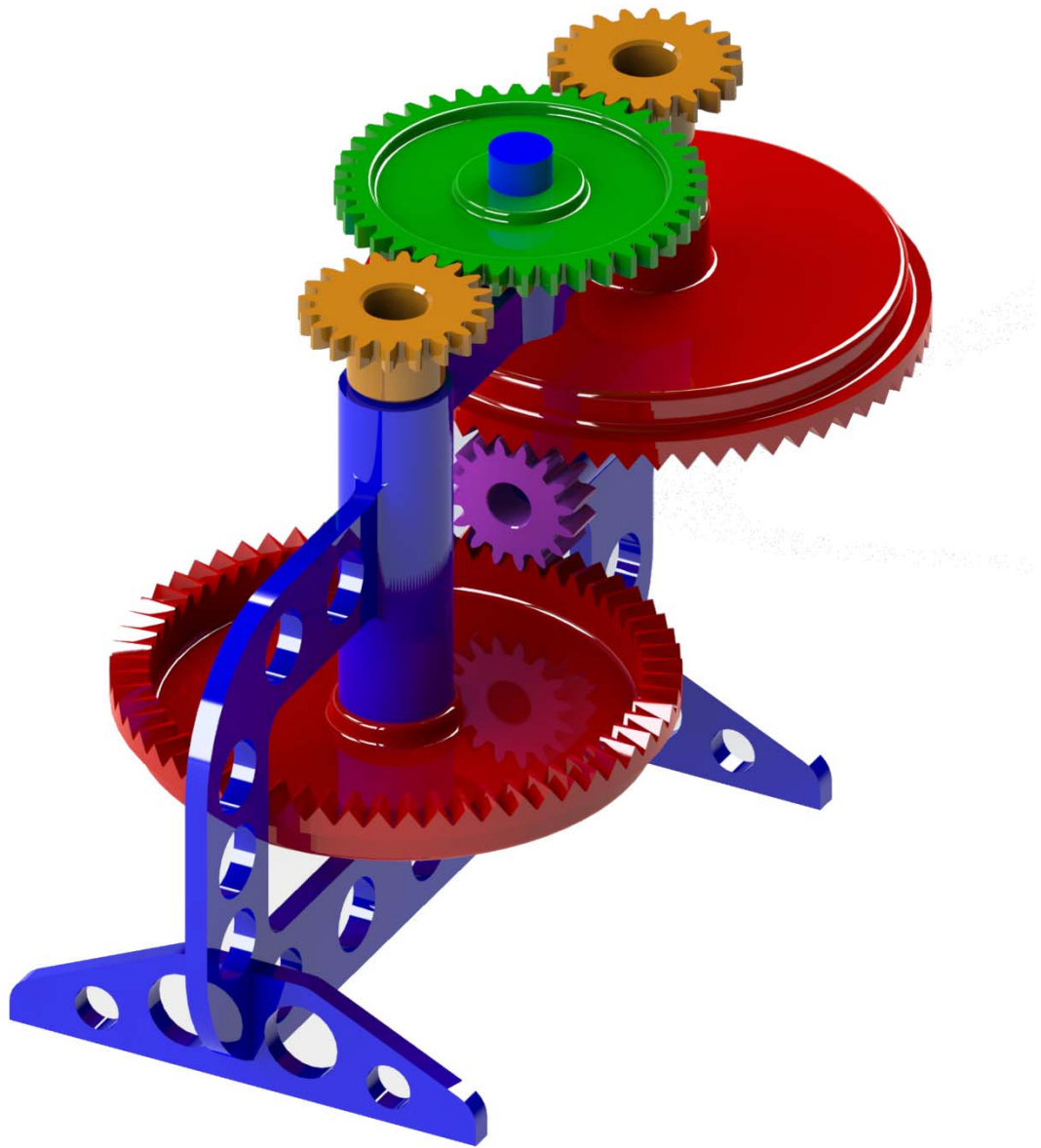


GearTeq



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Introduction

GearTeq is a property driven gear design program. It is not the intent of GearTeq 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.

A GearTeq component can be a CAD part by itself. The component can be part of a set of components that make up a single CAD part. For example, a spur gear is a single component but this spur gear may have an internal spline as a child mate that is defined as a bore. When created in CAD, the spur gear will be created and the internal spline will be added features of the spur gear. This is also true for mating parts that are defined as joined.

GearTeq 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

Assembly is a group of GearTeq assembly positions or a group of CAD parts.

Assembly Position (AP)

An assembly can contain any number of assembly positions. A single component is specified for each AP. A number of different APs can contain the same component. This way a component can be located on the assembly or multiple assemblies, each with different mating parameters.

Component

Component is a single GearTeq gear, sprocket, pulley, etc. Multiple components can define a single CAD part.

Part

Part is a single CAD part document file that may contain one or more GearTeq components. For example, a spur gear may have an internal spline as a bore, each being a GearTeq component.

Model

Model is a CAD part document.

Annotation

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

Model

Model is a CAD part document.

Starting GearTeq

GearTeq is compiled and published as a Click-once application. To have GearTeq available on a computer, each user of that computer must install GearTeq. Use the Windows Start menu to start GearTeq by clicking on Start>All Programs>Camnetics, Inc>GearTeq*.

GearTeq Menus

File

New

New removes all components and assembly positions. GearTeq is ready to start a new assembly or component.

Open Components

Open Components opens a previously saved component document. Component documents have .gtc as the file extension.

Open Assemblies

Open Assemblies opens a previously saved assembly document. Assembly documents have .gtqasm as the file extension.

Open Legacy Assemblies

Open Legacy Assemblies opens older GearTeq assembly files created in the 2011 and earlier versions. Legacy Assembly files have .gta as the extension.

Save Component

Save Component saves the active component. The component name will change to reflect then name of the file. Component files have .gtc as the extension.

Save Component As

Save Component As saves and changes the name of the active component. The component name will change to reflect the name of the file. Component files have .gtc as the extension.

Save All

Save All saves the assembly. The assembly will change to reflect the name of the file. Assembly files have .gtqasm as the extension and component files have.gtc as the extension.

Save All As

Save All As saves and changes the name of the assembly and components. The assembly will change to reflect the name of the file. Assembly files have .gtqasm as the extension and component files have .gtc as the extension.

Save Assembly As

Save Assembly As saves and changes the name of the assembly. The assembly will change to reflect the name of the file. Assembly files have .gtqasm as the extension.

Lock/Unlock Active Component

Lock/Unlock Active Component opens a dialog box that allows the user to lock or unlock a component. The pass code is simply an integer with a value between 200 and 32,000. When a component is locked the property background color will be gray.

Remove Active Component

Remove Active Component removes the active component from the assembly. The component must not be a parent to any other components.

Remove Unused Components

Remove Unused Components removes any orphaned components that may have been left in the assembly if there were changes to the components specified in an AP. A message box will prompt the user with which components can be removed.

Print the GearTeq Window

Print GearTeq Window prints the GearTeq window, including borders.

Capture GearTeq screen to a file

Capture GearTeq screen to file captures the GearTeq screen, including borders, to a bit map file.

Capture GearTeq screen to the clipboard

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

Capture GearTeq screen and open in bit map editor

Capture GearTeq screen and open in bit map editor captures the GearTeq screen, including borders, to the default bit map editor.

Empty Clipboard

Empty Clipboard clears the clipboard of any data. This frees up any memory the screen capture is using.

Convert GearTrax file

Convert GearTrax file opens an existing GearTrax file. Save it as a GearTeq component to complete the conversion to GearTeq.

Recent files

Recent files displays recently used files. Click on the recent file to open it in GearTeq.

Exit

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

CAD**Create CAD Models and Assembly**

Create CAD Models and Assembly opens the Create CAD Models dialog window.

Quick Create > Create Active Component

Quick Create>Create Active Component creates a CAD Model of the active component without any options.

Quick Create > Insert Tooth Cut Entities

Quick Create>Insert Tooth Cut Entities draws the tooth cut entities of the active spur or spline in the active sketch in CAD.

Edit CAD Models

Edit CAD Models opens the dialog window for editing CAD Models.

Component Template File

Component Template File opens a dialog box for specifying the CAD template to be used when creating new CAD models.

Assembly Template File

Assembly Template File opens a dialog box for specifying the CAD template to be used when creating new CAD assemblies.

Annotation Selector

Annotation Selector loads the annotation definitions into the property grid for the active component. Items set to true will be used when annotating a drawing file.

Insert Annotations

(SolidWorks users only)

Insert Annotations will insert a note into a view of a drawing document.

To insert an annotation a few things must be set:

First, the annotation selector must be active. It doesn't need to be the selector for the current active CAD file but it should be the same type of component. If the same annotation properties are to be used for a variety of components, then a generic component can always be used to create the annotation. This will relieve the user

of the task of having to set each of the properties to true or false for each new component.

The CAD document must be a drawing.

A drawing view must be active (a green box highlights the view).

When inserting an annotation there is a choice of adding it as text or as a table. In either case, the information is retrieved from the part file. If the part file data changes, this will be updated automatically. All information is related to the part file's Menu>Properties...>Configuration Specific>Apply to:> option.

If inserting as text, a font such as "Lucida Console" may be desired because it will align the columns with its even character spacing.

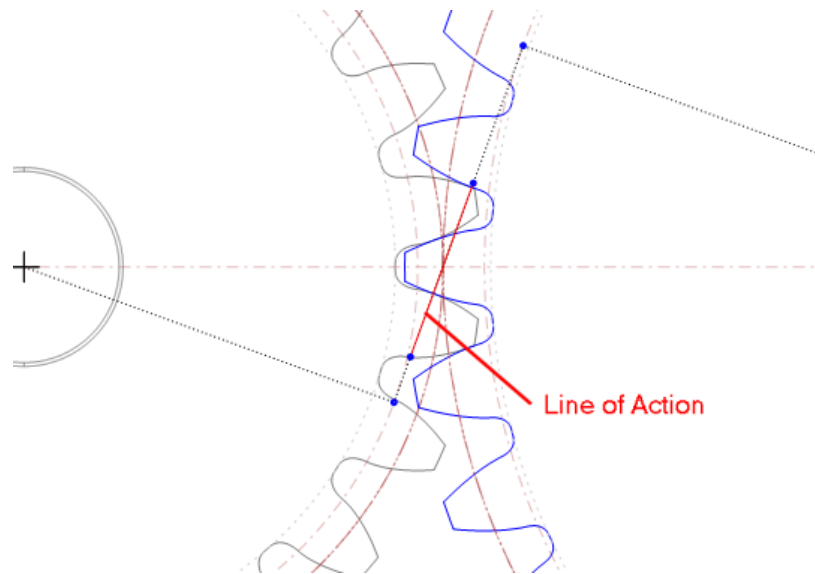
View

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.

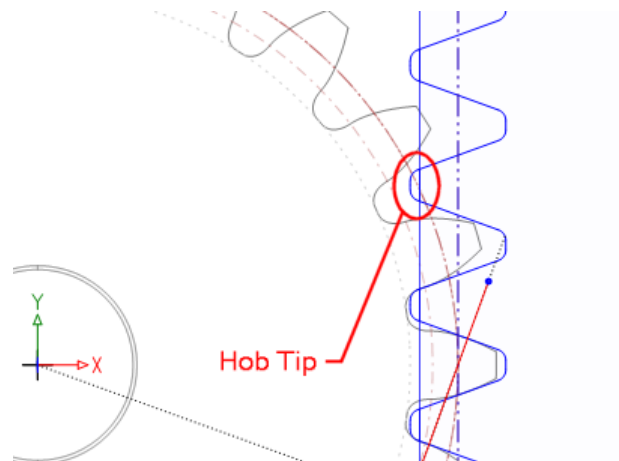


Show Over/Under Pins

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

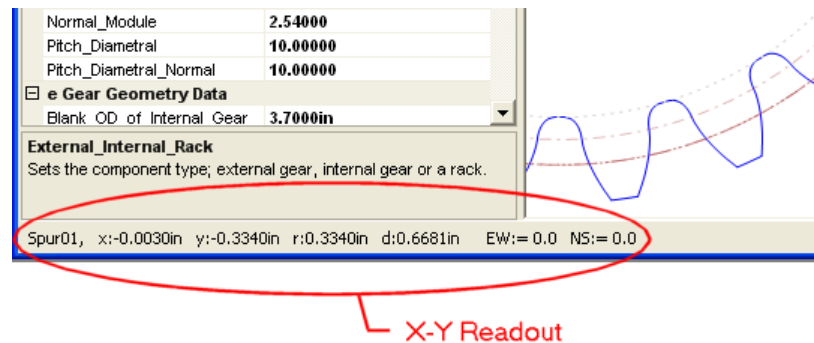
Show Hob Tip on Racks

Show Hob Tip on Rack, if checked, will display the hob tip on all rack type components.



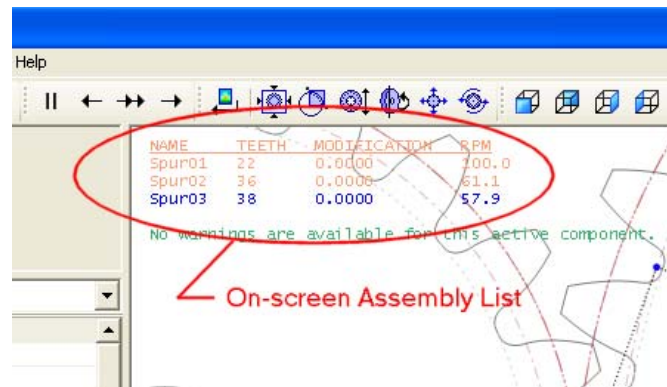
Show XY Readouts

Show XY Readouts, if checked, will display the x, y, r and d in the task bar for the cursor position in the current screen plane.



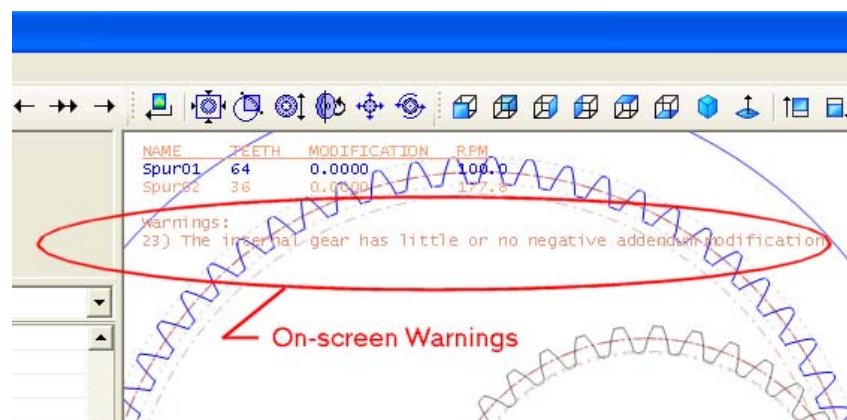
Show Assembly On-screen List

Show Assembly On-Screen List, if checked, will display the assembly list in the upper left hand corner of the drawing screen.



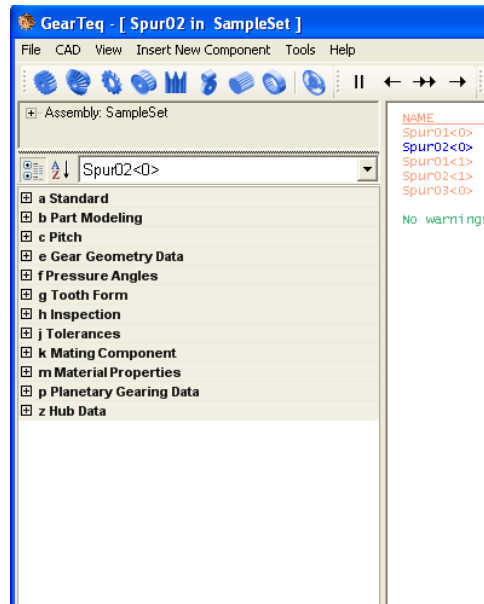
Show Warnings On-screen

Show Warnings On-screen, if checked, will display any available warnings in the upper left hand corner of the drawing screen.



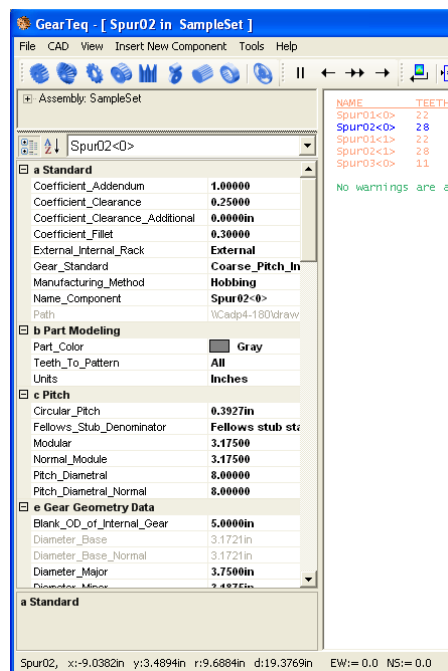
Collapse Grid

Collapse Grid collapses the property grid to the categories. The short cut key is F11.

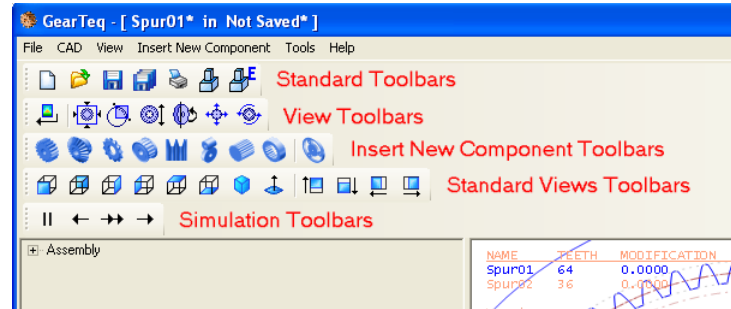


Expand Grid

Expand Grid expands the property grid to display all items. The short cut key is F12.



Toolbars



Standard

If checked, the Standard tool bar will be displayed.

View

If checked, the View tool bar will be displayed.

Standard Views

If checked, the Standard Views tool bar will be displayed.

Simulation

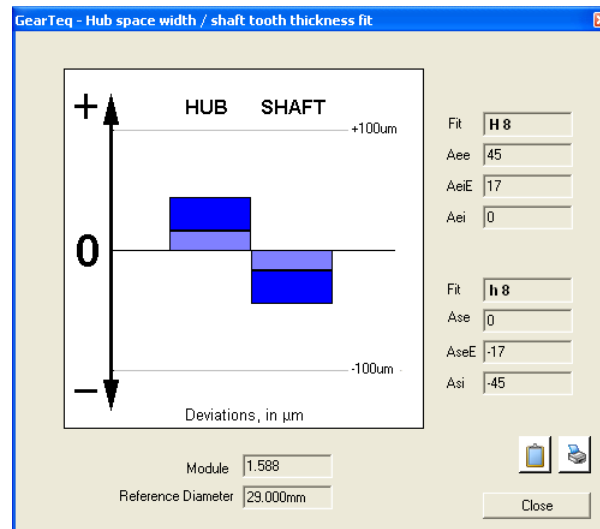
If checked, the Simulation tool bar will be displayed.

Insert New Component

If checked, the Insert New Component tool bar will be displayed.

Deviation Chart

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



Values of the components can be changed in the property grid and the chart is updated automatically.

CPI (Common Properties Interface)

The shortcut key to open the CPI is "F2"

The CPI is an interface that allows the user to modify more than one component without switching the active component. The CPI will change for different types of components or planetary gear sets.

The CPI is not available for chain sprockets, timing belt pulleys and Vee belt pulleys.

SI - Ring_Gear_Fixed01

	--- SUN ---	--- PLANET ---	--- PLANET ---	--- RING ---
Diametral pitch, normal:	10.00000		10.00000	
Number of teeth (z):	27	37	11	75
Type of gear:	External	External	External	Internal
Pitch diameter:	2.7000in	3.7000in	1.1000in	7.5000in
Major diameter:	2.9000in	3.9000in	1.3000in	7.7500in
Minor diameter:	2.4500in	3.4500in	0.8500in	7.3000in
Addendum:	0.1000in	0.1000in	0.1000in	0.1000in
Dedendum:	0.1250in	0.1250in	0.1250in	0.1250in
Add. mod. coef. (x):	0.0000	0.0000	0.0000	0.0000
Addendum modification (x*m)	0.0000in	0.0000in	0.0000in	0.0000in
Whole depth:	0.2250in	0.2250in	0.2250in	0.2250in
Pressure angle, transverse :	20.0000deg		20.0000deg	
Base diameter:	2.5372in	3.4769in	1.0337in	7.0477in
Helix angle:	0.0000deg		0.0000deg	
Helix direction:	Right Hand	Right_Hand	Right Hand	Right Hand
Circular pitch:	0.314159in		0.314159in	
Fillet radius:	0.0300in	0.0300in	0.0300in	0.0300in
Backlash:	0.0000in	0.0000in	0.0000in	0.0000in
Tooth thickness:	0.15708in	0.15708in	0.15708in	0.15708in
Face width:	0.7500in	0.7500in	0.7500in	0.7500in

Balance Addendum Modification Reload

Schematic

There are generic schematics available by clicking on this menu item.

YourGear Interface

The YourGear Interface gives the user a method for creating custom interfaces. See a PDF in the YourGear Interface help menu for more detailed information.

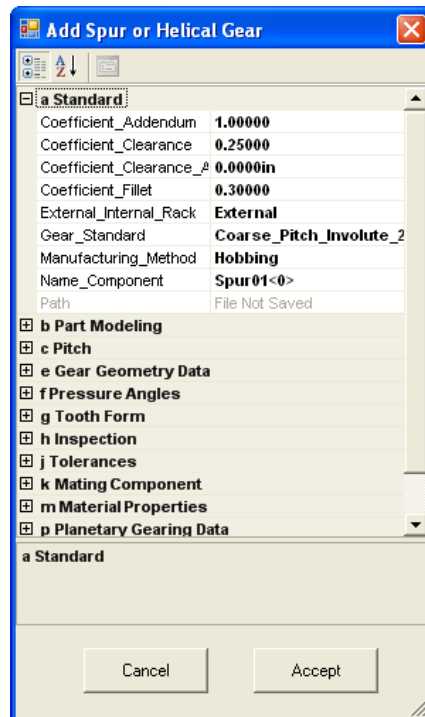
Insert New Component

Insert a New Assembly Position

Insert a New Assembly Position adds a blank assembly position into the assembly.

New Spur or Helical...

Clicking ‘New Spur or Helical...’ opens a dialog box to insert a new spur or helical gear into the assembly.

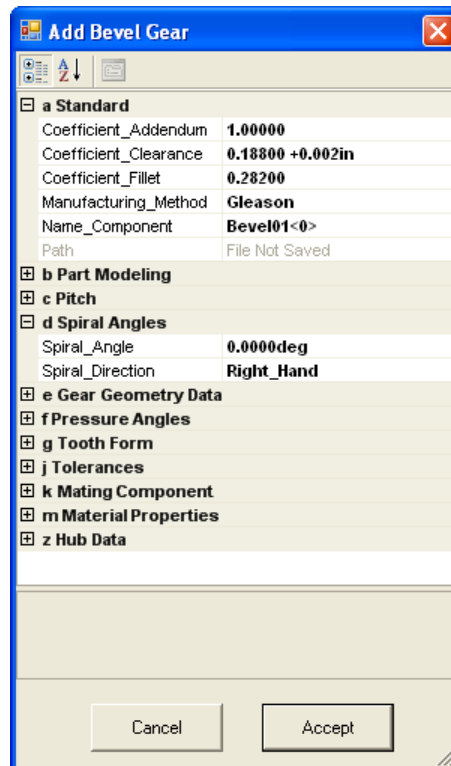


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

The short cut key for this menu item is Ctrl+S.

New Bevel Gear...

Clicking ‘New Bevel Gear...’ opens a dialog box to insert a new bevel gear into the assembly.

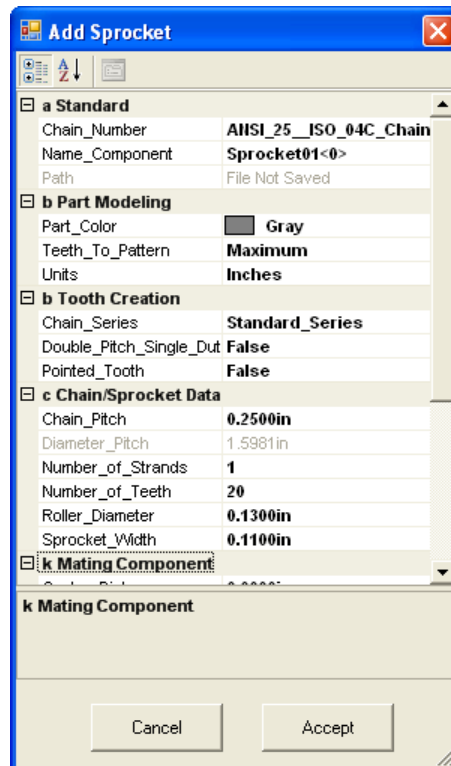


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

The short cut key for this menu item is Ctrl+B.

New Chain Sprocket...

Clicking ‘New Chain Sprocket...’ opens a dialog box to insert a new chain sprocket component into the assembly.

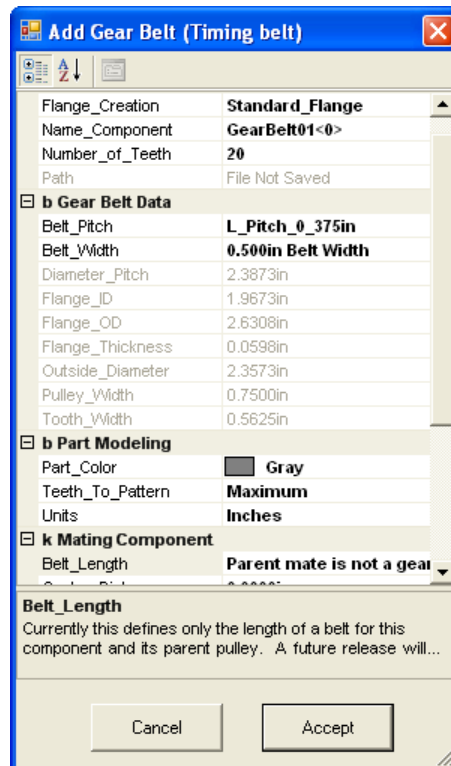


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

The short cut key for this menu item is Ctrl+C.

New Gear Belt Pulley...

Clicking ‘New Gear Belt Pulley...’ opens a dialog box to insert a new gear belt pulley component into the assembly.

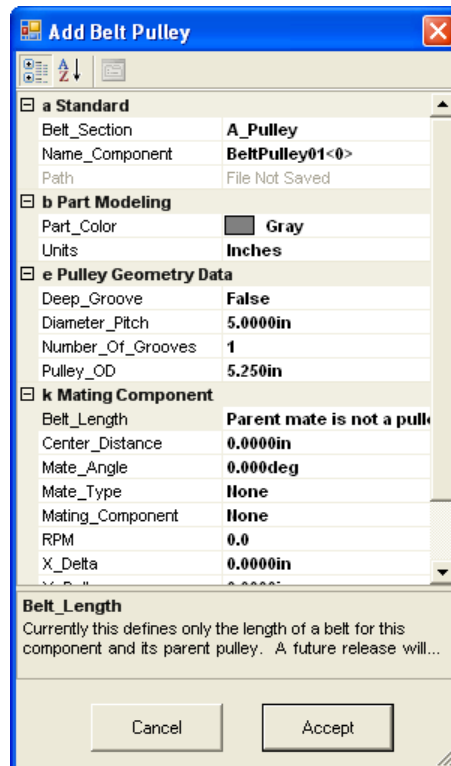


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

The short cut key for this menu item is Ctrl+G.

New Belt Pulley...

Clicking ‘New Belt Pulley...’ opens a dialog box to insert a new belt pulley component into the assembly.

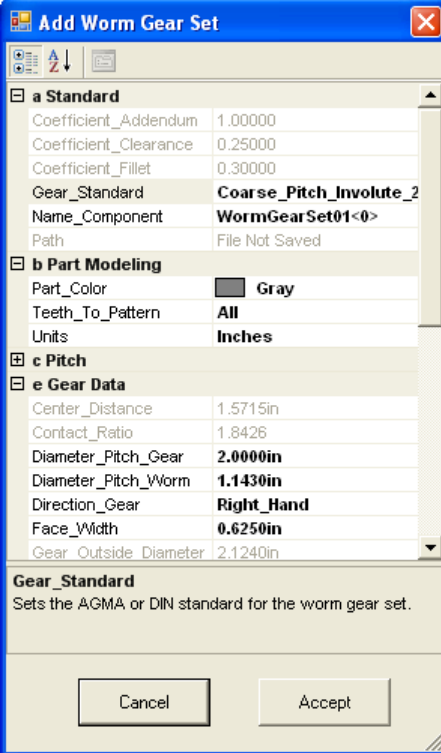


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

The short cut key for this menu item is Ctrl+P.

New Worm Gear Set...

Clicking ‘New Worm Gear Set...’ opens a dialog box to insert a new worm gear pair.



The 'Add Worm Gear Set' dialog box is shown with the following parameters:

a Standard	
Coefficient_Addendum	1.00000
Coefficient_Clearance	0.25000
Coefficient_Fillet	0.30000
Gear_Standard	Coarse_Pitch_Involute_2
Name_Component	WormGearSet01<0>
Path	File Not Saved

b Part Modeling	
Part_Color	Gray
Teeth_To_Pattern	All
Units	Inches

c Pitch	

e Gear Data	
Center_Distance	1.5715in
Contact_Ratio	1.8426
Diameter_Pitch_Gear	2.0000in
Diameter_Pitch_Worm	1.1430in
Direction_Gear	Right_Hand
Face_Width	0.6250in
Gear_Outside_Diameter	2.1240in

Gear_Standard
Sets the AGMA or DIN standard for the worm gear set.

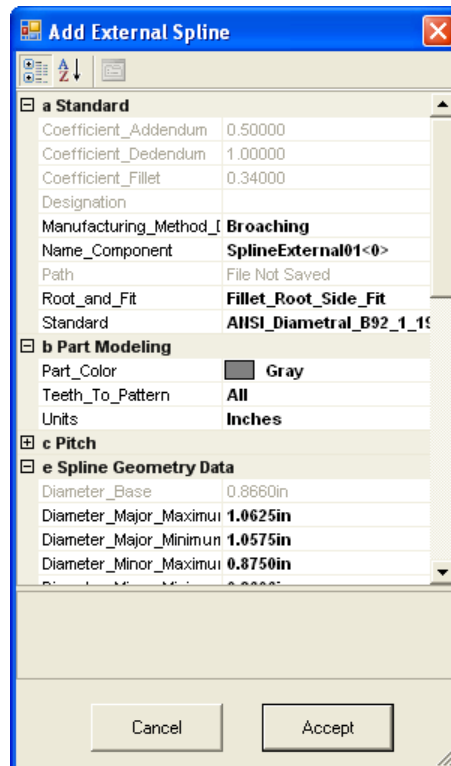
Buttons: Cancel, Accept

Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

The short cut key for this menu item is Ctrl+W.

New External Spline...

Clicking ‘New External Spline...’ opens a dialog box to insert a new external spline component into the assembly.

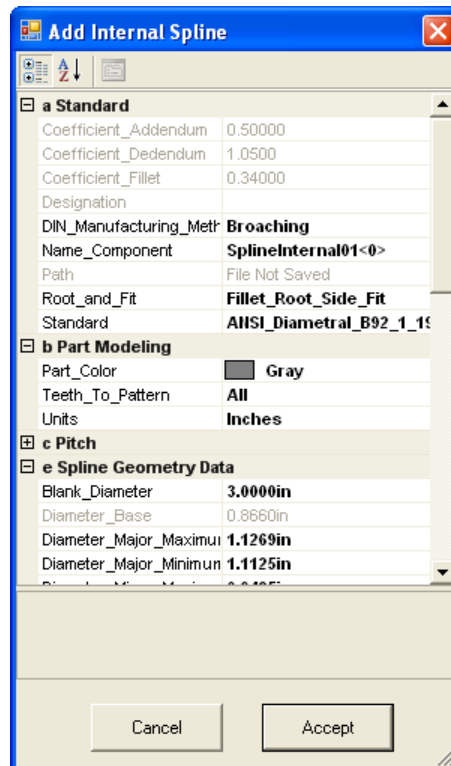


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

The short cut key for this menu item is Ctrl+E.

New Internal Spline...

Clicking ‘New Internal Spline...’ opens a dialog box to insert a new internal spline component into the assembly.

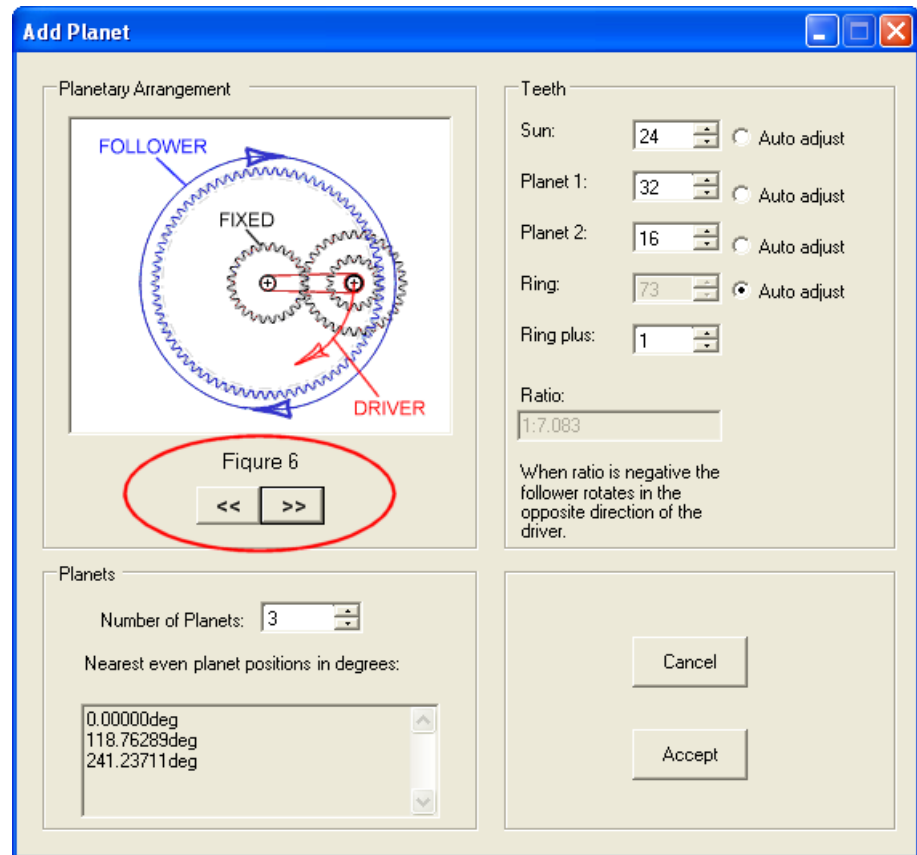


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

The short cut key for this menu item is Ctrl+I.

New Planetary Set...

Clicking “New Planetary Set...” opens a dialog box that can be used to create a single stage planetary gear set.



Use the index buttons to cycle between the available planetary arrangements. These figure numbers correspond to the figure numbers in Machinery’s Handbook.

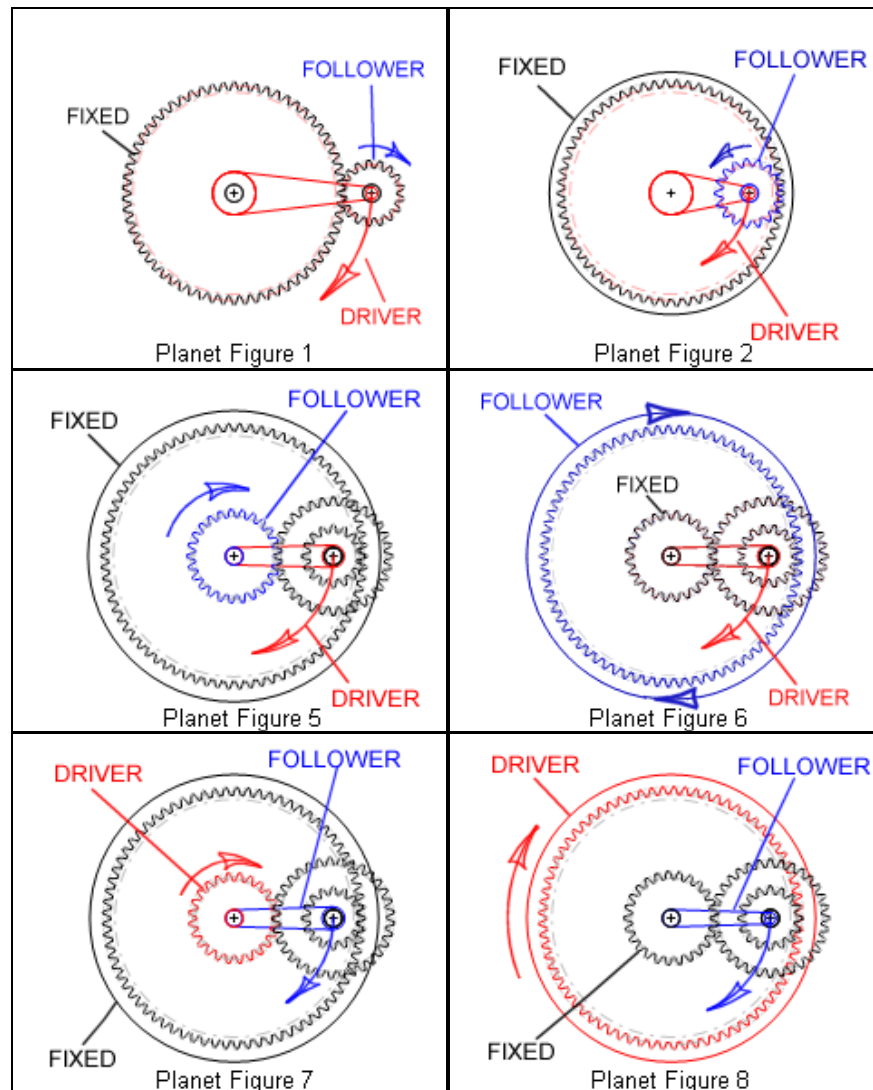
Select the number of planets for the set. The angular placement of the planets is displayed. The possible number of planet positions is equal to the sum of teeth in the sun and the ring. GearTeq displays the planet position of the nearest equally spaced position.

The user may change the number of teeth using the dialog box, or later, in the property grid for each of the components. GearTeq will only automatically adjust the number of teeth for one of the components as the other components are changed in the Add Planet dialog window. Select which component you would like GearTeq to change automatically by selecting the **Auto Adjust** bullet. After the parts are displayed in GearTeq, the user will be

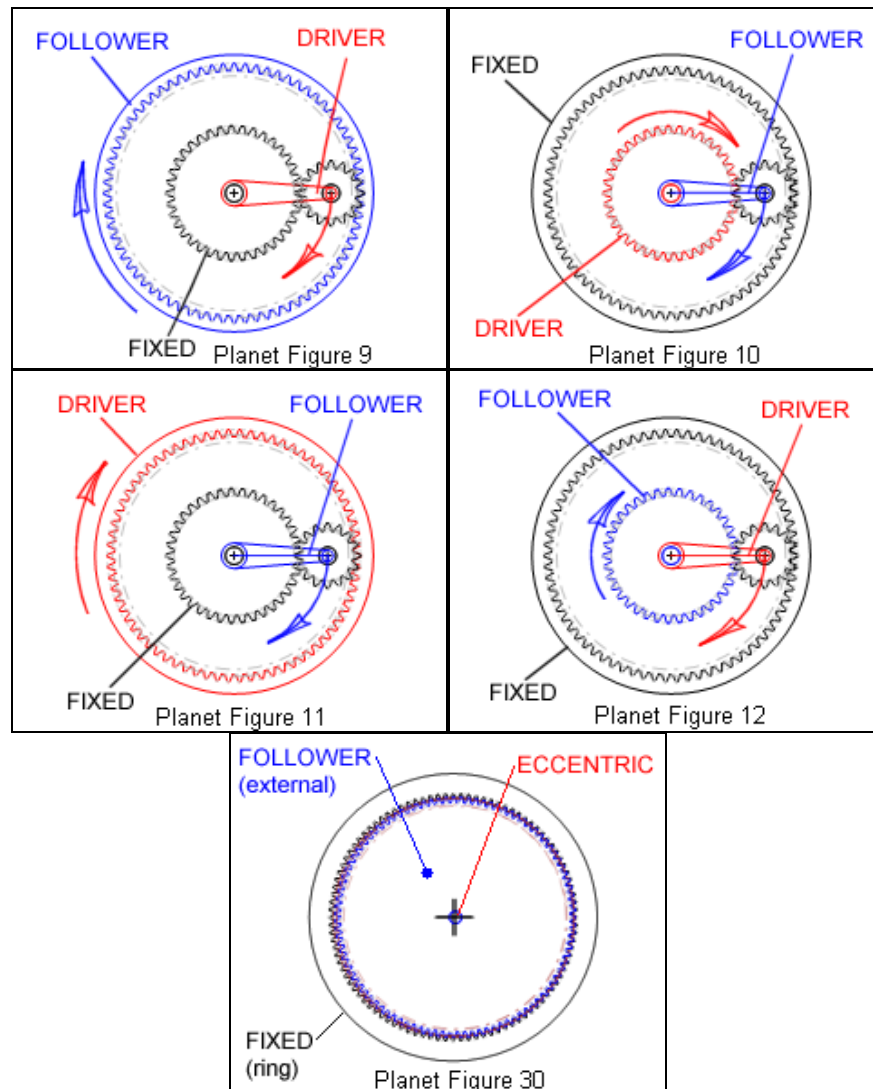
responsible for assuring that the number of teeth is appropriate if any one component's number of teeth is changed.

The **Ring Plus** option will add the number specified to the number of teeth of the ring.

Individual planetary components cannot be opened in GearTeq unless they are in the original assembly.



Standard Planet Layouts Available in GearTeq



Standard Planet Layouts Available in GearTeq

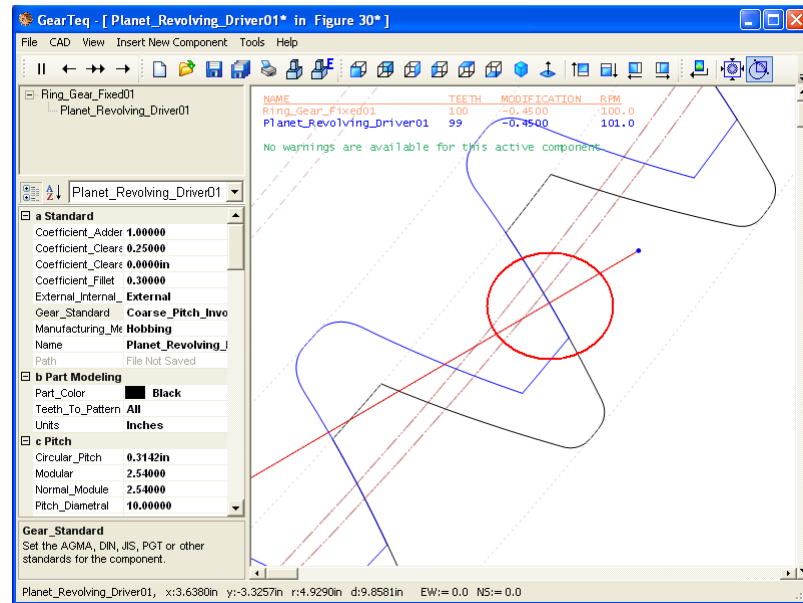
Cycloid Gear Sets

Cycloidal gear sets are very high ratio assemblies. Ratios between 30:1 and 350:1 are obtainable.

To create a cycloid (cycloidal) gear set, use the Add Planetary Set module. Select 'Figure 30' which will create an internal and external gear set. The default values are for a set with a ratio of 99:1. The number of teeth for the ring gear is 100 and the external gear is 99. The Ring plus is set to 0 and cannot be changed. The number of planets is set to 1 and should not be changed.

The default addendum modification coefficient is -0.450 for both components. After accepting the settings 'Zoom Area' the tooth

mesh at the 3:00 position. You will notice that the teeth are not in contact. This is NORMAL.



Zoom the area where the line of action intersects the teeth. Notice the contact for the gear teeth. The corresponding contact for the other side of the tooth is vertical of this position. The default backlash is 0.000, which can be changed to suit the application.

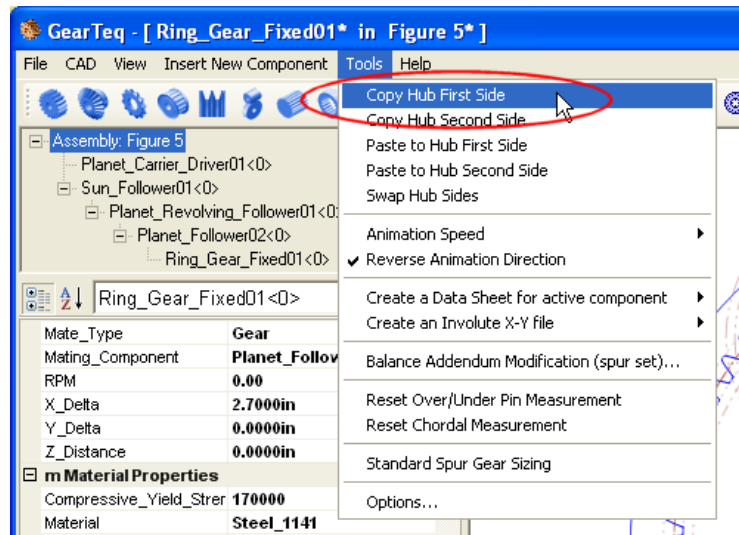
The carrier plate is created by the CAD system. The offset of the eccentric should be equal to the center distance displayed for the external component. It is up to the designer to select the best method to tie the external component to an output device, such as a plate. This plate may contain a number of protrusions, such as cam followers, which roll in the same number of holes in the external component. The size of these holes should be equal to the diameter of the follower plus twice the eccentric dimension.

The output plate is concentric to the input of the eccentric and ring gear.

Tools

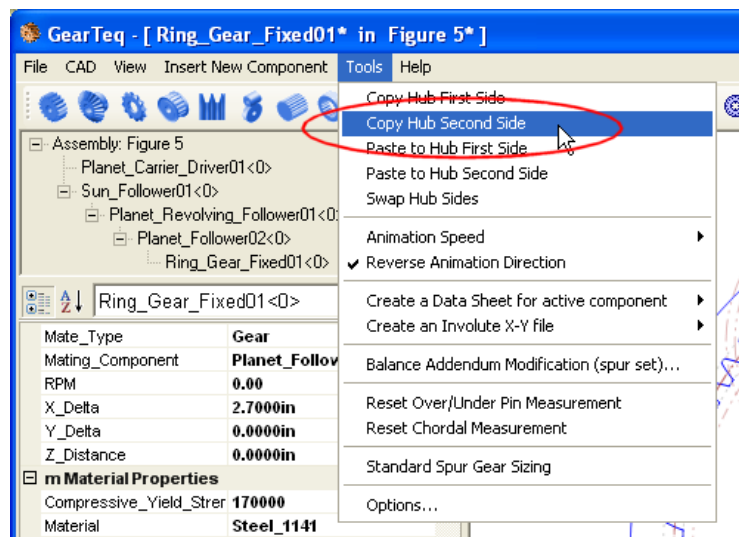
Copy Hub First Side

Clicking 'Copy Hub First Side' copies the hub first side data to the copy hub clipboard for pasting to the second side hub or to either side of other components.



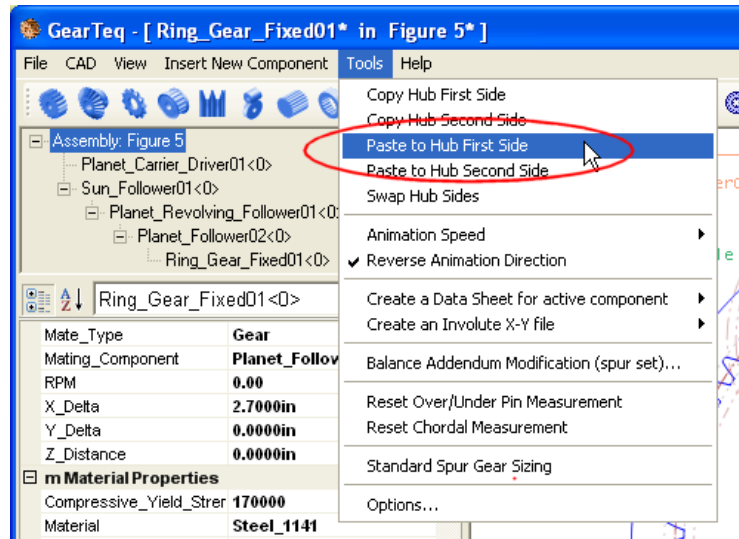
Copy Hub Second Side

Clicking 'Copy Hub Second Side' copies the hub second side data to the copy hub clipboard for pasting to the first side hub or to either side of other components.



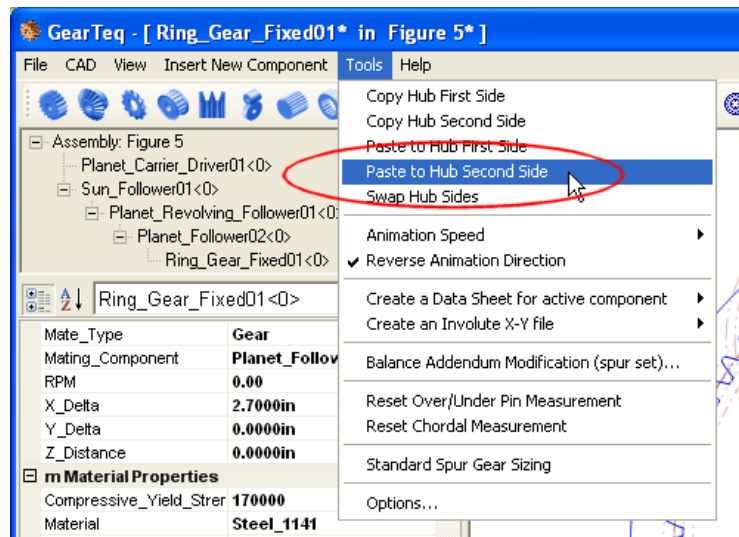
Paste to Hub First Side

Clicking 'Paste to Hub First Side' pastes the data stored in the hub copy clipboard to the hub first side of the current active component.



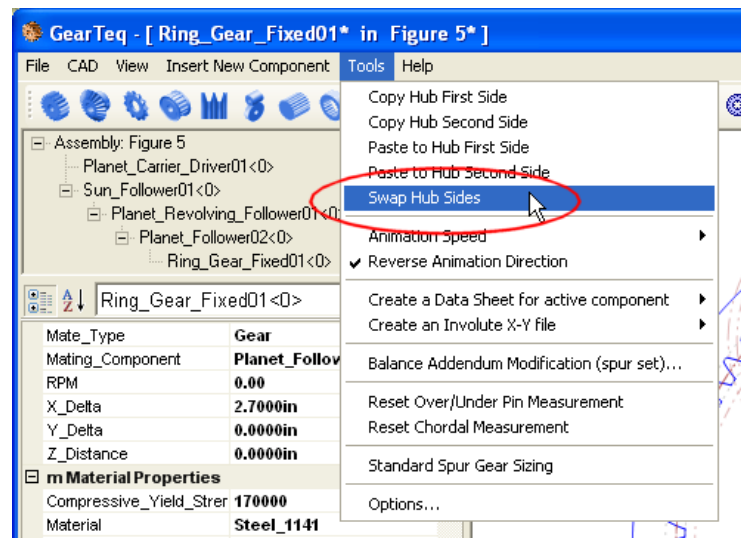
Paste to Hub Second Side

Clicking 'Paste to Hub Second Side' pastes the data stored in the hub copy clipboard to the hub second side of the current active component.



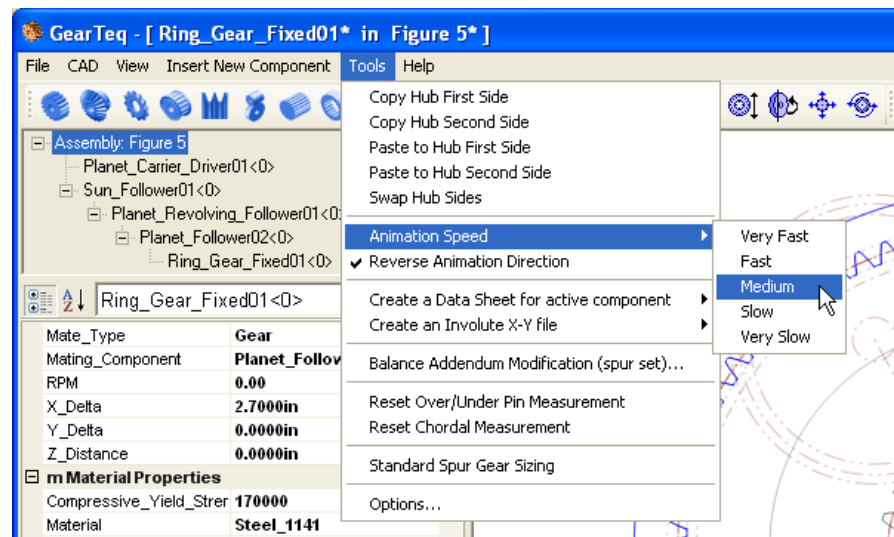
Swap Hub Sides

Clicking 'Swap Hub Sides' swaps the hub data of the two sides for the current active component.



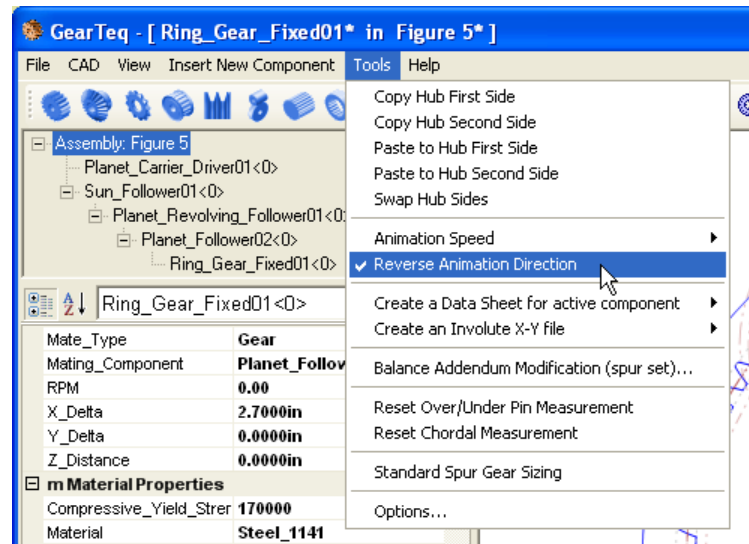
Animation Speed

Animation Speed sets the animation speed for the simulation. The slower the speed the finer the angular steps in the simulation. Select between Very Fast, Fast, Medium, Slow and Very Slow.



Reverse Animation Direction

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



Balance Addendum Modification...

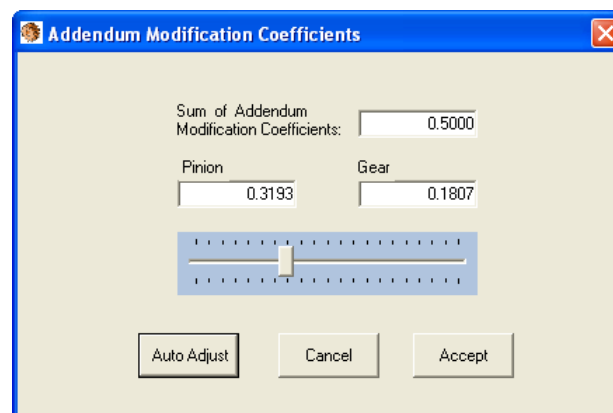
The Balance Addendum Modification menu option opens a dialog window that can adjust the addendum modification for the active gear and its mate. The mate must be a spur gear with a gear mate.

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:

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

where u is the gear ratio

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.

Create a Data Sheet for active component

Excel file

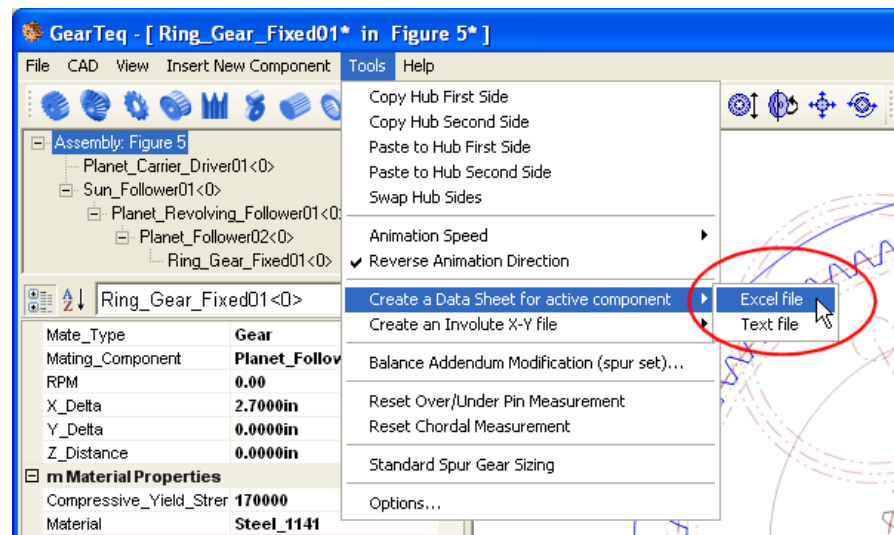
Clicking 'Excel file' creates a data sheet of the active component in Microsoft Excel, if available.

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

File Edit Format View Help

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
ø	3.175000		Modular Pitch
ø	25.0000	deg	Normal Pressure Angle
ø	25.0000	deg	Pressure Angle
mg	0.0000	deg	Helix Angle
C	0.3929		Ratio, i:x
MA	2.4516	in	Center Distance
MR	0.2368	in	Approach Length
mp	0.2436	in	Recess Length
Np	1.3577		Contact Ratio
Op	11		Number of Teeth
Opn	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
	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
	0.20967	in	Tooth Thickness, Normal
dw	0.7500	in	Face Width
M			Size Between Pins
	0.2160	in	Pin Diameter
	1.6794	in	Measurement Over Pins
	1.6710	in	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

Create an Involute X-Y file

This option is only available for spur gears and splines.

Excel file

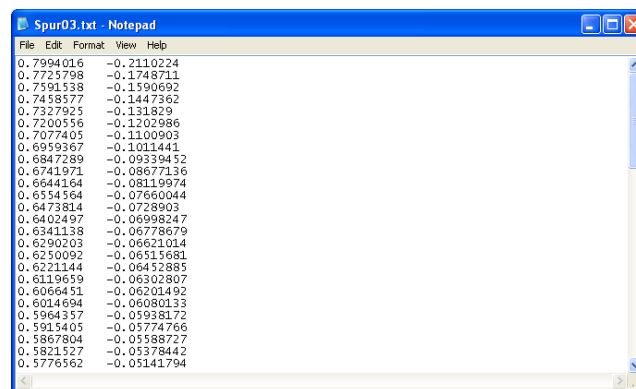
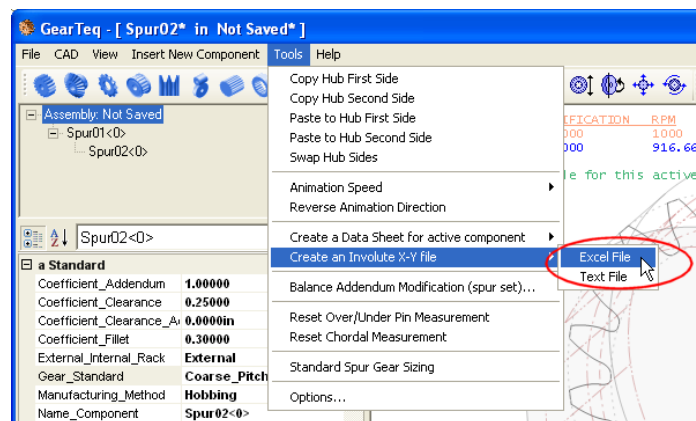
Clicking 'Excel file' creates an X-Y file of the points along the involute tooth in Microsoft Excel, if available.

Text file

Clicking 'Text file' creates an X-Y file of the points along the involute tooth in Microsoft Notepad, if available.

Comma Separated Values (CSV) file

Clicking 'Comma Separated Values' creates an XYZ file of the points along the involute tooth in Microsoft Notepad, if available. The Z column will have a 0.000 value.



Sample of an XY file created as a text file

Create a DXF File

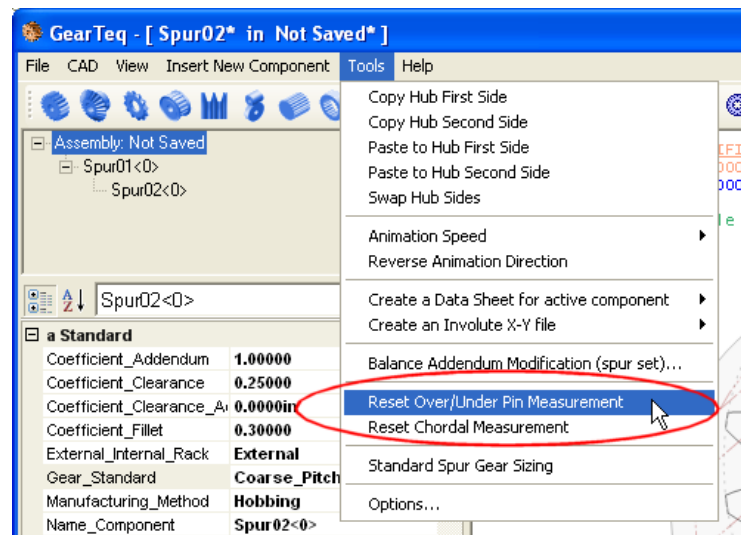
Depending on the type of active component, the Create a DXF file menu command can create a DXF file of the revolve geometry or the tooth cut entities. A message box will allow the user to open the file in the default DXF program.

Create a XYZ Surface File

GearTeq can create a XYZ surface file of points in 3d space that are used to create the loft tooth cuts of spur/helical, bevel and worm wheels. For spur/helical gears, a crowning drop must be specified otherwise the tooth cut is a simple extrusion.

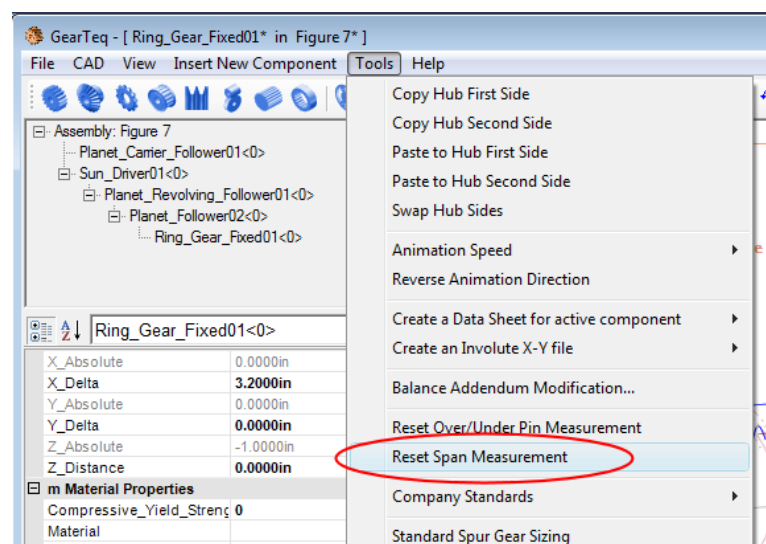
Reset Over/Under Pin Measurement

Clicking 'Reset Over/Under Pin Measurement' resets the over/under pin diameter if the user has manually changed the value. The over/under measurements are updated accordingly.



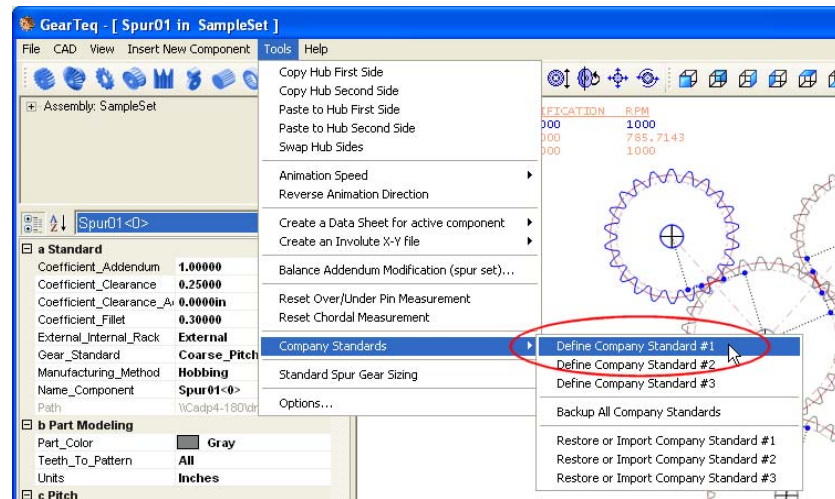
Reset Span Measurement

Clicking 'Reset Span Measurement' resets the number of teeth to gage over if the user has manually changed the value. The span measurement is updated accordingly.

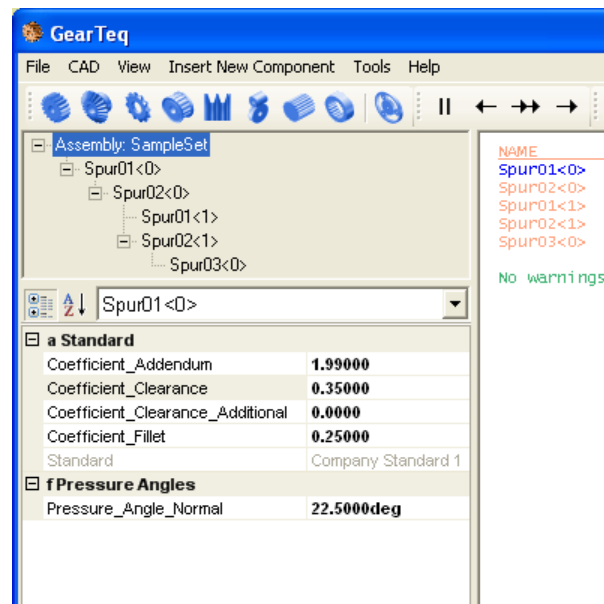


Company Standards

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

$\text{addendum} = \text{coefficient addendum} / \text{diametral pitch}$
(inches)

$\text{addendum} = \text{coefficient addendum} * \text{module (millimeters)}$

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.

$\text{dedendum} = (\text{coefficient addendum} + \text{coefficient clearance}) / \text{diametral pitch (inches)}$

$\text{dedendum} = (\text{coefficient addendum} + \text{coefficient clearance}) * \text{module (millimeters)}$

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.

$\text{The hob tip fillet radius} = \text{coefficient fillet} / \text{diametral pitch}$
(inches)

$\text{The hob tip fillet radius} = \text{coefficient fillet} * \text{module}$
(millimeters)

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

Standard

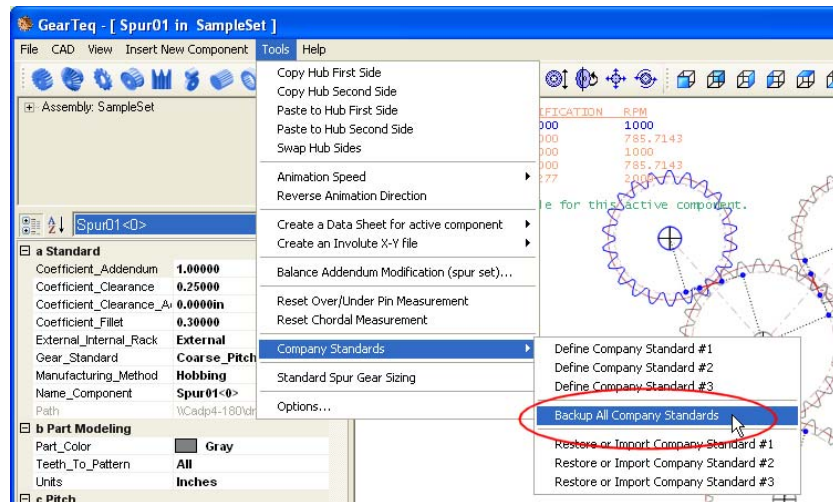
Standard is a read only display of which standard is being edited.

Pressure Angle Normal

Pressure Angle Normal defines the pressure angle for the standard.

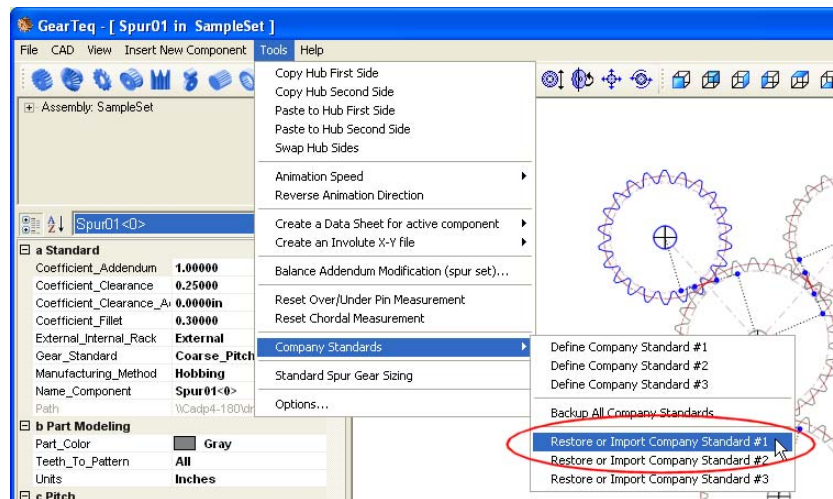
Backup All Company Standards

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 GearTeq. This can also be used to store the standards so that other users can have access to them.



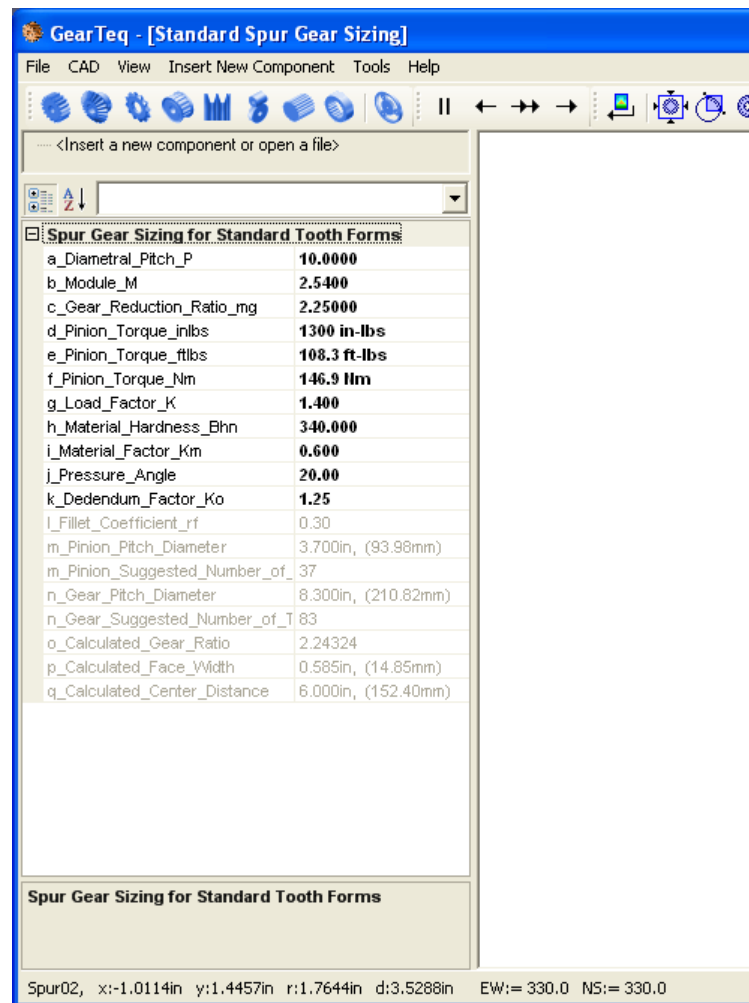
Restore or Import Company Standards

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



Standard Spur Gear Sizing

Standard Spur Gear Sizing loads the spur gear sizing module into the property grid.



The user input parameters are:

- Diametral pitch P
- Module M
- Gear reduction ratio mg, must be greater or equal to 1.0
- d, e, f) Pinion torque, in-lbs, ft-lbs or Nm
- Load factor K
- Material hardness Bhn, Brinell hardness number, the material must be steel
- Material factor Km, 0.6 for through-hardening steels, 0.9 for case-hardening steels
- 20 or 25 degree pressure angle
- Dedendum factor Ko, values of 1.25 or 1.35 are allowed
- Fillet coefficient rf, Assumed to be 0.30 or 0.21 for 25 degree pressure angle with 1.35 dedendum factor gears

The calculated values are:

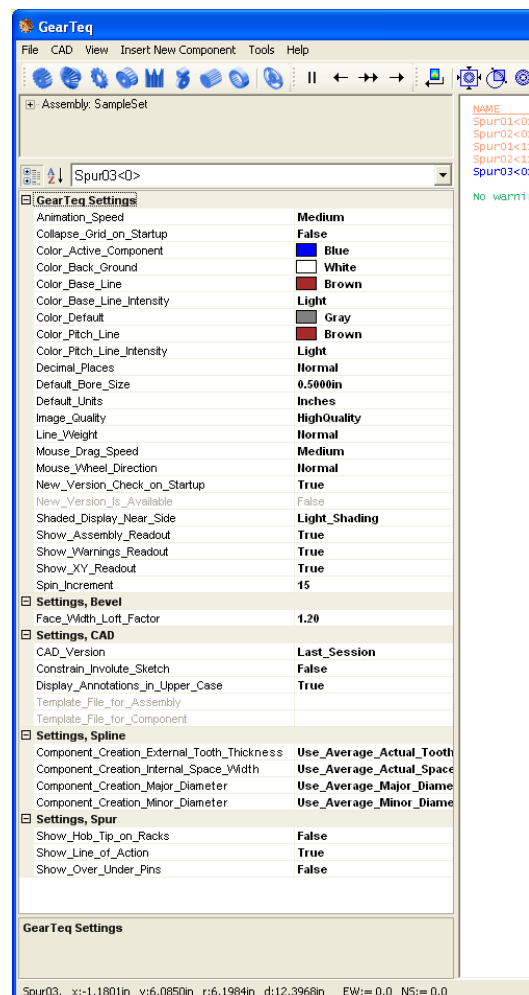
- m) Pinion Pitch Diameter
- m) Pinion, suggested number of teeth
- n) Gear Pitch Diameter
- n) Gear, suggested number of teeth
- o) Calculated Gear Ratio
- p) Calculated Face Width
- q) Calculated Center Distance

Open install folder using Explorer

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

Options...

The Options settings are made available through the GearTeq menu item Tools>Options.



Help

GearTeq Help Topics

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

GearTeq Quick Start Guide

Clicking 'GearTeq Quick Start Guide' opens a PDF of the quick start guide. Adobe Acrobat must be installed on the computer.

Registration...

Clicking 'Registration' loads the registration information into the property grid.

Create Email with registration information

Clicking 'Create Email with registration information' opens the default email with the information specified.

License

Clicking 'License' opens Notepad with the license agreement.

Check for Updates

Check for Updates will check the Camnetics' web site for a newer version of GearTeq. A message box will tell the user whether a newer version is available or not. The new version will not be downloaded or installed.

About GearTeq...

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

Options

The Options settings are made available through the GearTeq menu item Tools>Options.

GearTeq Settings

Animation Speed

Animation Speed sets the animation speed for the simulation. Slower speed settings will produce finer steps in the simulation. Select between Very Fast, Fast, Medium, Slow and Very Slow. This value is saved as a default for the next time GearTeq is started.

Collapse Grid on Startup

Collapse Grid on Startup defines whether the property grid should be collapsed on startup. This value is saved as a default for the next time GearTeq is started.

Color Active Component

Color Active Component defines the color of the active component. If the color is set to transparent, then the component color will be used. This value is saved as a default for the next time GearTeq is started.

Color Back Ground

Color Back Ground defines the color to be used as the background on the GearTeq drawing window. This value is saved as a default for the next time GearTeq is started.

Color Base Line

Color Base Line defines the color used to display the base line in the GearTeq window. This value is saved as a default for the next time GearTeq is started.

Color Base Line Intensity

Color Base Line Intensity defines the intensity of the color used to display the base line in the GearTeq window. This value is saved as a default for the next time GearTeq is started.

Color Default

Color Default defines the color of a new component. This value is saved as a default for the next time GearTeq is started.

Color Pitch Line

Color Pitch Line defines the color used to display the pitch line in the GearTeq window. This value is saved as a default for the next time GearTeq is started.

Color Pitch Line Intensity

Color Pitch Line Intensity defines the intensity of the color used to display the pitch line in the GearTeq window. This value is saved as a default for the next time GearTeq is started.

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 Bore Size

Default Bore Size defines the default diameter size of the bore for a new GearTeq component. This value is saved as a default for the next time GearTeq is started.

Default Units

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

Image Quality

Image Quality defines the image quality of the components that are displayed in the GearTeq window. This value is saved as a default for the next time GearTeq is started. The image quality options are:

- High Speed
- High Quality
- Anti Alias

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

Line Weight

Line Weight defines the thickness of the lines used to draw the components in the GearTeq window. This value is saved as a default for the next time GearTeq is started.

Mouse Drag Speed

Mouse Drag Speed defines the impact of mouse movements in conjunction with the Drive and Rotate the View toolbar controls. This value is saved as a default for the next time GearTeq is started.

Mouse Wheel Direction

Mouse Wheel Direction determines the zoom direction when zooming in or out in the GearTeq display window.

Shaded Display Near Side

Shaded Display Near Side defines the intensity of shading, if any, on the near side surface of the components in the GearTeq window. This value is saved as a default for the next time GearTeq is started.

Show Alerts Readout

Show Alerts Readout defines if the alert text will be displayed in the upper left hand corner of the drawing screen. This value is saved as a default for the next time GearTeq is started.

Show Assembly Readout

Show Assembly Readout defines if the assembly list will be displayed in the upper left hand corner of the drawing screen. This value is saved as a default for the next time GearTeq is started.

Show XY Readout

Show XY Readout defines if the XY readout will be displayed in the status bar at the bottom of the GearTeq window. This value is saved as a default for the next time GearTeq is started.

Spin Increment

Spin Increments defines the angle of increment when clicking on the simulation toolbar's forward and back buttons. Integers between 1 and 30 are valid. This value is saved as a default for the next time GearTeq is started.

Settings, Bevel**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.

Settings, CAD

CAD Version

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

Constrain Involute Sketch

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

Display Annotations in Upper Case

Display Annotations in Upper Case determines if annotations created in CAD are all upper case or a mix of case. This value is saved as a default for the next time GearTeq 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 GearTeq menu item CAD>Assembly Template File... This value is saved as a default for the next time GearTeq 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 GearTeq menu item CAD>Component Template File... This value is saved as a default for the next time GearTeq 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 GearTeq is started.

Settings, Spline

Component Creation, External Tooth Thickness

Component Creation External Tooth Thickness defines how to create the tooth thickness of external splines when creating a CAD model. This value is saved as a default for the next time GearTeq is started.

Component Creation, Internal Space Width

Component Creation, Internal Space Width defines how to create the space width of internal splines when creating a CAD model. This value is saved as a default for the next time GearTeq is started.

Component Creation, Major Diameter

Component Creation, Major Diameter defines how to create the major diameter of splines when creating a CAD model. This value is saved as a default for the next time GearTeq is started.

Component Creation, Minor Diameter

Component Creation, Minor Diameter defines how to create the minor diameter of splines when creating a CAD model. This value is saved as a default for the next time GearTeq is started.

Create Expanded Data Sheets

Spline data sheets created with the Create Expanded Data Sheets option set to true will include additional properties not normally included with a spline data sheet. Additional data included:

- Addendum, dedendum and fillet coefficients
- Addendum and dedendum
- Fillet radius
- Circular pitch
- Face width

This value is saved as a default for the next time GearTeq is started.

Settings, Spur

Show Hob Tip on Racks

Show Hob Tip on Racks defines if the hob tips should be displayed on all racks. This value is saved as a default for the next time GearTeq is started.

Show Line of Action

Show Line of Action defines if the line of action should be displayed on spur gear pairs with gear mates. This value is saved as a default for the next time GearTeq is started.

Show Over Under Pins

Currently this is unused. This value is saved as a default for the next time GearTeq is started.

Assemblies

Assemblies contain a collection of assembly positions (AP) and planetary information should the assembly consist of a planetary gear set. Each Assembly has two categories, General and Planetary Gearing Data.

General Category

Name Assembly

The Name Assembly property contains the name of the assembly and is also used for the file name.

Path Assembly

Path Assembly displays the computer path for the assembly file.

Planetary Gearing Data

Mode Planet

Planet Mode as defined in Machinery's Handbook. Modes 1, 2, 5 through 12 are available. This is a read only property that can be set only by using the insert New Planetary Set wizard.

Number of Planets

Number of Planets defines the number of planets in the set. There are a finite number of positions for planets running inside of ring gears. The possible positions equal the number of teeth in the ring and the sun gear. GearTeq displays the angular positions of the planet to the closest position evenly divided in 360 degrees.

Planet 1

Every planetary set has a planet 1 component. This is a read only property created in the Insert New Planetary Set wizard.

Planet 2

Only planetary arrangements 5 through 8 have a second planet component. This is a read only property created in the Insert New Planetary Set wizard.

Planet Carrier

Every planetary gear set has a planet carrier.

Planet Positions

Planet Positions displays the angular positions of the planets to the closest position evenly divided in 360 degrees. There are a finite number of positions for planets running inside of ring gears. The possible positions equal the number of teeth in the ring and the sun gear. This is a read only property that is updated if the number of

teeth of any of the planet components or the number of planets change.

Ring

Every planetary set has a ring component except arrangement 1. This is a read only property created in the Insert New Planetary Set wizard.

Sun

Every planetary set has a sun component except arrangements 2 and 30. This is a read only property created in the Insert New Planetary Set wizard.

Assembly Positions (AP)

Assembly positions (AP) determine the location and type of mating for each of the components. The APs also contain information about mating sets of components. For example, the center distance will be displayed for a set of spur gears. An assembly may contain any number of APs with each AP specifying the same gear component. Each AP could specify a different method for controlling the center distance.

There are two ways to activate an AP and have its properties displayed in the property grid. The APs are listed in the component select drop down box that is just above the property grid. Each AP is preceded by a <XYZ> designation. Drop down the list and click on the desired AP. The second method for selecting an AP is to click on the AP listed in the assembly tree view, then right click on the item to drop down a menu, which will allow the selection of the assembly, component or the AP.

Each AP has a set of categories. What categories are used depends on the type of component that is selected for that AP.

a General - Assembly Position **Component at this Location**

Click this box to open a drop down list of the components in the assembly.

Name Position

The name of this position used in the Component Selector and the Assembly Tree. Other positions can also specify this position as their Assembly Mating Position.

b Mating Component **Gear Ratio**

Gear Ratio is the rotational ratio between a parent component and the active component.

Mate Angle

Mate Angle is the angle of the part to its gear mated parent in the transverse plane. The angle is counter clockwise starting at the 3:00 position.

Mate Type

Click Mate Type to open a drop down list of the type of mates available. This specifies the type of mate a part has to its parent part. Each part can have only one parent mate while a part may have a number of child mates.

Mating Assembly Position

Mating Assembly Position is a user selectable drop down list of all the APs. The current AP is on the list but cannot be selected. Also, any AP that would result in a circular mating cannot be selected.

RPM

RPM is the revolutions per minute of the active component.

X Absolute

X Absolute is the distance between the X origin of the current component and the X origin of the assembly.

X Delta

X Delta defines the X distance from the mating component. Entering a value will react differently depending on the type of component and the type of mating relationship it has to its parent component. For example, changing the value of a spur gear with a gear relationship to its parent spur gear will also change the Y delta. The value entered must be less than the center distance for the pair. Changing the value for a belt pulley with a belt relationship to its parent belt pulley will not change the Y delta but will change the center distance.

Y Absolute

Y Absolute is the distance between the Y origin of the current component and the Y origin of the assembly.

Y Delta

Y Delta defines the Y distance from the mating component. Entering a value will react differently depending on the type of component and the type of mating relationship it has to its parent component. For example, changing the value of a spur gear with a gear relationship to its parent spur gear will also change the X delta. The value entered must be less than the center distance for the pair. Changing the value for a belt pulley with a belt relationship to its parent belt pulley will not change the X delta but will change the center distance.

Z Absolute

Z Absolute is the distance between the current component and its parent component on the transverse plane. Normally this is zero, as for a pair of spur gears whose faces are running on center with each other.

Z Delta

Z Delta is the distance between the Z origin of the current component and the Z origin of the assembly.

Belt Pulley**Center Distance Belt Pulley**

Center Distance Belt Pulley is the distance between the centers of two belt mated pulleys.

Vee Belt Length

Vee Belt Length is the length of the belt along the pitch line of this component and its mating belt pulley parent component. Idlers and other belt pulley components are not taken into consideration.

Bevel Gears**Sum of Pitch Angles**

Sum of Pitch Angles defines the sum of the pitch angles for this component and its parent component.

Chain Sprocket**Center Distance Chain**

Center Distance Chain is the distance between the centers of two chain mated sprockets.

Chain Length

Chain Length is the length of the chain along the pitch line of this component and its mating chain sprocket parent component. Idlers and other chain sprocket components are not taken into consideration.

Even Full Pitches

Even Full Pitches defines the number of full pitches in the length of chain.

Gear Belt**Center Distance Gear Belt**

Center Distance Gear Belt is the distance between the centers of two belt mated pulleys.

Gear Belt Length

Gear Belt Length defines the simple length of a belt between two gear belt pulleys. Entering a value will change the center distance. Changing the center distance will change this value.

Spline Tolerance ANSI

Clearance Actual Maximum

The actual clearance maximum is based on the size and class of the spline. The mate of the component must be a complimentary spline with a shaft mate for values other than 0 to be displayed.

Clearance Actual Minimum

The actual clearance minimum is based on the size and class of the spline. The mate of the component must be a complimentary spline with a shaft mate for values other than 0 to be displayed.

Clearance Effective Maximum

The effective clearance maximum is based on the size and class of the spline. The mate of the component must be a complimentary spline with a shaft mate for values other than 0 to be displayed.

Clearance Effective Minimum

The effective clearance minimum is based on the size and class of the spline. The mate of the component must be a complimentary spline with a shaft mate for values other than 0 to be displayed.

Spur

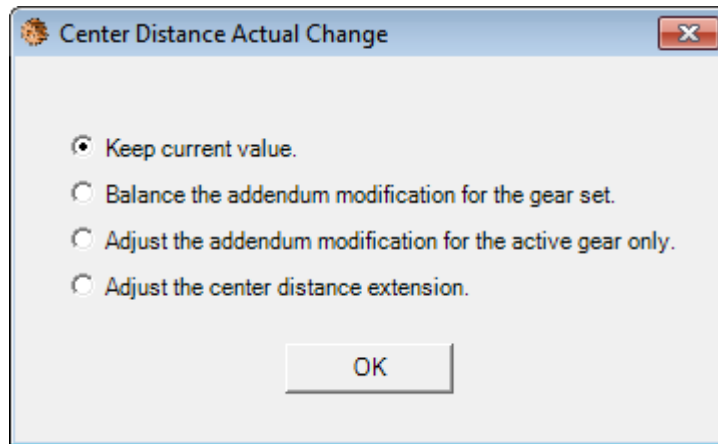
This section is for a spur and its gear mate.

Center Distance Actual

Center Distance Actual is the distance between this component and the parent after any changes.

If the center distance locked property is set to true, then the value entered will be used whether it is valid or not.

If the active component is a spur gear with a gear mate to another spur gear then changing the center distance will allow the user to change the addendum modification of the active component and the mating component. The following dialog box is displayed:



Select the "Keep current value" option to keep the value before the new value was entered.

Select the "Balance the addendum modification for the gear set" option change the addendum modification coefficient for the active component and its mating gear by the following formula:

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

where u is the gear ratio

x1 and x2 are the addendum modification coefficients of the pinion and gear, respectively

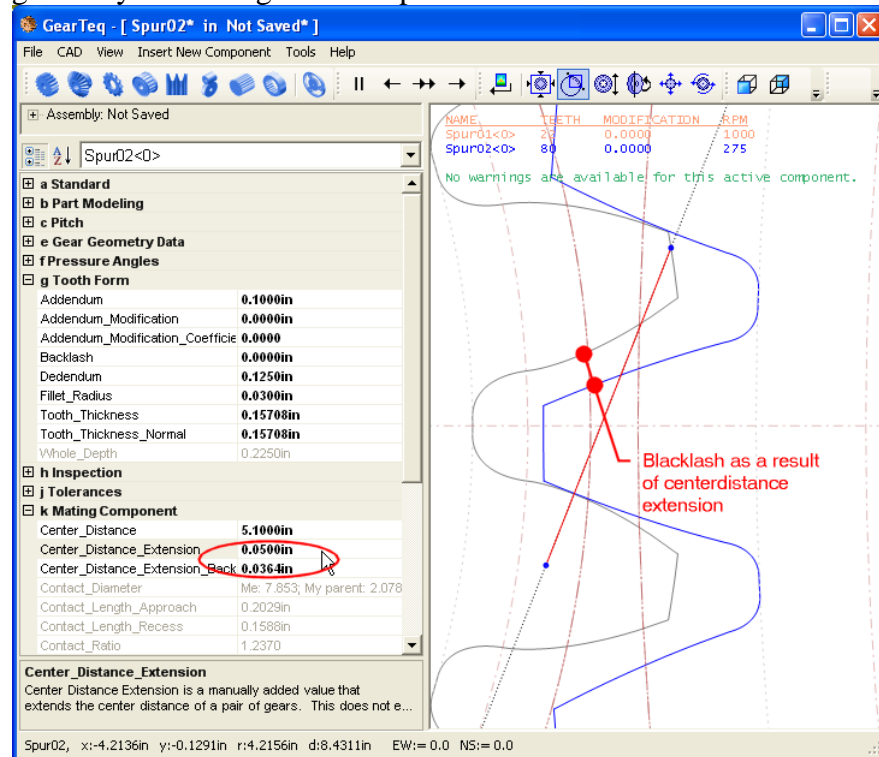
Select the "Adjust the addendum modification for the active gear only" option to change only the addendum modification of the active component.

Select the "Adjust the center distance extension" option to change the center distance extension. The center distance actual value will also change accordingly. No changes are made to the geometry of either gear, only the position on their assembly.

Tip: Change the center distance and accept the addendum modification to the active component. Then use the Tools>Balance Addendum Modification to redistribute the modification between the gear set (pair).

Center Distance Extension

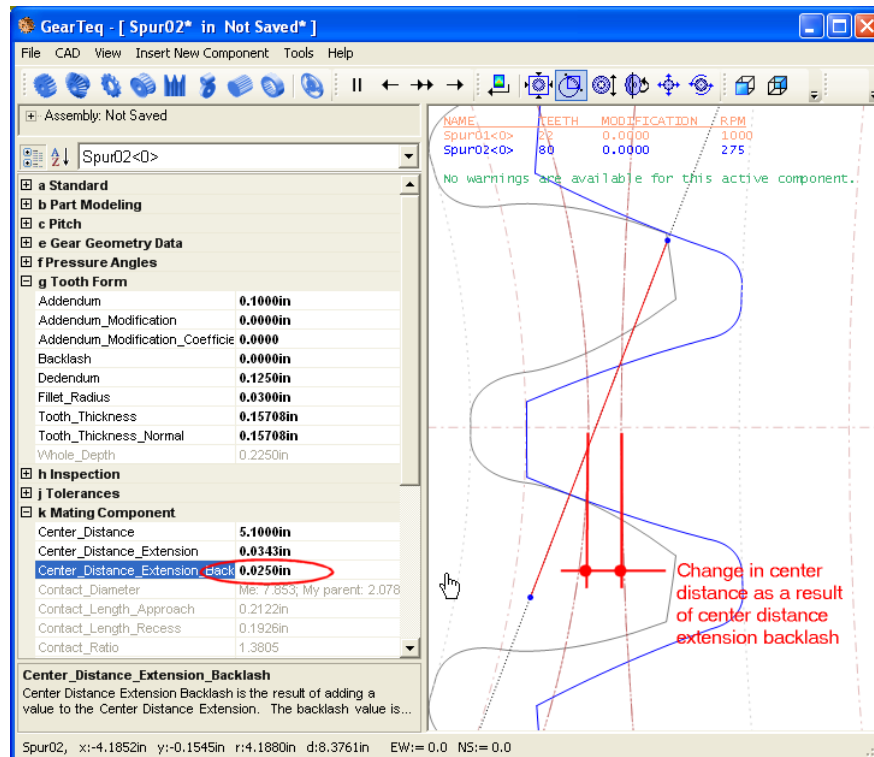
Center Distance Extension is a manually added value that extends the center distance of a pair of gears. This does not affect the geometry of either gear in the pair.



The increase in backlash is a result of the enter distance extension. The value displayed for the center distance extension backlash is approximate.

Center Distance Extension Backlash

The Center Distance Extension Backlash is a manually added value that increases the backlash on a set of gears without affecting the individual gear geometry. The change in the center distance is approximate relative to the change in backlash.



Center Distance Locked

Center Distance Locked is a true or false value determining if the center distance should be calculated or fixed at a user input value as specified in the Center Distance Actual property.

Center Distance Standard

Center Distance Standard is the calculated distance between this component and its parent after any addendum modification and before any center distance extension.

Contact Length Approach

Contact Length Approach is the approach path of contact in involute gears. It is a straight line passing through the pitch point and the contact diameter of the active component.

Contact Length Recess

Contact Length Recess is the recess path of contact in involute gears. It is a straight line passing through the pitch point and the contact diameter of the mating component.

Contact Ratio

Contact ratio is the ratio of the arc of action to the circular pitch.

Contact Ratio Face

Contact Ratio Face is the face contact ratio for a helical gear set. A value is displayed for a helical gear set when the parent component is a helical gear with a gear mate.

Contact Ratio Total

Contact Ratio Total is the total contact ratio for a helical gear set. The total contact ratio equals the contact ratio plus the contact face ratio. A value is displayed for a helical gear set when the parent component is a helical gear with a gear mate.

Diameter, Start of Active Profile

Start of Active Profile diameter is the diameter at which a gear comes in to contact with its mating gear. The active profile is displayed for the current gear and its parent mate.

Efficiency

The Efficiency is valid for a simple set of spur or helical gears, internal or external. The mating component must be a gear with a gear mate.

Flip Backlash

Flip Backlash is a true or false property that allows the user to show the backlash of a set of gears on the other side of the tooth contact.

Flip Line of Action

Flip Line of Action is a true or false property that allows the user to show the line of action of a set of gears on the other side of the tooth contact.

Friction Coefficient

The Coefficient of Friction is valid for a simple set of spur or helical gears. The mating component must be a gear with a gear mate. The user should provide a valid value between 0 and 1.000. The default value is 0.0000, which calculates to an efficiency of 100%.

Gear Type

Gear Type is a read only value and is used internally in GearTeq for positioning the gears when a planetary system is being used.

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.

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.

Pressure Angle Working

Pressure Angle Working is the angle of the tooth at the pitch diameter. This is normally the same as pressure angle unless either gear in the set has been modified or the center distance has been changed.

Working Depth

Working Depth is the length of the tooth that engages the mating gear.

Spur Gears

Addendum

Addendum is the length of the tooth from the pitch diameter to the major diameter. This value cannot be changed directly unless GearTeq 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 GearTeq 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 GearTeq 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.

Backlash Arc Minute

Backlash Arc Minute is the angular value in minutes (1/60 of a degree) of the backlash value.

Blank OD of Internal Gear

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

Chordal Tooth Height

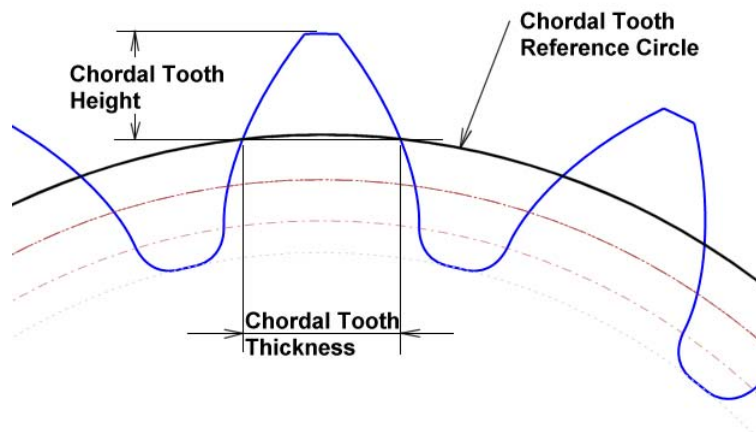
Chordal Tooth Height is the length of the tooth portion to be measured at which the chordal tooth thickness is calculated. Twice the chordal tooth height plus the chordal tooth reference circle should equal the outside diameter of the gear.

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.

Circular Pitch

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

Code Tooth Thickness

Code Tooth Thickness sets the tooth thickness code class for this component.

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, Clearance Additional

Coefficient, Clearance Additional is a linear value (not a coefficient) that is added to the dedendum to increase the amount of clearance between the tooth tip of the mating gear and the root of this gear. This is used with the AGMA fine pitch standards only.

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 Diameter

Contact Diameter is the diameter where the mating gear tip first makes contact with the component.

Contact Length, Approach

Contact Length, Approach and the recess length equal the total contact for a pair of gear teeth.

Contact Length, Recess

Contact Length, Recess and the approach length equal the total contact for a pair of gear teeth.

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.

Crowning

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.

Crowning Drop

Crowning Drop is the amount of change to the tooth thickness on one side of the tooth at the faces. Changing this value will change the crowning radius value. Changing the face width will not affect this value.

Crowning Radius

Crowning Radius is the radius used to create the drop to achieve the specified drop. Changing this value will change the drop value. Changing the face width will affect this value.

Crowning Sketches

Crowning Sketches is the number of sketches used in CAD to create the tooth loft cut. The number of sketches is limited to at least 5 and no more than 21. The sketches are evenly distributed along the face width. An odd number of sketches is recommended so a sketch is placed at the center of the gear. If half radial or half tapered crowning type is used it is recommended to have 9 or 17 sketches so a sketch is at the start of the crowning transition.

Crowning Type

Crowning Type defines the type of crowning.

- None, no crowning is used.
- Full Radial, the full length of the tooth is modified. If an odd number of sketches is used, the center sketch will have no modification.
- Half Radial, the center 1/2 of the tooth has no modification. Using at least 9 sketches is recommended with the half radial type. Using 9 or 17 sketches places one sketch at the transition at the start of the half radial crowning.
- Half Tapered, the center 1/2 of the tooth has no modification. Using at least 9 sketches is recommended with the half tapered type. Using 9 or 17 sketches places one sketch at the transition at the start of the half tapered crowning.

Dedendum

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

Diameter, Base

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

Diameter, Base Normal

Diameter, Base Normal is a diameter that is tangent to the normal pressure angle.

Diameter, Major

Major Diameter is the outside diameter of a gear.

Diameter, Minor

Minor Diameter is the root diameter of a gear.

Diameter, Pitch

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.

Diameter, Pitch, Normal

Normal Pitch Diameter is the theoretical diameter of the gear normal to the cutter.

Diameter, Pitch, 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.

Diameter, True Involute Form

The true involute form (TIF) diameter is the smallest diameter of the involute curve.

External Internal Rack

External Internal Rack defines the component as an external gear, internal gear, a rack or a face gear.

Face Gears

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

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.

Helix Angle

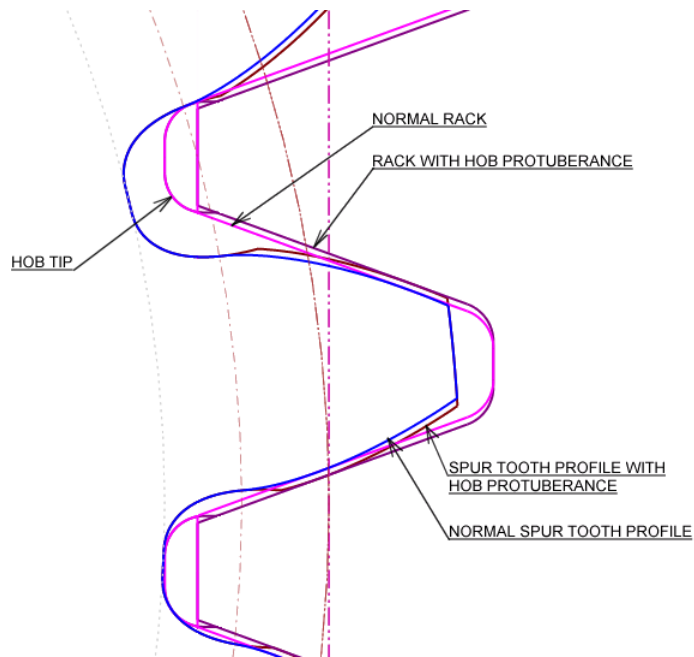
Helix Angle is the angle of the tooth from the shaft. A zero angle would be parallel to the shaft and would define a spur gear. An angle other than zero would define a helical gear.

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



Keyway

Keyway defines if a standard keyway is to be created. Select none or the standard. The key size is determined by the bore size. The part must have a valid bore for the keyway to be created.

Keyway Position

Keyway Position defines the angular position of the keyway relative to the Tooth Center or the Space Center.

Lead

Lead of the tooth helix. Lead is equal to the pitch diameter multiplied by PI times the Cotangent of the helix angle.

Manufacturing Method

Manufacturing Method defines the method to be 'used' for manufacturing the tooth cut. Select Hobbing or Full Fillet Radius.

Mate Angle

Mate Angle is the angle of the part to its gear mated parent in the transverse plane. The angle is counter clockwise starting at the 3:00 position.

Max Runout

Runout is the maximum variation of the distance between a surface of revolution and a datum surface, measured perpendicularly to the datum surface, per AGMA 2000-A88.

Measurement Over/Under 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. GearTeq 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.

Measurement Over/Under Pins Minimum

Measurement Over/Under 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.

Modification Profile APL Amount

Modification Profile APL Amount is the amount of modification to the involute profile above the pitch line. The modification can be linear or parabolic.

Modification Profile APL Diameter

Modification Profile APL Diameter is the diameter above the pitch line at which the profile modification starts. The value must be greater than or equal to the pitch diameter.

Modification Profile APL Length

Modification Profile APL Length is the length of the modification above the pitch line. The length is limited to the length of the addendum. It is prohibited from being within the pitch diameter. The modification can be linear or parabolic.

Modification Profile APL Type

Modification Profile APL Type is the type of modification to the involute profile above the pitch line. Choose between None, Linear and Parabolic.

Modification Profile BPL Amount

Modification Profile BPL Amount is the amount of modification to the involute profile below the pitch line. The modification can be linear or parabolic.

Modification Profile BPL Diameter

Modification Profile BPL Diameter is the diameter below the pitch line at which the profile modification starts. The value must be less than or equal to the pitch diameter.

Modification Profile BPL Length

Modification Profile BPL Length is the length of the modification below the pitch line. The length is limited to the length of the dedendum. It is

prohibited from being outside of the pitch diameter. The modification can be linear or parabolic.

Modification Profile BPL Type

Modification Profile BPL Type is the type of modification to the involute profile below the pitch line. Choose between None, Linear and Parabolic.

Module (displayed as Modular)

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.
 $\text{Module} = 25.4 / \text{diametral pitch}$.

Mounting Bushing Modeling

Mounting Bushing Modeling specifies if the bushing should be created as a separate part or as part of the component.

Mounting Bushing Side

Mounting Bushing Side specifies on which side of the component the bushing is located.

Name

Name defines the name of the part.

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.

Part Color

Part Color sets the color of the component. If the component is the active component then the color used for display is determined by the Color_Active_Component option in the Options menu. To get an active component to be displayed in its specified color, set the Color_Active_Component option to Transparent.

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 GearTeq menu.

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.

Pitch, Diametral

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

Pitch, Diametral, 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.

Pressure Angle, Working

Pressure Angle, Working is the angle of the tooth at the pitch diameter. This is normally the same as pressure angle unless either gear in the set has been modified or the center distance has been changed.

Roll Angle at Custom Diameter

Roll Angle at Custom Diameter provides the user with a method to calculate a roll angle at any valid diameter. Entering a new diameter in this property will calculate the roll angle for the entered value. The value must be greater than the base diameter.

Roll Angle at Major Diameter

Roll Angle at the Major Diameter is displayed.

Roll Angle at Profile Modification Above Pitch Line

Roll Angle at the start of the profile modification above the pitch line is displayed if a modification is specified.

Roll Angle at Profile Modification Below Pitch Line

Roll Angle at the start of the profile modification below the pitch line is displayed if a modification is specified.

Roll Angle at TIF Diameter

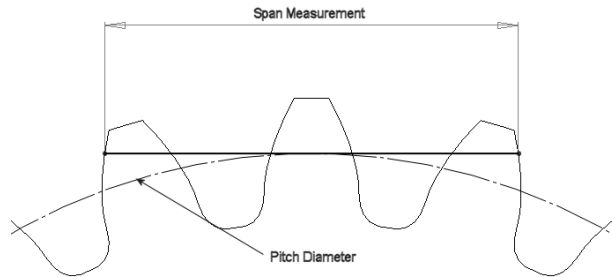
Roll Angle at the TIF Diameter is displayed.

Shrinkage Rate

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%), $0.0025 = 0.25\%$

Span Measurement

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



Span 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.

Span Measurement, Teeth to Gage Over

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

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, GearTeq 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.

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, GearTeq 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, GearTeq 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.

Tolerance, 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.

Tolerance, 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.

Tolerance, Tooth Thickness

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

Tolerance, 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.

Tolerance, Total Index

Total Composite Index.

Tooth Thickness

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

Tooth Thickness at Major Diameter

Tooth Thickness at Major Diameter is the arc thickness of the tooth at the tip of the gear.

Tooth Thickness, Normal

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

Tooth Thickness Tolerance

The Tooth Thickness Tolerance is a user input property that defines the tolerance according to DIN 3967.

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.

Units

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

Upper Tooth Thickness Allowance

The Tooth Thickness Allowance is a user input property that defines the allowance according to DIN 3967.

Variation, Pitch

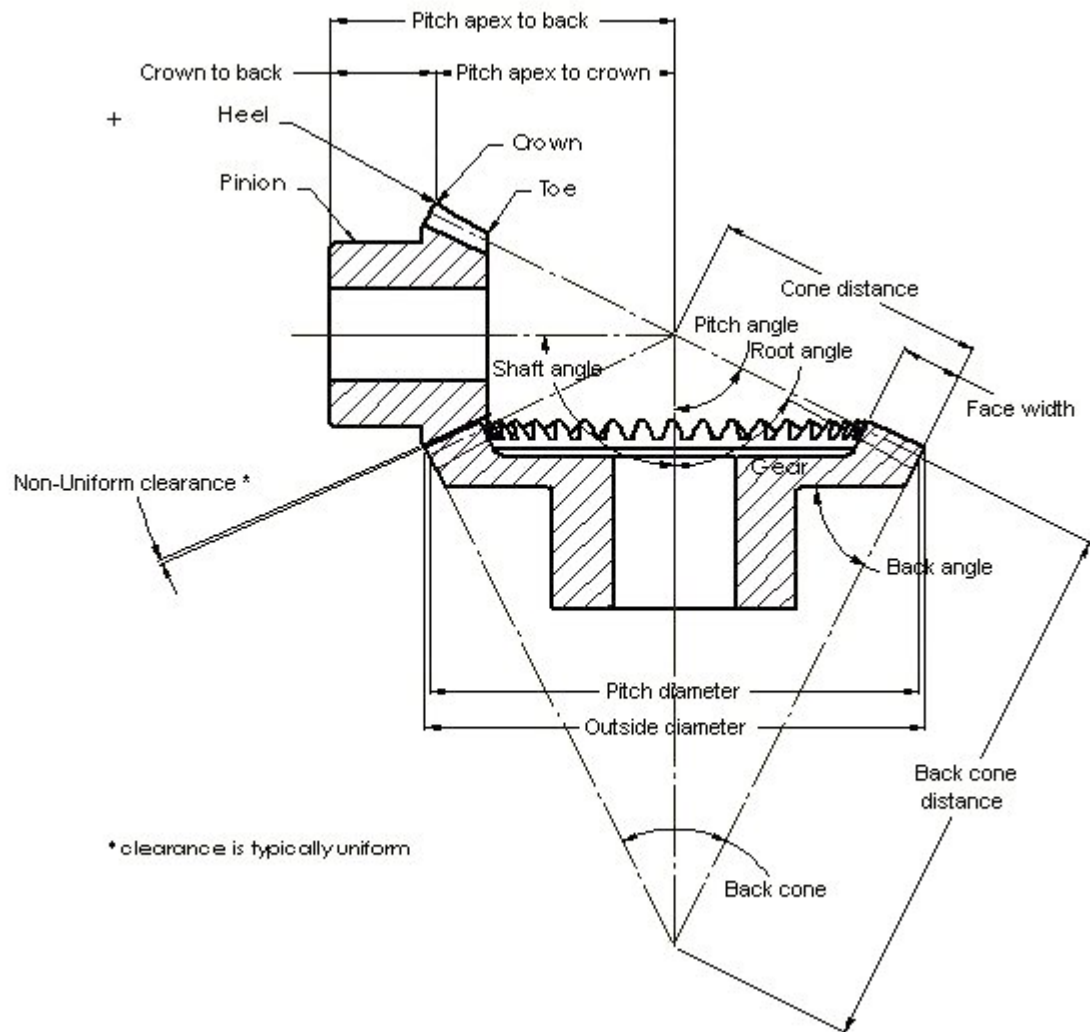
Pitch Variation is the algebraic (+ or -) plus or minus difference in the transverse plane, between the true position pitch and an actual pitch measurement. If measured in a plane other than the transverse plane, a correction using the appropriate helix angle must be applied to the measured value according to AGMA 2000-A88.

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.

Bevel Gears



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.

AGMA Class

AGMA Class sets the AGMA Class for this component.

Angle, Back

The Back Angle is perpendicular to the pitch cone at the pitch diameter measured perpendicular to the shaft.

Angle, Face

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

Angle, Front

Front Angle is parallel to the back angle at the inside surface of the face width. Miter gears do not have this geometry.

Angle, Pitch

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

Angle, Root

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

Angles, Sum of Pitch

Sum of Pitch Angles defines the sum of the pitch angles for this component and its parent 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.

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, Addendum Modification

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.

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.

Diameter, Base

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

Diameter, Outside

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.

Diameter, Pitch

Pitch Diameter is a theoretical diameter of the gear.

Diametral Pitch

Diametral Pitch is used in imperial system gears

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 can be found in Machinery's Handbook in the 24th and earlier editions. The user must specify this value for spiral bevel gears. It is automatically generated for Gleason straight bevel gear sets.

Manufacturing Method

Manufacturing Method defines how the gear is to be manufactured. Select Gleason or Non-standard. Selecting Non-standard allows the user more control of the parameters of the geometry.

Module (displayed as Modular)

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

Mounting Distance

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

Name

Name defines the name of the part.

Number of Teeth

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

Part Color

Part Color sets the color of the component. If the component is the active component then the color used for display is determined by the

Color_Active_Component option in the Options menu. To get an active component to be displayed in its specified color, set the Color_Active_Component option to Transparent.

Path

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

Pitch Variation Accumulated

Currently not used

Pitch Variation Allowable

Currently not used

Pressure Angle

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

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.

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.

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.

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.

Working Depth

Working Depth is the length of the tooth that engages the mating gear at the pitch line.

Chain Sprockets

Chain Length

Chain Length is the length of a chain around this component and its parent sprocket.

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.

Even Full Pitches

Even Full Pitches displays the number of even full pitches of chain for this component and its parent sprocket.

Name

Name defines the name of the part.

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.

Part Color

Part Color sets the color of the component. If the component is the active component then the color used for display is determined by the Color_Active_Component option in the Options menu. To get an active component to be displayed in its specified color, set the Color_Active_Component option to Transparent.

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, GearTeq 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.

Diameter, Pitch

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.

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.

Name

Name defines the name of the part.

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.

Part Color

Part Color sets the color of the component. If the component is the active component then the color used for display is determined by the Color_Active_Component option in the Options menu. To get an active

component to be displayed in its specified color, set the Color_Active_Component option to Transparent.

Path

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

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, GearTeq 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.

Diameter, Pitch

Pitch Diameter defines the pitch diameter for the component.

Name

Name defines the name of the part.

Number of Grooves

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

Part Color

Part Color sets the color of the component. If the component is the active component then the color used for display is determined by the Color_Active_Component option in the Options menu. To get an active component to be displayed in its specified color, set the Color_Active_Component option to Transparent.

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

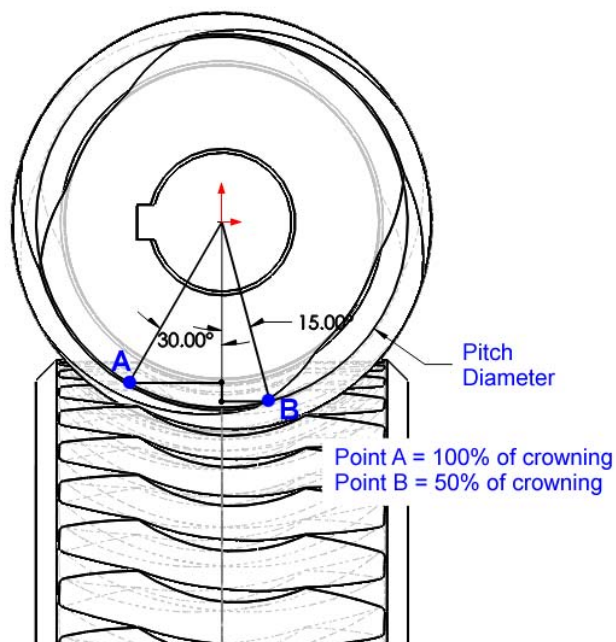
Contact Ratio defines the contact ratio for the worm gear set.

Create Worm and or Wheel

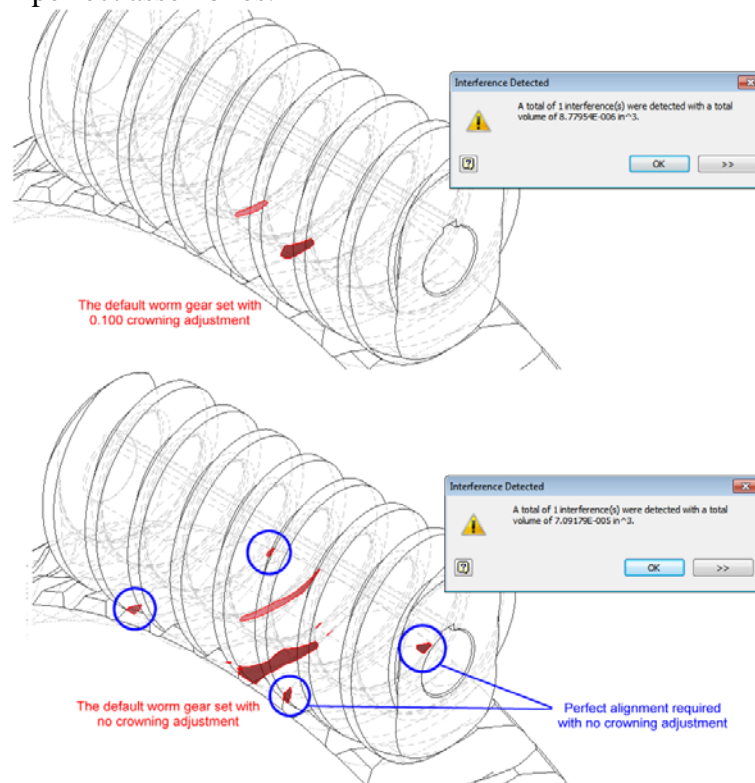
Select between Both, Worm Only and Wheel Only.

Crowning Drop

Crowning Drop defines the amount of drop at 30 degrees from the center of the worm axis. At 15° from the worm axis 50% of the drop is used. This provides a crown type that is close to a full radial type.



Crowning is important to worm gear sets to avoid abnormal tooth contact due to imperfect assemblies.



Crowning will also provide for the forming of an oil film which enhances the endurance and transmission efficiency. The standard methods for worm wheel crowns are derived by the limits of manufacturing capabilities of the gear machines. GearTeq uses a method that is more like those used for spur, helical and bevel gears.

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 worm 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, Gear

Lead, Gear 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, Worm, Angle

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

Length, Worm

Length, Worm defines the axial length of the worm.

Module (Displayed as Modular)

Module (transverse) is used in metric 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.

Name, Component

Name, Component defines the name of the worm set.

Number of Sketches

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”.

Part Color

Part Color sets the color of the component. If the component is the active component then the color used for display is determined by the Color_Active_Component option in the Options menu. To get an active component to be displayed in its specified color, set the Color_Active_Component option to Transparent.

Path

Path defines the name of the folder where the component files are stored on the computer or network.

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, GearTeq 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

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.

Involute 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 GearTeq menu item Tools>Reset Chordal Measurement.

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.

Face Width

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

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.

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. GearTeq 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. GearTeq 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. GearTeq 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. GearTeq 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. GearTeq 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. GearTeq 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.

Name

Name defines the name of the part.

Number of Teeth

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

Part Color

Part Color sets the color of the component. If the component is the active component then the color used for display is determined by the Color_Active_Component option in the Options menu. To get an active component to be displayed in its specified color, set the Color_Active_Component option to Transparent.

Path

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

Pitch, Circular

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

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 GearTeq menu.

Pitch, Diametral

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

Pitch, Diametral, 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.

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
JIS B 1603

Tact

(DIN 5480) Actual tooth thickness (or space width) tolerance

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, GearTeq 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) Effective tooth thickness (or space width) tolerance

TG

(DIN 5480) Total tooth thickness (or space width) tolerance

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) 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.

ACME Screws

ACME Class

AMCE Classes 2G, 3G, 4G and 5G are available.

ACME Standards

AMCE Threads B1-5-1977 and Standards Stub Threads B1-5-1977 are available.

ACME Class

AMCE Classes 2G, 3G, 4G and 5G are available.

Addendum

Addendum is the radial length of the tooth between the pitch diameter and the major diameter. This value is read only and controlled by the standard selected and the pitch of the screw.

Allowance on Major Diameter

A.

Allowance on Minor Diameter

A.

Allowance on Pitch Diameter

A.

Angle of Thread

The angle of thread is read only and is always 29 degrees.

Basic Thread Height

A.

Basic Thread Thickness

The basic thread thickness is equal to 1/2 of the pitch.

Basic Tooth Height

The basic tooth height is equal to twice the addendum.

Coefficient Addendum

The pitch times the addendum coefficient equals the addendum height of the thread.

Dedendum

Dedendum is the radial length of the tooth between the pitch diameter and the minor diameter. This value is read only and controlled by the standard selected and the pitch of the screw.

Diameter Nut Blank

A.

Hand Direction

The hand direction determines whether the screw is left or right hand threads.

Lead

The lead is equal to the thread pitch times the number of starts. For example, a 0.250 pitch screw with 4 starts will have a lead of 1.000

Length Nut

The nut length defines the length of the nut blank.

Length Screw

The screw length defines the length of the screw blank.

Major Diameter Max Ext

A.

Major Diameter Min Ext

A.

Minor Diameter Max Ext

A.

Minor Diameter Min Ext

A.

Major Diameter Max Int

A.

Major Diameter Min Int

A.

Minor Diameter Max Int

A.

Minor Diameter Min Int

A.

Name Component

Name component is the name of the part used by GearTeq.

Number of Starts

The number of starts defines the number of independent threads on the screw.

Nut Inside Diameter

The nut inside diameter equals the pitch diameter minus twice the dedendum. This value is read only and cannot be changed.

Path

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

Part Color

Part Color sets the color of the component. If the component is the active component then the color used for display is determined by the Color_Active_Component option in the Options menu. To get an active component to be displayed in its specified color, set the Color_Active_Component option to Transparent.

Pitch

Pitch is the distance from one thread to the next in the axial direction. It is not uncommon to see a 0.250 pitch screw referred to as a "4 thread screw".

Pitch Diameter

The pitch diameter of a screw is the diameter where the thread thickness and thread space are equal.

Pitch Diameter Max Ext

The pitch diameter maximum external is a read only value that is controlled by the specified pitch diameter and ACME class.

Pitch Diameter Min Ext

The pitch diameter minimum external is a read only value that is controlled by the specified pitch diameter and ACME class.

Pitch Diameter Max Int

The pitch diameter maximum internal is a read only value that is controlled by the specified pitch diameter and ACME class.

Pitch Diameter Min Int

The pitch diameter minimum internal is a read only value that is controlled by the specified pitch diameter and ACME class.

Screw Outside Diameter

The screw outside diameter equals the pitch diameter plus twice the addendum. This value can be changed and the pitch diameter will change accordingly.

Units

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

Material Property

Each component has a Material Property category.

Compressive Yield Strength PSI

Defines the Compressive Yield Strength, PSI for this component. Select material type of 'Other' to manually change this value.

Material

Defines the type of material to be used with this component.

Modulus of Elasticity KSI

Defines the Modulus of Elasticity, KSI for this component ($KSI = PSI * 1000$). Select material type of 'Other' to manually change this value.

Poisson's Ratio

Defines the Poisson's Ratio for the component. Select material type of 'Other' to manually change this value.

Safe Bending Strength PSI

Defines the Safe Bending Strength, PSI for this component. The safe stress is assumed to be 1/3 of the Tensile Strength.

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 (0.0025 = 0.25%)

Tensile Strength Ultimate PSI

Defines the Tensile Strength, Ultimate PSI for this component. Select material type of 'Other' to manually change this value.

Hub Data

Each Component has a Hub Data category

Bore

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.

Mounting Bushing Modeling

Mounting Bushing Modeling determines if the bushing should be created as part of the component or as a new part. Select 'Create as component

feature’ to have the bushing created as a feature in the CAD model. Select ‘Create as new part’ to have the bushing created as a new CAD model.

Mounting Bushing Side

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

Mounting Split Taper Bushing

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

Standard Set Screw

Standard Set Screw 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.

Tool Bars

Insert New Components

Inserting a new component automatically inserts a new assembly position for the component.



Insert New Spur or Helical Gear



Insert New Bevel Gear



Insert New Chain Sprocket



Insert New Gear Belt Pulley



Insert New V-Belt Pulley



Insert New Worm Gear Set



Insert New External Spline



Insert New Internal Spline



Insert New Planetary Gear Set

Standard



New



Open



Save Active Component



Save All



Print the GearTeq Window



Create CAD Models and Assembly



Edit CAD Models



Common Properties Interface (CPI)

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.



Index Up

Clicking the Index Up tool bar button changes the view orientation up by the spin increment specified in the Options.



Index Down

Clicking the Index Down tool bar button changes the view orientation down by the spin increment specified in the Options.



Index Left

Clicking the Index Left tool bar button changes the view orientation to the left by the spin increment specified in the Options.



Index Right

Clicking the Index Right tool bar button changes the view orientation to the right by the spin increment specified in the Options.

View



Last View

Clicking the Last View tool bar button changes the view orientation to previous 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

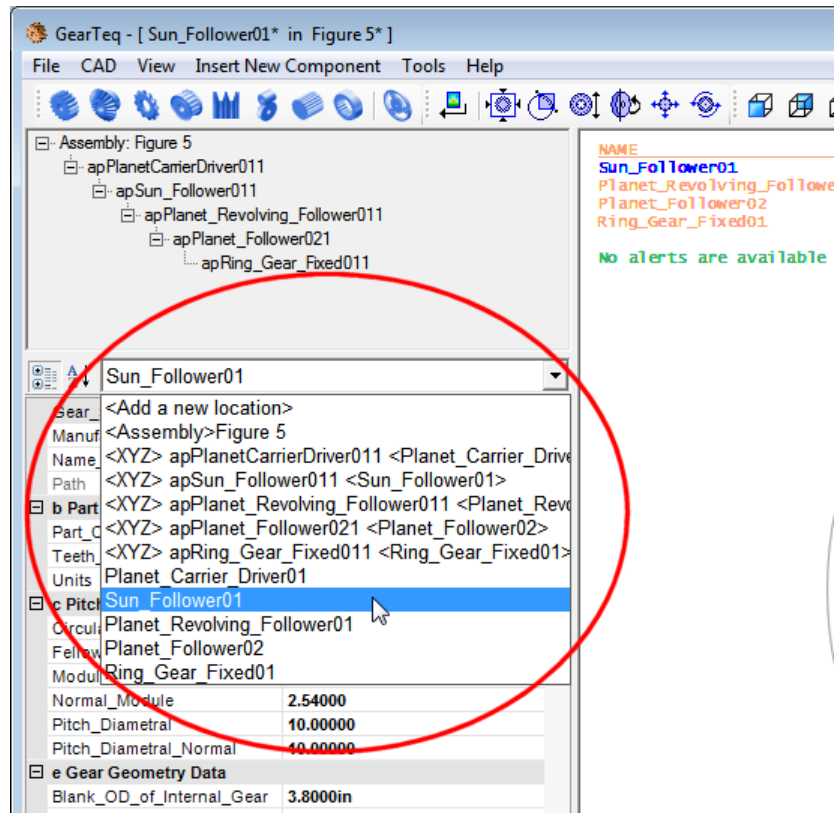
Clicking the Play Reverse tool bar button will drive the components in the opposite direction for a short simulation.



Forward

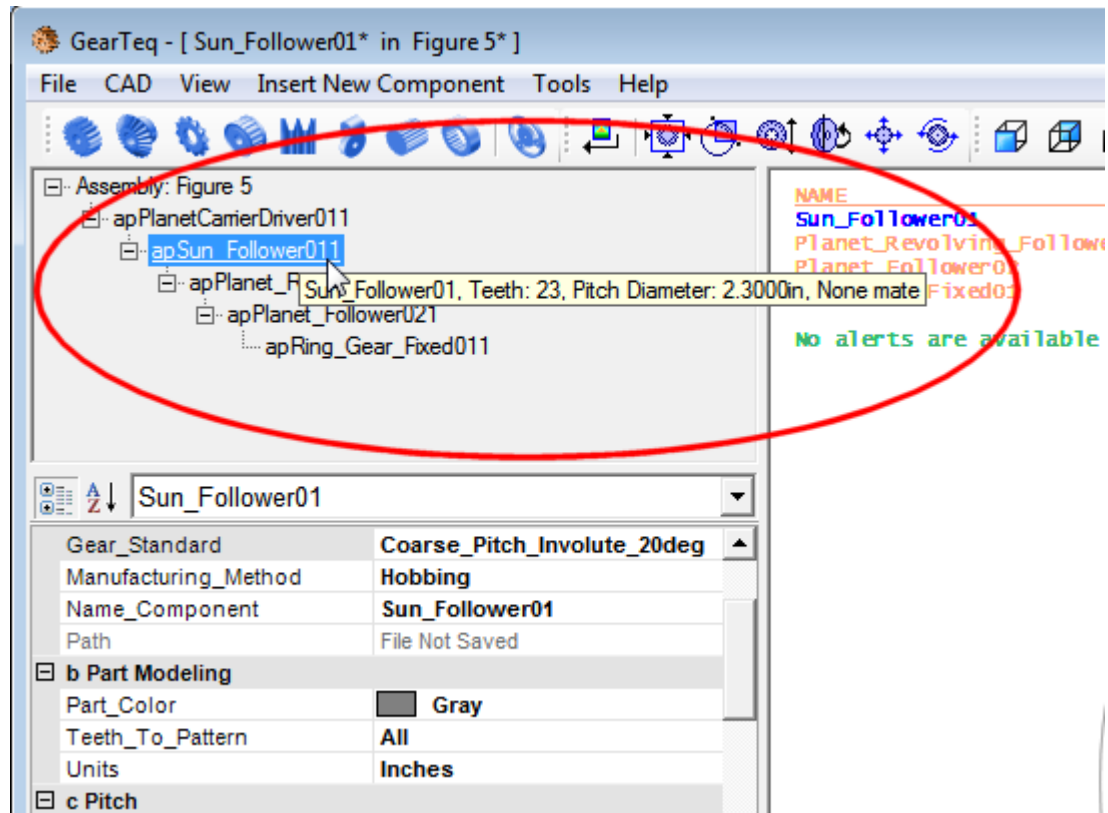
Clicking the Forward tool bar button will drive the components forward one spin increment. Set the spin increment in the options menu.

Active Feature Selector



The Active Feature Selector provides a method to switch between the assembly, assembly positions and components.

Assembly Tree

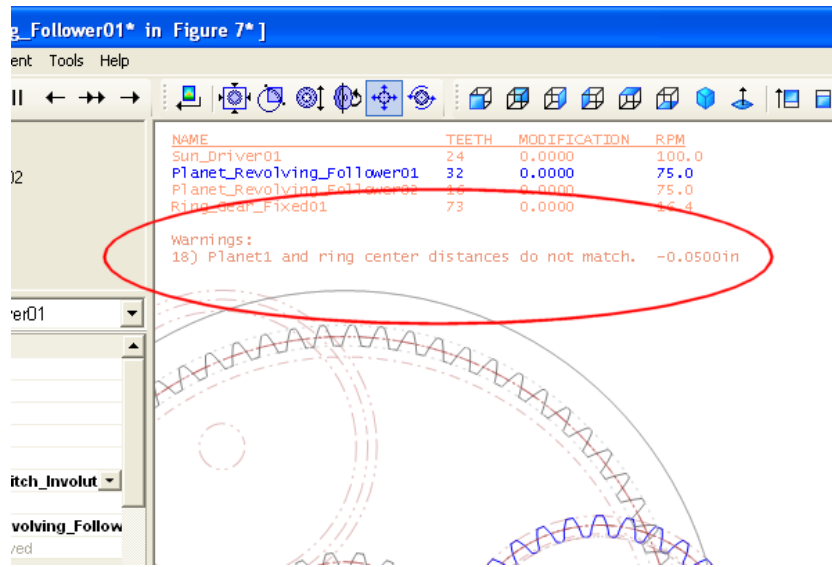


The assembly tree displays a list of assembly positions in the assembly. Move the mouse over an AP name will display information about the component associated with that AP.

Double clicking on an assembly position will activate the associated component.

Click on a name then right click to display a menu to activate the assembly, the assembly position or the associated component.

Alerts



GearTeq provides some alerts when it detects that something might be amiss with the current component's design or with a set of components. This is not meant to be an all inclusive check. There may be problems with the design that GearTeq does not detect or alert the user about.

1 through 11 are not used.

12. The gear has no addendum modification. This may cause undercutting and/or tip interference with the mating gear.

13. The spiral angle of the bevel gear does not equal 0 or 35 degrees.

14. The bore diameter may be too large for this gear.

15. The backlash is greater than 1/4 of the circular pitch.

16. The pressure angle does not match mating parent.

17. Not used

18. The planet center distances are not matching.

19. The base diameter of the internal gear is larger than the minor diameter.

20. The pitch depth of rack is less than or equal to the dedendum.

21. The blank diameter of the internal gears is less than or equal to the major diameter.

- 22. Spiral bevel CAD models may be suitable for manufacturing. Please inspect them carefully using clearance and interference checks at different rotations.
- 23. The internal gear has little or no negative addendum modification. This may cause tip interference with the mating gear.
- 24. A crowning drop has not been specified. A crowning drop should be specified if the models will be used for manufacturing.
- 25. There is a hub projection on side 1 but no diameter.
- 26. There is a hub projection on side 2 but no diameter.
- 27. There is a hub projection on side 1 specified but no diameter.
- 28. There is a hub diameter on side 2 specified but no projection.
- 29. The course pitch standard is selected but a diametral pitch finer than 20 is specified. The fine pitch standard is selected but a diametral pitch courser than 20 is specified.
- 30. The Module for the spline is not a standard increment.
- 31. A non-AGMA class is specified for an AGMA Spline.
- 32. The blank diameter of the internal spline is less than or equal to the major diameter.
- 33. The actual outside diameter is less than the specified diameter because of tooth shortening.
- 34. The rack addendum modification does not compliment the mating gear's addendum modification.
- 35. The web thickness is outside of the acceptable range.
- 36. Pinion number of teeth is less than that permitted by the Gleason standard.
- 37. Gear number of teeth is less than that permitted by the Gleason standard.
- 38. The total backlash for this component and its parent is outside of the range recommended by Gleason.
- 39. The hand of the mating gear does not compliment the current helical gear.

- 40. The standard set screw is set to true but a valid hub and a keyed bore are not specified.
- 41. Not used
- 42. Not used
- 43. The bore is larger than the minor diameter.
- 44. Timing belt CAD model of the active component is not suitable for manufacturing. Please use it for documentation purposes only.
- 45. The tooth thickness is less than recommended by the PGT standard.
- 46. The rack will not be created with the basic PGT rack radius ('sled runner' effect) added to the outer half of the addendum.
- 47. Special geometry ASA is specified but the chain pitch is set to 0.0000
- 48. Special geometry ASA is specified but the roller diameter is set to 0.0000
- 49. Heavy duty series sprockets with multi-strand are not available with this chain number.

Mouse controls

Current Mouse Position

The status bar at the bottom of the screen displays the current mouse position relative to the origin of the currently active component. All coordinates are normal to the screen.

On Screen Measurements

Right click and move for on screen measurements. This is an easy way to get an approximate measurement from the drawing window. All measurements are normal to the screen. The status bar at the bottom of the GearTeq window displays all measurements.

Mouse wheel

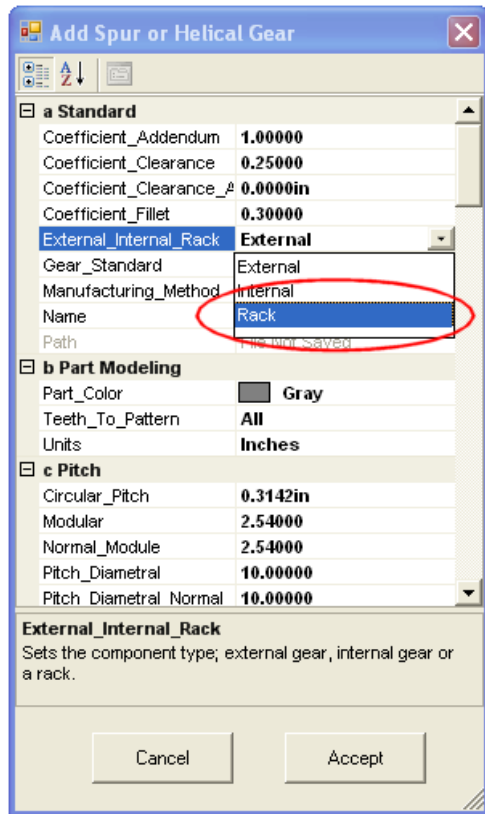
Wheel up and down scales the view of the components. All scaling is done from the center of the screen, not the mouse position. Pan the geometry to the center of the screen then use the mouse wheel to enlarge or reduce the view scale. The direction of the mouse wheel is controllable by a setting in the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

Creating a Rack

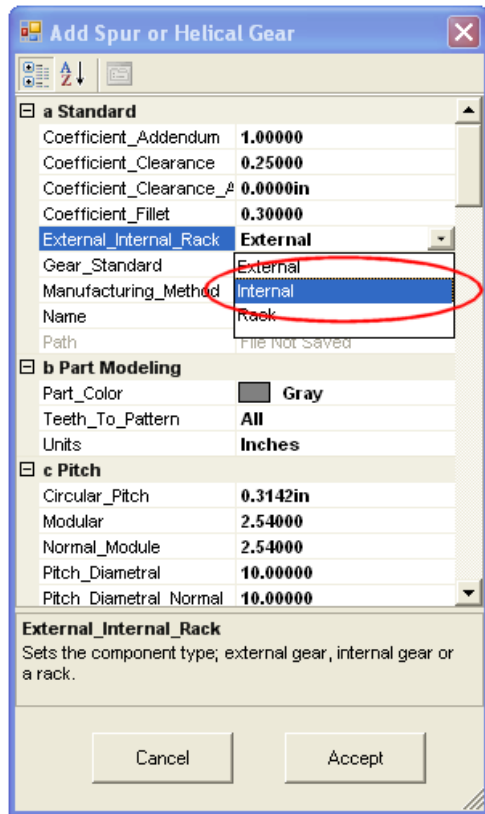
To create a rack, select from the GearTeq menu **Insert New Component>New Spur or Helical Gear** menu option, or press the control button and the S key at the same time to open the **Add New Spur or Helical Gear** dialog window.



The user may change the rack to an external gear after the component is displayed in the GearTeq window.

Creating an Internal Gear

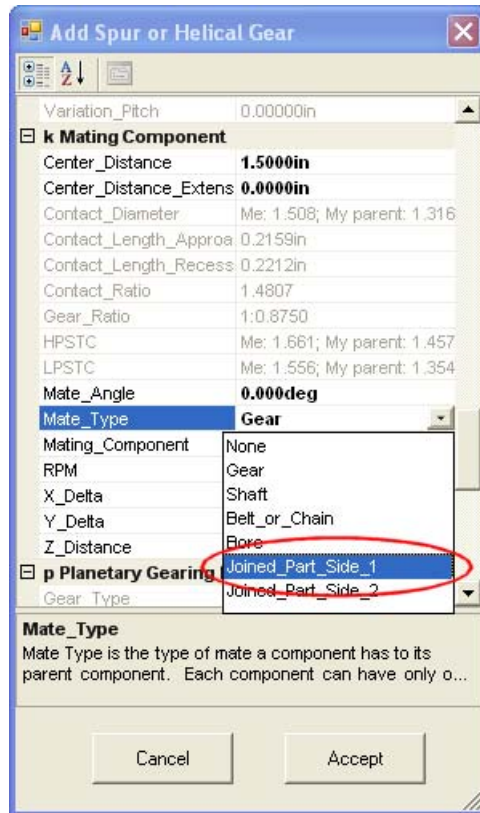
To create an internal gear, select from the GearTeq menu **Insert New Component>New Spur or Helical Gear** menu option, or press the control button and the S key at the same time to open the **Add New Spur or Helical Gear** dialog window.



The user may change the internal gear to an external gear after the component is displayed in the GearTeq window.

Joined parts, side one or two

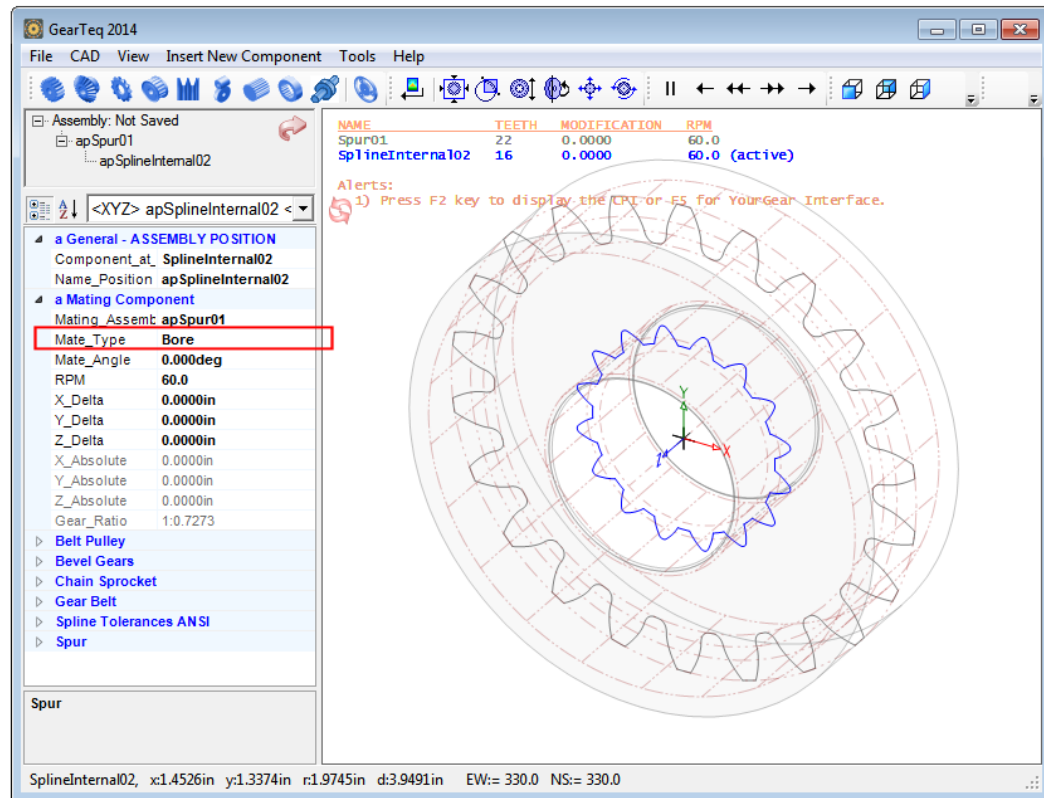
Two components may be joined together to create one CAD model. Add a new spur gear to GearTeq. You may change the parameters of the new gear at any time or stay with the default settings for this example. Open the Insert New Spur or Helical Gear dialog window. You may change any of the parameters of the new gear before you accept it. Scroll down to the Mating Component section and change the Mate Type to either **Joined Part Side 1** or **Joined Part Side 2**.



These options may be changed after the components are displayed in GearTeq or changed to another valid mating type. If a joined spur gear is changed to a gear type mate, the diametral pitch (or module) will be changed to match the parent component. Switching back to a joined mate will not change the diametral pitch back to the previous diametral pitch.

Adding an internal spline as a bore to another component

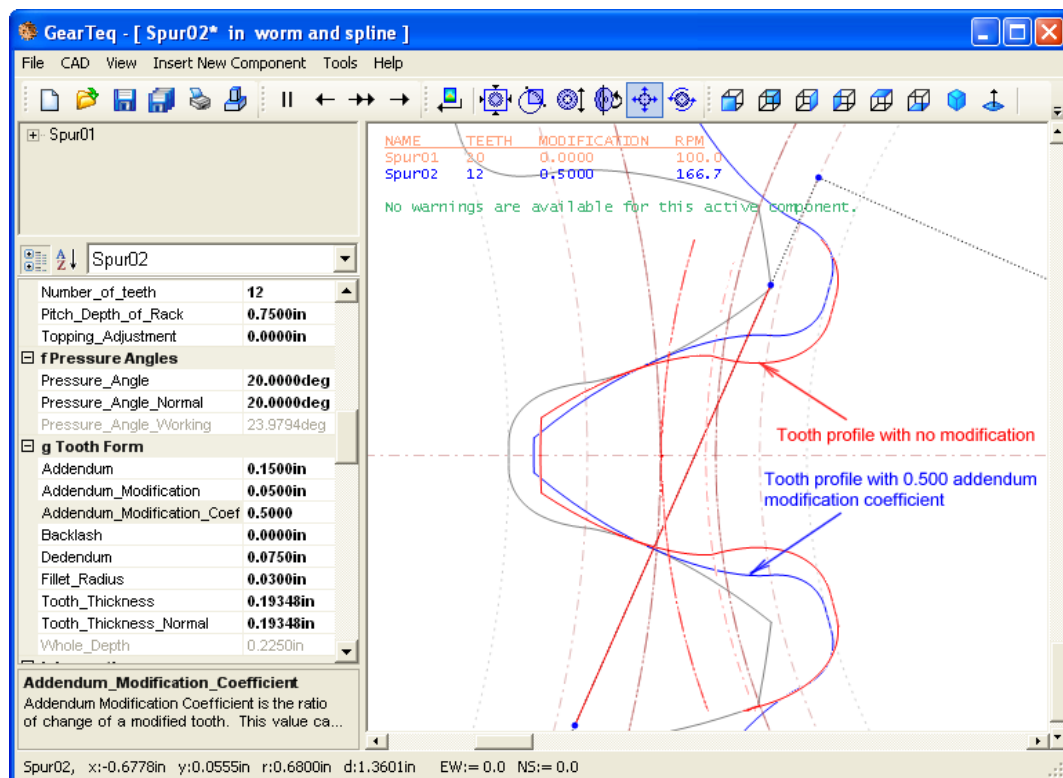
To add an internal spline as a bore to another component, have the component active in GearTeq. Then add a new internal spline to the assembly. Inserting a new internal spline with an external spline active will default to a shaft mate. Inserting a new internal spline to any other type of component will default to a bore mate.



The Bore mate type defines the internal gear as a bore to its mating component.

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.

Also see the Balance Addendum Modification section.

Custom Coefficient Method

Addendum

Clearance (clearance plus addendum equals dedendum)

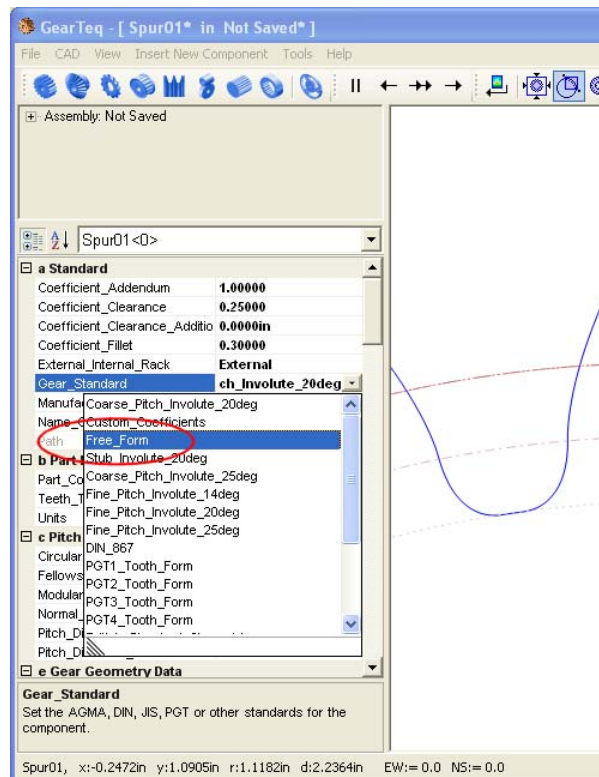
Clearance Addition, which is a linear dimension, not a coefficient

Fillet



Free Form Method

Selecting Free Form as the Gear Standard type will allow the user to create a non-standard spur gear. This allows the user to specify almost every parameter of the spur gear. With this increased flexibility also comes the responsibility of making certain the data makes sense. For example, you can specify a major diameter that is less than the minor diameter. GearTeq does not check the integrity of the data even though it is obviously wrong.



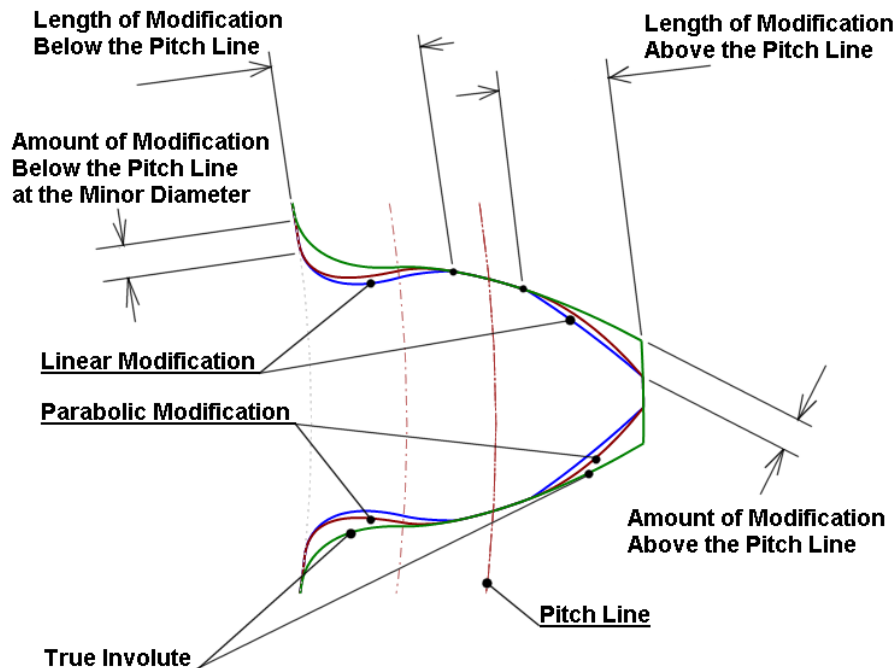
Modifying the Involute Profile on a Spur Gear

The involute profile may be modified using a linear or parabolic deviation from the true involute profile. The tooth form may be modified above and below the pitch line. The modification always removes material.

The length of modification above the pitch line (APL) cannot be greater than the addendum. The length of modification below the pitch line (BPL) cannot be greater than the dedendum.

The start (nearest the pitch line) of a parabolic modification is tangent to the involute.

For more information on profile modifications search the Internet for the technical publication NASA/TM-2000-210061.



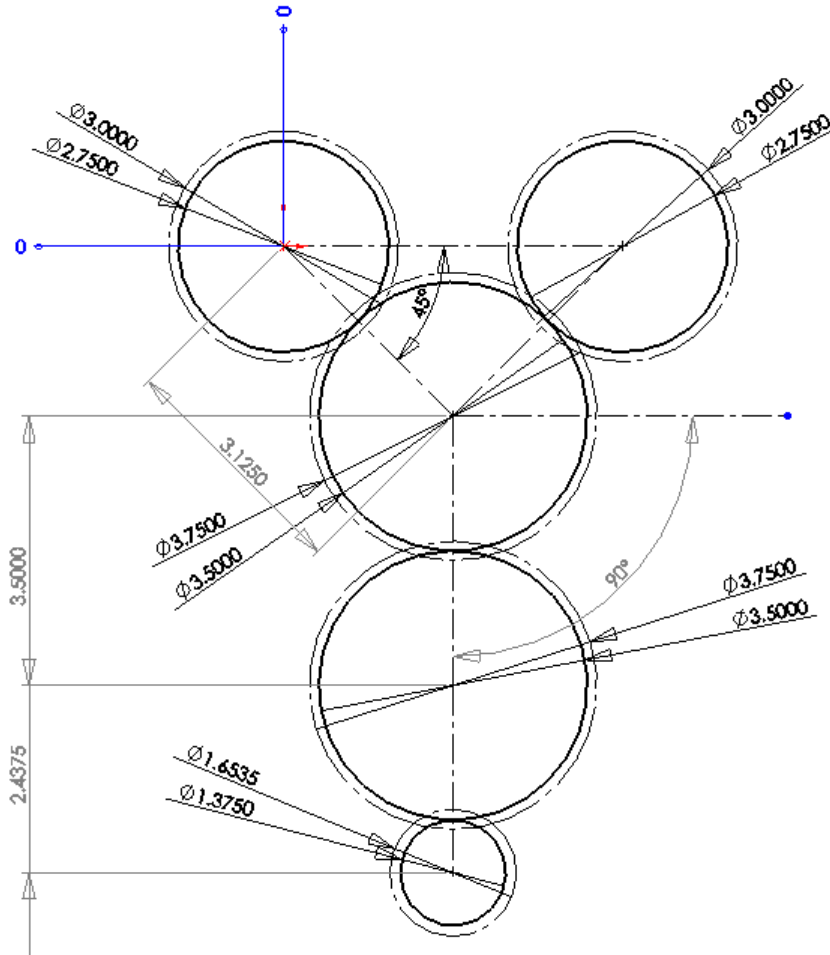
A Profile Modified 20 Tooth Spur Gear with no Addendum Modification

Tutorial – Spur Gear Assembly


The task is to create an assembly with 5 gears. Two pairs of gears are identical; a 22-tooth gear is mated to a 28-tooth gear and a 28-tooth gear is mated to the other 28-tooth gear. The last gear is an 11-tooth gear that has its addendum modified so the center distance is 2.4516" to its mating 28-tooth gear.

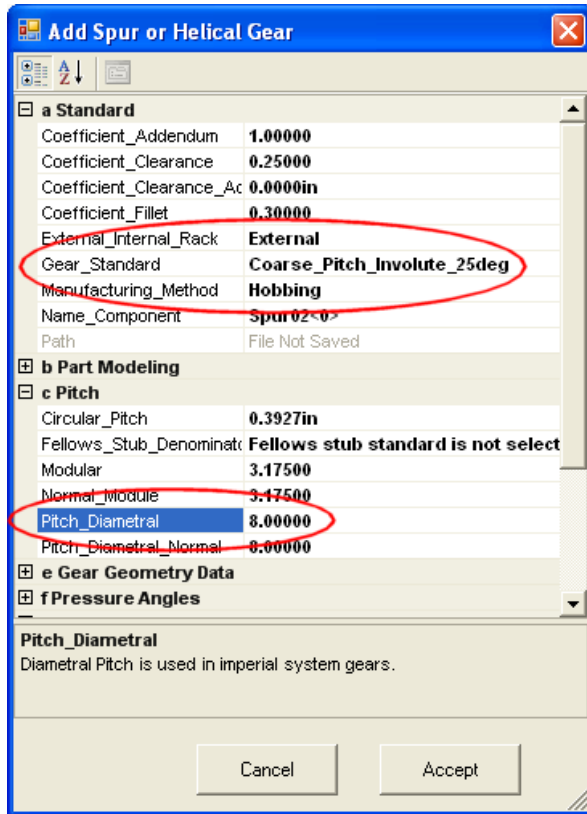
The pressure angle is 25 degrees.

The diametral pitch is 8.



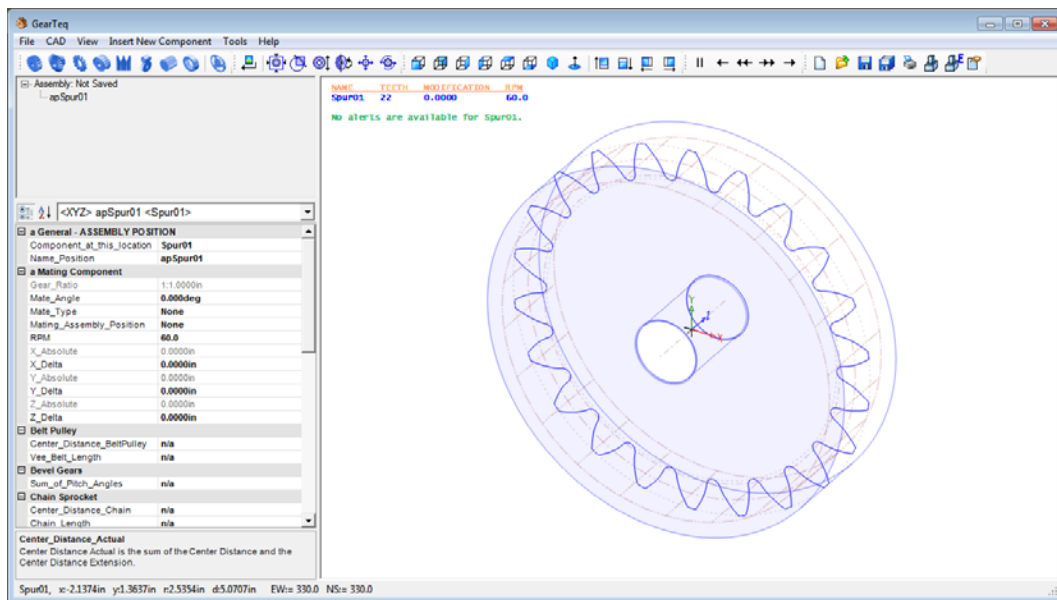
Sketch layout of sample assembly

Open the Add New Spur or Helical dialog window by clicking on the  icon, pressing Ctrl-S or selecting from the GearTeq menu Insert New Component>New Spur or Helical.



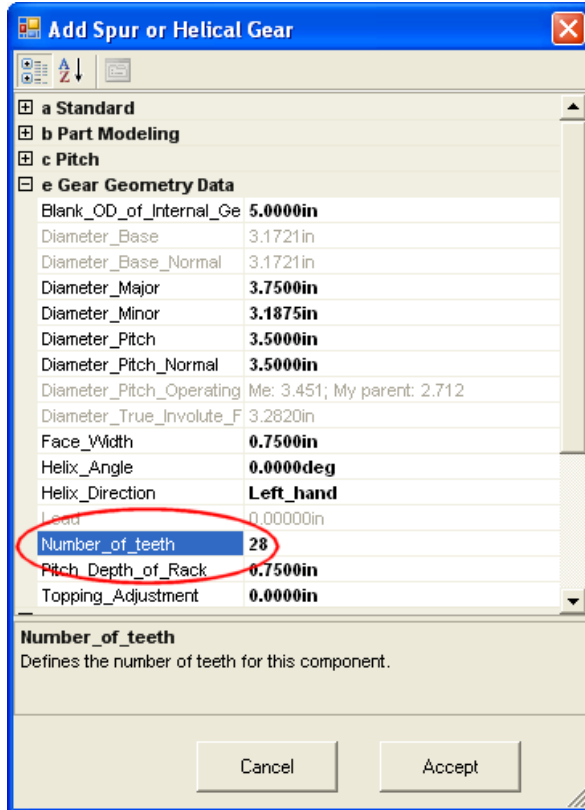
Change the Gear Standard to Coarse Pitch Involute 25deg and Diametral Pitch to 8.0.

Click Accept to add the component into the assembly.

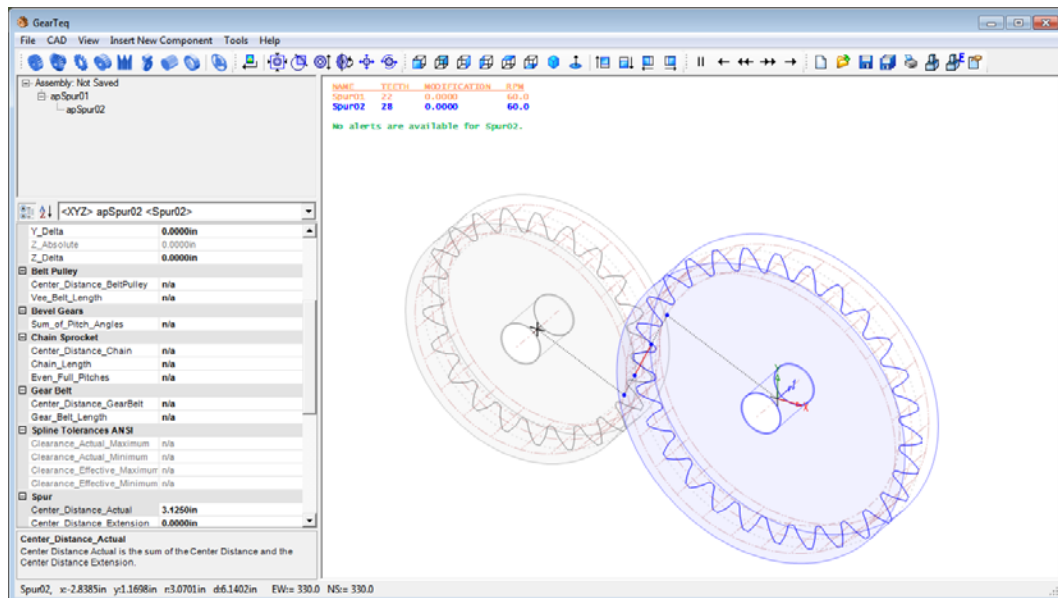


Assembly with first component

Open the dialog box to add another spur gear.



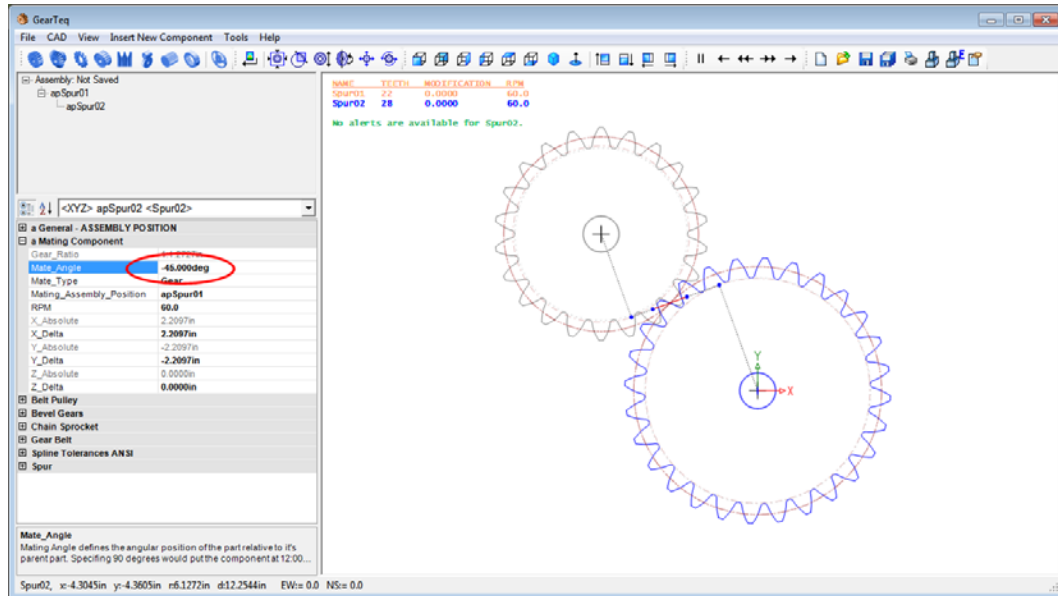
Change the number of teeth to 28 and click the accept button to add the component to the assembly.



Assembly with second component added

In the property grid, change the Mate Angle to -45.0 degrees.
Click on the Front and the Zoom to Fit icons.

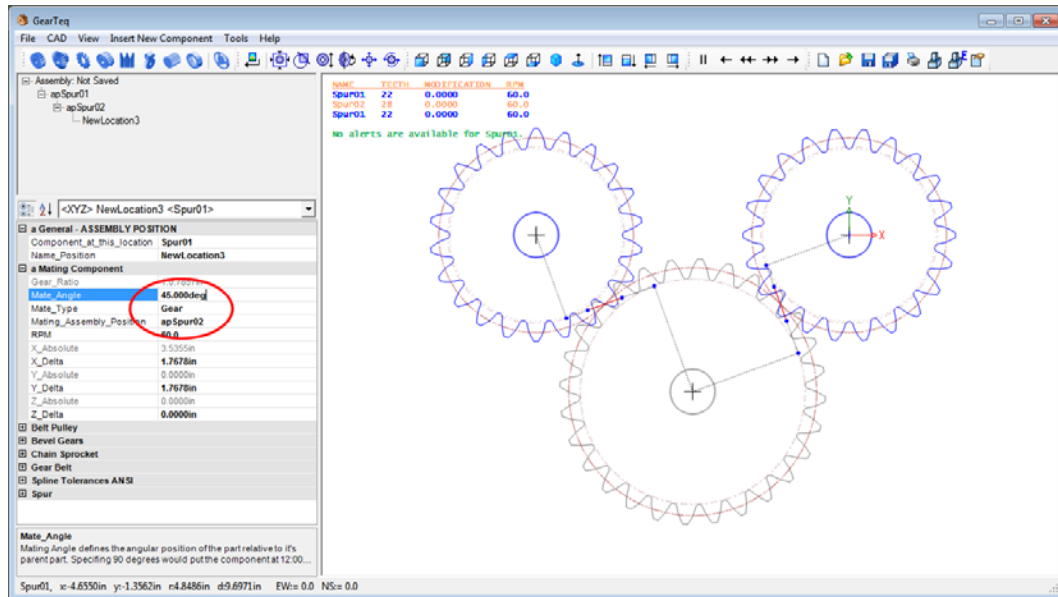
Tip: Press the F11 key to collapse the categories, then press the "+" button to the left of the "a Mating Component" category to expand just the mating category.



The next task is to add an instance of the first spur gear so select “Spur01” in the component selector to make it the active component.

From the GearTeq menu click Insert New Component>Insert a New Assembly Position or from the Active Feature Selector click the <Inset New Assembly Position> option.

If required, expand the General - ASSEMBLY POSITION category. The component at the location property contains a list of the components in the assembly. Drop down the property and select Spur0.



The Mating Component> Mating Assembly Position property contains a list of the assembly positions. Drop down the list and select apSpur02.

Change the Mate Type property to Gear and the Mate Angle to 45.

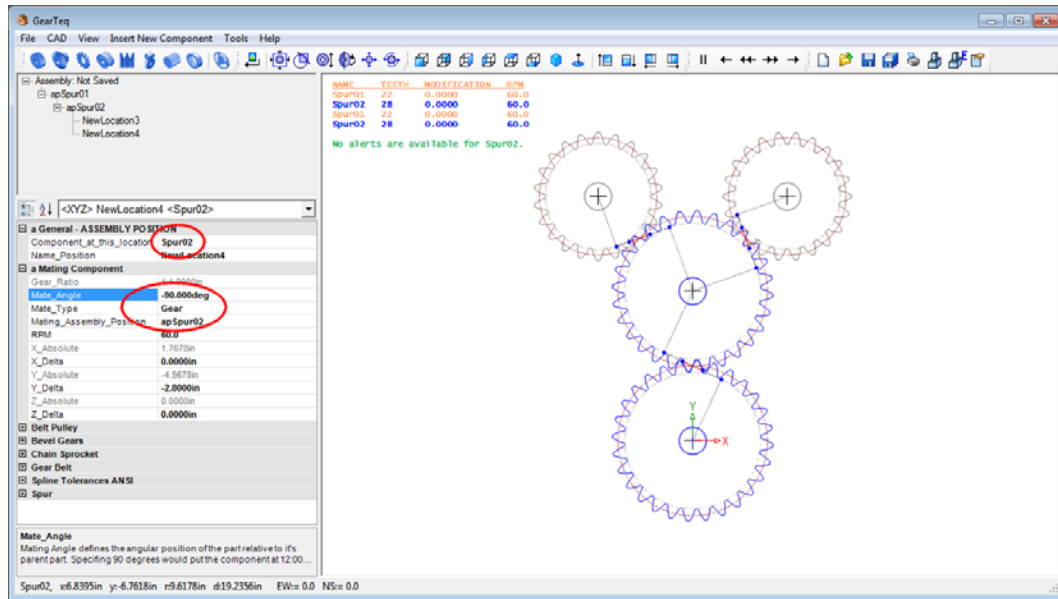
The next task is to add a new instance of the 28-tooth gear. From the GearTeq menu click Insert New Component>Insert a New Assembly Position or from the Active Feature Selector click the <Inset New Assembly Position> option.

In the "Component at this location" property select the component "Spur02".

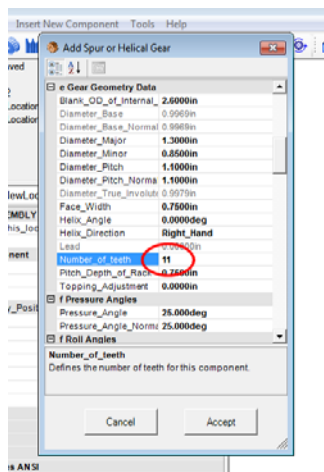
In the "Mating Assembly Position" select the assembly position "apSpur02".

For the Mate Type select "Gear" and for the Mate Angle enter -90degrees.

Select Spur02<0> as the mating component and Mate Type of Gear. Click OK to add the new instance to the assembly.



The next task is to add a new spur gear.

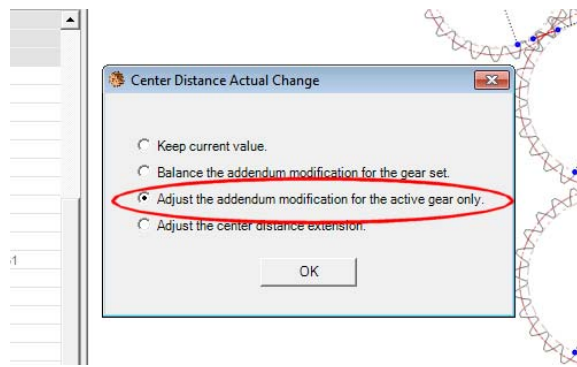


Open the Add Spur or Helical Gear dialog box.

Change the Number of Teeth to 11.

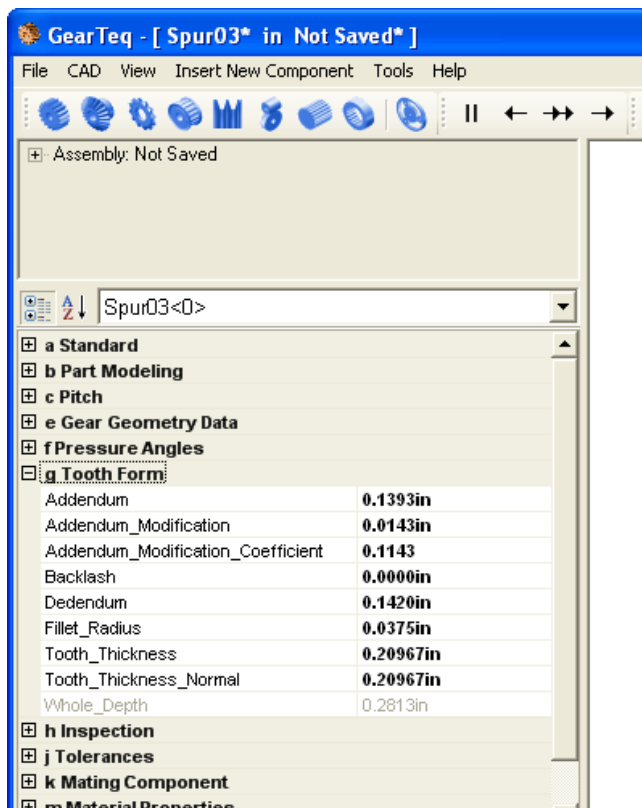
Click the Accept button to add the new component to the assembly.

Change the Mate Angle to -90.0 degrees.



The next task is to change the center distance.

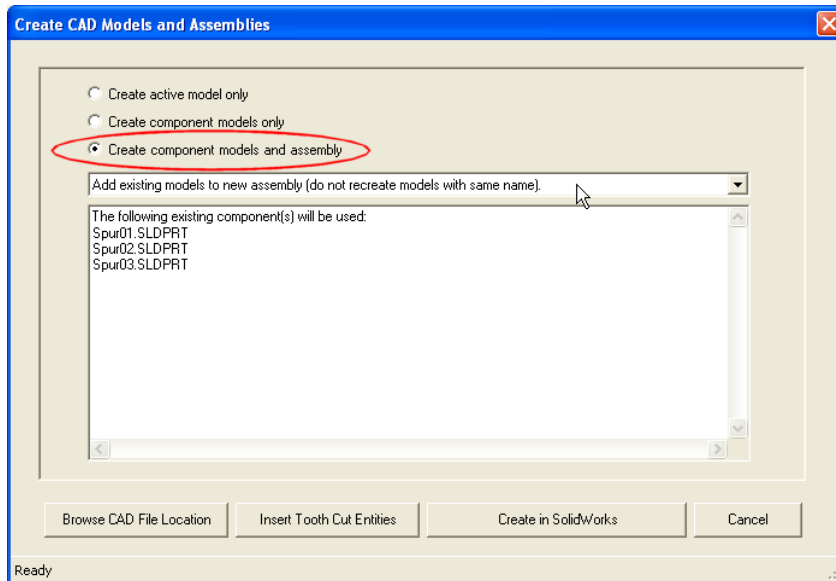
In the Spur category, Center Distance Actual property, change the Center Distance to 2.4516 inches. A dialog box will provide a few options on how GearTeq is to deal with this change in center distance. Select the third option so the change in addendum only affects the new component.



Activate component "Spur05". In the "g Tooth Form" category, note how GearTeq changed the addendum, dedendum and tooth thickness parameters for the new component. The mating component's addendum, dedendum and tooth thickness parameters remain unchanged.

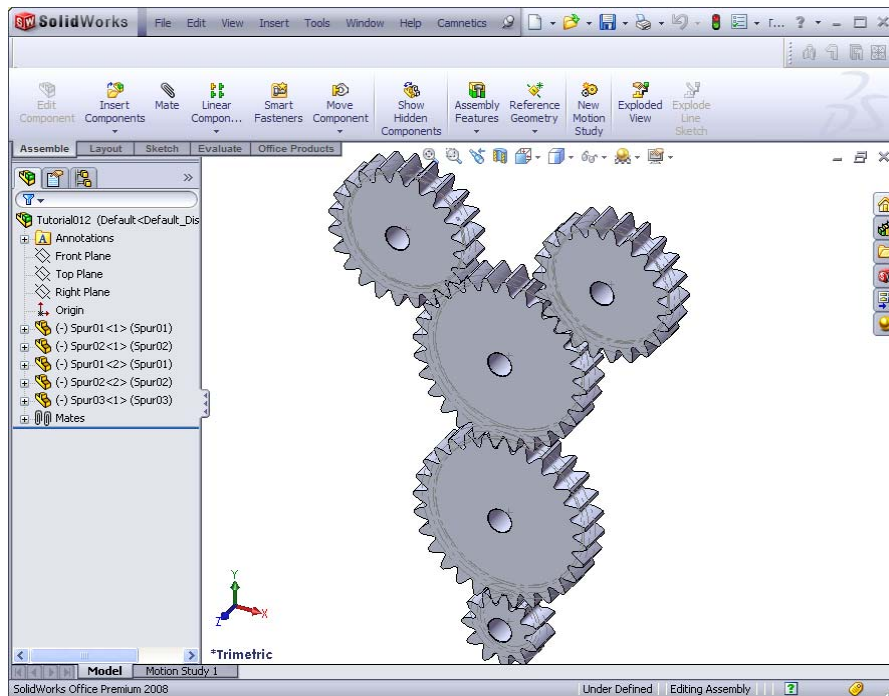
Save the assembly and components.

The last task is to create the components and assembly in CAD. From the GearTeq menu select CAD>Create CAD Models and Assemblies to open a dialog box.



Select the Create component models and assembly option bullet.

Click “Create in <your CAD system>” button.



The completed gear assembly created in SolidWorks

Tutorial – Creating a Face Gear with a mating pinion

In this tutorial the task will be to create an 80 tooth face spur gear with a 14 tooth pinion. The face width of the gears will be .50 inches. The default diametral pitch of 10 with a 20-degree pressure angle will be used. 0.010" crowning and a 0.350 addendum modification coefficient will be specified for the pinion.

Add a spur gear, change number of teeth to 80 and "External Internal Rack" property to Face.

Add a 2nd spur, change number of teeth to 14. Change addendum modification coefficient to 0.350. The pinion settings do not have any effect on the face gear. There are two face gear properties that will need to be changed which will be addressed in the next paragraph.

Why have addendum modification?

- To avoid under cutting of the pinion teeth
- To take advantage of full line of action
- To add strength to the pinion
- Does not weaken the face gear

Why not have addendum modification?

- It will change the center distance
- Makes the pinion slightly more complicated

Switch to the face gear.

Set the "Number of teeth face gear mate" property to 14.

Change the "Face gear mate addendum modification coefficient" property to 0.350.

Inspect the parts before creating in CAD.

Click on the Right view tool bar button.

Use the Zoom Area to zoom in on the pinion tooth that mates with the face gear.

The lines that develop the face gear tooth profile should be tangent to the involute flank of the pinion tooth.

Use the Drive tool bar button or the VCR buttons to rotate the gear mesh.

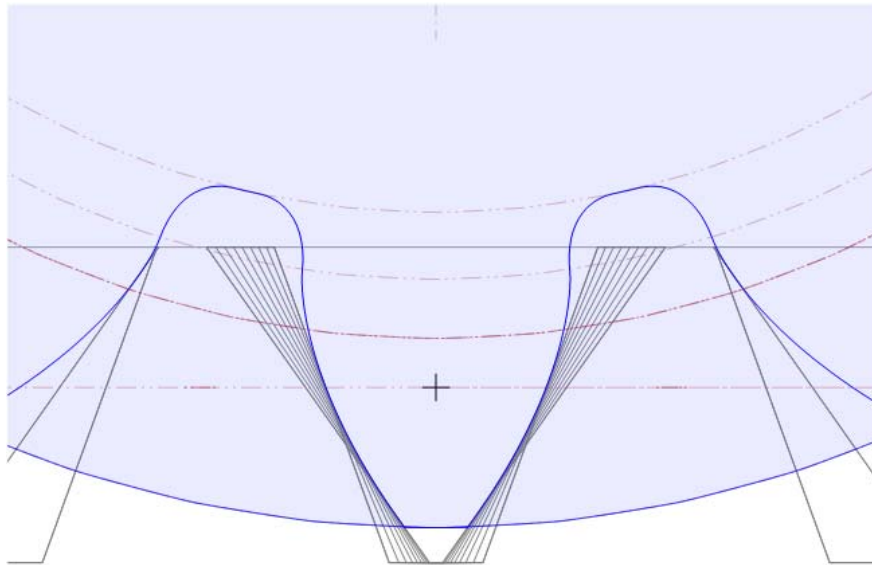
The face gear tooth lines should remain tangent to the pinion tooth. If not, check that the pinion matches the two properties (number of teeth and addendum modification) in the face gear.

Save the GearTeq components and assembly and create all in CAD.

If available, run the interference detection in your CAD system. Note any interferences. Rotate the gear mesh about 1/2 tooth and run the interference detection again. Feel free to run more detections at different positions. You may always find a very slight interference. To test even further, edit the first distance mate by adding 0.0005" and run the interference detection again. You might not find any interferences.

Notes:


- The face gear should always be the first gear in the assembly
- Any additional components after pinion gear may not be placed in the correct position.
- The face gear and the pinion need to be a set. Any changes to the pinion could affect the tooth profile of the face gear. The user must manually set two of the properties of the face gear for this to happen. Those properties are "Number of teeth face gear mate" and "Face gear mate addendum modification coefficient."
- This tutorial did not include any backlash for reasons of demonstrating the accuracy of the models. Add backlash to the pinion or the assembly position as required.
- GearTeq does not create the root fillet. That can be manually added to the loft cut or the individual sketches in CAD.
- The shaft angle is always 90 degrees and the axis of the face gear and pinion gear are coincident.

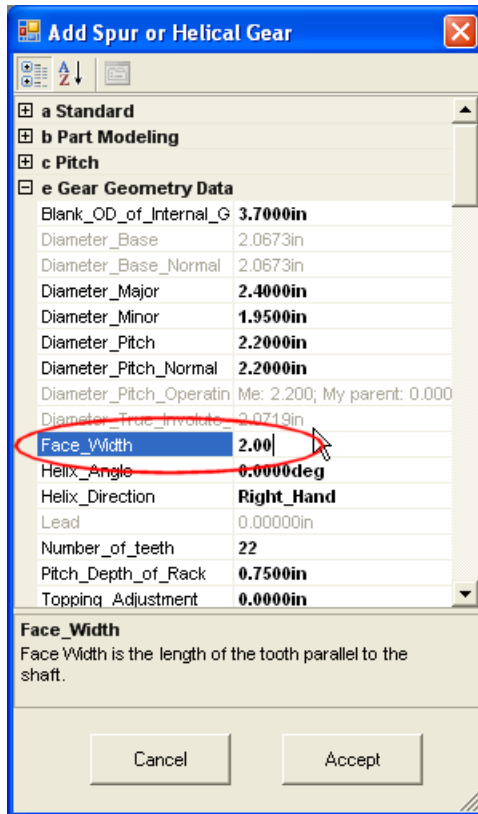


View looking down the axis of the pinion
Face gear tooth generating lines are tangent to the pinion involute

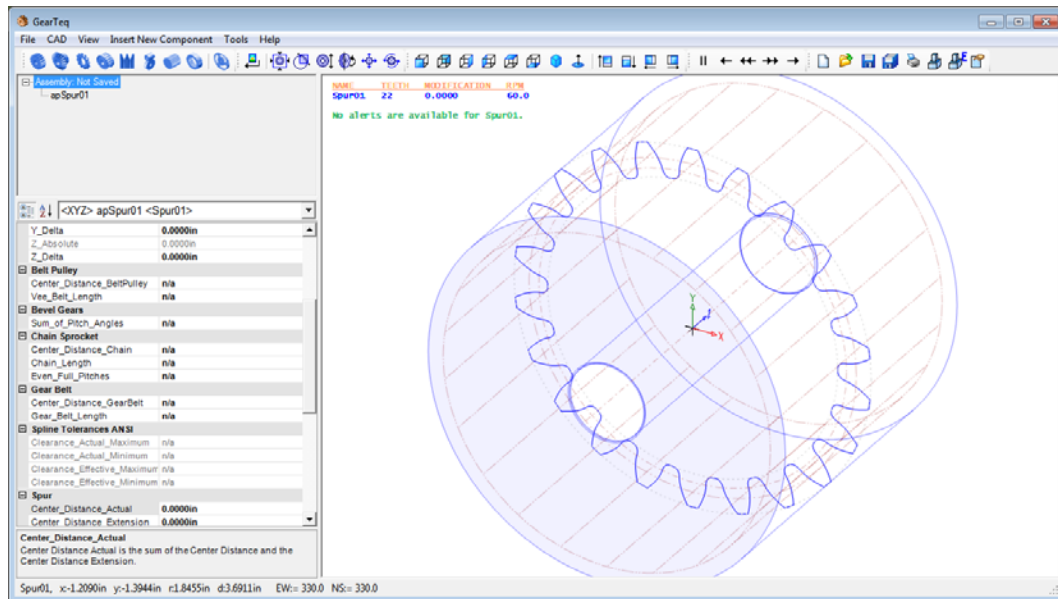
Tutorial – Creating a Spur Gear with two Racks

In this tutorial the task will be to create a single spur gear with a rack and an instance of the rack at 90 degrees. The face width for the spur gear will be 2.0 inches. The two racks will be centered on the spur gear at 1.0 between the rack centers. We will use the default diametral pitch of 10 with a 20-degree pressure angle.

Open the Add New Spur or Helical dialog window by clicking on the  icon, pressing Ctrl-S or selecting from the GearTeq menu Insert New Component>New Spur or Helical.

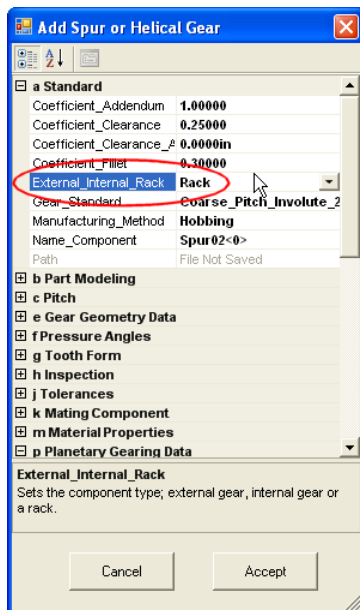


Change the face width to 2.00” and click Accept to add the component to the GearTeq assembly.

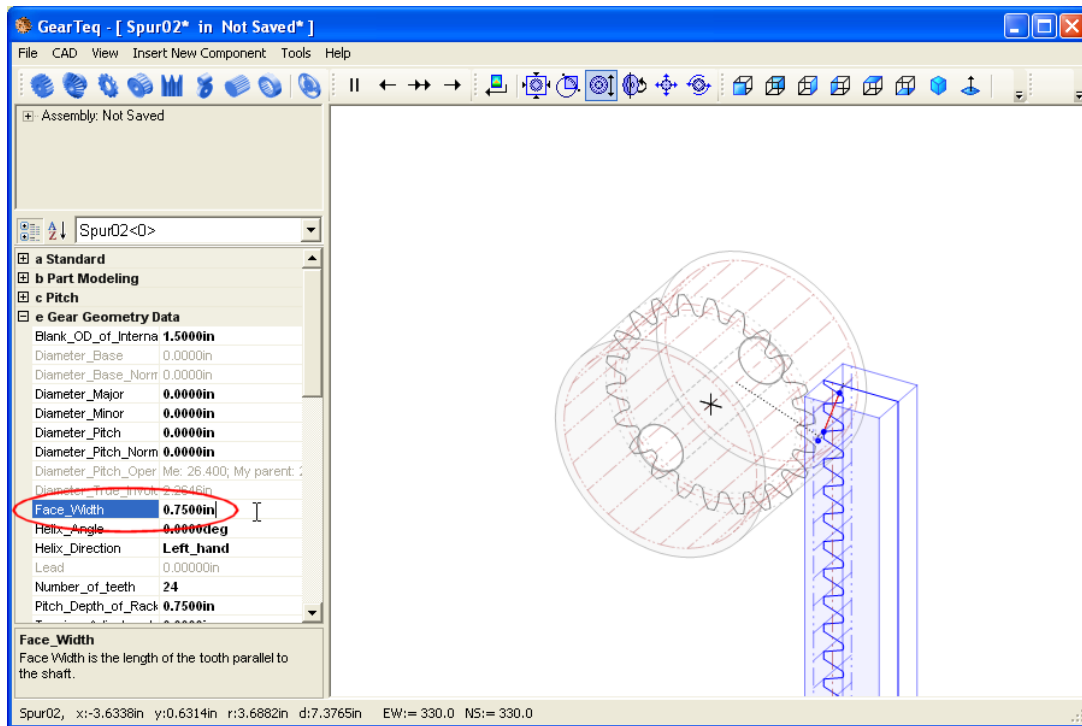


GearTeq with spur gear

Open the Add Spur or Helical Gear dialog window again to add the rack to the assembly.

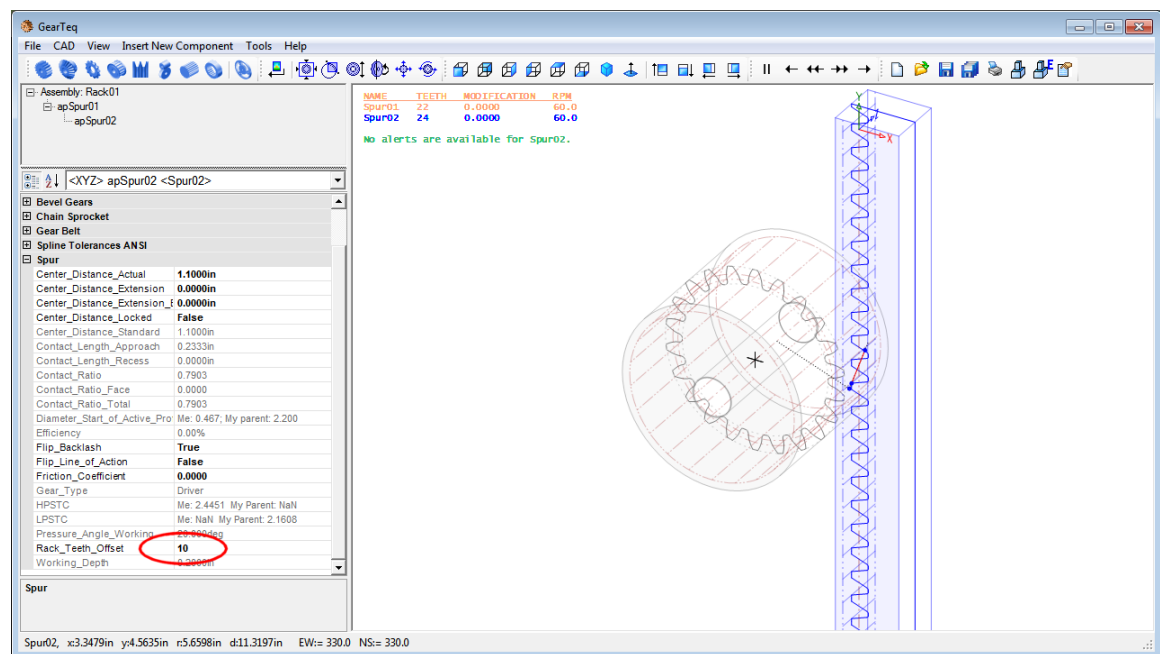


Change the External, Internal or Rack option to “Rack” and click Accept to add the rack to the assembly.



In the Mating Component category, change the Z Delta to 0.50 inches.

Activate Spur2 and change the Face Width of the rack to 0.750 inches.



Activate the apSpur02 Assembly Position, and in the Spur category, change the Rack Teeth Offset to 10 to center the rack on the spur gear.

Next a new position will be added to the assembly and it will use the same rack component. From the GearTeq menu click Insert New Component>Insert a New Assembly Position or from the Active Feature Selector click the <Inset New Assembly Position> option.

In the "Component at the location" property select Spur02.

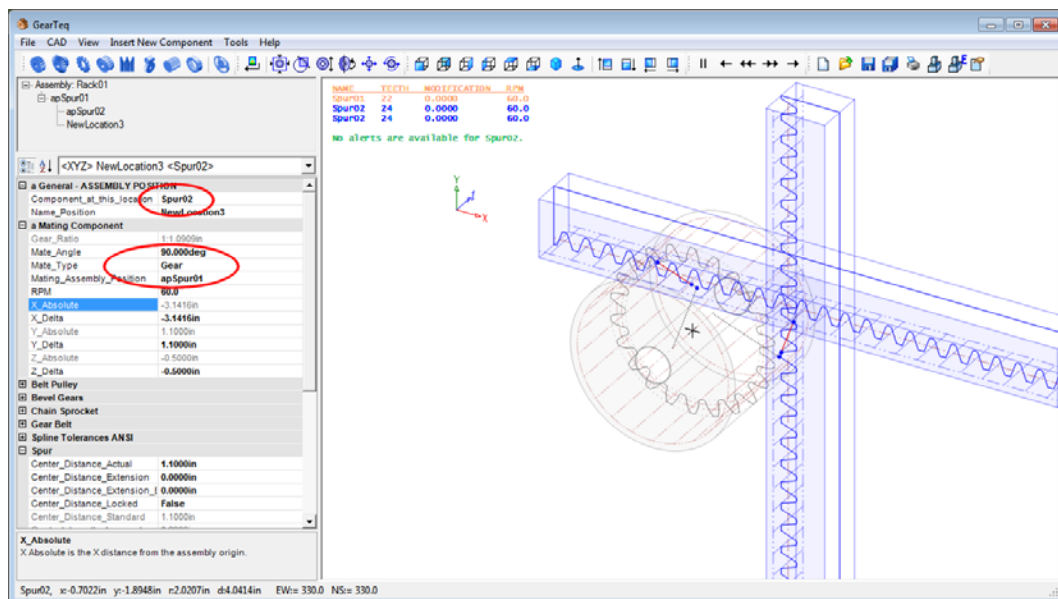
Set the Mate Angle to 90 degrees.

Set the Mate Type to Gear

Set the Mating Assembly Position to apSpur01.

Set the Z Delta property to -0.500in.

Set Rack Teeth Offset to 10 to center the rack on the spur gear to complete this tutorial.



Tutorial – Creating a Worm and Helical Gear Set

In this tutorial the task will be to create a worm and a helical gear set.

Step 1 is to add a worm gear set to GearTeq. Then configure it as if the worm wheel is the helical gear. Later we will replace the worm wheel with a helical gear that will use the parameters of the wheel.

After the worm gear set is added to GearTeq make any desired changes or leave the default values for the purpose of this tutorial.

Next add a new spur gear to the assembly. GearTeq will change the parameters of the gear to match that of the worm wheel. GearTeq will also set the "Create Worm and or Wheel" property to "Worm Only". GearTeq will add full radial crowning to the helical that is equal to 10% of the addendum length. Crowning on the helical gear is not a requirement.

If any changes are made to the worm gear set after the helical gear was added will NOT update the helical gear. Manual changes to the helical gear may be required. Another option would be to remove the helical gear from the assembly and add a new one back in.

Before using these models for fabrication do a thorough interference check. Be careful of long double-enveloping worms. The way the helical angles interact between the helical and the worm it is very possible that there will interference at the ends of the worm. The amount of interference will depend on the pitch diameter of the worm, number of starts or threads and the number of teeth on the helical.

The following properties are inherited by the helical gear from the worm wheel:

- Gear Standard
- Number of Teeth
- Helical Angle
- Normal Diametral Pitch
- Gear Direction
- Backlash
- Face Width
- Addendum Modification

Again, if any changes are made to the worm gear set those values may need to be changed in the helical gear as it is not updated automatically.

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
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


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