooth at the pitch diameter. This is sometimes referred to as the transverse pressure angle.

Pressure Angle, Normal

Pressure Angle, Normal is the angle of the tooth at the pitch diameter normal to the cutter.

Single Tooth Contact

Single tooth contact. To view these read-only values, click on the small button next to the Pitch Diameter text boxes.

Start of Active Profile

Start of Active Profile is the diameter at which a gear comes in to contact with its mating gear. To view these read-only values, click on the small button next to the Pitch Diameter text boxes.

Shrinkage Rate

Shrinkage Rate is not available in GearTrax but is available in our GearTeq product.

Shrinkage Rate defines the shrinkage rate for plastic gears. This is only used when inserting a tooth cut profile. The value entered must be less than 0.0100 (1%), 0.0025 = 0.25%

Tooth Pattern in CAD

Tooth Pattern in CAD selects the number of teeth to pattern when creating the model in CAD. The options are All, None, or First 10. If this is set to All, GearTrax will create a pattern that contains all the teeth. This might be very time consuming if there are a large number of teeth and/or the component is a helical gear. If this is set to none, then a pattern will not be created. Set this to 'First 10' to create a pattern with 10 teeth that can easily be expanded in the CAD system.

Test Master Gear Pitch Diameter

Enter the pitch diameter of the master test gear, if available. By itself, this value has no effect on the actual geometry of the gear. To enter or view this value click on the inspection button (picture of a caliper).

Test Radius

The test radius of the master gear plus the test radius of this gear equals the setup center distance for a composite action test. If a test radius for the master gear has been entered, GearTrax will calculate the test radius of this gear.

The test radius can also be used to calculate the tooth thickness of a gear. If a test diameter for the master has been entered and a test radius for this gear is then entered, GearTrax will calculate the addendum modification coefficient for this gear to achieve the proper tooth thickness in conjunction with the perfect master gear. The user will be prompted to use this calculated value or not. To enter or view this value click on the inspection button (picture of a caliper).

Tooth Thickness (Normal)

Tooth Thickness is the arc thickness of the tooth at the pitch diameter normal to the cutter.

Topping Adjustment

Topping Adjustment will shorten the length of the addendum by this value and the major diameter by twice this value. This value must be equal to or greater than zero and less than the unadjusted addendum. To view and edit these values, click on the small button next to the Major Diameter text boxes.

Units

Units sets the measurement units for the component; select Inches or Metric.

Whole Depth

Whole Depth is the depth of the tooth (from the major diameter to the minor diameter).

The whole depth is equal to the sum of the clearance coefficient and twice the addendum coefficient divided by the diametral pitch.

Spur Tolerances ANSI/AGMA 2000-A88

Open the Tolerances window by clicking on the small button to the right of the Gear Standards drop down selection box.

Tolerances ANSI/AGMA

AGMA_Q7

Pinion
 Gear

ANSI/AGMA 2015-1-A01

Cumu. Pitch Deviation Tol.: n/a

Diameter Tolerance: n/a

Helix Form Tolerance: n/a

Helix Slope Tolerance: n/a

Helix Tolerance Total: n/a

Profile Form Tolerance: n/a

Profile Slope Tolerance: n/a

Profile Tolerance Total: n/a

Single Flank Comp. Tol. T2T: n/a

Single Flank Comp. Tol. Total: n/a

Single Flank Deviation Tol.: n/a

ANSI/AGMA 2000-A88

Code Tooth Thickness: A

Radial Runout: 0.00310in

Pitch Variation: 0.00099in

Profile: 0.00120in

Tooth Alignment: 0.00000in

Total Composite: 0.00540in

Tooth to Tooth Composite: 0.00220in

Tolerance Tooth Thickness: 0.15708in

Code Tooth Thickness

A user selectable between A, B, C and D.

Radial Runout

Please refer to ANSI/AGMA 2000-A88 standard for a detailed description.

Pitch Variation

Please refer to ANSI/AGMA 2000-A88 standard for a detailed description.

Profile

Profile Tolerance is the permissible amount of profile variation in the functional profile, designated by a specified “K” chart envelope. Plus material at the tip, which increases the amount of variation outside the functional profile, is not acceptable. Minus material beyond the start of tip break can be disregarded.

Tooth Alignment

Tooth Alignment Tolerance is the permissible amount of tooth alignment variation, designated by the specified “K” chart envelope. Tolerance values in this standard are normal to the tooth surface.

Total Composite

Total Composite Tolerance is the permissible amount of total composite variation, which is the total change in center distance while the gear being tested is rotated one complete revolution during double flank composite action test.

Tooth to Tooth Composite

Please refer to ANSI/AGMA 2000-A88 standard for a detailed description.

Tooth Thickness

Tooth Thickness Tolerance is the permissible amount of tooth thickness variation.

ANSI/AGMA 2015-1-A01

Open the Tolerances window by clicking on the small button to the right of the Gear Standards drop down selection box.

Cumulative Pitch Deviation Tolerance

Please refer to ANSI/AGMA 2015-1-A01 standard for a detailed description.

Diameter Tolerance

Please refer to ANSI/AGMA 2015-1-A01 standard for a detailed description.

Helix Form Tolerance

Please refer to ANSI/AGMA 2015-1-A01 standard for a detailed description.

Helix Slope Tolerance

Please refer to ANSI/AGMA 2015-1-A01 standard for a detailed description.

Helix Tolerance Total

Please refer to ANSI/AGMA 2015-1-A01 standard for a detailed description.

Profile Form Tolerance

Please refer to ANSI/AGMA 2015-1-A01 standard for a detailed description.

Profile Slope Tolerance

Please refer to ANSI/AGMA 2015-1-A01 standard for a detailed description.

Profile Tolerance Total

Please refer to ANSI/AGMA 2015-1-A01 standard for a detailed description.

Single Flank Composite Tolerance Tooth to Tooth

Please refer to ANSI/AGMA 2015-1-A01 standard for a detailed description.

Single Flank Composite Tolerance Total

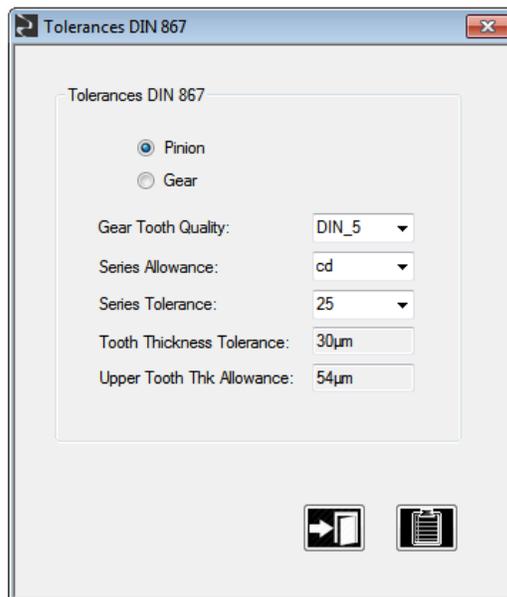
Please refer to ANSI/AGMA 2015-1-A01 standard for a detailed description.

Single Flank Deviation Tolerance

Please refer to ANSI/AGMA 2015-1-A01 standard for a detailed description.

DIN 867

Open the Tolerances window by clicking on the small button to the right of the Gear Standards drop down selection box.

**Gear Tooth Quality**

User selectable between DIN 1 and DIN 12. Please refer to DIN 867 standard for a detailed description.

Series Allowance

User selectable a, ab, b, bc, c, cd, d, e, f, g and h series allowance.
Please refer to DIN 867 standard for a detailed description

Series Tolerance

User selectable between 21 and 30. Please refer to DIN 867 standard for a detailed description.

Tooth Thickness Tolerance

Please refer to DIN 867 standard for a detailed description.

Upper Tooth Thickness Allowance

Please refer to DIN 867 standard for a detailed description.

Spur Inspection**Measurement Over or Between Pins**

| Measurement Over or Between Pins | | |
|----------------------------------|--|--|
| | PINION | GEAR |
| Pin Diameter: | <input type="text" value="4.50000mm"/> | <input type="text" value="4.50000mm"/> |
| Over Pins: | <input type="text" value="52.0278mm"/> | <input type="text" value="77.6208mm"/> |
| Over Pins Minimum: | <input type="text" value="51.9616mm"/> | <input type="text" value="77.6208mm"/> |

Pin Diameter

Pin Diameter defines the diameter of the pin or ball (helical gears).

Over Pins

Measurement Over/Under Pins is the measurement over two pins diametrically opposed in a gear and is normally used for inspection. The measurement is over pins for external gears and under pins for internal gears. GearTrax will calculate the tooth thickness for an external gear and the space width for internal gears if a value is entered for the over/under pin measurement.

Over Pins Minimum

Over Pins Minimum is the measurement over two pins diametrically opposed in a gear and is normally used for inspection. Measures the tooth thickness minus the total tooth thickness tolerance.

Span Measurement

| Span Measurements | | | |
|----------------------|---------------------------------------|---------------------------------------|---|
| Teeth to Gage Over: | <input type="text" value="2"/> | <input type="text" value="4"/> |  |
| Measurement: | <input type="text" value="11.837mm"/> | <input type="text" value="27.241mm"/> | |
| Measurement Minimum: | <input type="text" value="11.809mm"/> | <input type="text" value="27.241mm"/> | |

Teeth to Gage Over

Span Measurement, Teeth to Gage Over define the number of teeth that are used in conjunction with Span Measurement.

Measurement

Span Measurement is a straight-line measurement across the Number of Teeth to Gage Over. The measurement line is normally tangent to the base line. This value is read-only.

Measurement Minimum

Span Measurement Minimum is the minimum straight-line measurement across the Number of Teeth to Gage Over using the tooth thickness minimum value. This value is read-only.

Chordal Measurements

| Chordal Measurements | | |
|----------------------|---------------------------------------|---------------------------------------|
| Reference Circle: | <input type="text" value="42.969mm"/> | <input type="text" value="67.492mm"/> |
| Tooth Height: | <input type="text" value="4.006mm"/> | <input type="text" value="4.413mm"/> |
| Thickness: | <input type="text" value="4.3320mm"/> | <input type="text" value="4.7260mm"/> |
| Thickness Minimum: | <input type="text" value="4.3039mm"/> | <input type="text" value="4.7260mm"/> |

Chordal Tooth Height

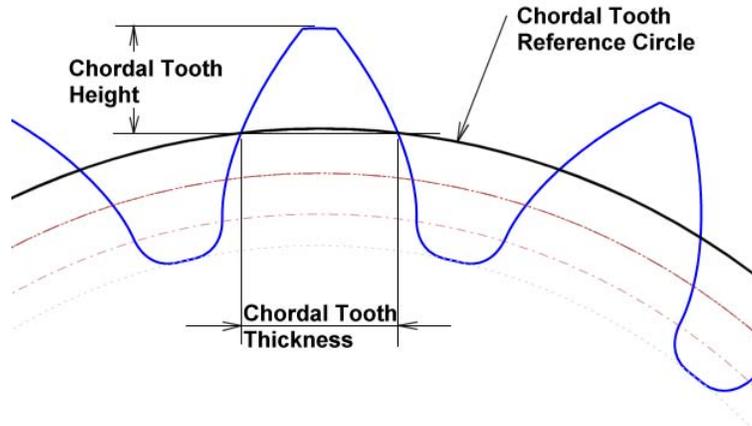
Chordal Tooth Height is the length of the tooth portion to be measured at which the chordal tooth thickness is calculated. Twice the chordal tooth height plus the chordal tooth reference circle should equal the outside diameter of the gear.

Chordal Tooth Reference Circle

Chordal Tooth Reference Circle is the diameter of the datum circle at which the chordal tooth thickness is measured. The chordal tooth reference circle should be equal to the outside diameter of the gear minus twice the chordal tooth height.

Chordal Tooth Thickness

Chordal Tooth Thickness is a straight-line measurement across a single tooth at the chordal tooth reference circle.



Chordal Tooth Thickness Minimum

Chordal Tooth Thickness is a straight-line measurement across a single tooth (maximum actual tooth thickness minus tooth thickness tolerance) at the chordal tooth reference circle.

Master Gear Test

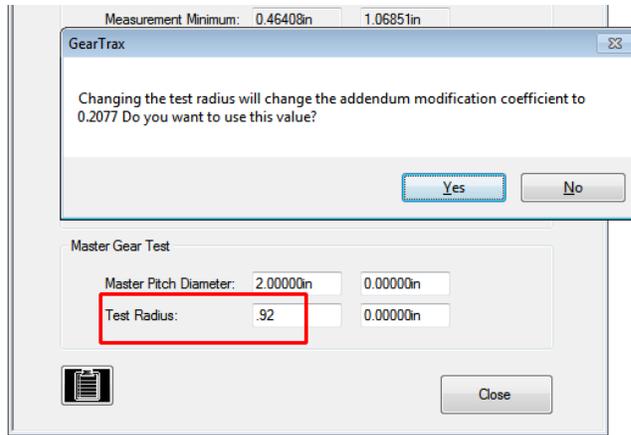
| Master Gear Test | | |
|------------------------|---|---|
| Master Pitch Diameter: | <input type="text" value="50.80000mm"/> | <input type="text" value="50.80000mm"/> |
| Test Radius: | <input type="text" value="22.78538mm"/> | <input type="text" value="35.56000mm"/> |

Master Pitch Diameter

After entering the master pitch diameter, the test radius will be calculated for the pinion or gear. For example, if the gears are 10 Diametral Pitch (2.54 Module) and 50.0mm is entered, GearTrax will calculate the nearest master gear size with a whole number of teeth.

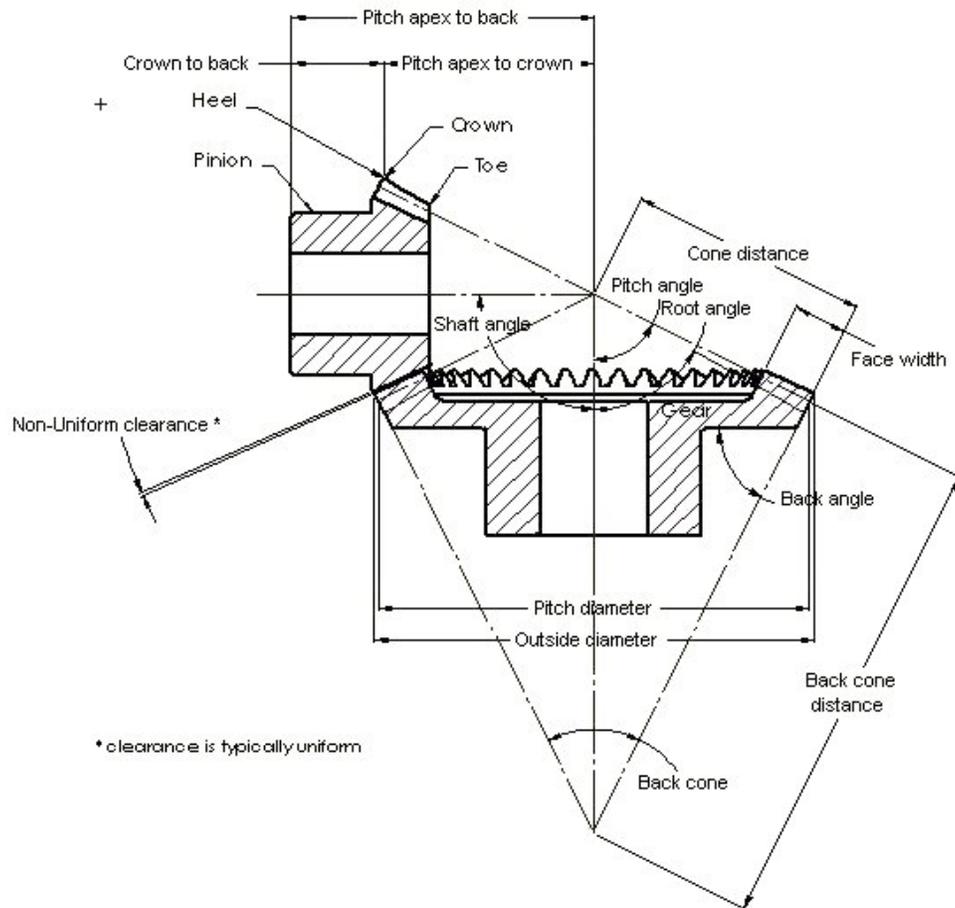
Test Radius

Enter a value for the test radius. If the master pitch diameter does not equal 0, GearTrax will calculate the change to the addendum modification and display a message box with that change and allow the user to make that change to the gear.



Master Gear Test

Bevel Gears



Addendum

Addendum is the length of the tooth from the pitch diameter to the major diameter.

Addendum Mod. (Addendum Modification)

Addendum Modification is the amount of addendum change of a modified tooth. This value can be positive or negative. The user value can only adjust this value in the Non standard mode. In the Gleason and DIN mode it is automatically calculated. In the Free Form mode it is not used.

Add. Mod. Coef. (Addendum Modification Coefficient)

Addendum Modification Coefficient is the ratio of change of a modified tooth. This value can be positive or negative. The user value can only adjust this value in the Non standard mode. In the Gleason and DIN mode it is automatically calculated. In the Free Form mode it is not used.

AGMA Class

AGMA Class sets the AGMA Class for this component.

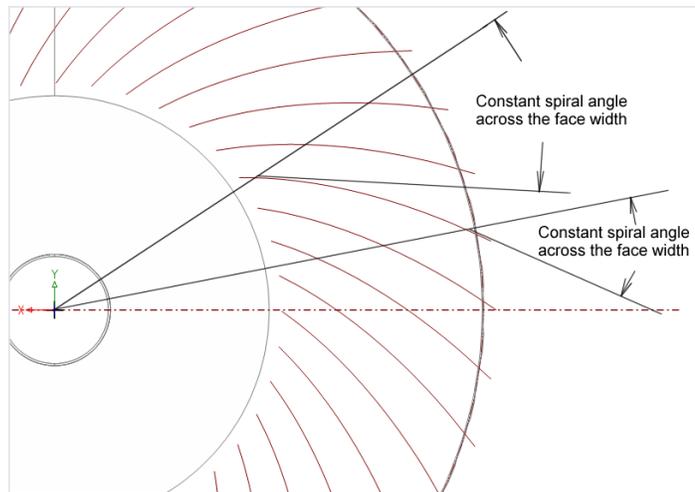
Backlash

Backlash is the thinning (or thickening, if a negative value) of the tooth profile after any modification to the tooth form.

Bevel Type

Camnetics TruSpiral

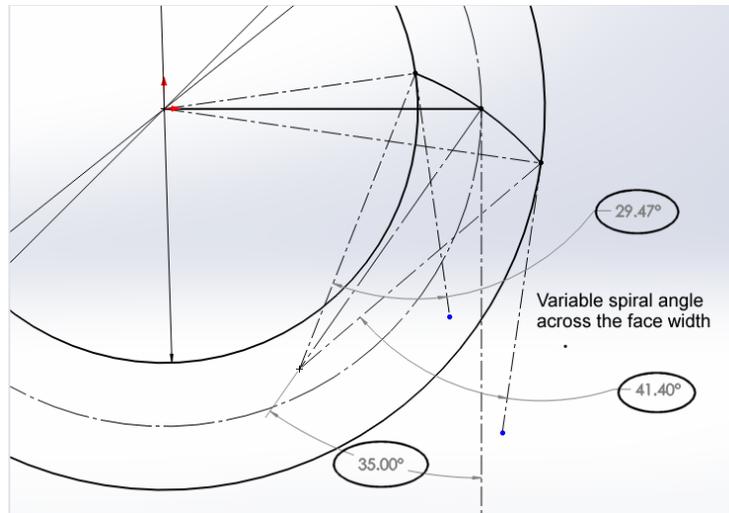
The Camnetics TruSpiral is an equiangular spiral on the pitch cone through the length of the tooth. As a set, the models are very accurate.



Straight

The straight tooth has no curvature over the length of its tooth.

Spiral



The spiral tooth is defined by the cutter diameter whose center is at the specified spiral angle from the tooth midpoint. This is the standard method that gear machines use. Use extreme caution using a gear created with this program and one created on a gear machine. It is unknown at this time how close the geometry matches.

Zerol

The Zerol tooth is similar to the spiral except the spiral angle is always zero.

Circular Pitch

Circular Pitch is the arc length of a single pitch at the pitch diameter.

Coefficient, Addendum

Addendum Coefficient is a constant that is divided by the diametral pitch to determine the length of the addendum before any modification. The addendum length of a 10 diametral-pitch gear with an addendum coefficient of 0.8 is 0.080in.

The whole depth is equal to the sum of twice the addendum coefficient and the clearance coefficient divided by the diametral pitch.

Coefficient, Clearance

Clearance Coefficient is a constant that is divided by the diametral pitch to determine the clearance in the root of the gear tooth.

The whole depth is equal to the sum of the clearance coefficient and twice the addendum coefficient divided by the diametral pitch.

Coefficient, Fillet

Fillet Coefficient is a constant that is divided by the diametral pitch to determine the radius of the hob tip that creates the gear tooth root.

Crown to Back

Crown to Back defines the distance from the crown to the back of the gear including any hub projection.

Crown to Apex

Crown to Apex defines the distance from the crown to the gear apex. This is a read-only value.

Dedendum

Dedendum is the radial length of the tooth between the pitch diameter and the minor diameter.

Diametral Pitch

Diametral Pitch is used in imperial system gears

Face Angle

The Face Angle of a blank is made parallel to the root cone of the mating gear.

Face to Back

Face to Back is a read only value that defines the overall length of the gear along the axis.

Face Width

Face Width defines the length of the tooth along the pitch cone.

Fillet Radius

Fillet Radius defines the radius on the tip of the cutter, which forms a trochoidal curve tangent to the tooth root.

K Factor

Circular Thickness Factors K for Gleason system spiral bevel gears. This value is automatically generated for Gleason straight bevel gear sets. The user must specify this value for spiral bevel gears, see the chart below:

Equations for Chordal Thickness of Gleason Spiral Bevel Gears

| No. | Item | Symbol | Formula | Example |
|-----|---------------------------------|----------------|--|---|
| 1 | Circular Tooth Thickness Factor | K | Obtain from chart | $\Sigma = 90^\circ$ $m = 3$ $\alpha_n = 20^\circ$ $z_1 = 20$ $z_2 = 40$ $\beta_m = 35^\circ$ |
| 2 | Circular Tooth Thickness | s_1 s_2 | $p - s_2$ $\frac{p}{2} - (h_{a1} - h_{a2}) \frac{\tan \alpha_n}{\cos \beta_m} - Km$ | $h_{a1} = 3.4275$ $K = 0.060$ $p = 9.4248$ $s_1 = 5.6722$ $s_2 = 3.7526$ |

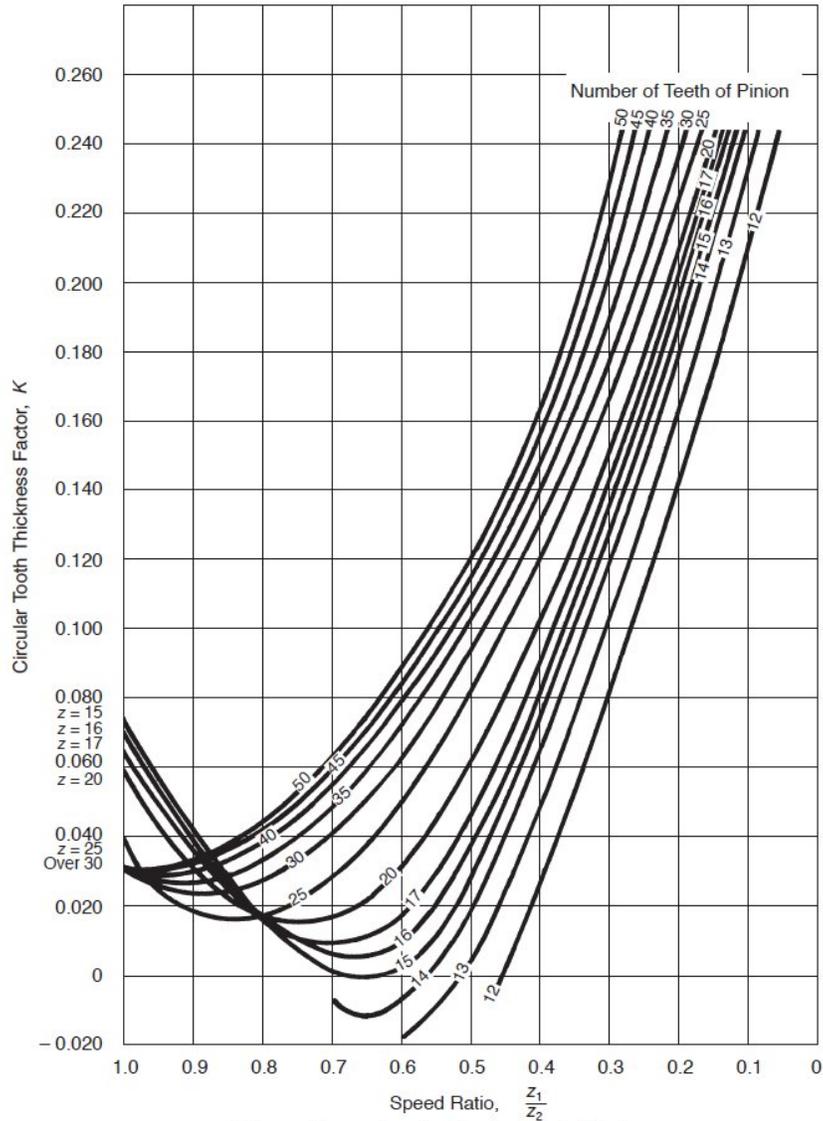


Chart to Determine the Circular Tooth Thickness Factor K for Gleason Spiral Bevel Gears

Manufacturing Method

Manufacturing Method defines the manufacturing standard for the gear. Select Gleason, DIN 3971, Non-standard or Free-Form.

Selecting Non-standard allows the user control over these parameters:

- Coefficients of addendum, clearance and fillet radius
- Pressure angle

The Free-Form standard allows the user control over these parameters:

- Pressure angle
- Fillet radius
- Face angle
- Pitch angle
- Addendum
- Dedendum

Module

Module is used in metric system gears. $\text{Module} = 25.4 / \text{Diametral Pitch}$

Mounting Distance

Mounting Distance defines the distance from the apex to the end of the bevel gear. This dimension will include the hub projection.

Number of Teeth

Number of Teeth defines the number of teeth for the component.

Outside Diameter

Outside Diameter displays the outside diameter of the bevel gear. This value cannot be changed directly and is a result of the pitch diameter, addendum and back angle.

Pitch Angle

The Pitch Angle is from the apex point to a point on the pitch diameter measured from the shaft.

Pitch angles, sum

Pitch Angles, sum defines the shaft angle for the bevel gear set.

Pitch Diameter

Pitch Diameter is a theoretical diameter of the gear.

Pressure Angle

Pressure Angle is the angle of the tooth at the pitch diameter.

Root Angle

The Root Angle is from the apex point to the bottom of the dedendum.

Spiral Angle

Spiral Angle defines the angle of the spiral. Specify 0 degrees for a straight bevel gear. Normally 35 degrees is specified for spiral bevel gears.

Spiral Direction

Spiral Direction defines the direction of the spiral for spiral bevel gears. Select right hand or left hand.

Sum of Pitch Angles

Sum of Pitch Angles defines the sum of the pitch angles for the set of bevel gears.

Tooth Thickness

Tooth Thickness is the arc thickness of the tooth at the pitch diameter.

Units

Units sets the measurement units for the component; select Inches or Metric for millimeters.

Web Thickness

Web Thickness defines the thickness between the face of the bevel gear and the back face, not including the hub projection. For a miter type gear, set this value to 0.0.

Sprockets**Chain Number**

Chain Number defines the ANSI, ISO or DIN specification number for the sprocket.

Chain Pitch

Chain Pitch defines the linear dimension of a single chain pitch. This is a read only value unless the Special ASA standard is selected.

Chain Series

Chain Series defines if the sprocket is standard series or heavy series.

Double Pitch Single Duty

Double Pitch Single Duty defines if the component is double pitch, single duty. Select True or False.

Number of Strands

Number of Strands defines the number of strands to be created.

Number of Teeth

Number of Teeth defines the number of teeth for the component.

Path

Path defines the name of the folder where the component files are stored.

Pitch Diameter

Pitch Diameter is a read only value that defines the value of the pitch diameter. The pitch diameter of chain sprockets equals the chain pitch divided by the sine of PI divided by the number of teeth.

Pointed Tooth

Pointed Tooth defines if the sprocket should be created with a pointed tooth. Select True or False.

Roller Diameter

Roller Diameter defines the diameter of the roller. This is a read only value unless the Special ASA standard is selected.

Sprocket Chamfer Depth

Sprocket Chamfer Depth defines the depth of the chamfer at the tooth tip. This is a read only value unless the Special ASA standard is selected.

Sprocket Outside Diameter

Sprocket Outside Diameter displays the outside diameter of the sprocket. This is a read only value.

Sprocket Chamfer Width

Sprocket Chamfer Width defines the width of the chamfer at the tooth tip. This is a read only value unless the Special ASA standard is selected.

Sprocket Width

Sprocket Width defines the width of the sprocket. In the case of a multi-strand sprocket, it defines the width of each plate. This is a read only value unless the Special ASA standard is selected.

Teeth to Pattern

Teeth to Pattern defines the number of teeth to pattern when creating the model in CAD. The options are All, None, or First 10. If this is set to All, GearTrax will create a pattern that contains all the teeth. This might be very time consuming if there are a large number of teeth and/or the component is a helical gear. If this is set to none, then a pattern will not be created. Set this to 'First 10' to create a pattern with 10 teeth that can easily be expanded in the CAD system.

Units

Units sets the measurement units for the component; select Inches or Metric.

Gear Belt Pulleys**Belt Pitch**

Belt Pitch defines the belt series and pitch.

Belt Width

Belt Width defines the width of belt to be used with this component.

Flange Creation

Flange Creation defines the type of flange to be created, if any.

Flange ID

Flange ID defines the inside diameter of the flange. This is a read only value and cannot be changed. This is a reference dimension; different manufacturers may use other values.

Flange OD

Flange OD defines the outside diameter of the flange. This is a read only value and cannot be changed. This is a reference dimension; different manufacturers may use other values.

Flange Thickness

Flange Thickness defines the thickness of the flange. This is a read only value and cannot be changed. This is a reference dimension; different manufacturers may use other values.

Number of Teeth

Number of Teeth defines the number of teeth for the component.

Outside Diameter

Outside Diameter defines the diameter of the pulley body, not the flange diameter. This is a read only value and cannot be changed.

Pitch Diameter

Pitch Diameter is a read only value that defines the value of the pitch diameter. The pitch diameter of gear belt pulleys equals the belt pitch multiplied by the number of teeth divided by PI.

Pulley Width

Pulley Width defines the width of the pulley at the bore not including any hub projections.

Teeth to Pattern

Teeth to Pattern defines the number of teeth to pattern when creating the model in CAD. The options are All, None, or First 10. If this is set to All, GearTrax will create a pattern that contains all the teeth. This might be very time consuming if there are a large number of teeth and/or the component is a helical gear. If this is set to none, then a pattern will not be created. Set this to 'First 10' to create a pattern with 10 teeth that can easily be expanded in the CAD system.

Tooth Width

Tooth Width is the width of the pulley between the flanges. This is a read only value and cannot be changed.

Units

Units sets the measurement units for the component; select Inches or Metric.

Belt Pulleys

Belt Section

Belt Section defines the belt section for the component.

Deep Groove

Deep Groove defines if the component is to have a deep groove. Select True or False.

Number of Grooves

Number of Grooves defines the number of grooves for the component.

Pitch Diameter

Pitch Diameter defines the pitch diameter for the component.

Pulley OD

Pulley OD defines the outside diameter of the pulley. Changing this value will change the pitch diameter.

Units

Units sets the measurement units for the component, select Inches or Metric.

Worm Gears

Addendum, Gear

Addendum, Gear is the length of the gear tooth from the pitch diameter to the major diameter.

Addendum Modification

Addendum Modification is the amount of addendum change of a worm tooth. A positive value increases the addendum of the worm and decreases the addendum of the gear (wheel).

Addendum Modification Coefficient

Addendum Modification Coefficient is the ratio of change of a worm tooth. A positive value increases the addendum of the worm and decreases the addendum of the gear (wheel).

Addendum, Worm

Addendum, Worm is the length of the worm tooth from the pitch diameter to the major diameter.

Backlash, Gear

Backlash, Gear is the thinning (or thickening, if a negative value) of the gear tooth profile after any modification to the tooth form.

Backlash, Worm

Backlash, Worm is the thinning (or thickening, if a negative value) of the worm tooth profile after any modification to the tooth form.

Bore

Bore defines the diameter of the bore for the gear (wheel) component.

Bore, Chamfer

Bore, Chamfer defines the chamfer size for the bore of the gear (wheel) component.

Bore, Worm

Bore, Worm defines the diameter of the bore for the worm component.

Bore, Worm Chamfer

Bore, Worm Chamfer defines the chamfer size for the bore of the worm component.

Cavity cut tooth adjustment

The Cavity cut tooth adjustment gives the user a method to tweak how the "hobs" are places on the wheel for cutting. Depending on the number of teeth, starts and pitch diameter of the worm this value may need to be adjusted to properly create the cavity cut. Values between 0.5 and 2.0 are valid. Typically, the greater the number of teeth the higher the value. Recommended starting values are < 50 teeth, .75; < 100 teeth, 1.0; < 200 teeth, > 200 teeth, 2.0

Center Distance

Center Distance defines the distance between the centers of the worm and the wheel gear. This is a read only value and cannot be directly changed.

Coefficient, Addendum

Addendum Coefficient is a constant that is divided by the diametral pitch to determine the length of the addendum before any modification. The addendum length of a 10 diametral-pitch gear with an addendum coefficient of 0.8 is 0.080 in.

Coefficient, Clearance

Clearance Coefficient is a constant that is divided by the diametral pitch to determine the clearance in the root of the gear tooth.

Coefficient, Fillet

Fillet Coefficient is a constant that is divided by the diametral pitch to determine the radius of the hob tip that creates the gear tooth root.

Contact Ratio

Contact Ratio defines the contact ratio for the worm gear set.

Create Worm and or Wheel

Select between Both, Worm Only and Wheel Only.

Dedendum, Gear

Dedendum, Gear is the radial length of the gear tooth between the pitch diameter and the minor diameter.

Dedendum, Worm

Dedendum, Worm is the radial length of the worm tooth between the pitch diameter and the minor diameter.

Diameter, Pitch, Gear

Gear Pitch Diameter is a theoretical diameter of the gear.

Diameter, Pitch, Worm

Worm Pitch Diameter is a theoretical diameter of the worm.

Diametral Pitch

Diametral Pitch (transverse) is used for imperial system gears.

Diametral Pitch, Normal

Select between Profile I and Profile II designations for the DIN standard only. A Profile I has a clearance coefficient of 0.167 and Profile II has a clearance coefficient of 0.250

DIN 3972 Profile

Normal Diametral Pitch is used for imperial system gears. Normal diametral pitch = diametral pitch / cosine(lead angle).

Direction, Gear

Direction, Gear determines whether the teeth on the gear are right hand or left hand.

Enveloping System

Select between Single and Double Enveloping. The worn of a single enveloping system is basically a straight screw whereas the double enveloping system worm wraps partially around the wheel. At this time, double enveloping worms should only be used with helical gears.

Face Width

Face Width (rim width) defines the width of the gear wheel parallel to the shaft without any hub projections.

Fillet Radius

Fillet Radius defines the radius on the tip of the cutter, which forms a trochoidal curve tangent to the tooth root.

Gear, Outside Diameter

Gear, Outside Diameter is the maximum diameter of the gear wheel.

Gear Standard

Gear Standard sets the AGMA or DIN standard for the component.

Gear System

Specify either Axial or Normal systems. These two systems determine how the addendum and dedendum are calculated.

Lead Wheel

Lead Wheel defines the length of one pitch of the gear wheel. The lead of the gear equals the lead of the worm divided by the number of worm threads (starts).

Lead Worm

Lead Worm defines the length of one pitch of the worm. The lead of the worm equals the lead of the gear wheel multiplied by the number of worm threads (starts).

Lead Angle, Worm

Lead Angle, Worm defines the angle of the worm lead.

Worm Length

Worm Length defines the axial length of the worm.

Module

Module (transverse) is used in imperial system gears.

Module, Normal

Module, Normal is used in metric system gears. Module equals the normal module multiplied by the cosine of the worm angle.

Number of Sketches

This option is only available in GearTeq, it is fixed at 9 in GearTrax. Number of Sketches defines the number of sketches that will be used to construction the wheel tooth loft cut. Values between 5 and 21 are valid. An odd number of sketches is recommended so a sketch will be positioned at the center of the tooth.

Number of Teeth

Number of Teeth defines the number of teeth for the gear (wheel).

Number of Threads

Number of Threads defines the number of threads of the worm. This is also referred to as “starts”.

Pressure Angle

Pressure Angle is the angle of the tooth at the pitch diameter in the transverse plane.

Pressure Angle Normal

Pressure Angle Normal is the angle of the tooth at the pitch diameter normal to the cutter.

Teeth to Pattern

Teeth to Pattern defines the number of teeth to pattern when creating the model in CAD. The options are All, None, or First 10. If this is set to All, GearTrax will create a pattern that contains all the teeth. This might be very time consuming if there are a large number of teeth and/or the component is a helical gear. If this is set to none, then a pattern will not be created. Set this to ‘First 10’ to create a pattern with 10 teeth that can easily be expanded in the CAD system.

If the advanced worm gear tooth shape is used, the tooth cut will not be patterned in CAD. Creating a pattern of an advanced worm tooth on a wheel with many teeth could take more than a few minutes.

Units

Units sets the measurement units for the component; select Inches or Metric.

Wheel Cut Method

GearTeq and SolidWorks users can select between Loft and Cavity cut methods. The loft cut method which is available to all CAD systems uses a series of sketches to create the tooth cut. The cavity method available to SolidWorks users create the tooth cut using a series of "hobs" to create half of the tooth cut. This cut is then circled patterned to complete the whole tooth cut.

Whole Depth

Whole Depth is the depth of the tooth (from the major diameter to the minor diameter).

Worm Outside Diameter

Worm Outside Diameter defines the maximum diameter of the worm unless it is a double-enveloping worm which then determines the outside diameter at the axial center.

Splines

Addendum

Addendum is the length of the tooth from the pitch diameter to the major diameter.

Addendum Modification

Addendum Modification is the amount of addendum change of a modified tooth. This value can be positive or negative. If the sum of change to a pair of gears equals zero then there is no change to the center distance.

Addendum Modification Coefficient

Addendum Modification Coefficient is the ratio of change of a modified tooth. This value can be positive or negative. If the sum of change to a pair of gears equals zero then there is no change to the center distance.

As

(DIN 5480) Upper tooth thickness deviation

Chordal Measurement

Chordal Measurement is a straight-line measurement across a specified number of teeth. The range is based on the maximum effective and the maximum actual tooth thickness.

Chordal Measurement Teeth to Gage Over

Teeth to Gage Over defines the number of teeth that are used with the chordal measurement. This value is automatically generated but the user may input another value. The value can be reset by clicking on the GearTrax menu item Tools>Reset Chordal Measurement.

Circular Pitch

Circular Pitch is the arc length of a single pitch at the pitch diameter.

Coefficient, Addendum

Addendum Coefficient is a constant that is divided by the diametral pitch to determine the length of the addendum before any modification. The addendum length of a 10 diametral-pitch gear with an addendum coefficient of 0.8 is 0.080 in.

Coefficient, Dedendum

Dedendum Coefficient is a constant that is divided by the diametral pitch to determine the length of the dedendum before any modification.

Coefficient, Fillet

Fillet Coefficient is a constant that is divided by the diametral pitch to determine the radius of the hob tip that creates the gear tooth root.

Dedendum

Dedendum is the radial length of the tooth between the pitch diameter and the minor diameter.

Designation

(DIN 5480) Designation is a read-only property that displays the parameters of the individual spline.

For example, DIN 5480-W35x2x30x16x8h:

DIN 5480, the standard used
W, W for a shaft, N for a hub
35, reference diameter
2, module
30, pressure angle
16, number of teeth
8h, tolerance grade and deviation

Deviation

(DIN 5480) Tolerance Position. Select "a" through "v" for the DIN standard. If "no class" is selected then the individual properties of the gear can be controlled by the user.

Diameter, Base

Base Diameter is a diameter that is tangent to the pressure angle. The involute curve cannot be within this diameter.

Diameter, Major, Maximum

Diameter, Major, Maximum defines the maximum major diameter. This is normally read only unless “No Class” is selected for the Tolerance Class, ANSI and JIS splines only.

Diameter, Major, Minimum

Diameter, Major, Minimum defines the minimum major diameter. This is normally read only unless “No Class” is selected for the Tolerance Class, ANSI and JIS splines only.

Diameter, Minor, Maximum

Diameter, Minor, Maximum defines the maximum minor diameter. This is normally read only unless “No Class” is selected for the Tolerance Class, ANSI and JIS splines only.

Diameter, Minor, Minimum

Diameter, Minor, Minimum defines the minimum minor diameter. This is normally read only unless “No Class” is selected for the Tolerance Class, ANSI and JIS splines only.

Diameter, Nominal SAE

Defines the SAE Nominal Diameter.

Diameter, Pitch

Pitch Diameter is the theoretical diameter of the spline.

Diameter, True, Involute Form

True Involute Form diameter (TIF) is the diameter of the circle beyond which the tooth profile must conform to the specified involute curve.

Diametral Pitch

Diametral Pitch is used in imperial system gears. This defines the diametral pitch in the transverse plane.

Diametral Pitch, Stub

Stub Diametral Pitch is used in imperial system gears. This defines the stub diametral pitch in the transverse plane and is normally equal to twice the diameter pitch.

Fillet Radius

Fillet Radius defines the radius on the tip of the cutter, which forms a trochoidal curve tangent to the tooth root.

Form Clearance

Form clearance is the radial depth of the involute profile beyond the depth of engagement with the mating spline component.

Length

Length is the length of the tooth parallel to the shaft.

Manufacturing Method DIN

Manufacturing Method DIN sets the manufacturing method for the component. The manufacturing method has a direct effect on the dedendum depth. For external splines the options are broaching, hobbing, shaping and cold rolling. For internal the options are broaching, hobbing and shaping.

Measurement Between Pins; Actual Maximum



Measurement Between Pins is the measurement between two pins diametrically opposed in an internal spline and is normally used for inspection. GearTrax will calculate the space width for internal spline if a value is entered for the between pin measurement.

Measurement Between Pins; Actual Minimum



Measurement Between Pins is the measurement between two pins diametrically opposed in an internal spline and is normally used for inspection. GearTrax will calculate the space width for internal spline if a value is entered for the between pin measurement.

Measurement Between Pins; Effective Minimum



Measurement Between Pins is the measurement between two pins diametrically opposed in an internal spline and is normally used for inspection. GearTrax will calculate the space width for internal spline if a value is entered for the between pin measurement.

Measurement Over Pins; Actual Maximum



Measurement Over Pins is the measurement between two pins diametrically opposed in an external spline and is normally used for inspection. GearTrax will calculate the tooth thickness for an external spline entered for the over pin measurement.

Measurement Over Pins; Actual Minimum



Measurement Over Pins is the measurement between two pins diametrically opposed in an external spline and is normally used for inspection. GearTrax will calculate the tooth thickness for an external spline if a value is entered for the over pin measurement.

Measurement Over Pins; Effective Maximum



Measurement Over Pins is the measurement between two pins diametrically opposed in an external spline and is normally used for inspection. GearTrax will calculate the tooth thickness for an external spline if a value is entered for the over pin measurement.

Module

Module (transverse) is used in metric system gears. Module equals the normal module divided by the cosine of the helical angle.

Number of Teeth

Number of Teeth defines the number of teeth for the component.

Pin Diameter

Pin Diameter defines the diameter of the pins or wires used with Measurement of Pins. A user defined pin diameter may be entered. To reset the value to the standard value select Tools>Reset Over/Under Pin Measurement in the GearTrax menu.

Pressure Angle

Pressure Angle is the angle of the tooth at the pitch diameter.

Root and Fit

Root and Fit defines the type of fit for the component. Select fillet root side fit, flat root side fit or flat root major diameter fit.

Space Width; Actual Maximum



The actual space width is the circular width on the pitch circle of any single space. Please consult other gear manuals for a fuller description.

Space Width; Actual Minimum



The actual space width is the circular width on the pitch circle of any single space. Please consult other gear manuals for a fuller description.

Space Width; Basic



The basic space width is the basic circular width on the pitch circle of any single space. Please consult other gear manuals for a fuller description.

Space Width; Effective Maximum



Space Width Effective Maximum of an internal spline is equal to the circular tooth thickness on the pitch circle of an imaginary perfect external spline, which would fit the internal spline without looseness or interference, considering the engagement of the entire axial length of the spline. Please consult other gear manuals for a fuller description.

Space Width; Effective Minimum



Space Width Effective Minimum of the internal spline is always basic.

Spline Fit Class

Spline Fit Class (ANSI) is a user selectable property. For internal splines, the class is always "H". For external splines the options are "H_d", "H_e", "H_f" and "H_h".

Standard

Standard, select between 4 different standards:

- ANSI Diametral B92.1 1996
- ANSI Module B92.2M 1980 R1989
- DIN 5480 and 5482
- JIS B 1603

Tact

(DIN 5480 and 5482) Actual tooth thickness (or space width) tolerance. To view this value click on the Chart button.

Teeth to Pattern

Teeth to Pattern defines the number of teeth to pattern when creating the model in CAD. The options are All, None, or First 10. If this is set to All, GearTrax will create a pattern that contains all the teeth. This might be very time consuming if there are a large number of teeth and/or the component is a helical gear. If this is set to none, then a pattern will not be created. Set this to 'First 10' to create a pattern with 10 teeth that can easily be expanded in the CAD system.

Teff

(DIN 5480 and 5482) Effective tooth thickness (or space width) tolerance. To view this value click on the Chart button.

TG

(DIN 5480 and 5482) Total tooth thickness (or space width) tolerance. To view this value click on the Chart button.

Tolerance Class

(ANSI and JIS) Specifies the tolerance class for ANSI and JIS splines. If "No Class" is selected then the user may enter values for individual parameters of the component

Tolerance Grade

(DIN 5480 and 5482) Values between 4 and 12 inclusive are valid.

Tooth Thickness; Actual Maximum

Actual Maximum Tooth Thickness is the maximum circular thickness on the pitch circle of any single tooth.

Tooth Thickness; Actual Minimum

Actual Minimum Tooth Thickness is the minimum circular thickness on the pitch circle of any single tooth.

Tooth Thickness; Basic

Tooth Thickness Basic is the basic circular thickness on the pitch circle of any single tooth.

Tooth Thickness; Effective Maximum

The effective tooth thickness of an external spline is equal to the circular space width on the pitch circle of an imaginary perfect internal spline, which would fit the external spline without looseness or interference, considering engagement of the entire axial length of the spline. Please consult other gear manuals for a fuller description.

Tooth Thickness; Effective Minimum



The effective tooth thickness of an external spline is equal to the circular space width on the pitch circle of an imaginary perfect internal spline, which would fit the external spline without looseness or interference, considering engagement of the entire axial length of the spline. Please consult other gear manuals for a fuller description.

Units

Units sets the measurement units for the component; select Inches or Metric. DIN and JIS are always metric units.

Hub Mounting



Hubs can be added to each part by clicking on the Hub Mounting button. Hub mounting is not available for splines.

Bore Diameter

Bore defines the diameter of the bore.

Bore Chamfer

Bore Chamfer defines the size of the chamfer at the ends of the bore.

Hub 1st Side Chamfer

Hub 1st Side Chamfer defines the size of the chamfer and fillet for the hub revolve on side one of the component.

Hub 1st Side Diameter

Hub 1st Side Diameter defines the size of the diameter of the hub revolve on side one of the component.

Hub 1st Side Projection

Hub 1st Side Projection defines the size of the projection of the hub revolve on side one of the component.

Hub 2nd Side Chamfer

Hub 2nd Side Chamfer defines the size of the chamfer and fillet for the hub revolve on side two of the component.

Hub 2nd Side Diameter

Hub 2nd Side Diameter defines the size of the diameter of the hub revolve on side two of the component.

Hub 2nd Side Projection

Hub 2nd Side Projections defines the size of the projection of the hub revolve on side two of the component.

Keyway

Keyway defines the type of keyway, if any, to be created with the component. A valid bore diameter must be specified.

Keyway Position

Keyway Position defines the keyway relative to the tooth; select On Tooth Center or On Space Center. If the component is part of a planetary set it should be left at the On Space Center option.

Bushing Side

Bushing Side defines on which side of the component the bushing will be mounted.

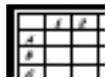
Split Taper Bushing

Mounting Split Taper Bushing defines, if any, the bushing to be used with the component.

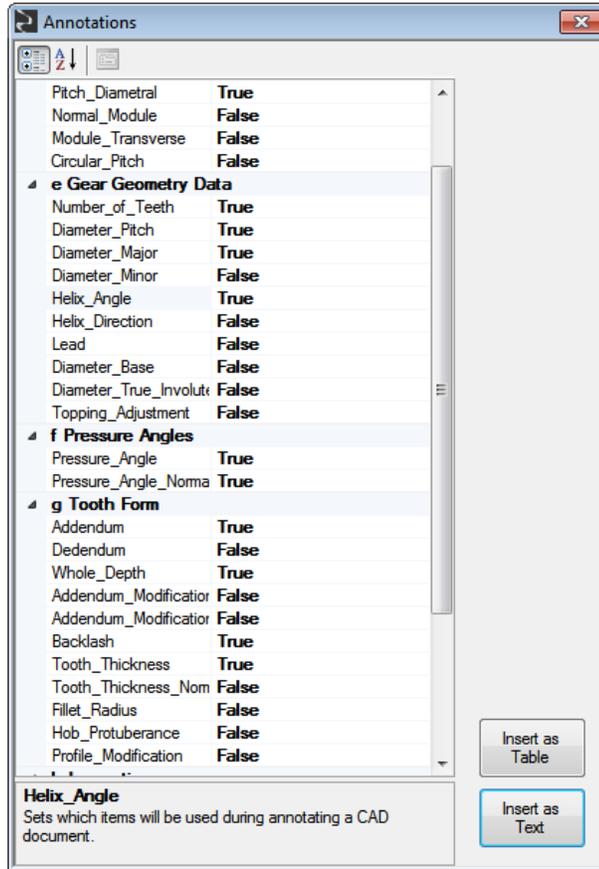
Standard Set Screw

Standard Set Screw check box defines if a standard set screw should be used with a keyway. The setscrew hole will only be created if a hub and keyway are also created.

Annotations



Annotations in the form of tables and text can be added to a CAD drawing document.



The Annotation window can be opened by clicking on the Annotation button. Set each of the properties to True items if you would like that item to be added to the table or text in the CAD drawing.

Insert as Table

Click on the Insert as Table button to have the table added to the CAD drawing document.

Insert as Text

Click on the Insert as Text button to have the text added to the CAD drawing document.

* Note to SolidWorks users:

- A view must be active in the drawing for the text or table to be added
- The parameters displayed in the text or table are derived from the Configuration Specific properties of the part document. If any of those values are change in the part file, the drawing will reflect those changes.

Tool Bars

Standard Views



Front

Clicking the Front tool bar button changes the view orientation to Front at the current view scale.



Back

Clicking the Back tool bar button changes the view orientation to Back at the current view scale.



Left

Clicking the Left tool bar button changes the view orientation to Left at the current view scale.



Right

Clicking the Right tool bar button changes the view orientation to Right at the current view scale.



Top

Clicking the Top tool bar button changes the view orientation to Top at the current view scale.



Bottom

Clicking the Bottom tool bar button changes the view orientation to Bottom at the current view scale.



Isometric

Clicking the Isometric tool bar button changes the view orientation to Isometric at the current view scale.



Normal To Cutter

Clicking the Normal To Cutter tool bar button changes the view orientation to normal to the cutter at the current view scale.

View



Zoom to Fit

Clicking the Zoom to Fit tool bar button changes the current view orientation to fit inside of the window.



Zoom Area

Clicking the Zoom Area tool bar button allows the user to draw a box over an area to be displayed at full screen.



Zoom In Out

Clicking the Zoom In Out tool bar button allows the user to use the mouse left button to zoom in or out by dragging the cursor on the screen. Dragging the cursor vertically up increases the scale of the display. Dragging the cursor down decreases the scale of the display. The amount of change can be increased by dragging from the upper left hand corner to the lower right hand corner. Conversely, the change can be decreased by dragging from the upper right hand corner to the lower left hand corner.



Rotate the View

Clicking the Rotate the View tool bar button allows the user to rotate the view by using either the mouse left or middle buttons to drag the cursor on the screen.



Pan

Clicking the Pan tool bar button allows the user to move the view by using either the mouse left or middle buttons to drag the cursor on the screen.



Drive

Clicking the Drive tool bar button allows the user to drive the components by using the mouse left button to drag the cursor on the screen.

Simulation



Stop

Clicking the Stop tool bar button will stop a running simulation.



Back

Clicking the Back tool bar button will drive the components back one spin increment. Set the spin increment in the options menu.



Play

Clicking the Play tool bar button will drive the components for a short simulation.



Play Reverse

Clicking the Play Reverse tool bar button will drive the components in the opposite direction for a short simulation.



Forward

Clicking the Forward tool bar button will drive the components forward one spin increment. Set the spin increment in the options menu.

Mouse controls

Mouse wheel

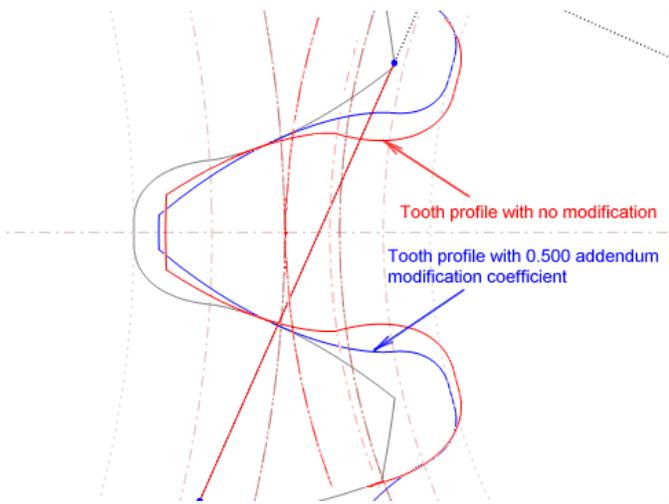
Wheel up and down scales the view of the components. All scaling is done from the center of the screen, not the mouse position. Pan the geometry to the center of the screen then use the mouse wheel to enlarge or reduce the view scale. The direction of the mouse wheel is controllable by a setting in the Options menu.

Middle Mouse Button (wheel)

Drag the view with the middle button held down. The wheel can be rotated at the same time to zoom the view.

Addendum Modification and Addendum Modification Coefficient.

With the pinion gear normally having fewer teeth than its mating gear, it has a disadvantage of being the weaker of the two components. To compensate for this inherent weakness, the addendum of the pinion can be made stronger by increasing its tooth thickness. The pinion's mating gear is normally adjusted to compensate, thereby making the gear's tooth thinner. The process that controls this adjustment also varies the length of the addendum and dedendum. If the pinion's mating gear does not compensate for this adjustment then the center distance of the pair will be enlarged or reduced. Under certain circumstances, this can be used to the designer's advantage.



It is recommended that the designer think in the terms of the coefficient rather than the simpler length adjustment. Once the designer is accustomed to thinking in terms of the coefficient, then the size of the diametral pitch (or Module) does not need to be considered. For example, a 0.25 addendum modification coefficient will affect a 12 diametral pitch gear the same way it will affect a 24 diametral pitch gear, proportionally for the size of diametral pitch. The simpler addendum modification is a derivative of the diametral pitch (or module) and the coefficient.

Tutorials

Instructional Videos

Instructional videos can be found on the Camnetics web site:

<http://www.camnetics.com/media.htm>

Creating Parts and Assemblies in CAD



Creating a parts and assemblies in CAD are easy:

On tabs with only one part clicking the Create Model in CAD will create a new model in CAD without any prompts.

On tabs with two parts the options are to create:

- Assembly and both models
- Both models
- Either of the two models
- In some case, the tooth profile can be added to an open sketch in CAD

When creating assemblies the part files must be saved. GearTrax always uses the same name for each of the different parts. It is required that if the parts were previously created in CAD they need to be closed. The part files will be overwritten when the new parts are created. It is highly recommended to save those CAD files right away with new names if they are to be preserved.

Inserting tooth cut entities into a open sketch in CAD



The tooth cut entities of following components can be created in an open sketch in CAD:

- Spur pinion
- Spur gear
- Chain sprocket
- Internal spline
- External spline

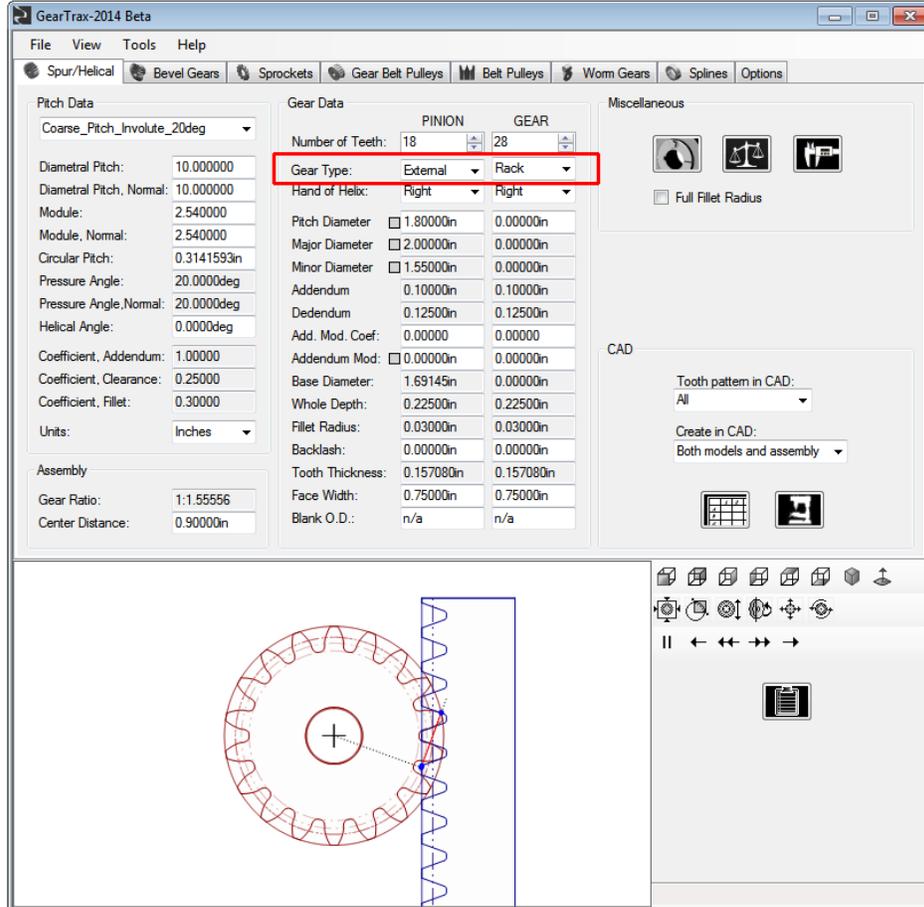
To create only the tooth cut entities select the appropriate option in the Create in CAD drop down box. Then click the create button.

Rack and Pinion

Creating a rack and pinion set in GearTrax is easy:

For the Gear, Gear Type select Rack

Set all the other required parameters like diametral pitch and number of teeth.



Internal Gear Set

Creating an internal gear set in GearTrax is easy:

For the Gear, Gear Type select Internal

Set all the other required parameters like diametral pitch and number of teeth.

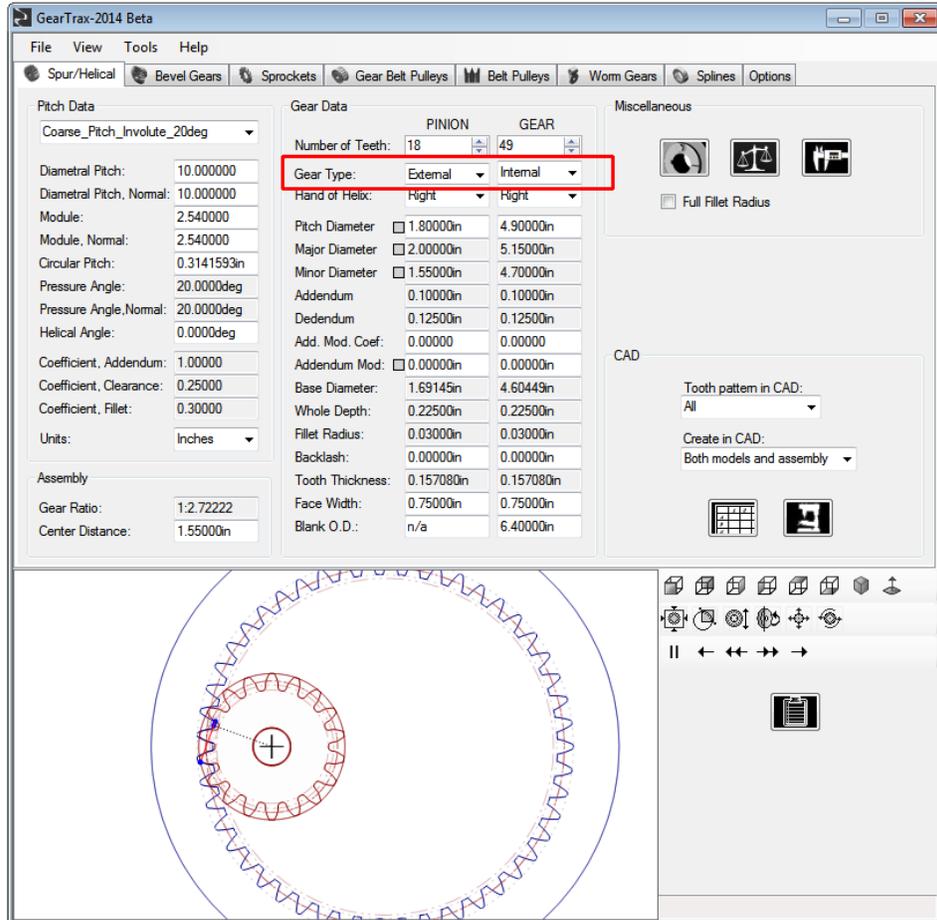


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