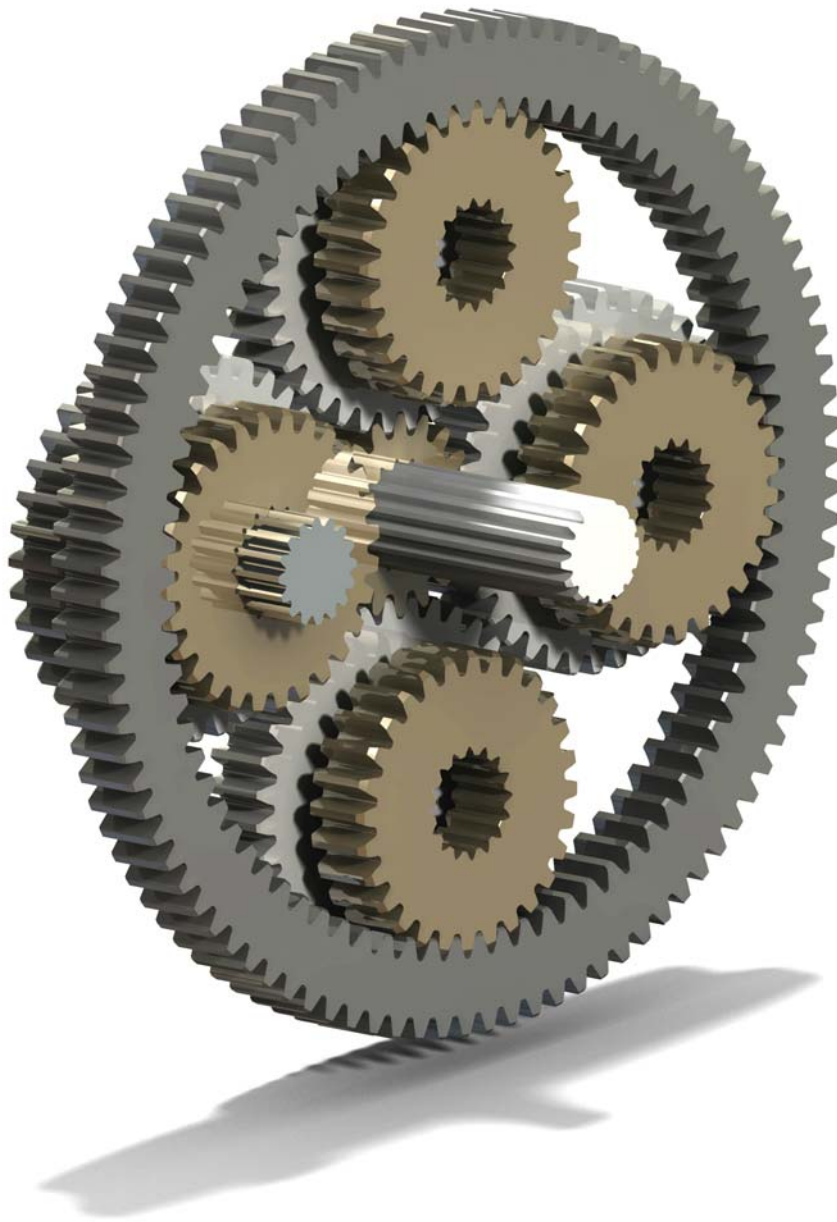


GearTeq



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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.

Component

Component is a single GearTeq item. 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.

Assembly

Assembly is a group of GearTeq components or a group of CAD parts.

Annotation

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

Model

Model is a CAD part document.

Instance

An instance of a component is an additional placement of a component in the assembly. For example, an assembly can have 6 gears, a single gear with 5 additional instances of itself.

Starting GearTeq

This release of GearTeq requires it to be started using the Windows Start menu.

GearTeq Menus

File

New

New removes all components and clears GearTeq to start a new assembly or component.

Open Components

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

Open Assemblies

Open Assemblies opens a previously saved assembly document. Component files have .gtc as the extension.

Insert Existing Component into Assembly

Inserts an existing component into the assembly. This may be turned off for the Beta release.

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 then name of the active component. The component name will change to reflect then 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 .gta as the extension and component files have the .gtc 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 .gta as the extension and component files have the .gtc 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 .gta as the extension.

Remove Component

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

Remove Instance of Component

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

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.

Recent files

Recent files are displayed. 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.

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.

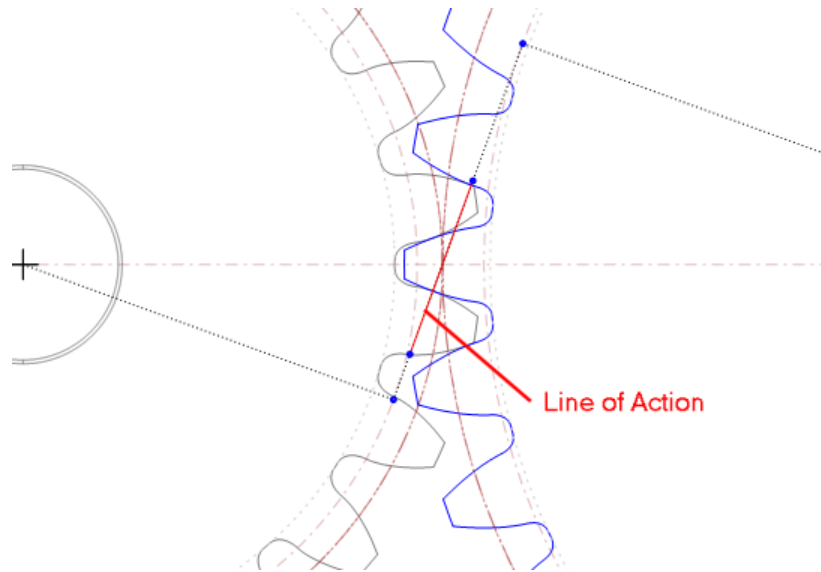
View

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.

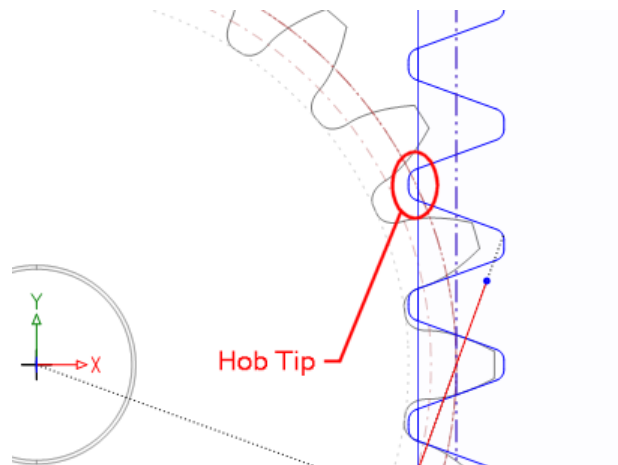
Show Line of Action

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



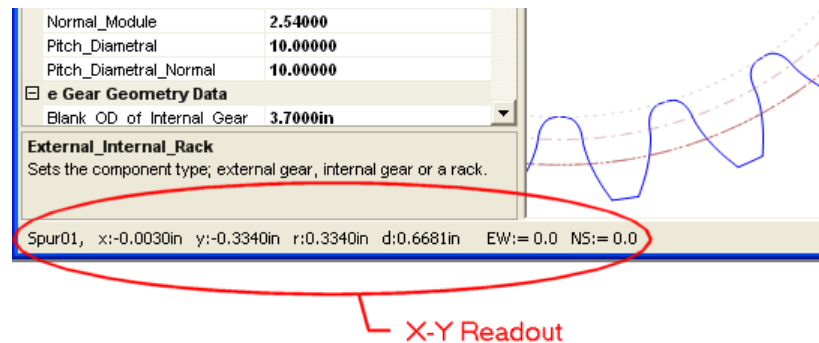
Show Hob Tip on Racks

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



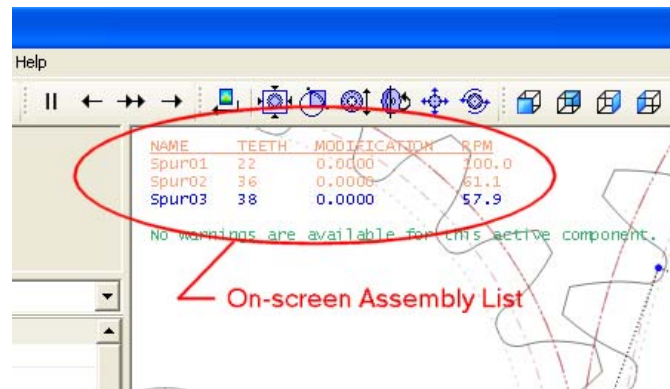
Show XY Readouts

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



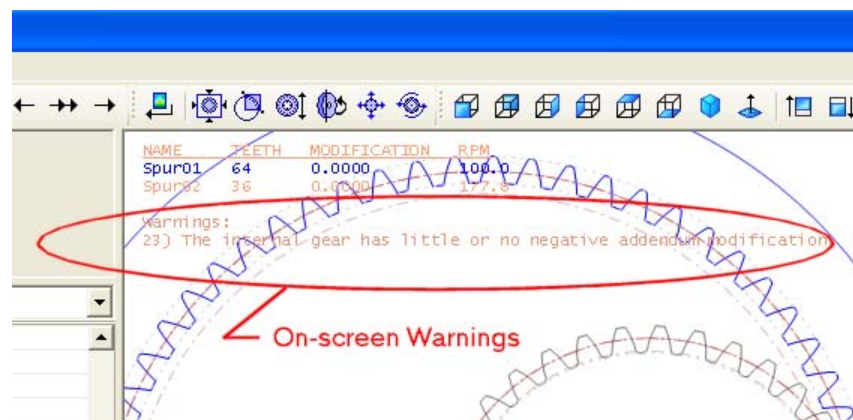
Show Assembly On-screen List

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



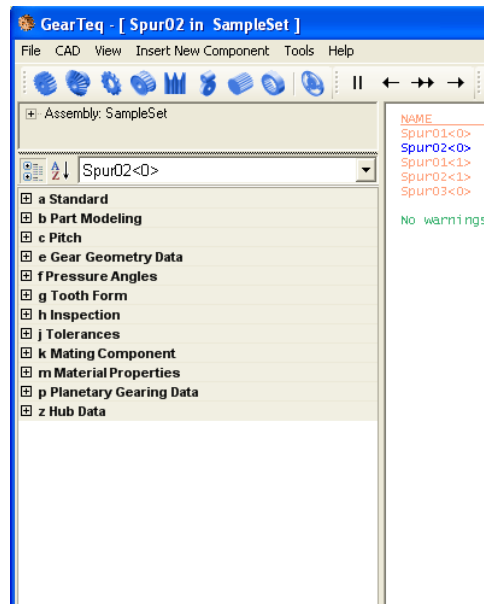
Show Warnings On-screen

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



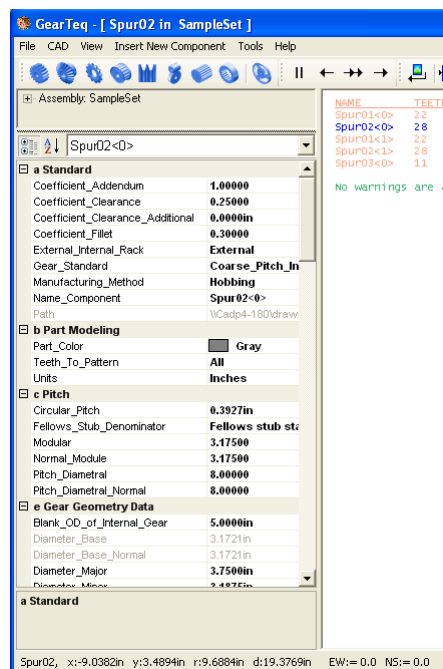
Collapse Grid

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

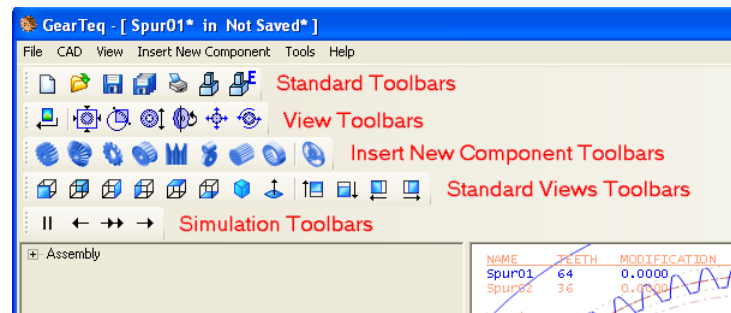


Expand Grid

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

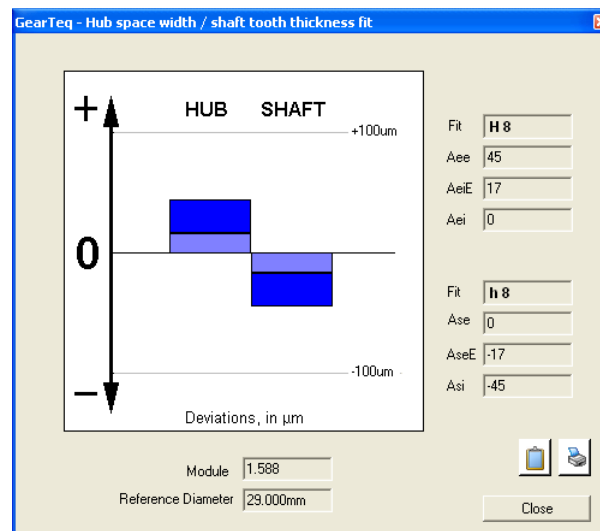
If checked, the Simulation tool bar will be displayed.

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.

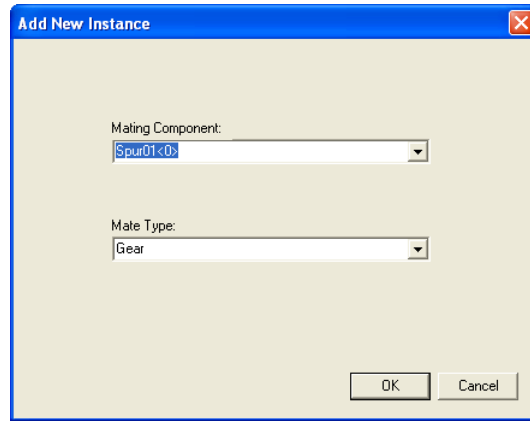


Values of the components can be changed in the property grid and the chart is updated automatically.

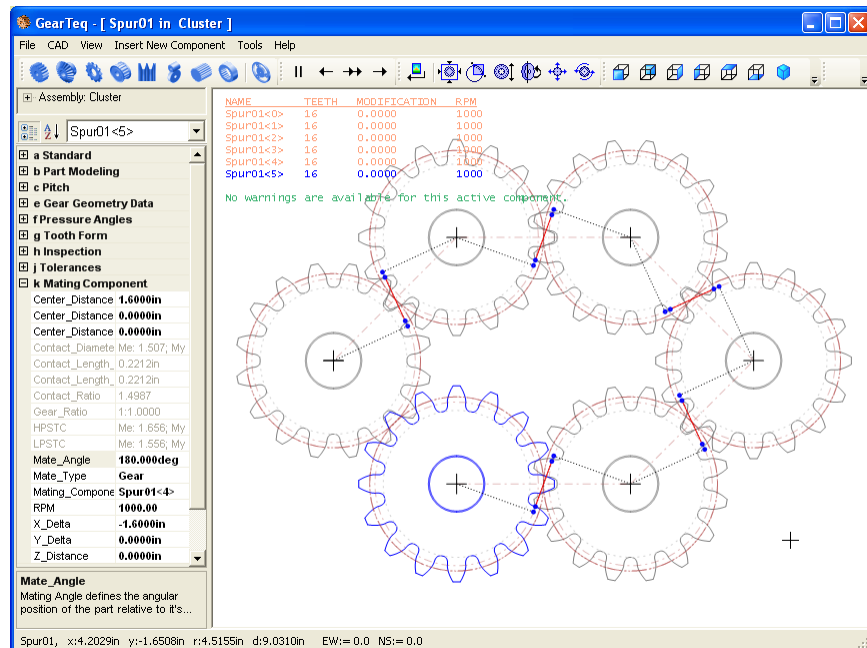
Insert New Component

Insert New Instance of the Active Component

Insert New Instance of the Active Component opens a dialog box to facilitate the inserting of a new instance of the active component.



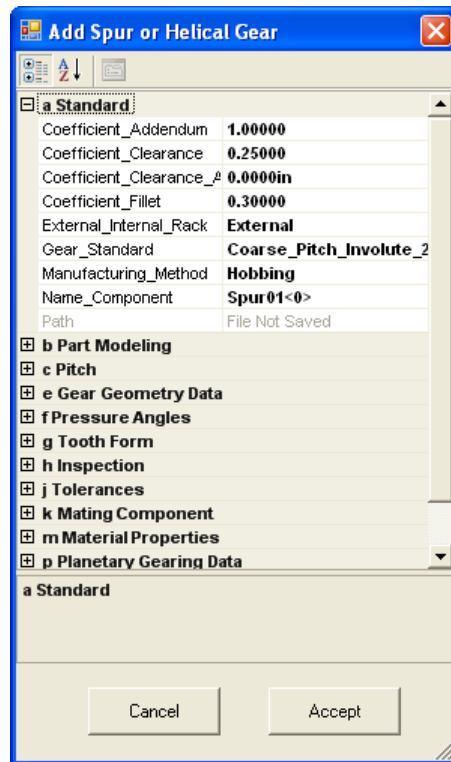
Select the mating component and the mate type. Click OK to add the new instance of the active component or Cancel to continue without adding the new instance. The mating component and the mate type can be changed after the new instance is added to the assembly.



Example of a single spur gear with 5 additional instances of that gear

New Spur or Helical...

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

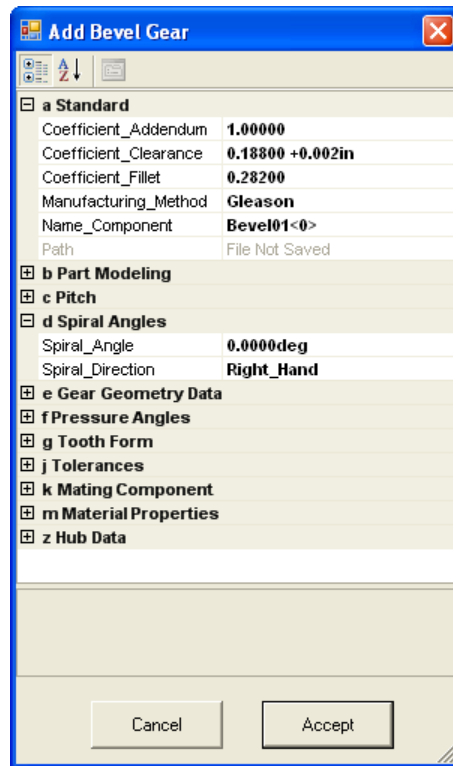


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

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.

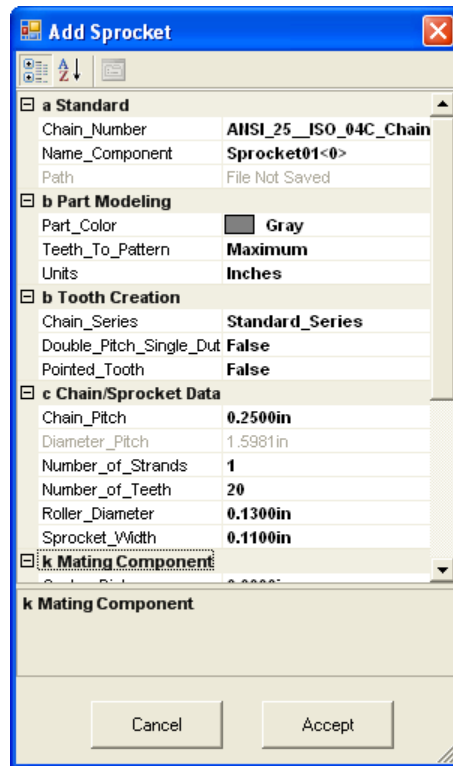


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

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.

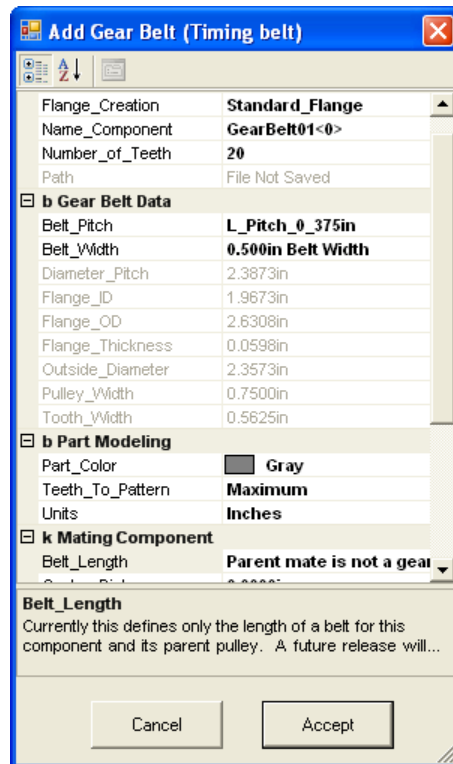


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

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.

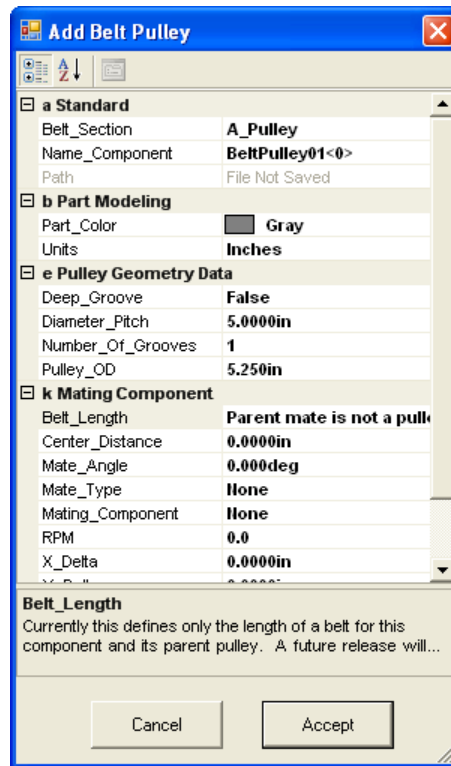


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

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.

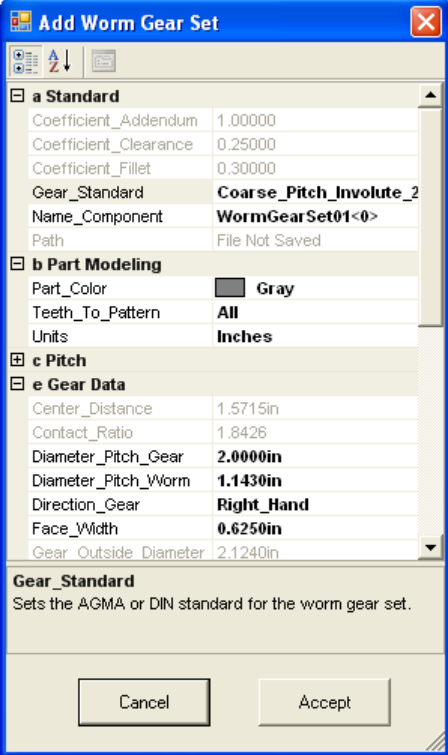


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

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.

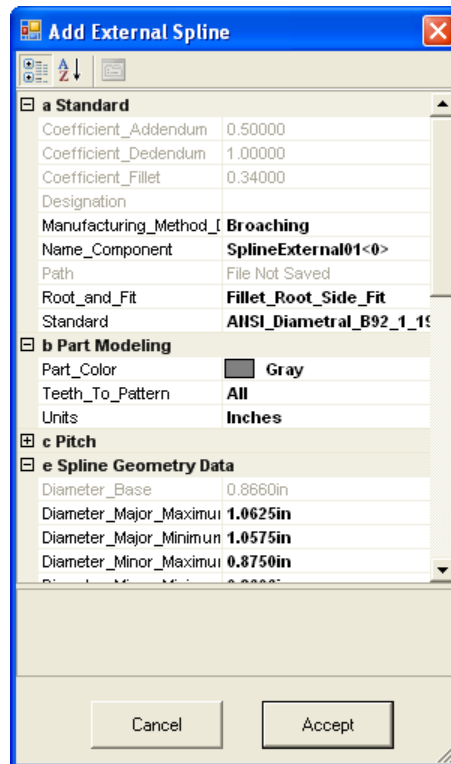
Buttons: Cancel, Accept

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.

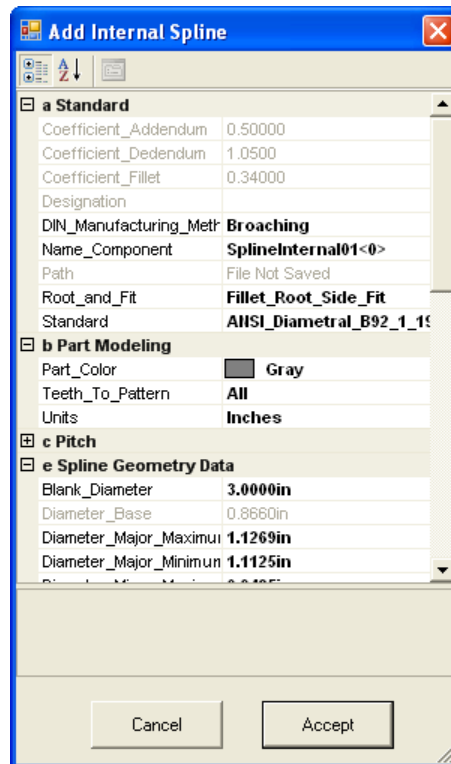


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

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.

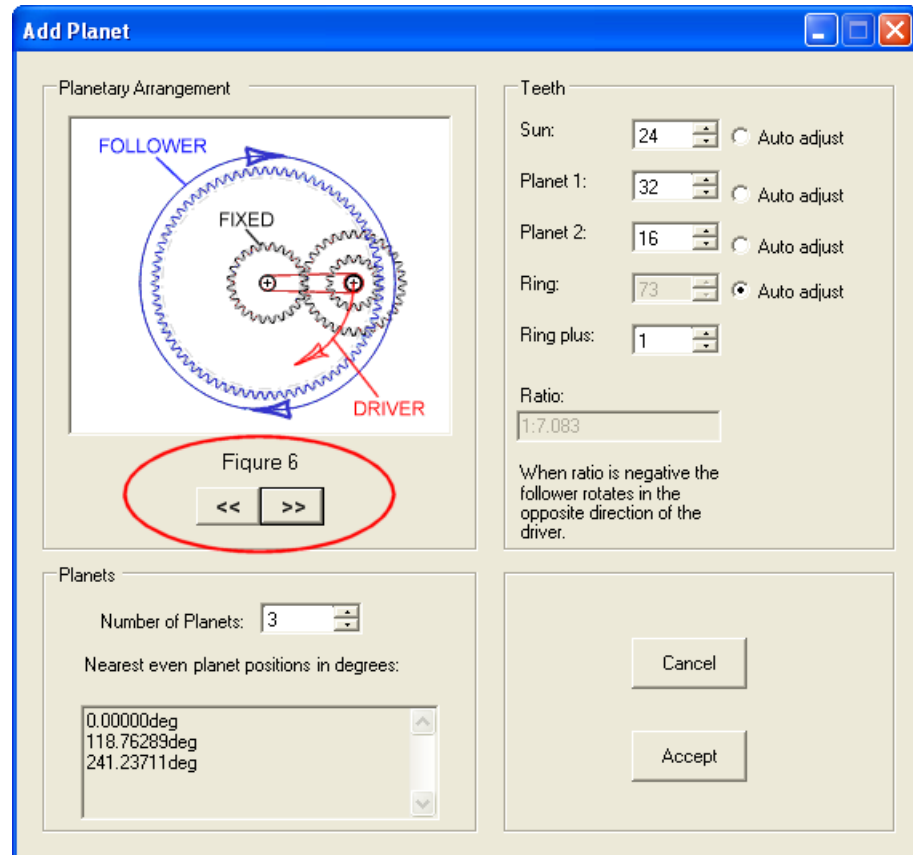


Click “Accept” to add the new component or “Cancel” to continue without adding the component. The parameters for the new component can be changed at any time, before or after adding it to the assembly.

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

New Planetary Set...

Clicking “New Planetary Set...” to create a single stage planetary gear set select from the GearTeq menu, Insert New Component>New Planetary Set to open the dialog box.



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

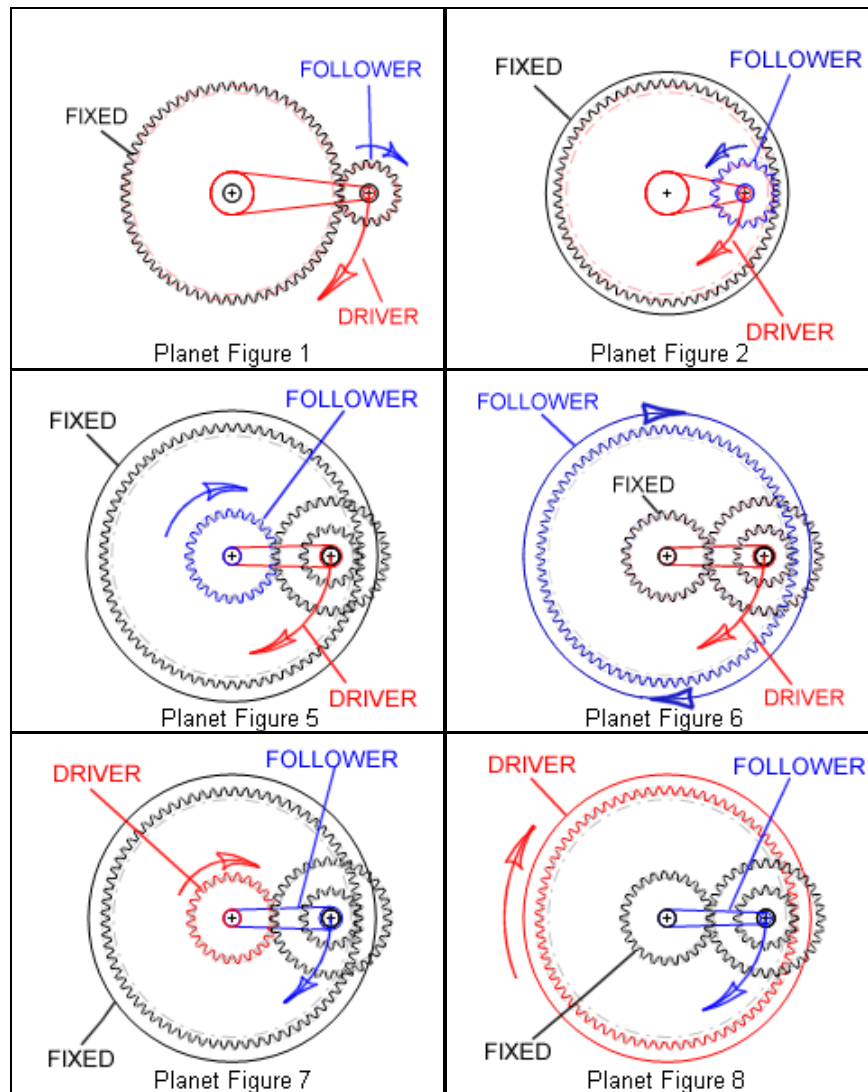
Select the number of planets for the set. The angular placement of the planets is displayed. The possible number of planet positions is equal to the sum of teeth in the sun and the ring. GearTeq displays the planet position of the nearest equally spaced position.

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

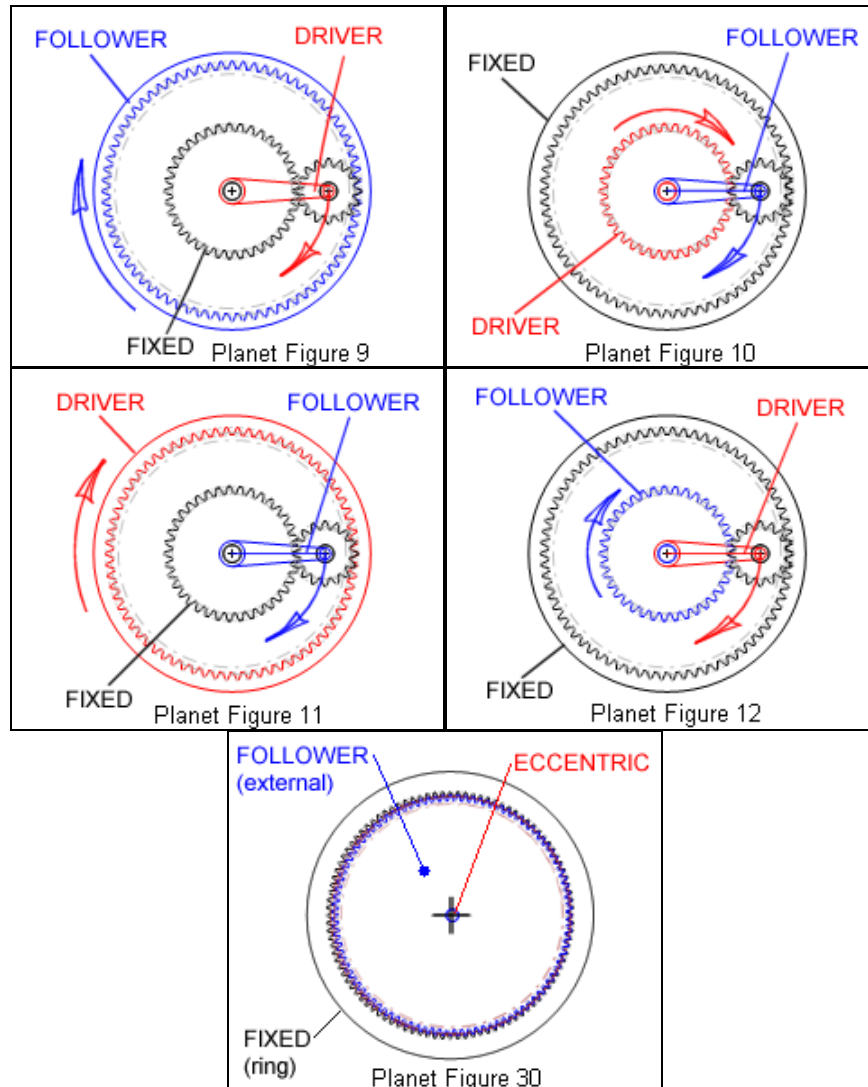
responsible for assuring the number of teeth are appropriate if any one component's number of teeth is changed.

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 it is in the original assembly.



Standard Planet Layouts Available in GearTeq



Standard Planet Layouts Available in GearTeq

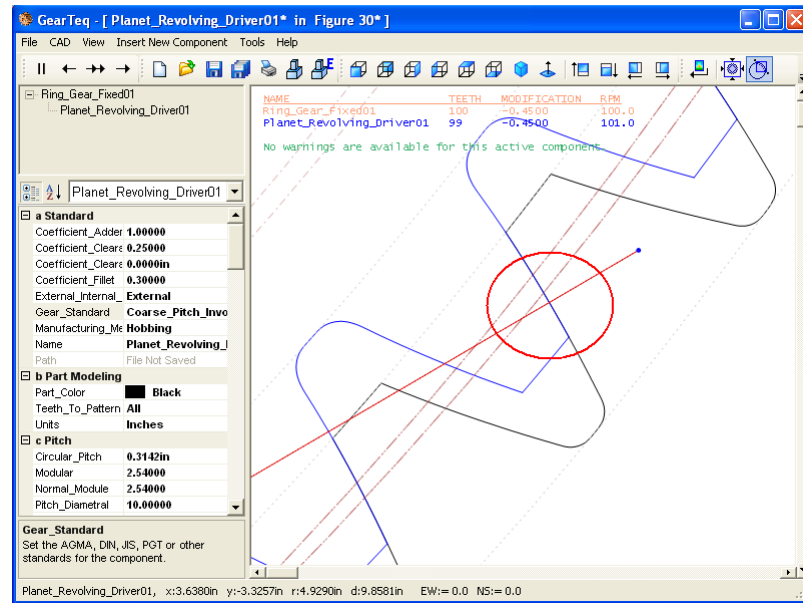
Cycloid Gear Sets

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.

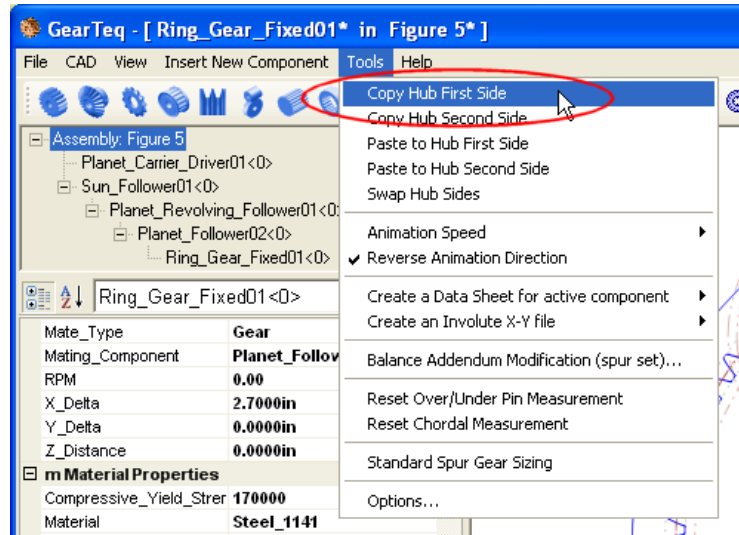
The carrier plate is created by the CAD system. The offset of the eccentric should be equal to the center distance displayed for the external component. It is up to the designer to select the best method to tie the external component to an output device, such as a plate. This plate may contain a number 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

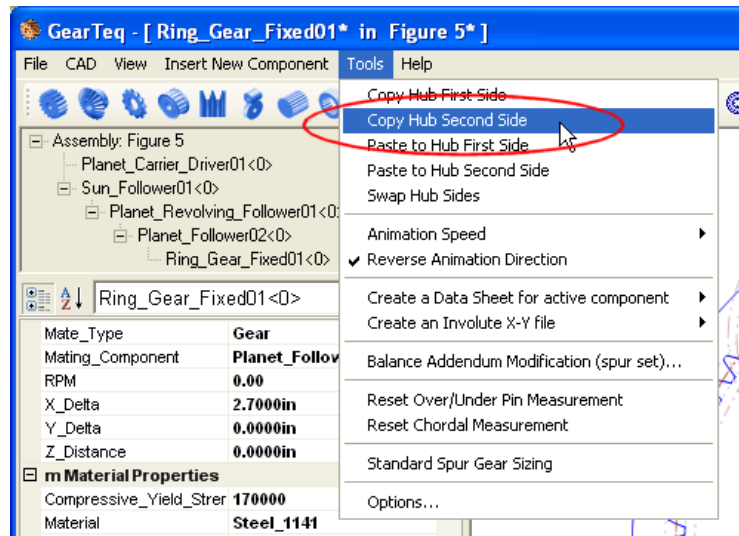
Copy Hub First Side

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.



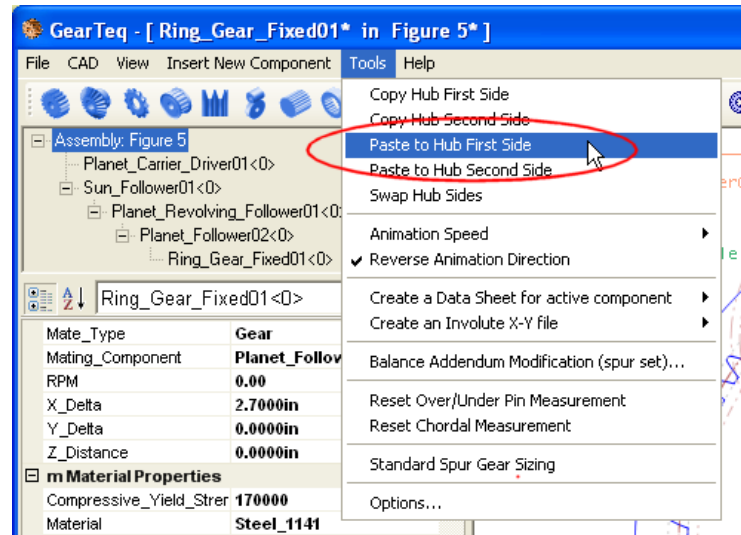
Copy Hub Second Side

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.



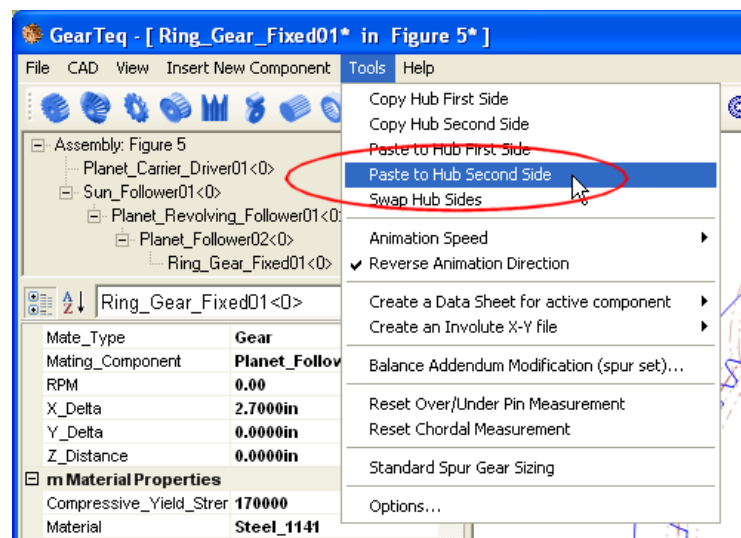
Paste to Hub First Side

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.



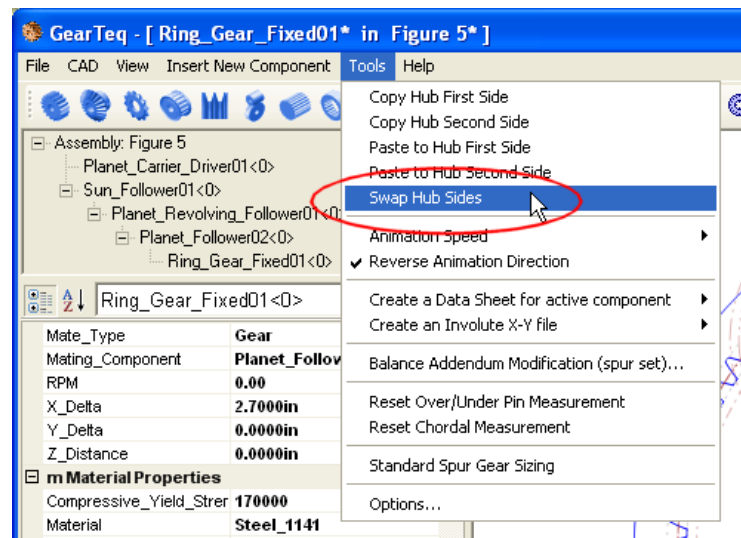
Paste to Hub Second Side

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.



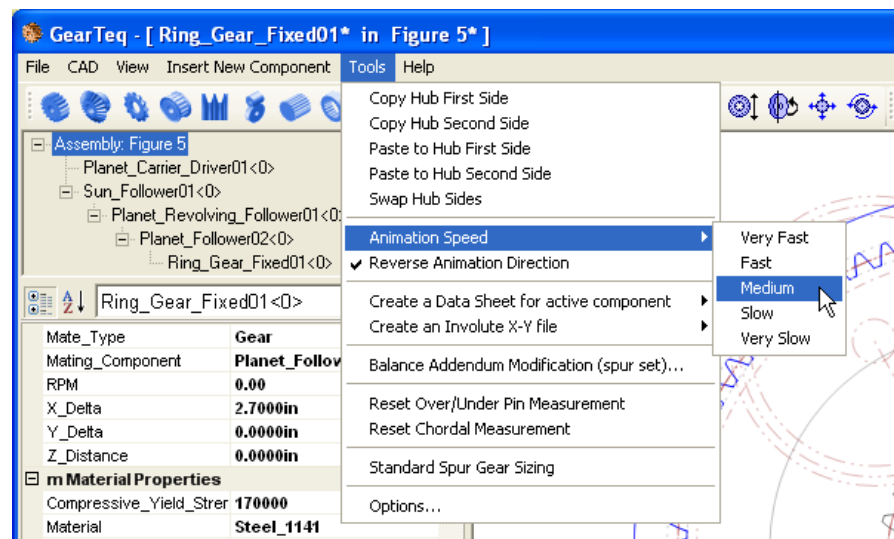
Swap Hub Sides

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



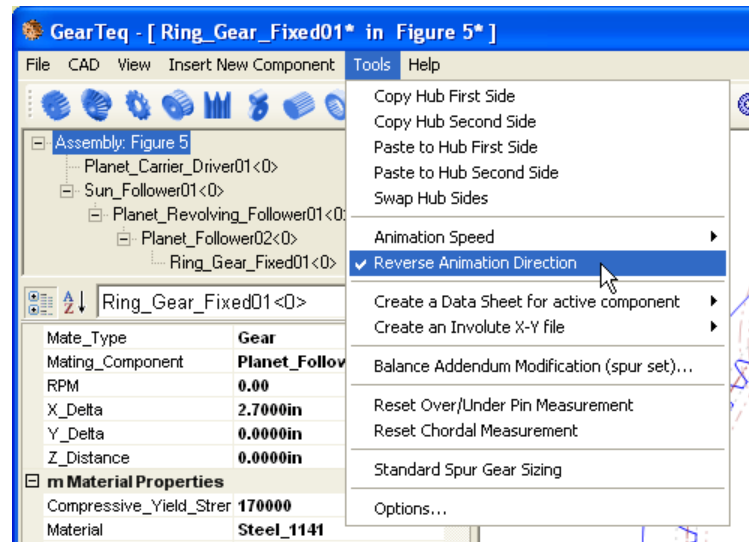
Animation Speed

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.



Mate_Type	Gear
Mating_Component	Planet_Follow
RPM	0.00
X_Delta	2.7000in
Y_Delta	0.0000in
Z_Distance	0.0000in

Material Properties	
Compressive_Yield_Str	170000
Material	Steel_1141

Balance Addendum Modification (spur set)...

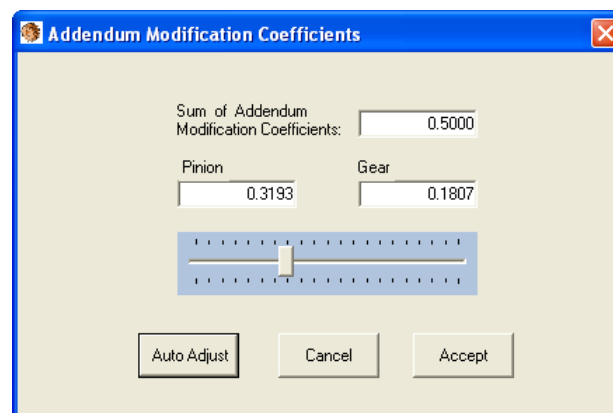
The Balance Addendum Modification menu option opens a dialog window that can adjust the addendum modification for the active gear and its mate. The mate must be a spur gear with a gear mate.

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.

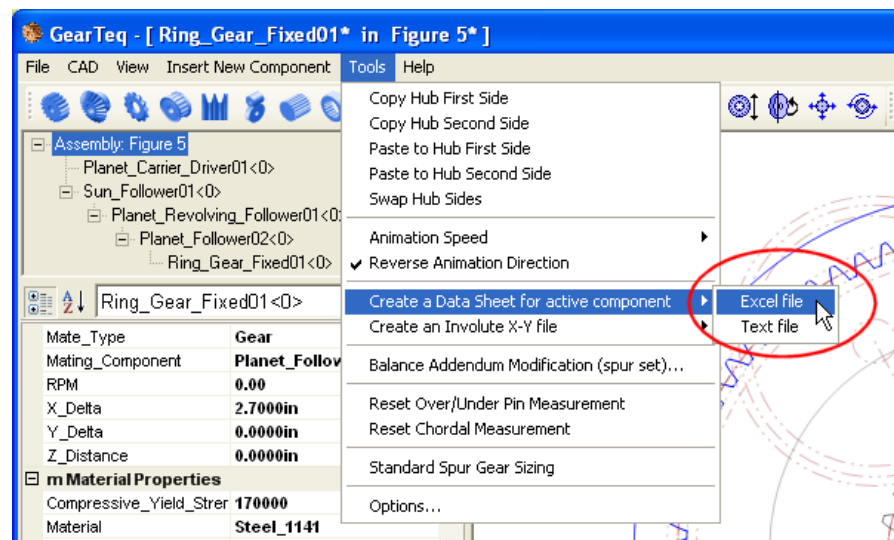
Create a Data Sheet for active component

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.



Spur03.txt - Notepad

Name: Spur03
Path:
Date:
Time:

SYMBOL	VALUE	UNIT	TERM
	Coarse_Pitch_Involute_25deg		Standard
Pdn	8.000000		Normal Diametral Pitch
Pd	8.000000		Diametral Pitch
m	3.175000		Normal Modular Pitch
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
	0		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

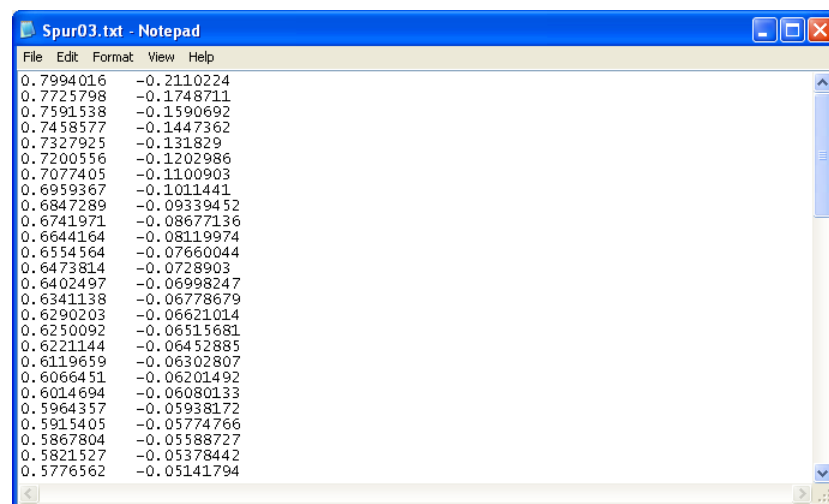
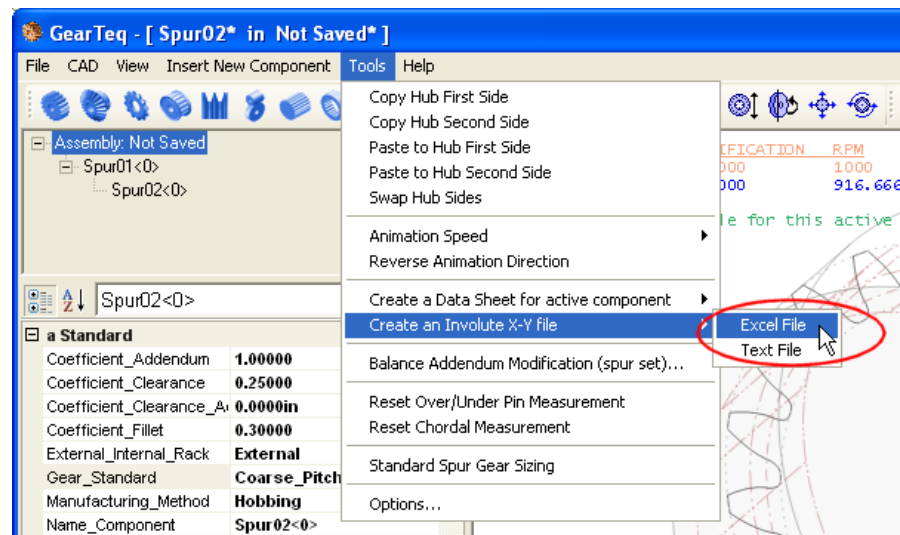
Create an Involute X-Y file

Excel file

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

Text file

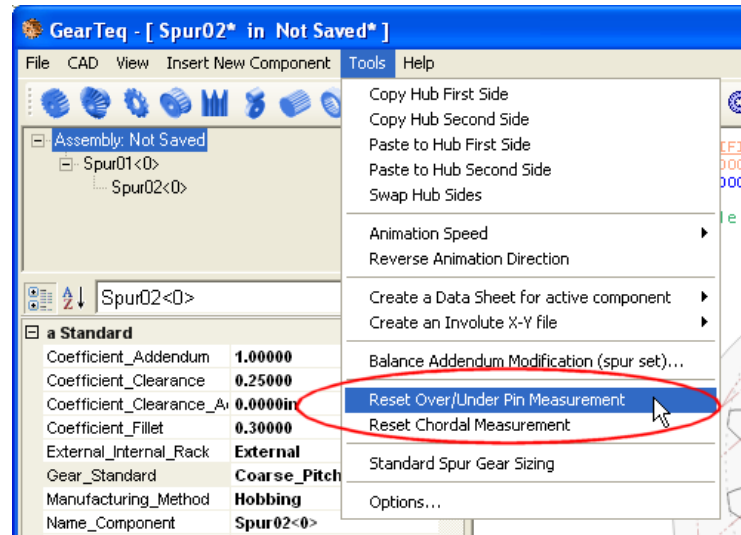
Clicking 'Text file' creates an X-Y file of the points along the tooth in Microsoft Notepad, if available.



Sample of an XY file created as a text file

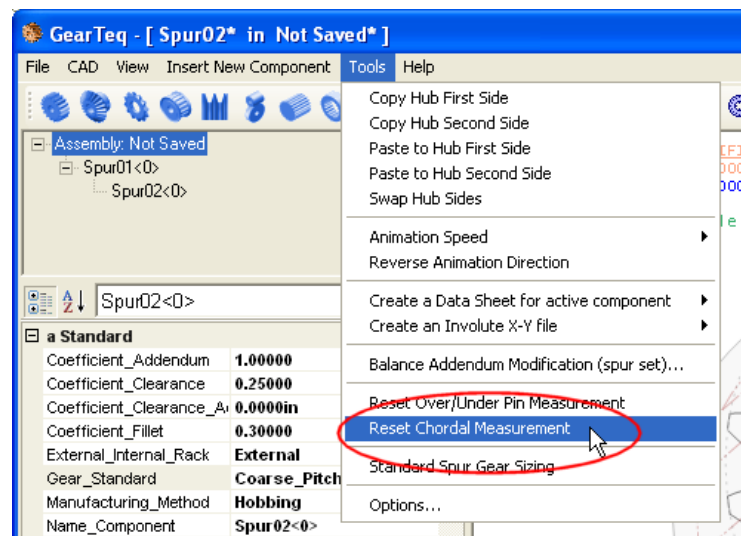
Reset Over/Under Pin Measurement

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.



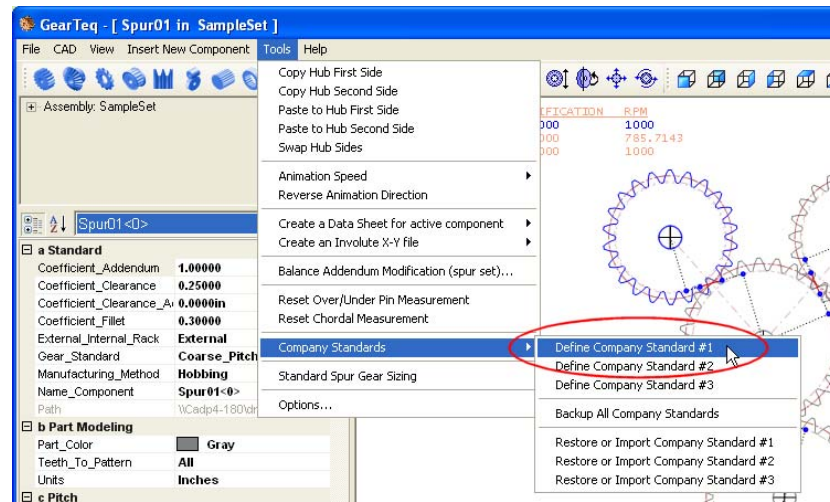
Reset Chordal Measurement

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

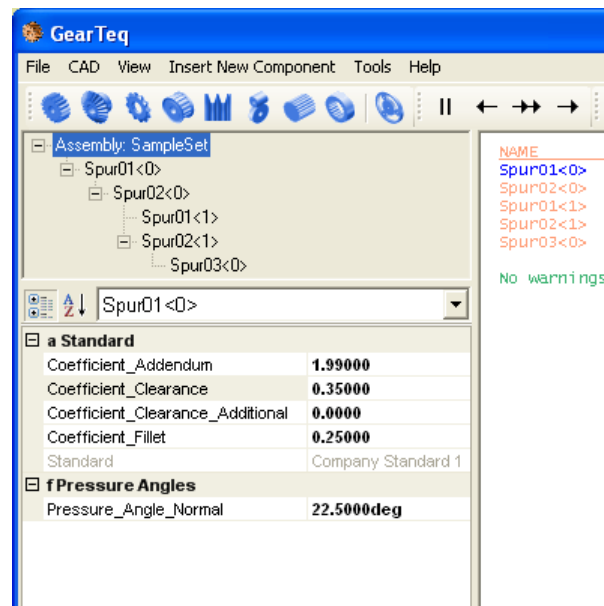


Define Company Standards

Define 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 and 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

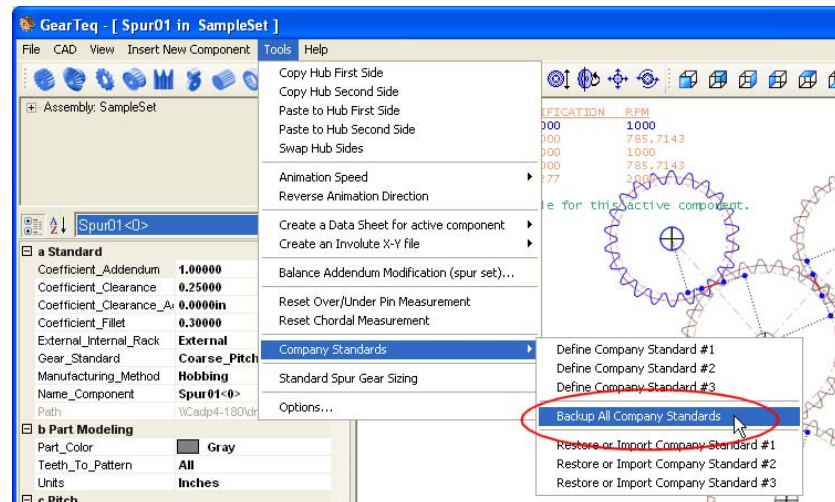
Standard is a read only display of which standard is being edited.

Pressure Angle Normal

Pressure Angle Normal defines the pressure angle for the standard.

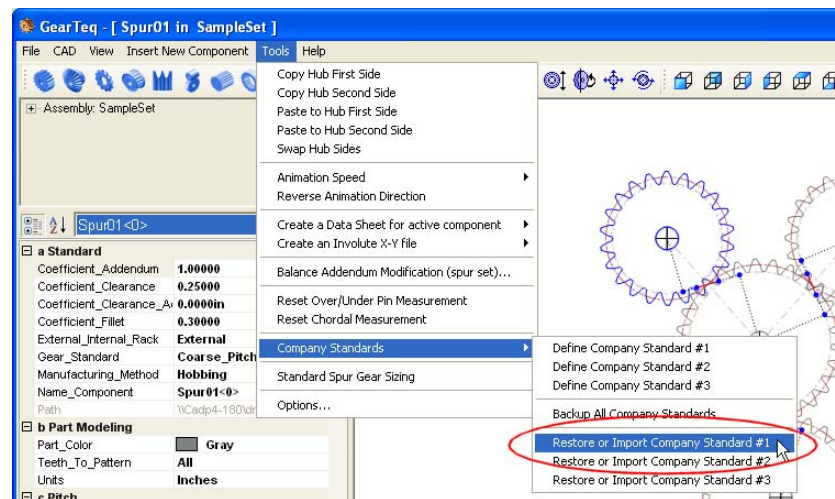
Backup All Company Standards

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 user can have access to them.



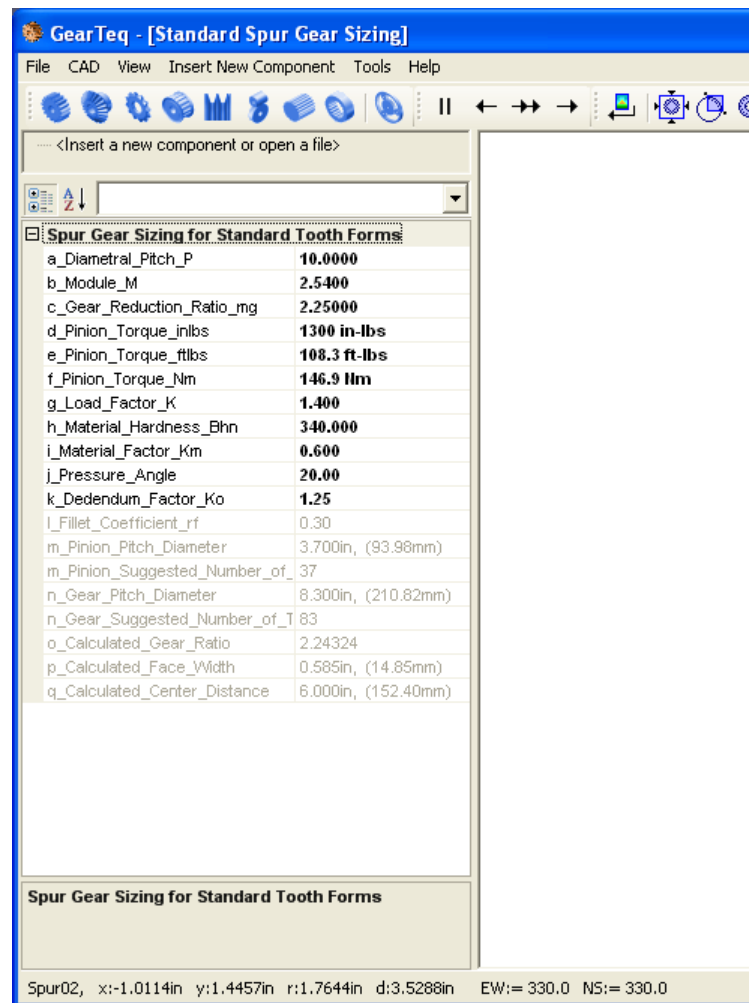
Restore or Import Company Standards

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:

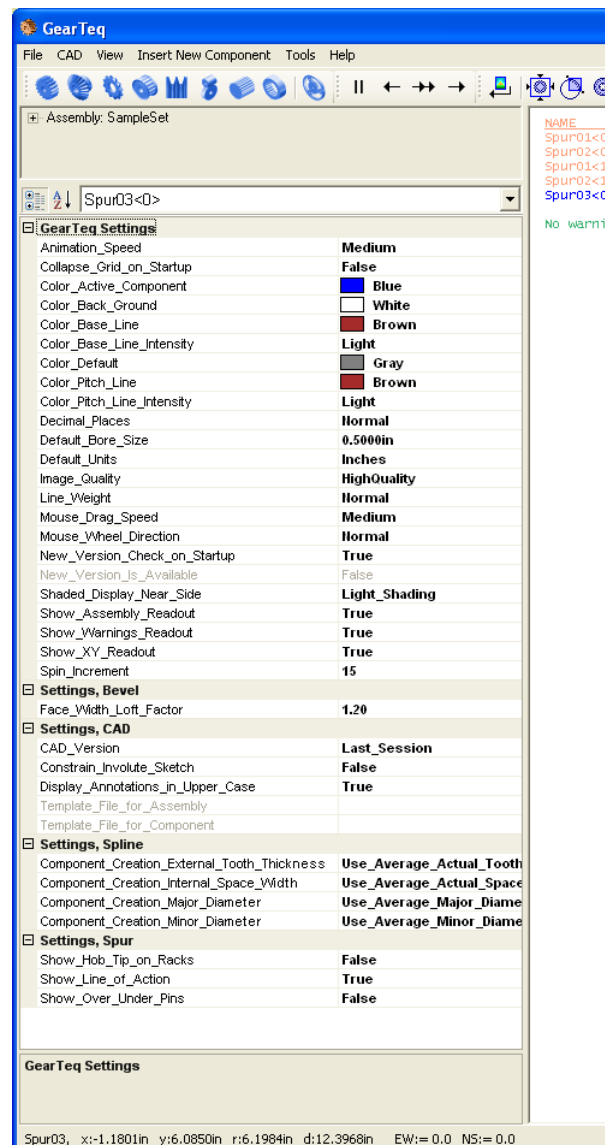
- Diametral pitch P
- Module M
- Gear reduction ratio mg, must be greater or equal to 1.0
- d, e, f) Pinion torque, in-lbs, ft-lbs or Nm
- Load factor K
- Material hardness Bhn, Brinell hardness number, the material must be steel
- Material factor Km, 0.6 for through-hardening steels, 0.9 for case-hardening 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

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

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

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

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 quality the components that are display in the GearTeq window. This value is saved as a default for the next time GearTeq is started.

Involute Precision

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

Low: 10

Medium: 20

High: 40

Very High: 80

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.

New Version Check on Startup

New Version Check on Startup defines if GearTeq should check if a newer version is available on the web site. This value is saved as a default for the next time GearTeq is started. Access to the Internet is a must for this option to function.

New Version is Available

New Version Is Available is a read only property that states if a newer version is available on www.camnetics.com. Access to the Internet is a must for this option to function.

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 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 Warnings Readout

Show Warnings Readout defines if the warnings 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

Face Width Loft Factor

Face width Loft Factor defines how much longer the loft should be to facilitate the complete lofting of the bevel tooth cut.

Settings, CAD

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.

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.

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.

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.

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.

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.

Blank OD of Internal Gear

Blank OD of Internal Gear is the diameter of the blank used to create the internal gear.

Chordal Measurement

Chordal Measurement is a straight-line measurement across the Number of Teeth to Gage Over.

Chordal Measurement Minimum

Chordal Measurement is the minimum straight-line measurement across the Number of Teeth to Gage Over.

Chordal Measurement, Teeth to Gage Over

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

Chordal Tooth Height

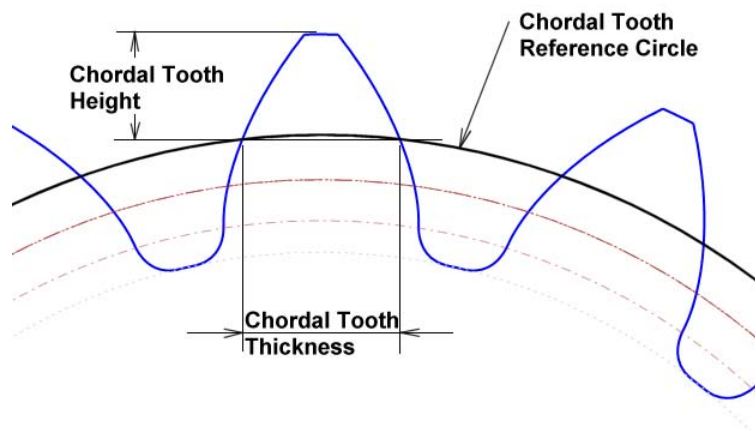
Chordal Tooth Height is the length of the tooth portion to be measured at which the chordal tooth thickness is calculated. Twice the chordal tooth height plus the chordal tooth reference circle should equal the outside diameter of the gear.

Chordal Tooth Reference Circle

Chordal Tooth Height 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.

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.

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.

Diameter, Pitch, Normal

The 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, 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.

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 or a rack.

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 Ratio

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

Gear Standard

Gear Standard sets the AGMA, DIN, JIS, PGT or other standards for the component.

Gear Type

Gear Type displays the type of gear when used in a planetary gear set.

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

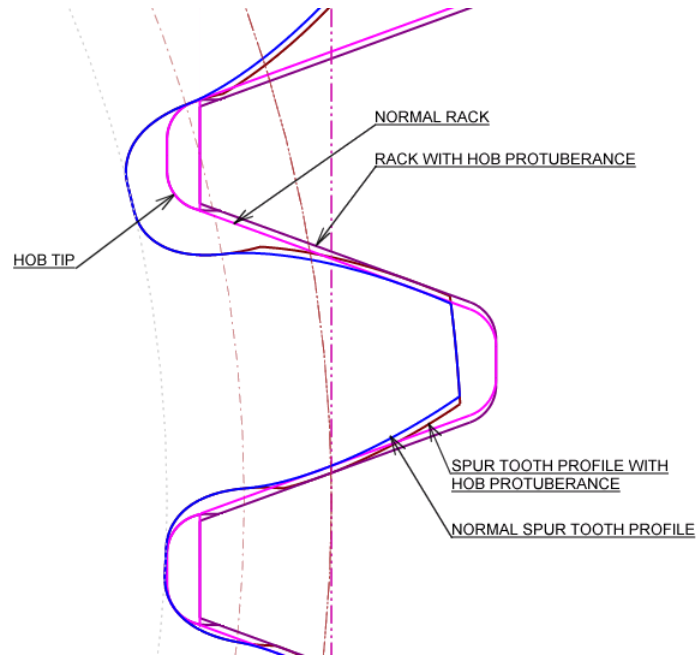
Helix Direction defines the direction of the angle. The teeth on a left hand helical gear on a horizontal surface lean to the left.

HPSTC (Highest Point of Single Tooth Contact)

HPSTC (Highest Point of Single Tooth Contact) The gear must have a mating parent for the value to be greater than zero.

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.

LPSTC (Lowest Point of Single Tooth Contact)

LPSTC (Lowest Point of Single Tooth Contact) The gear must have a mating parent for the value to be greater than zero.

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.

Mate Type

Mate Type is the type of mate a part has to its parent part. Each part can have only one parent while a part may have many child parts.

Mating Component

Mating Component is the name of the active component's parent. Only similar components can have gear, belt or chain mate types. Only internal splines can have a bore mate. Any component may have a joined mate to any other component.

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 BPL 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 APL 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 which side of the component the mounting is referenced.

Name

Name defines the name of the part.

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 closet position evenly divided in 360 degrees.

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.

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.

Planet Mode

Planet Mode as defined in Machinery's Handbook. Modes 1,2, 5 through 12 are available.

Planet Positions

Planet Positions displays the angular positions of the planets to the closet 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.

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. Normally the same as pressure angle unless either gear in the set has been modified or the center distance has been changed.

Ratio of Driver to Follower

Ratio of Driver to Follower is the speed ratio of the input gear to the output gear.

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

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%), $0.0025 = 0.25\%$

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.

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.

Topping Adjustment

Topping Adjustment will shorten then 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.

Variation, Pitch

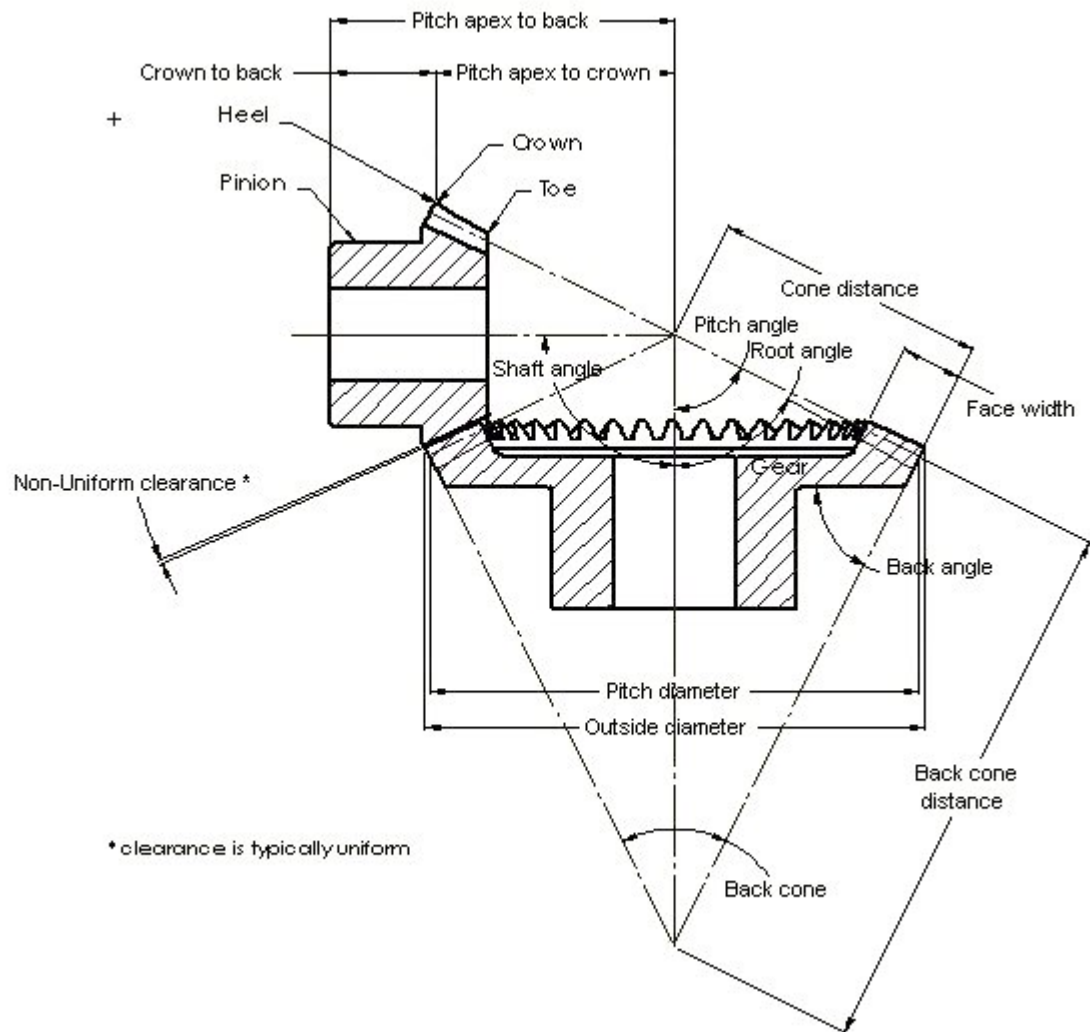
Pitch Variation is the algebraic (+ or -) plus or minus difference in the transverse plane, between the true position pitch, and an actual pitch measurement. If measured in a plane other than the transverse plane, a correction using the appropriate helix angle must be applied to the measured value according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

Backlash

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

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 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.

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 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.

Gear Ratio

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

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.

Mating Component

Mating Component is the name of the active component's parent. Only similar components can have gear, belt or chain mate types. Only internal splines can have a bore mate. Any component may have a joined mate to any other component.

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.

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.

Z Distance

Z Distance is the distance between the current component's apex point and it's parent component on the Z Axis.

Chain Sprockets

Chain Length

The 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

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.

RPM

RPM is the revolutions per minute of the current component.

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 Length

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.

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.

Center Distance

Center distance defines the distance between two gear belt pulleys.

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 Length

Belt length defines the simple length of a belt between two belt pulleys. Entering a value will change the center distance. Changing the center distance will change this value. This does not take into consideration multiple pulleys and/or idles.

Belt Section

Belt Section defines the belt section for the component.

Center Distance

Center distance defines the distance between two gear belt pulleys.

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.

RPM

RPM is the revolutions per minute of the current component.

Units

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

Worm 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.

Addendum Modification Coefficient

Addendum Modification Coefficient is the ratio of change of a modified wheel tooth. This value can be positive or negative. This value does not affect the worm.

Bore, Chamfer

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

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 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 0.080in.

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 gear tooth root.

Contact Ratio

Defines the contact ratio for the worm gear set.

Dedendum

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

Diameter, Pitch, Gear

Pitch Diameter is a theoretical diameter of the gear.

Diameter, Pitch, Worm

Pitch Diameter is a theoretical diameter of the worm.

Diametral Pitch

Diametral pitch is used for imperial system gears.

Direction, Gear

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

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 AGMS, DIN, JIS, PGT or other standards for the component.

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

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 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.

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 it's 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 normal to the cutter.

RPM

RPM is the revolutions per minute of the current component.

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.

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.

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.

Ase

(DIN 5480) Upper tooth thickness deviation

AseE

(DIN 5480) Upper tooth thickness deviation for individual measurement

Asi

(DIN 5480) Lower 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.

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.

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.

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 gear tooth root.

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, Major, Maximum

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

Diameter, Pitch

The pitch diameter is the theoretical diameter of the spline.

Diameter, True, Involute Form**DIN Manufacturing Method**

Sets the manufacturing method for the component. The manufacturing method has a direct effect on the dedendum depth.

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.

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 it's specified color, set the Color_Active_Component option to Transparent.

Path

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

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, Circular

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

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 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.

Standard

Standard, select between 4 different standards:

ANSI Diametral B92.1 1996

ANSI Module B92.2M 1980 R1989

DIN 5480

JIS B 1603

T

Dimensional tolerance

TE

Individual tolerance (for individual 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.

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.

Tolerance Position

(DIN 5480)

Tooth Thickness Actual Maximum

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

Tooth Thickness Actual Minimum

Actual tooth thickness is the minimum 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.

Mating

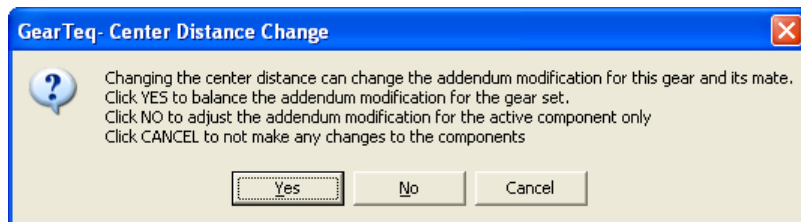
Belt Length

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

Center Distance

Center Distance is the distance between this component and it's parent.

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.



Click YES to 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

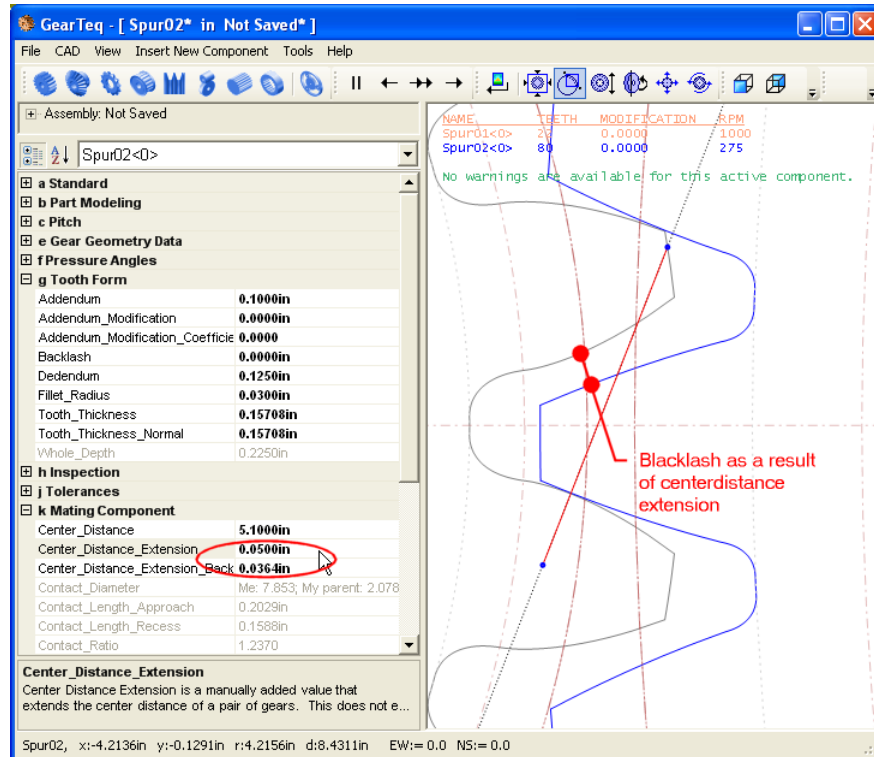
Click NO to change only the addendum modification of the active component.

Click CANCEL to exit without making any changes to the active component or its mate.

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

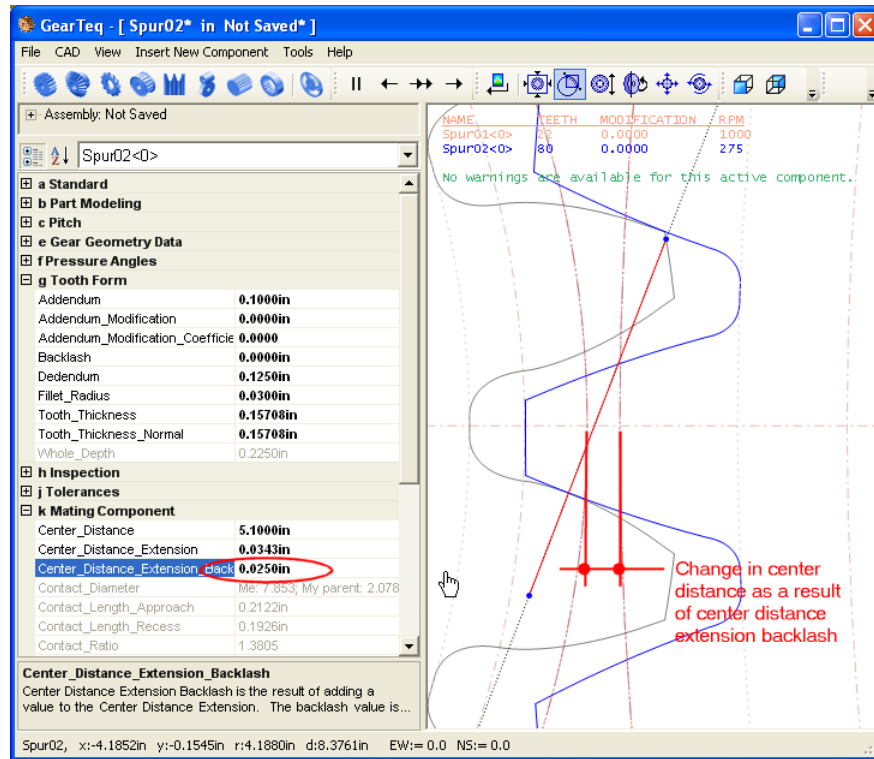
Center Distance Extension is a manually added value that extends the center distance of a pair of gears. This does not affect the geometry of either gear in the pair. The mesh may not be displayed correctly.



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.



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.

Contact Diameter

Contact Diameter is the smallest diameter on a gear tooth with which the mating component makes contact.

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.

Even Full Pitches

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

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.

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 Ratio

Gear Ratio is the speed ratio of the component relative to its parent component.

HPSTC

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 is the smallest diameter on a spur gear at which a single tooth is in contact with the mating gear.

Mate Angle

Mate Angle defines the angular relationship between the component and its parent component. A zero degree angle would place the component at 3:00 relative to its parent and a 270 degree angle would place it at 6:00.

Mate Type

Mate Type defines the type of mate relationship this component has with its parent. Each component may only have one parent component but a single component may have any number of child components.

Restrictions on the types of mates depend on the type of components that are to be mated.

Gear Mate can only be applied to spur gear sets or bevel gear sets.

Shaft mates can be applied to any pair of components.

Belt or Chain mates can only be applied to pulley sets, gear belt sets or chain sprocket sets.

A bore mate can only be applied to an internal spline as a mate to any other type of component.

Joined mate can only be applied to a pair of spur gears.

Mating Component

Mating Component is the name of the active component's parent. If "None" is selected then the component will be at X zero and Y zero. The Z distance can be defined.

Rack Teeth Offset

Rack Teeth Offset moves a rack the specified number of teeth in the assembly.

RPM

RPM is the revolutions per minute of the current component.

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 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 Distance

Z Distance 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.

X Absolute

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

Y Absolute

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

Z Absolute

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

Material Property

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

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 Diameter 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 Diameter 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



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

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 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 by dragging 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 by dragging 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 by dragging 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

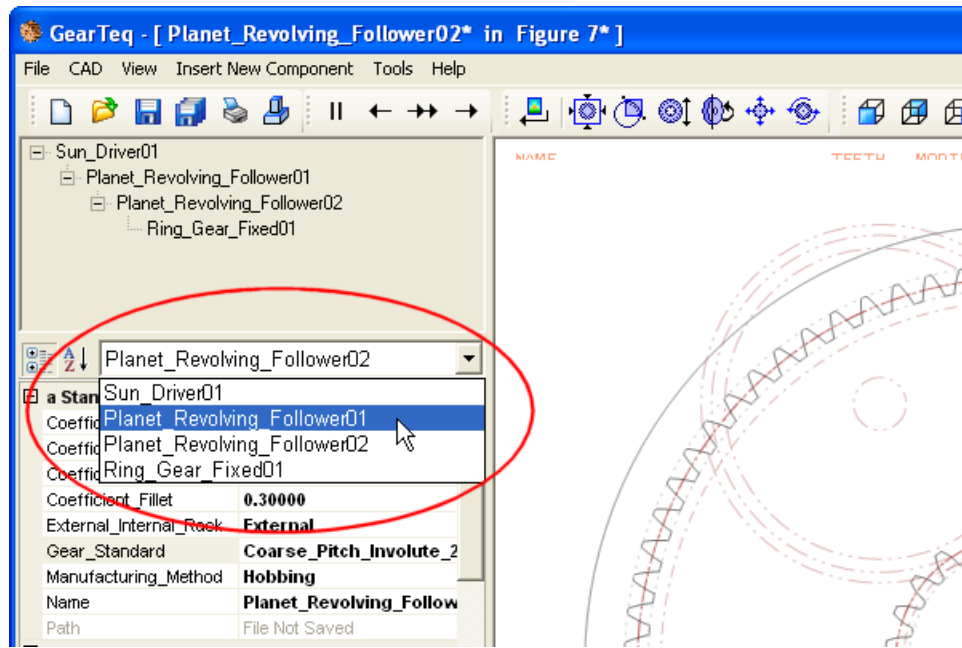
Clicking the Play tool bar button will drive the components for a short simulation.



Forward

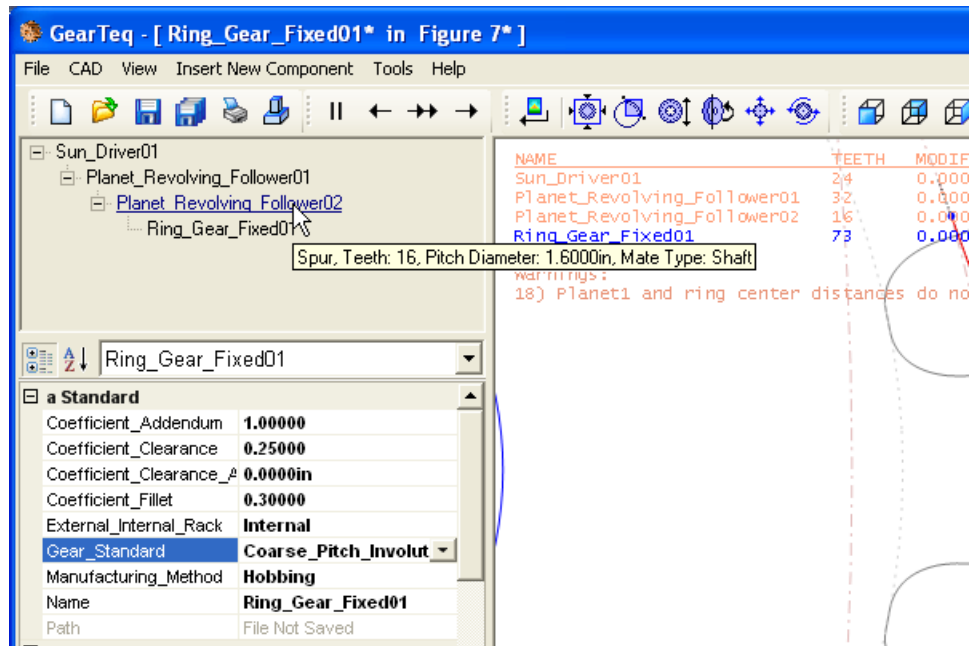
Clicking the Forward tool bar button will drive the components forward one spin increment. Set the spin increment in the options menu.

Component Selector



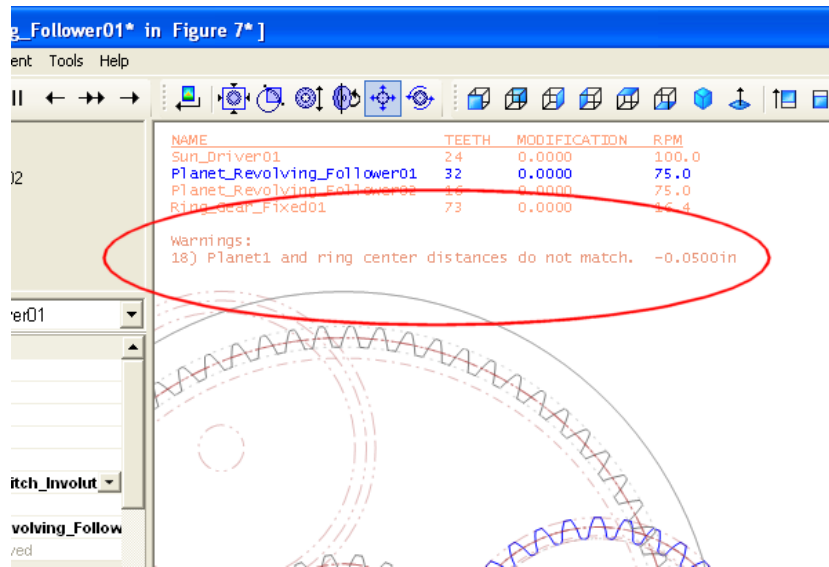
The component selector provides a method to switch between components in the assembly.

Part Tree



The part tree displays a list of the components in the assembly. Moving the mouse over a component name will display some information about the component. Double clicking on a component name will make that component the active component.

Warnings



GearTeq provides some warnings 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 warn the user about. We welcome your suggestions for additional warnings.

1. The addendum modification coefficient is outside of (+/-) .05 to .45
2. The spline pressure angle is not 30, 37.5 or 45 and/or does not conform with fillet or flat roots.
3. The blank outside diameter of the internal spline is too small.
4. Not used
5. The spiral angle of the bevel gear does not equal 0 or 35 degrees.
6. Not used
7. The face width is too large for bevel gear.
8. The web thickness is greater than the body thickness
9. The bore is larger than hub diameter.
10. The bore is larger than outside diameter or minor diameter
11. Invalid involute, the curve intersects itself.

12. The gear has no addendum modification. This may cause undercutting and/or tip interference with the mating gear.
13. The diametral pitch or the module is not a standard hob size.
14. Not used
15. The backlash is greater than 1/4 of the circular pitch.
16. The pressure angle does not match mating parent.
17. The standard does not match mating parent.
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 are not suitable for manufacturing. Please use them for documentation purposes only.
23. The internal gear has little or no negative addendum modification. This may cause tip interference with the mating gear.
24. Worm gear CAD models are not suitable 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 projection2 specified but no diameter.
28. There is a hub diameter2 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-DIN class is specified for a DIN 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 is not specified.
- 41. The Tooth Thickness Tolerance is not available for the specified Quality Number and/or Diametral Pitch.
- 42. The worm gear face width is greater than the recommended width.
- 43. The bore is larger than the minor diameter.
- 44. Timing belt CAD models 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

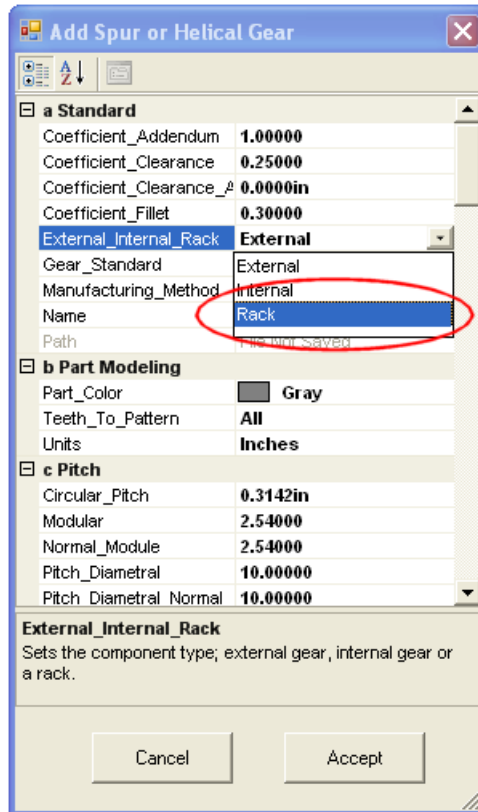
Wheel up and down scales the view of the components. All scaling is done from the center of the screen, not the mouse position. Pan the geometry to the center of the screen then use the mouse wheel to enlarge or reduce the view scale. The direction of the mouse wheel is controllable by a setting in the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

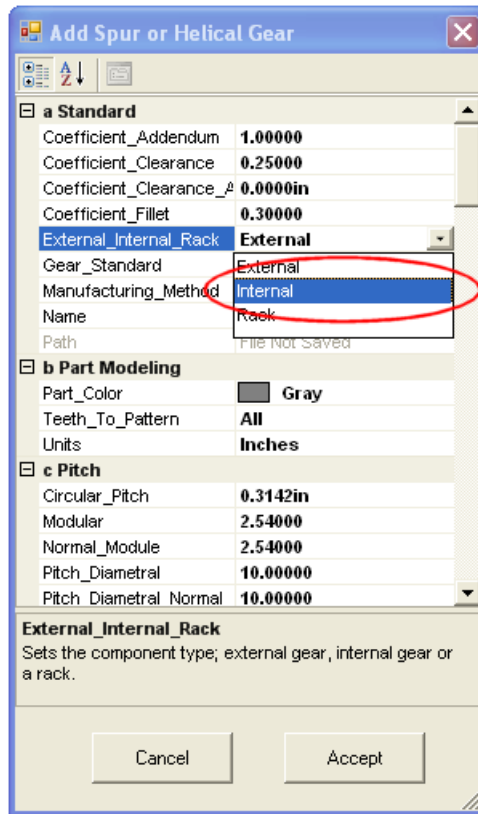
To create a rack select from the GearTeq menu **Insert New Component>New Spur or Helical Gear** menu option or press the control button and the S key at the same time to open the **Add New Spur or Helical Gear** dialog window.



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

Creating an Internal Gear

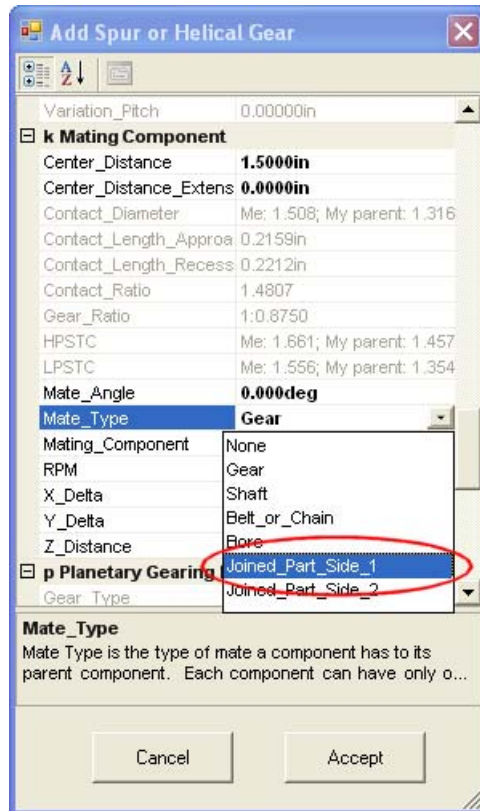
To create an internal gear select from the GearTeq menu **Insert New Component>New Spur or Helical Gear** menu option or press the control button and the S key at the same time to open the **Add New Spur or Helical Gear** dialog window.



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

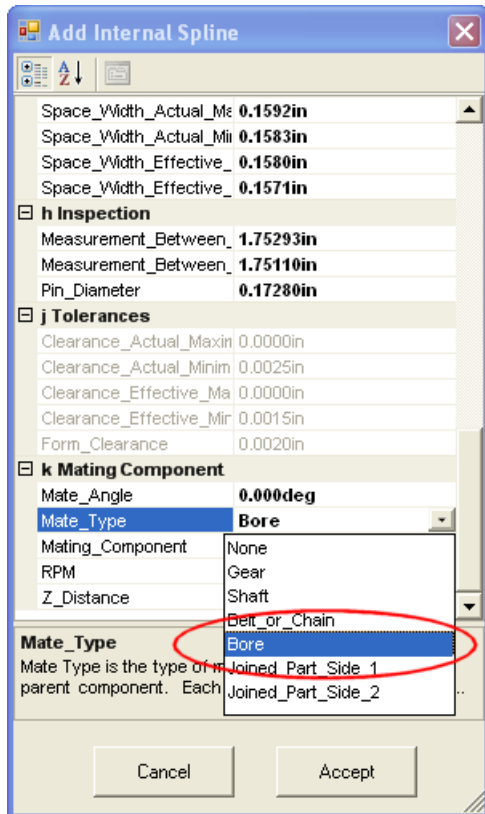
Two components may be joined together to create on CAD model. Add a new spur gear to GearTeq. You may change the parameters of the new gear at any time or stay with the default settings for this example. Open the Insert New Spur or Helical Gear dialog window. You may change any of the parameters of the 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

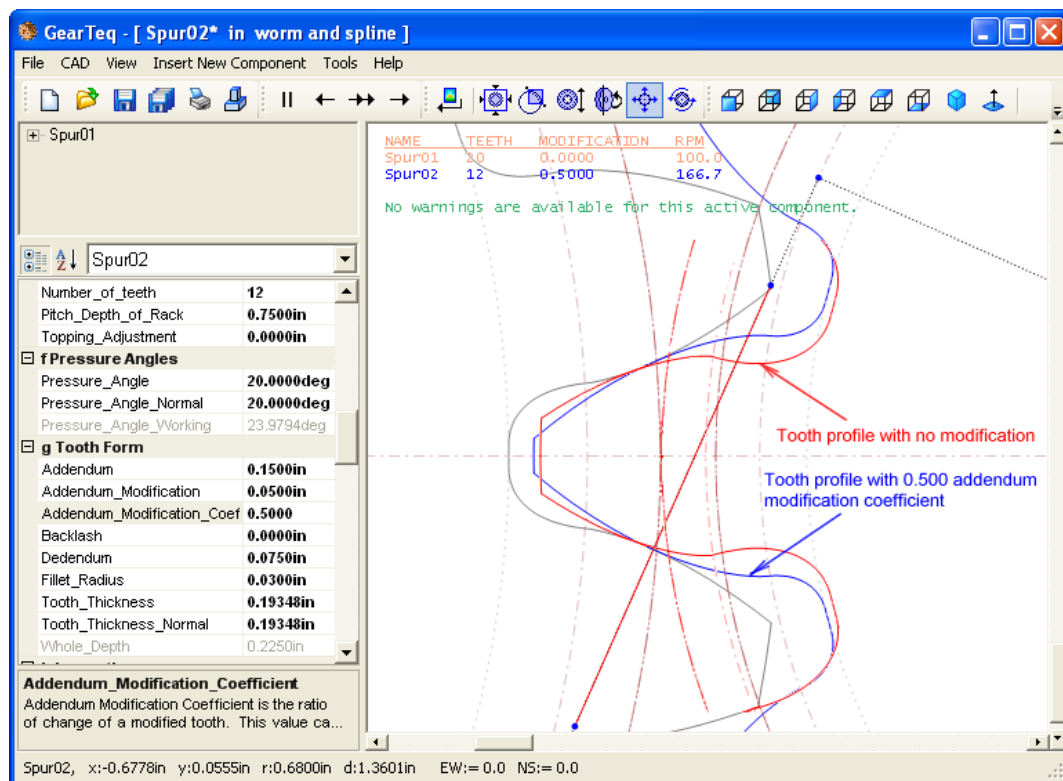
To add an internal spline as a bore to another component, start by adding a new internal spline to the assembly. Making the target component the active component first will simplify the process. Inserting a new internal spline with an external spline active will default to a shaft mate. Inserting a new 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 gears, 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.

Creating a Non-Standard Spur Gear Custom Coefficient Method

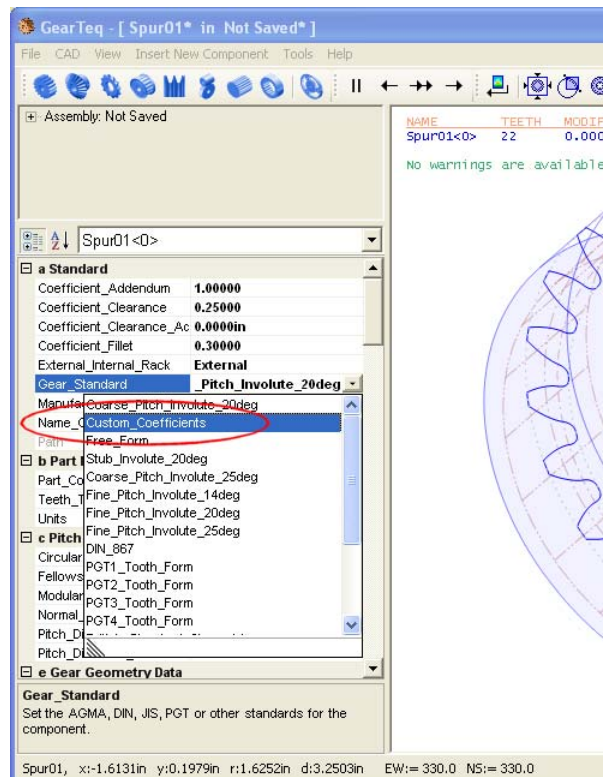
Selecting Custom Coefficients as the Gear Standard type can create a non-standard spur gear. This allows the user to specify the coefficients for:

Addendum

Clearance (clearance plus addendum equals dedendum)

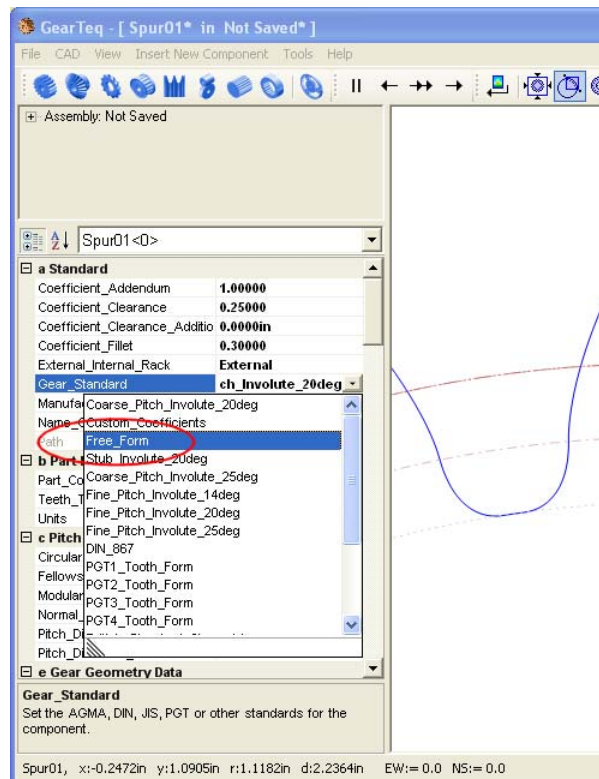
Clearance Addition, which is a linear dimension, not a coefficient

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.



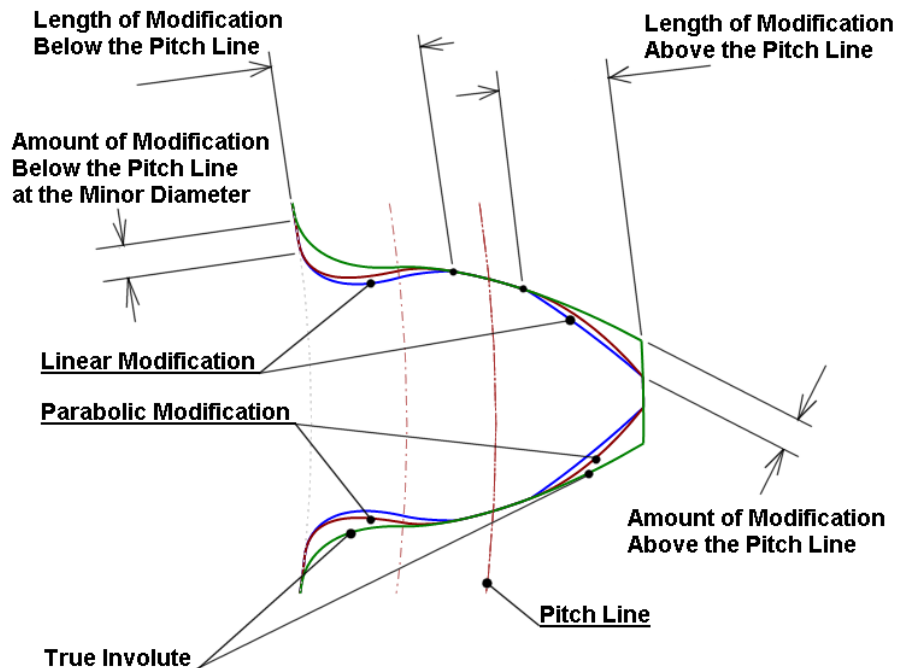
Modifying the Involute Profile on a Spur Gear

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

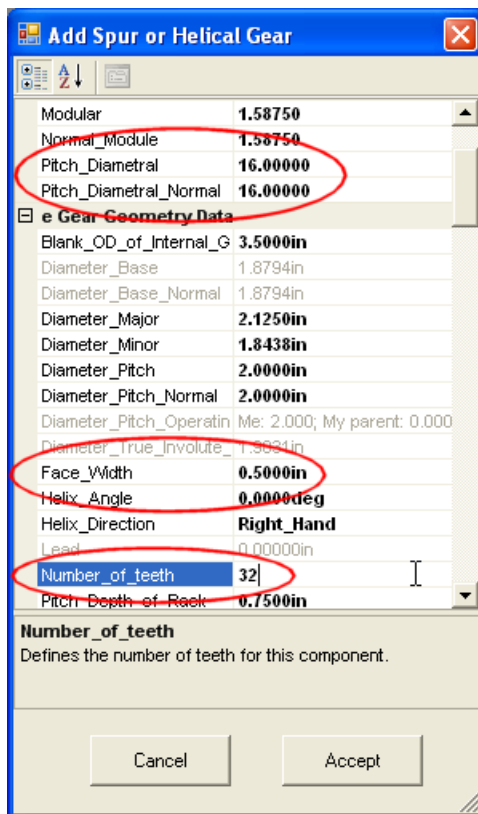
Tutorial – Creating a Spur Gear with an Internal Spline as a Bore

The Task has two parts. The first part is to create a spur gear with an internal spline as a bore. The second part is to edit the spur gear by changing the number of teeth to 36 and change the addendum modification coefficient to -0.120.

The spur gear will have a diametral pitch of 16 with 32 teeth and a face width of 0.500 inches.

The internal spline will have 20 teeth and a 16/32 diametral pitch.

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

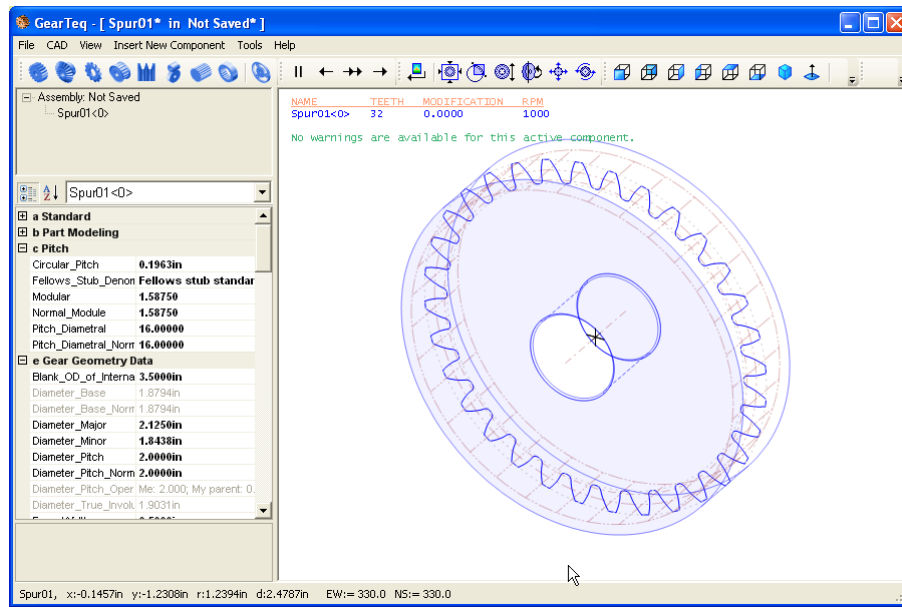


Change the Diametral Pitch to 16

Change the Face Width to 0.500in

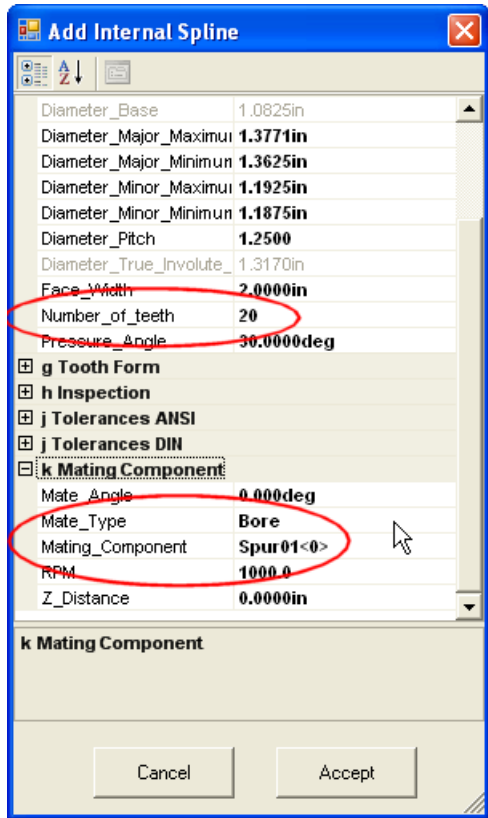
Change the number of teeth to 32

Click Accept to add the component to the GearTeq assembly.



GearTeq with a spur gear

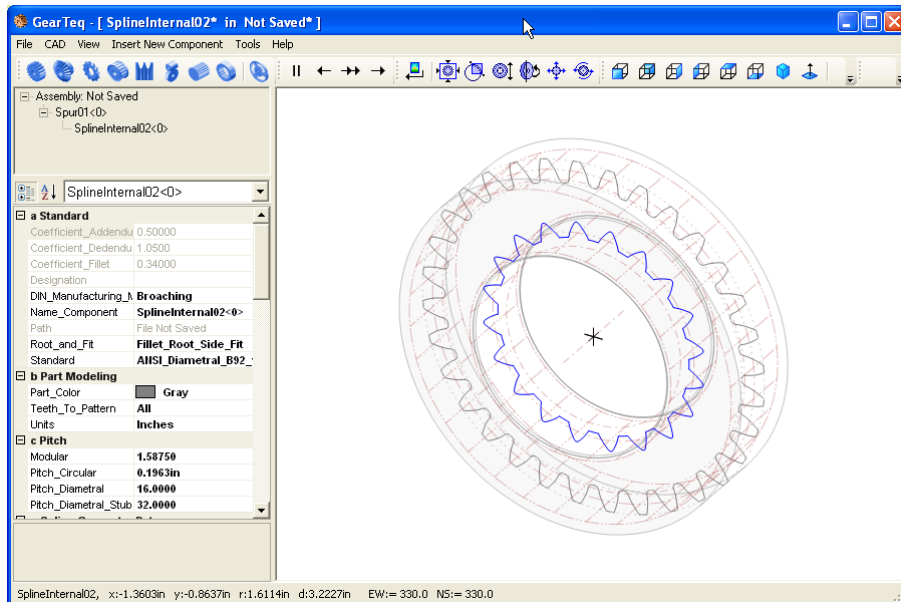
Open the Add Internal Spline dialog window by clicking on the  icon, pressing Ctrl-I or selecting from the GearTeq menu Insert New Component>New Internal Spline.



Change the Number of Teeth to 20

Note how the default Mate Type is a Bore and the Mating Component is the spur gear. Also note the face width, the face width for the spur gear will take president over the face width of the internal spline.

Click Accept to add the internal spline to the assembly as a bore of the spur gear.

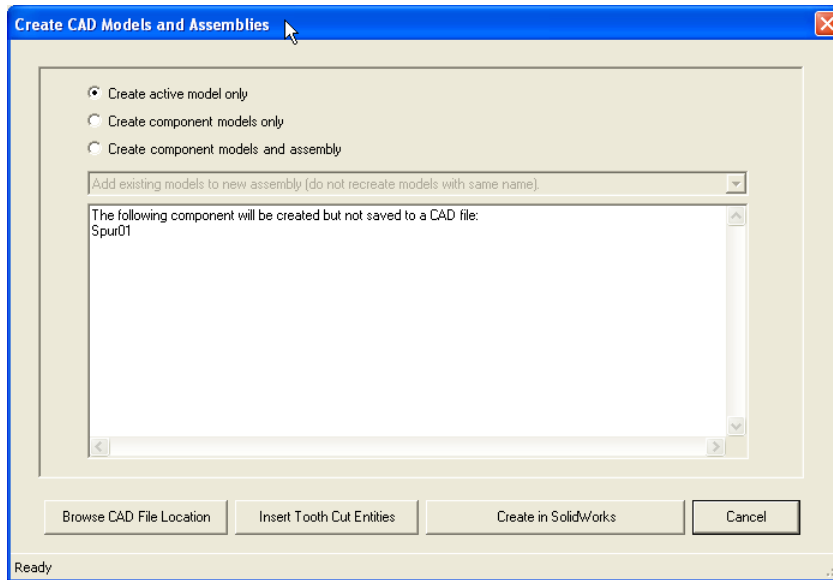


GearTeq with spur gear and the internal spline as a bore mate

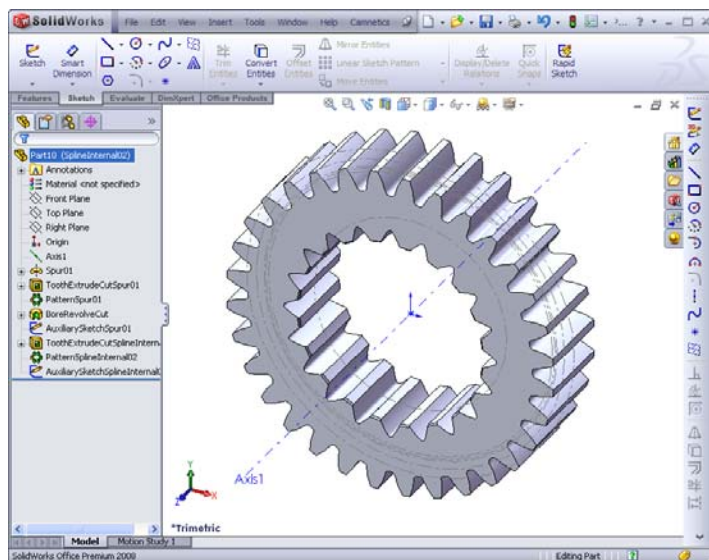
The next task is to create the components in CAD.

We will not create the assembly in this tutorial; only the spur with the internal spline bore. Using the component selector, switch so the spur gear (Spur01<0>) is the active component.

From the GearTeq menu select CAD>Create CAD Models and Assembly to open a dialog box.



There is no need to change any of the settings so click “Create in <your CAD system>” to create the models in CAD.



Spur gear with internal spline as created in CAD

Now that the model has been completed in CAD lets edit the CAD model.

Start by using selecting “Spur01<0>” in the component selector or double clicking on “Spur01<0>” in the assembly tree to make it the active component.

Next in the GearTeq menu select CAD>Edit CAD Models to open the Edit CAD Models dialog box. The information for each component in this model is stored by configuration. In the drop down box, select “Spur01”. The data loaded into the property grid comes from the active part in GearTeq, not from CAD so any changes you made in GearTeq before open this dialog box are in the property grid.

In paragraph e Gear Geometry Data change the number of teeth to 36

In paragraph g Tooth Form change the Addendum Modification Coefficient to -0.120

Click “Update CAD” to start the updating of the CAD model.

Edit CAD Models

e Gear Geometry Data

Blank_OD_of_Internal_Gear	3.5000in
Diameter_Base	1.8794in
Diameter_Base_Normal	1.8794in
Diameter_Major	2.1100in
Diameter_Minor	1.8288in
Diameter_Pitch	2.0000in
Diameter_Pitch_Normal	2.0000in
Diameter_Pitch_Operating	Me: 2.000; My parent: 0.000
Diameter_True_Involute_Form	1.8967in
Face_Width	0.5000in
Helix_Angle	0.0000deg
Helix_Direction	Right_Hand
Lead	0.00000in
Number_of_teeth	32
Pitch_Depth_of_Rack	0.7500in
Topping_Adjustment	0.0000in

f Pressure Angles

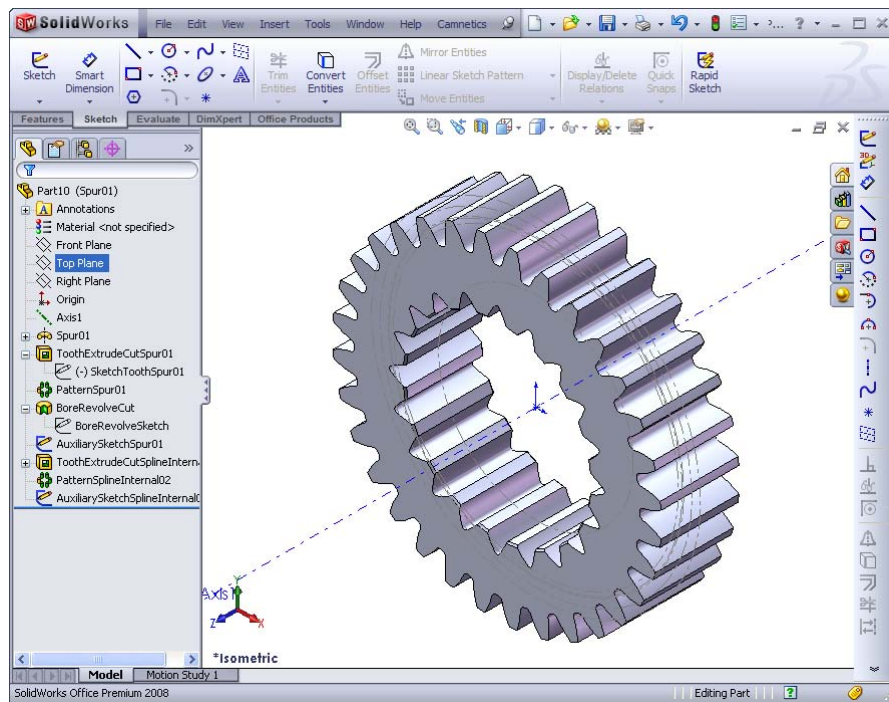
g Tooth Form

Addendum	0.0550in
Addendum_Modification	-0.0075in
Addendum_Modification_Coefficient	-0.1200
Backlash	0.0000in
Dedendum	0.0856in

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 gear...

Configurations available in the CAD document:
Spur01

Update CAD Reload Cancel OK



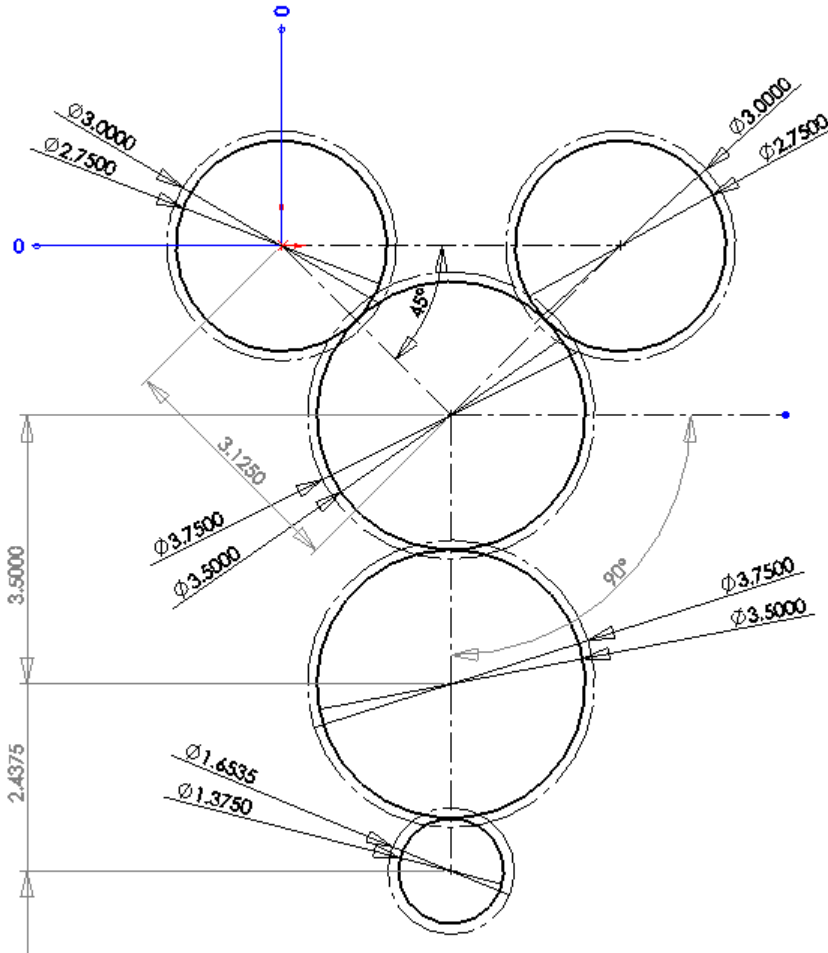
Completed CAD model after changes

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.

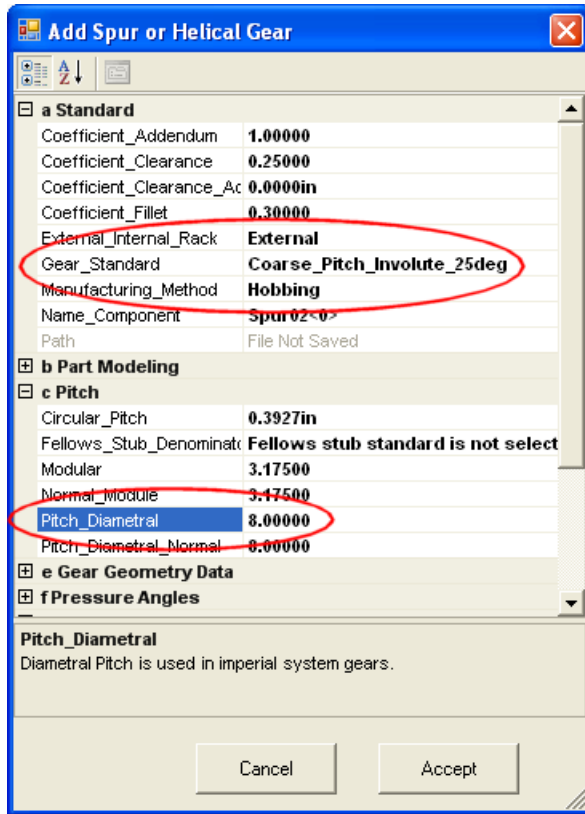
The pressure angle is 25 degrees

The diametral pitch is 8



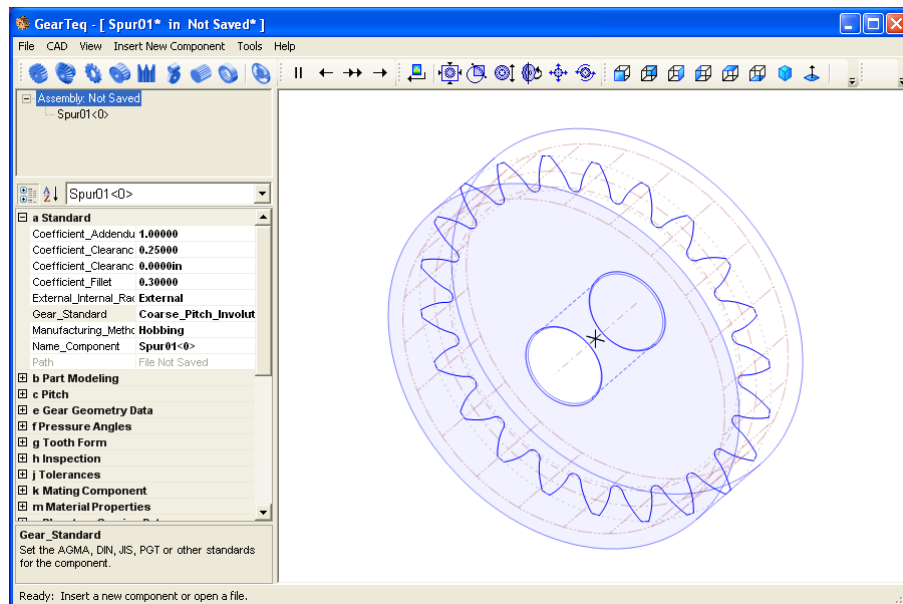
Sketch layout of sample assembly

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.



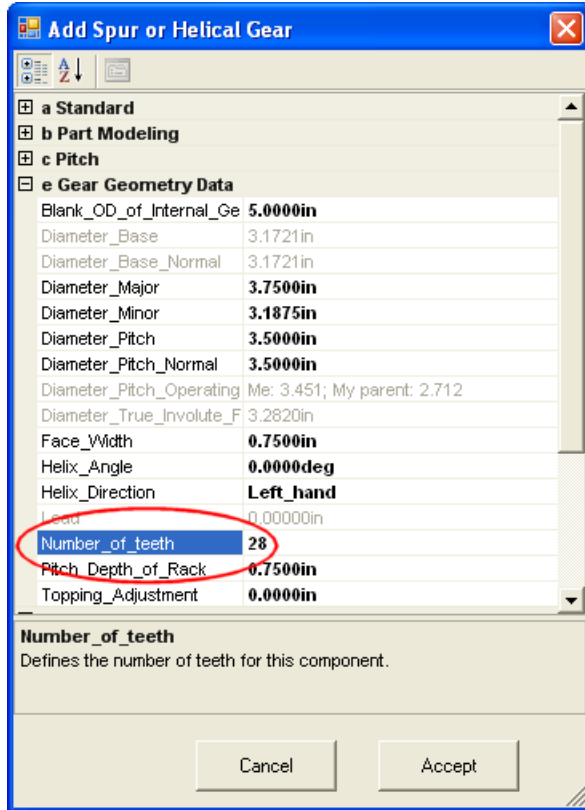
Change the Gear Standard to Coarse Pitch Involute 25deg and Diametral Pitch to 8.0

Click Accept to add the component into the assembly.

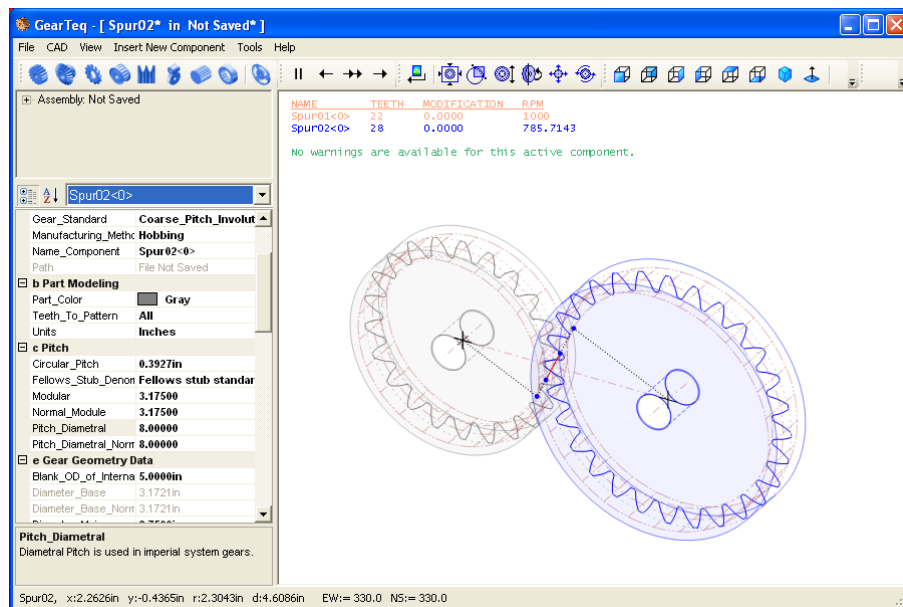


Assembly with first component

Open the dialog box to add another spur gear.

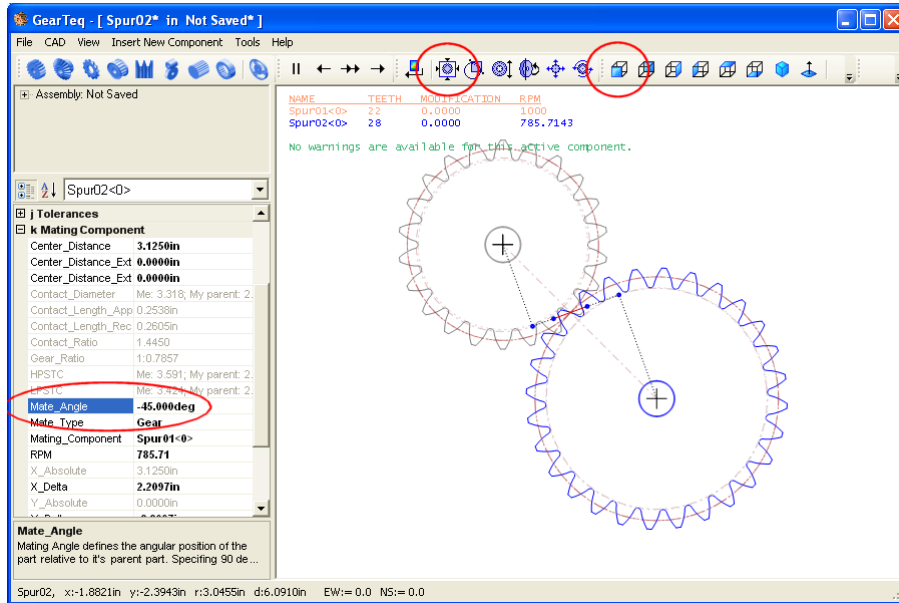


Change the number of teeth to 28 and click the accept button to add the component to the assembly.



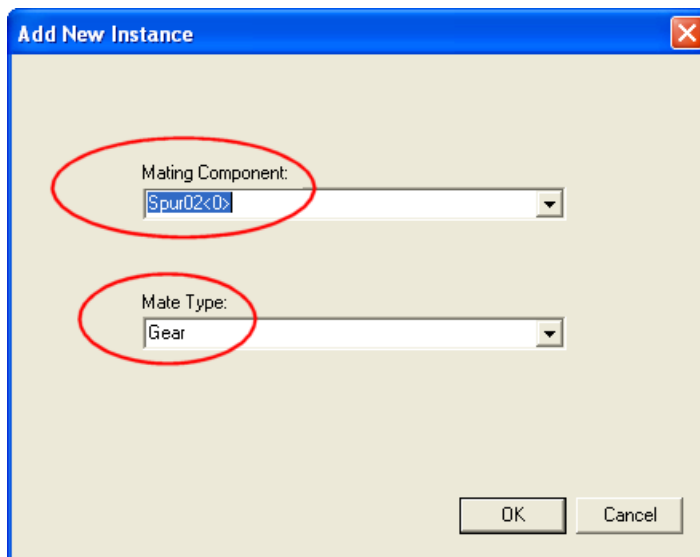
Assembly with second component added

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



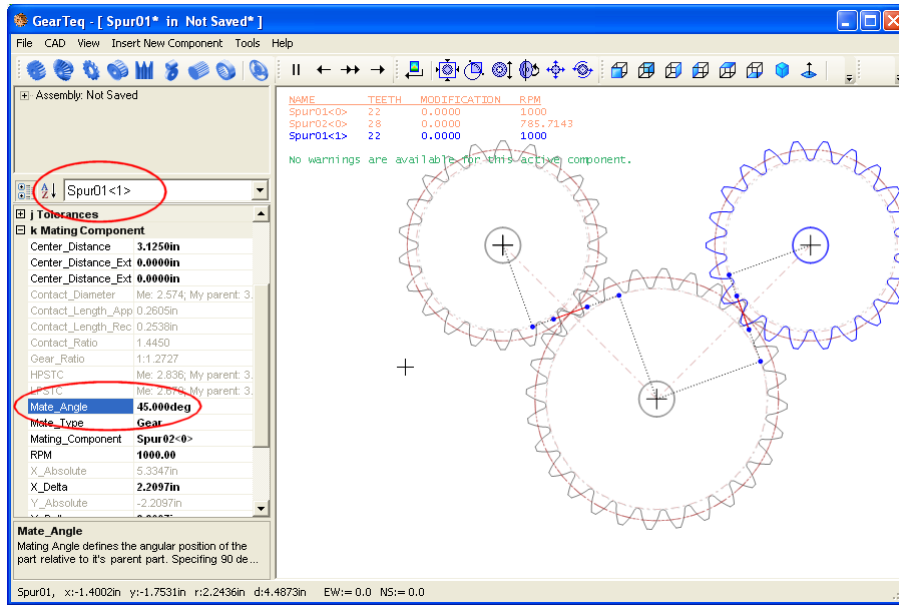
The next task is to add an instance of the first spur gear so select “Spur01<0>” in the component selector to make it the active component.

From the GearTeq menu select Insert New Component>Insert a new Instance of the Active Component... to open the Add New Instance dialog box.



Select Spur02<0> as the mating component and Mate Type Gear.

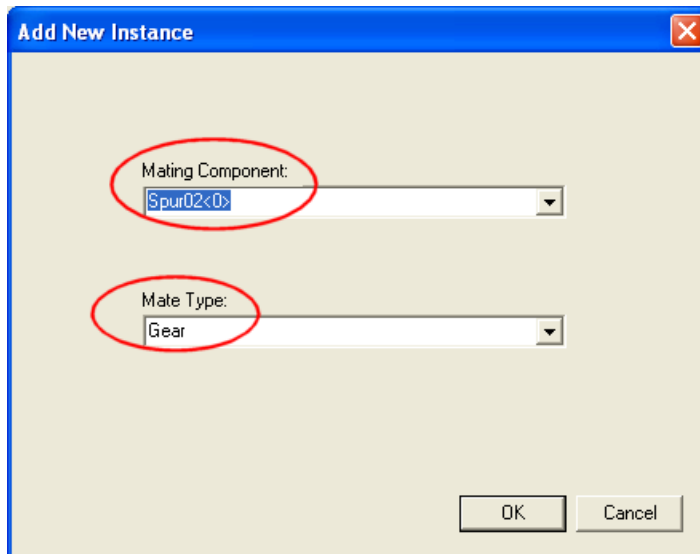
Click OK to add the new instance to the assembly



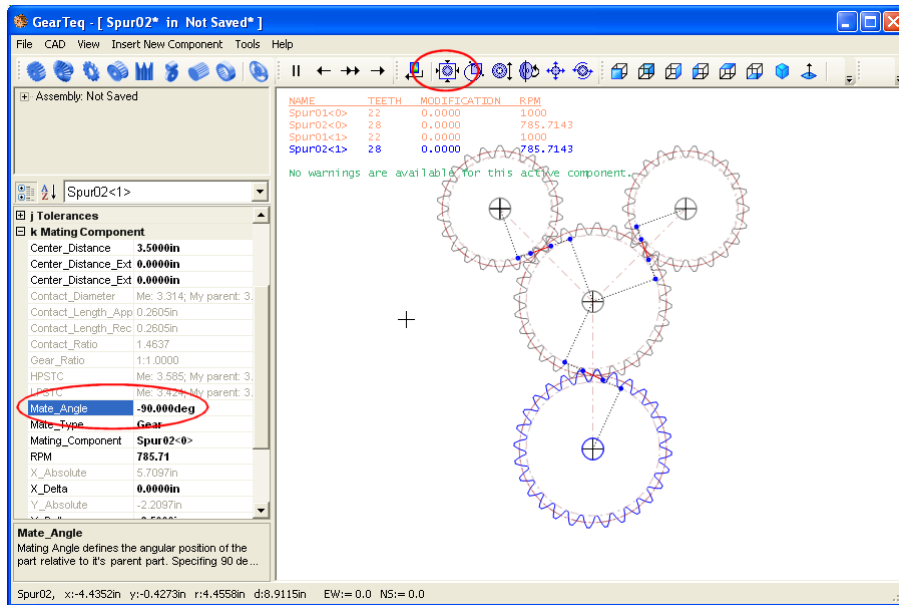
Change the active component to the new instance Spur01<1>.

Change the Mate Angle to 45.0 degrees

The next task is to add a new instance of the 28-tooth gear so make Spur02<0> the active component and open the Insert New Instance dialog box.

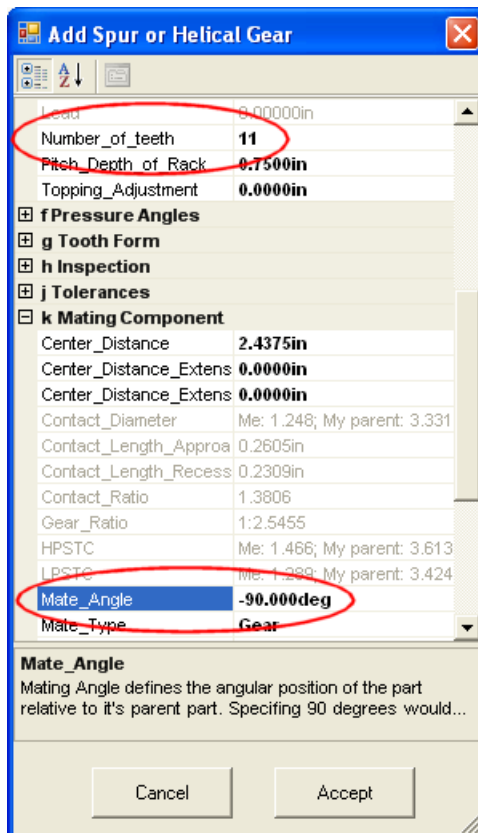


Select Spur02<0> as the mating component and Mate Type of Gear. Click OK to add the new instance to the assembly.



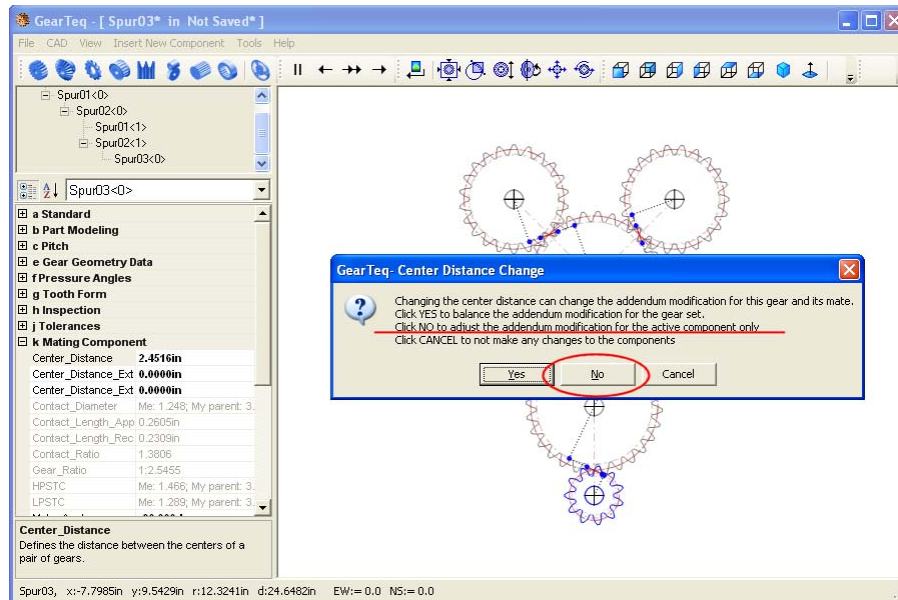
Set the mate angle to -90.0 degrees and click the Zoom All icon.

The next task is to add a new spur gear.

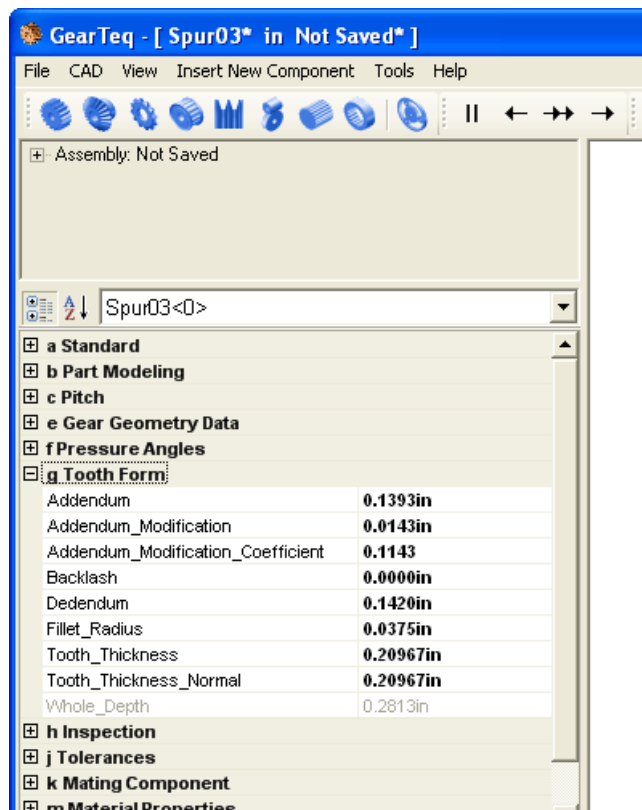


Open the Add Spur or Helical Gear dialog box.

Change the Number of Teeth to 11
Change the Mate Angle to -90.0 degrees
Click the Accept button to add the new component to the assembly.



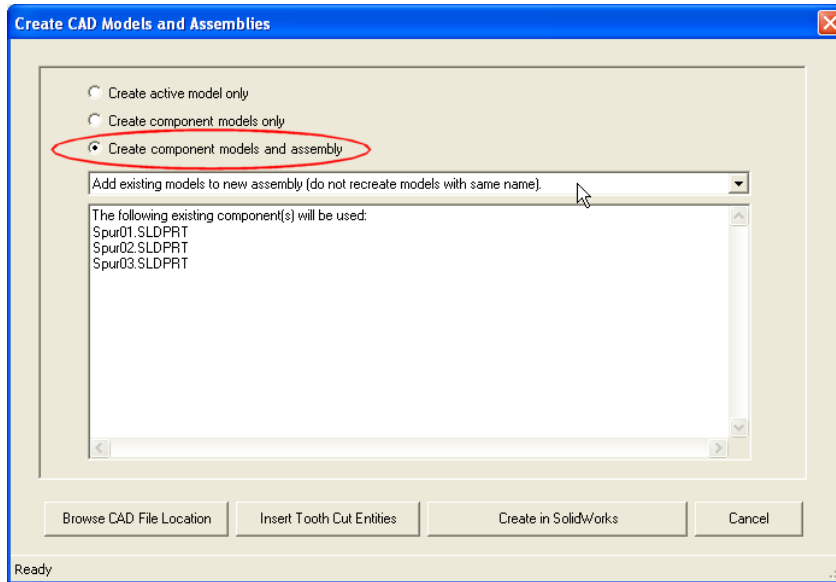
The next task is to change the center distance. In the k Mating Component paragraph change the Center Distance to 2.4516 inches. A prompt will provide a few options on how GearTeq is to deal with this change in center distance. Select No so the change in addendum only affects the new component.



Look in the “g Tooth Form” paragraph and 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.

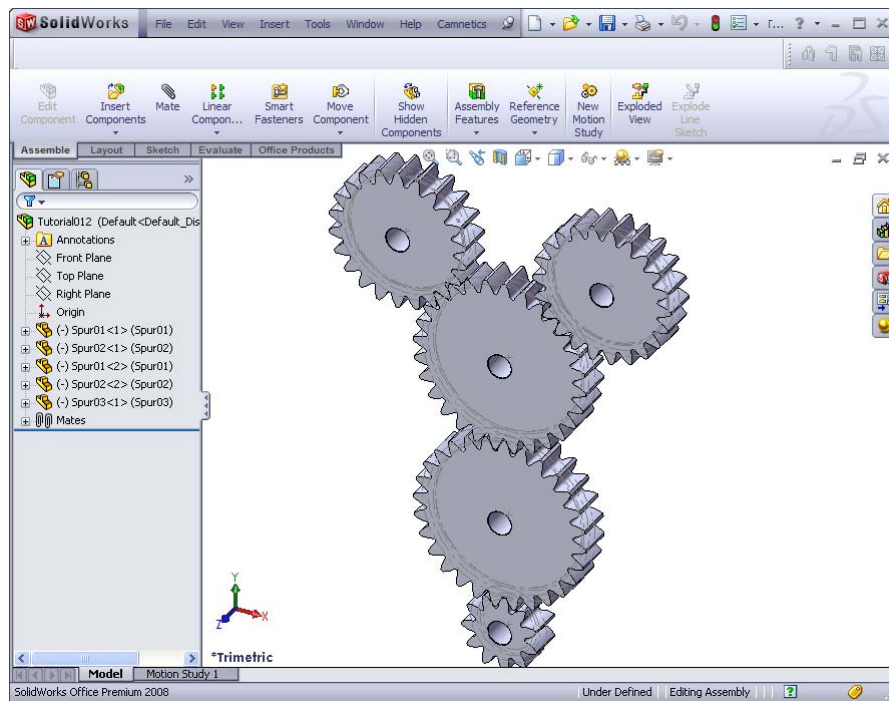
Save the assembly and components.

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.

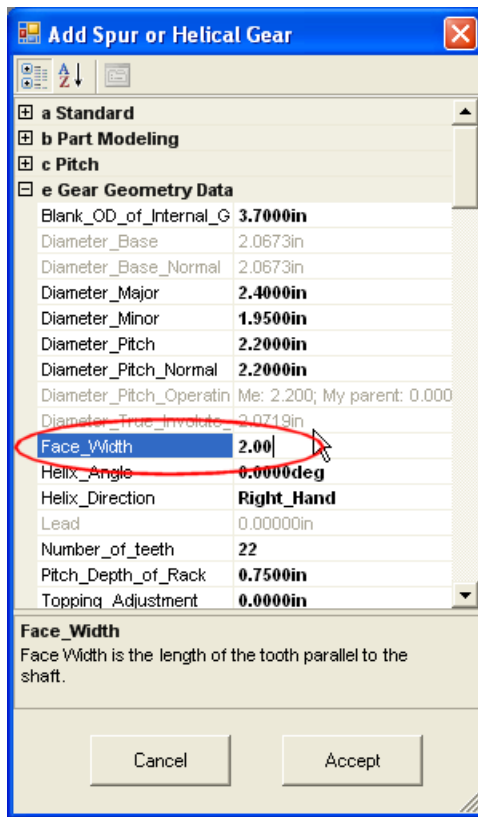


The completed gear assembly created in SolidWorks

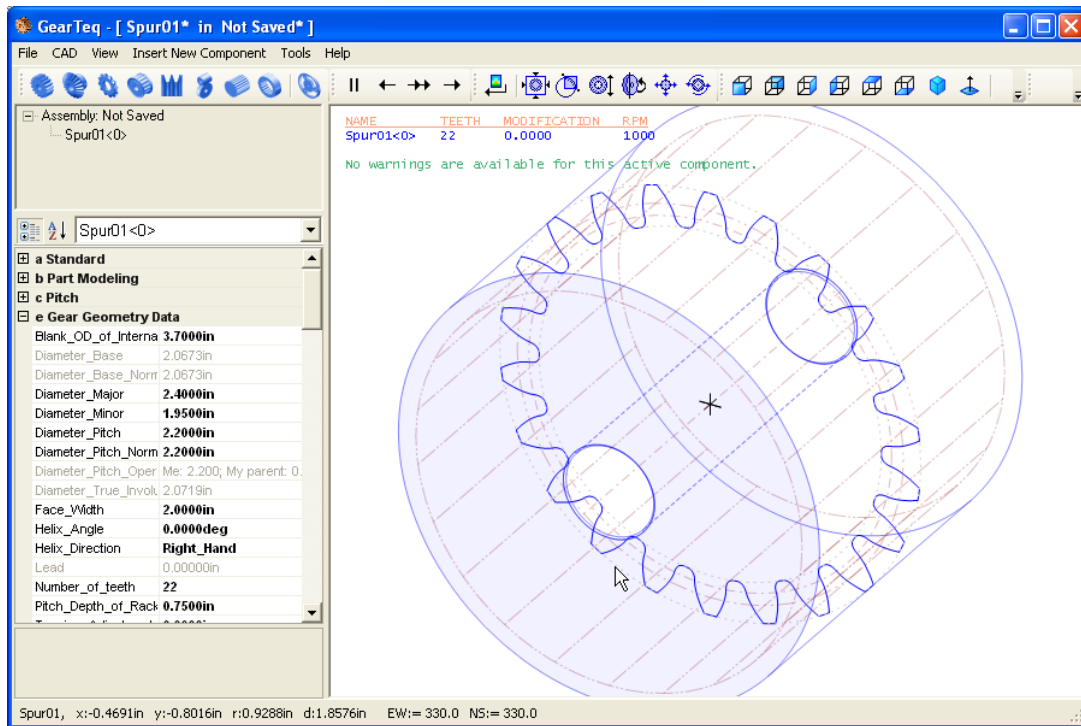
Tutorial – Creating a Spur Gear with two Racks

In the 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.

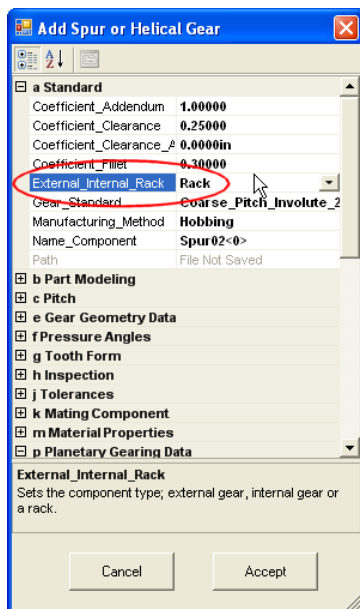


Change the face width to 2.00” and click Accept to add the component to the GearTeq assembly.

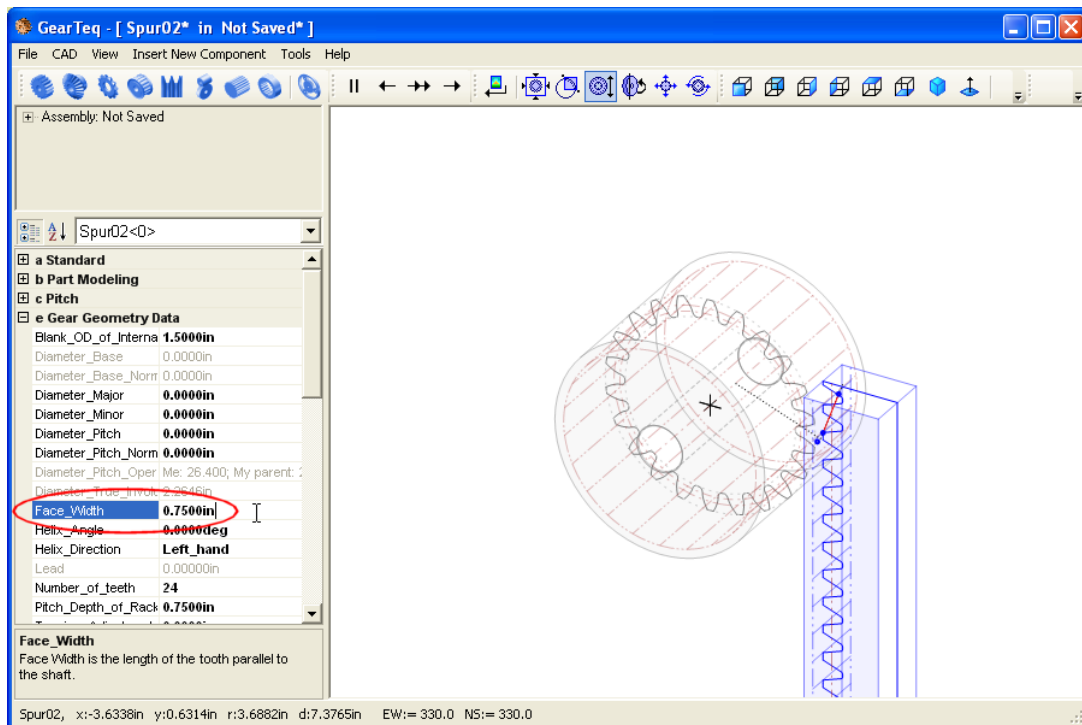


GearTeq with spur gear

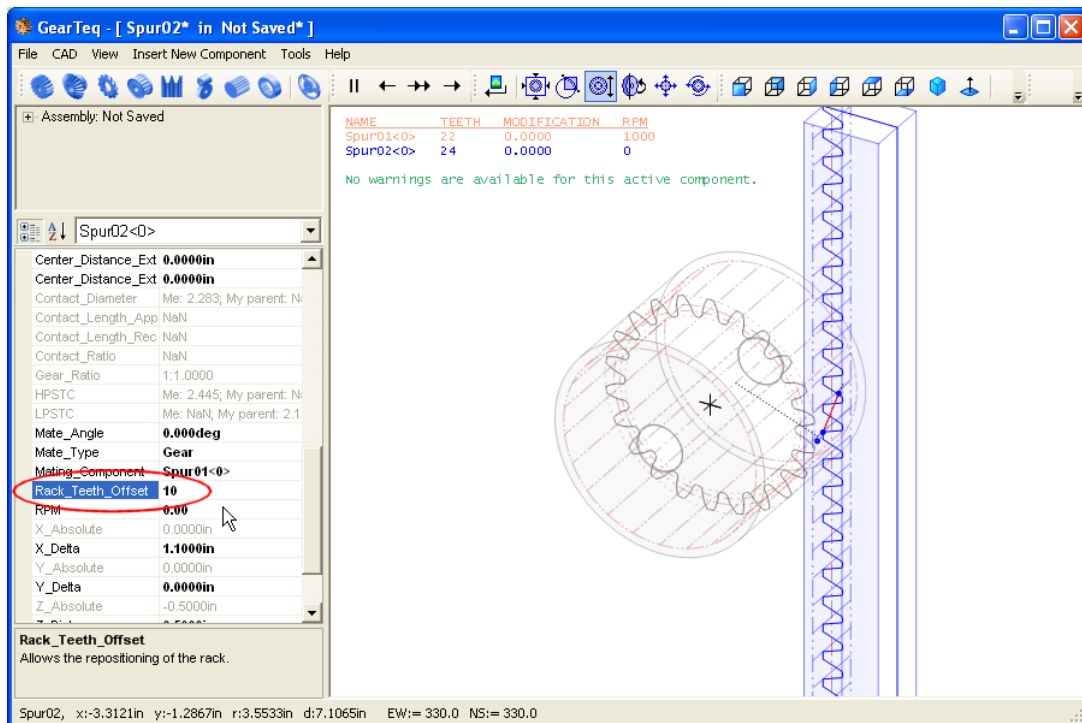
Open the Add Spur or Helical Gear dialog window again to add the rack to the assembly.



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

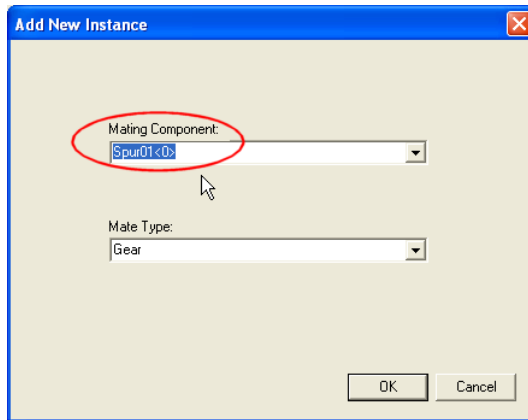


Change the Face Width of the rack to 0.750 inches.
Change the Z Distance to 0.50 inches (in the Mating Components paragraph).

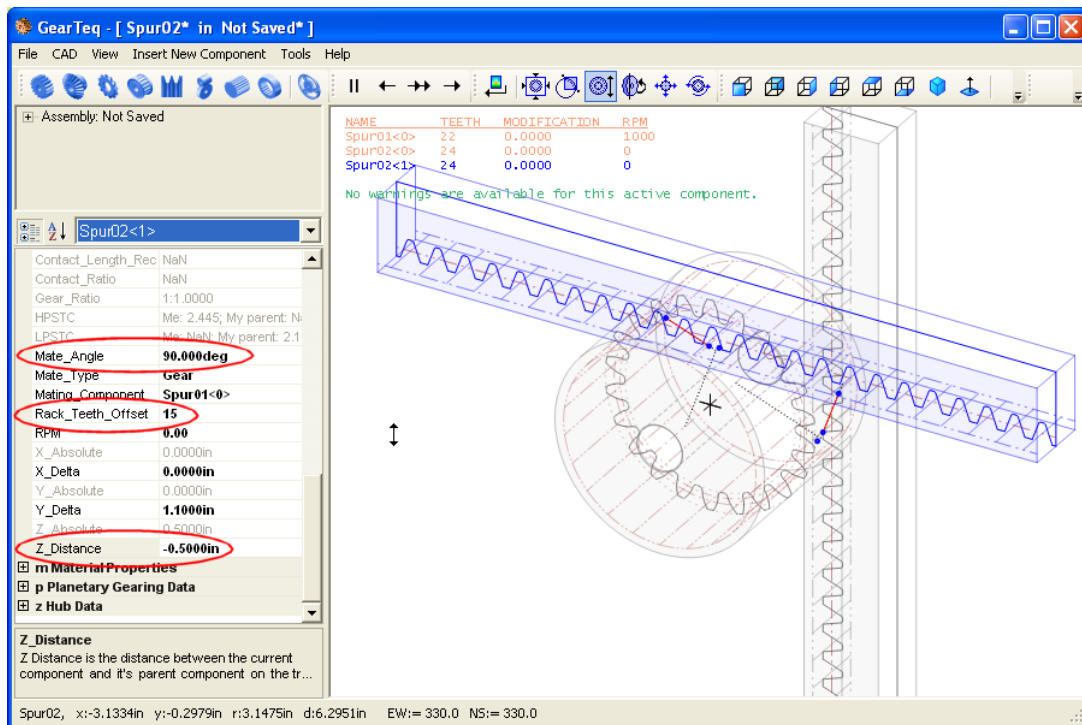


Change the Rack Teeth Offset to 10 to center the rack on the spur gear.

From the GearTeq menu select Insert New Component>Insert a new Instance of the Active Component... to open the Add New Instance dialog box.



Change the Mating Component to “Spur01<0>” and click OK to add the new instance of the rack to the assembly.



Change the Mate Angle to 90, the Rack Teeth Offset to 15 and the Z Distance to -0.50 to complete this tutorial.

Table of Contents

Copyright Notice.....	1
Introduction.....	2
Definitions:	3
Definitions:	3
CAD	3
Component.....	3
Part	3
Assembly.....	3
Annotation.....	3
Model	3
Instance	3
Starting GearTeq.....	3
GearTeq Menus.....	4
File	4
New	4
Open Components.....	4
Open Assemblies	4
Insert Existing Component into Assembly	4
Save Component	4
Save Component As.....	4
Save All.....	4
Save All As	4
Save Assembly As	4
Remove Component.....	5
Remove Instance of Component.....	5
Print the GearTeq Window	5
Capture GearTeq screen to a file	5
Capture GearTeq screen to the clipboard.....	5
Capture GearTeq screen and open in bit map editor.....	5
Empty Clipboard.....	5
Recent files.....	5
Exit.....	5
CAD	5
Create CAD Models and Assembly.....	5
Edit CAD Models	5
Component Template File.....	6
Assembly Template File	6
Annotation Selector	6
Insert Annotations.....	6
View	7
Redraw - fit all	7
Show Line of Action.....	7
Show Hob Tip on Racks	7
Show XY Readouts.....	8

Show Assembly On-screen List.....	8
Show Warnings On-screen	8
Collapse Grid	9
Expand Grid.....	9
Toolbars	10
Deviation Chart.....	10
Insert New Component	11
Insert New Instance of the Active Component	11
New Spur or Helical.....	12
New Bevel Gear.....	13
New Chain Sprocket... ..	14
New Gear Belt Pulley... ..	15
New Belt Pulley... ..	16
New Worm Gear Set.....	17
New External Spline... ..	18
New Internal Spline... ..	19
New Planetary Set.....	20
Cycloid Gear Sets	22
Tools	24
Copy Hub First Side.....	24
Copy Hub Second Side	24
Paste to Hub First Side.....	25
Paste to Hub Second Side	25
Swap Hub Sides	26
Animation Speed.....	26
Reverse Animation Direction	27
Balance Addendum Modification (spur set)... ..	27
Create a Data Sheet for active component.....	28
Reset Over/Under Pin Measurement	31
Reset Chordal Measurement	31
Define Company Standards	32
Backup All Company Standards.....	34
Restore or Import Company Standards.....	34
Standard Spur Gear Sizing.....	35
Options.....	36
Help.....	37
GearTeq Help Topics.....	37
GearTeq Quick Start Guide	37
Registration... ..	37
Create Email with registration information	37
License	37
Check for Updates.....	37
About GearTeq.....	37
Options.....	38
GearTeq Settings.....	38
Animation Speed.....	38

Collapse Grid on Startup.....	38
Color Active Component	38
Color Back Ground	38
Color Base Line	38
Color Base Line Intensity	38
Color Default	38
Color Pitch Line.....	38
Color Pitch Line Intensity.....	39
Decimal Places.....	39
Default Bore Size.....	39
Default Units.....	39
Image Quality.....	39
Involute Precision	39
Line Weight	39
Mouse Drag Speed.....	39
Mouse Wheel Direction	40
New Version Check on Startup.....	40
New Version is Available	40
Shaded Display Near Side	40
Show Assembly Readout	40
Show Warnings Readout.....	40
Show XY Readout	40
Spin Increment.....	40
Settings, Bevel	41
Face Width Loft Factor.....	41
Settings, CAD	41
Constrain Involute Sketch.....	41
Display Annotations in Upper Case.....	41
CAD Version	41
Template File for Assembly.....	41
Template File for Component.....	41
Settings, Spline	41
Component Creation, External Tooth Thickness.....	41
Component Creation, Internal Space Width	42
Component Creation, Major Diameter	42
Component Creation, Minor Diameter	42
Settings, Spur	42
Show Hob Tip on Racks	42
Show Line of Action.....	42
Show Over Under Pins.....	42
Spur Gears.....	43
Addendum.....	43
Addendum Modification.....	43
Addendum Modification Coefficient.....	43
AGMA Class.....	43
Backlash.....	43

Blank OD of Internal Gear.....	43
Chordal Measurement.....	43
Chordal Measurement Minimum.....	43
Chordal Measurement, Teeth to Gage Over	44
Chordal Tooth Height	44
Chordal Tooth Reference Circle	44
Chordal Tooth Thickness.....	44
Chordal Tooth Thickness is a straight-line measurement across a single tooth at the chordal tooth reference circle.	44
Chordal Tooth Thickness Minimum.....	44
Circular Pitch	44
Code Tooth Thickness	44
Coefficient, Addendum.....	45
Coefficient, Clearance.....	45
Coefficient, Clearance Additional	45
Coefficient, Fillet	45
Contact Diameter	45
Contact Length, Approach.....	45
Contact Length, Recess.....	45
Contact Ratio	45
Dedendum	45
Diameter, Base.....	46
Diameter, Base Normal.....	46
Diameter, Major.....	46
Diameter, Minor.....	46
Diameter, Pitch	46
Diameter, Pitch, Normal	46
Diameter, Pitch, Operating	46
Diameter, Start of Active Profile	46
Diameter, True Involute Form	46
External Internal Rack	46
Face Width.....	46
Fellows Stub Denominator	46
Fillet Radius	47
Gear Ratio	47
Gear Standard.....	47
Gear Type.....	47
Helix Angle.....	47
Helix Direction.....	47
HPSTC (Highest Point of Single Tooth Contact)	47
Hob Protuberance.....	48
Keyway	48
Keyway Position	48
Lead.....	48
LPSTC (Lowest Point of Single Tooth Contact)	48
Manufacturing Method	48

Mate Angle.....	49
Mate Type	49
Mating Component	49
Max Runout	49
Measurement Over/Under Pins.....	49
Measurement Over/Under Pins Minimum.....	49
Modification Profile APL Amount	49
Modification Profile APL Diameter	49
Modification Profile APL Length.....	49
Modification Profile APL Type	50
Modification Profile BPL Amount	50
Modification Profile BPL Diameter.....	50
Modification Profile BPL Length	50
Modification Profile BPL Type	50
Module (displayed as Modular).....	50
Module Normal	50
Mounting Bushing Modeling.....	50
Mounting Bushing Side	50
Name	50
Number of Planets.....	51
Number of Teeth	51
Path	51
Part Color	51
Pin Diameter	51
Pitch Depth of Rack	51
Pitch, Diametral	51
Pitch, Diametral, Normal	51
Planet Mode	51
Planet Positions.....	51
Pressure Angle	52
Pressure Angle, Normal	52
Pressure Angle, Working.....	52
Ratio of Driver to Follower.....	52
Roll Angle at Custom Diameter.....	52
Roll Angle at Major Diameter	52
Roll Angle at Profile Modification Above Pitch Line	52
Roll Angle at Profile Modification Below Pitch Line	52
Roll Angle at TIF Diameter	52
Shrinkage Rate	52
Teeth to Pattern	53
Tolerance, Profile.....	53
Tolerance, Tooth Alignment.....	53
Tolerance, Tooth Thickness.....	53
Tolerance, Total Composite.....	53
Tolerance, Total Index	53
Tooth Thickness.....	53

Tooth Thickness at Major Diameter	53
Tooth Thickness, Normal.....	53
Topping Adjustment	54
Units.....	54
Variation, Pitch	54
Whole Depth	54
Bevel Gears.....	55
Addendum.....	55
Addendum Modification.....	55
AGMA Class.....	55
Angle, Back.....	56
Angle, Face	56
Angle, Front	56
Angle, Pitch.....	56
Angle, Root.....	56
Angles, Sum of Pitch	56
Backlash.....	56
Circular Pitch	56
Coefficient, Addendum.....	56
Coefficient, Addendum Modification.....	56
Coefficient, Clearance.....	57
Coefficient, Fillet	57
Crown to Back	57
Dedendum	57
Diameter, Base.....	57
Diameter, Outside	57
Diameter, Pitch	57
Diametral Pitch	57
Face Width.....	57
Fillet Radius.....	57
Gear Ratio	57
K Factor	58
Manufacturing Method	58
Mating Component	58
Module (displayed as Modular).....	58
Mounting Distance.....	58
Name	58
Number of Teeth.....	58
Part Color.....	58
Path	58
Pitch Variation Accumulated.....	58
Pitch Variation Allowable.....	59
Pressure Angle	59
Spiral Angle	59
Spiral Direction.....	59
Tooth Thickness.....	59

Units.....	59
Web Thickness.....	59
Whole Depth.....	59
Z Distance.....	59
Chain Sprockets.....	60
Chain Length.....	60
Chain Number.....	60
Chain Pitch.....	60
Chain Series.....	60
Double Pitch Single Duty.....	60
Even Full Pitches.....	60
Name.....	60
Number of Strands.....	60
Number of Teeth.....	60
Path.....	60
Part Color.....	60
Pitch Diameter.....	61
Pointed Tooth.....	61
Roller Diameter.....	61
RPM.....	61
Sprocket Width.....	61
Teeth to Pattern.....	61
Units.....	61
Gear Belt Pulleys.....	62
Belt Length.....	62
Belt Pitch.....	62
Belt Width.....	62
Center Distance.....	62
Diameter, Pitch.....	62
Flange Creation.....	62
Flange ID.....	62
Flange OD.....	62
Flange Thickness.....	62
Name.....	62
Number of Teeth.....	62
Outside Diameter.....	63
Part Color.....	63
Path.....	63
Pulley Width.....	63
Teeth to Pattern.....	63
Tooth Width.....	63
Units.....	63
Belt Pulleys.....	64
Belt Length.....	64
Belt Section.....	64
Center Distance.....	64

Deep Groove	64
Diameter, Pitch	64
Name	64
Number of Grooves.....	64
Part Color	64
Pulley OD.....	64
RPM	64
Units.....	64
Worm Gears	65
Addendum.....	65
Addendum Modification	65
Addendum Modification Coefficient.....	65
Bore, Chamfer	65
Bore, Worm.....	65
Bore, Worm Chamfer.....	65
Center Distance	65
Coefficient, Addendum.....	65
Coefficient, Clearance.....	65
Coefficient, Fillet	65
Contact Ratio	66
Dedendum	66
Diameter, Pitch, Gear.....	66
Diameter, Pitch, Worm	66
Diametral Pitch	66
Direction, Gear.....	66
Face Width	66
Fillet Radius	66
Gear, Outside Diameter	66
Gear Standard.....	66
Lead, Gear	66
Lead, Worm	66
Lead, Worm, Angle.....	67
Length, Worm	67
Module (Displayed as Modular)	67
Name	67
Number of Teeth	67
Number of Threads	67
Part Color	67
Path	67
Pressure Angle	67
RPM	67
Teeth to Pattern	67
Units.....	68
Whole Depth	68
Worm Outside Diameter	68
Involute Splines	69

Addendum.....	69
Addendum Modification.....	69
Addendum Modification Coefficient.....	69
Ase	69
AseE.....	69
Asi	69
Chordal Measurement.....	69
Chordal Measurement Teeth to Gage Over	69
Clearance Actual Maximum	69
Clearance Actual Minimum.....	69
Clearance Effective Maximum	70
Clearance Effective Minimum.....	70
Coefficient, Addendum.....	70
Coefficient, Dedendum	70
Coefficient, Fillet	70
Dedendum	70
Diameter, Base.....	70
Diameter, Major, Maximum	70
Diameter, Major, Minimum.....	70
Diameter, Minor, Maximum	70
Diameter, Minor, Minimum.....	71
Diameter, Pitch	71
Diameter, True, Involute Form	71
DIN Manufacturing Method	71
Face Width.....	71
Fillet Radius	71
Form Clearance.....	71
Measurement Between Pins Actual Maximum.....	71
Measurement Between Pins Actual Minimum	71
Measurement Between Pins Effective Minimum	71
Measurement Over Pins Actual Maximum.....	72
Measurement Over Pins Actual Minimum	72
Measurement Over Pins Effective Maximum.....	72
Module	72
Name	72
Number of Teeth.....	72
Part Color.....	72
Path	72
Pin Diameter	72
Pitch, Circular	72
Pitch, Diametral	73
Pitch, Diametral, Stub	73
Pressure Angle	73
Root and Fit.....	73
Space Width Actual Maximum.....	73
Space Width Actual Minimum	73

Space Width Effective Maximum.....	73
Space Width Effective Minimum	73
Standard	73
T	73
TE.....	74
Teeth to Pattern	74
Tolerance Class.....	74
Tolerance Grade.....	74
Tolerance Position	74
Tooth Thickness Actual Maximum	74
Tooth Thickness Actual Minimum	74
Tooth Thickness Effective Maximum	74
Tooth Thickness Effective Minimum	74
Units.....	75
Mating.....	76
Belt Length.....	76
Center Distance.....	76
Center Distance Extension.....	77
Center Distance Extension Backlash	78
Chain Length.....	78
Contact Diameter	78
Contact Length, Approach.....	78
Contact Length, Recess.....	78
Contact Ratio	79
Contact Ratio Face.....	79
Contact Ratio Total	79
Even Full Pitches	79
Efficiency.....	79
Friction Coefficient.....	79
Gear Ratio	79
HPSTC	79
LPSTC.....	79
Mate Angle.....	79
Mate Type	80
Mating Component	80
Rack Teeth Offset	80
RPM	80
X Delta	80
Y Delta.....	80
Z Distance	81
X Absolute	81
Y Absolute	81
Z Absolute.....	81
Material Property	82
Compressive Yield Strength PSI	82
Material	82

Modulus of Elasticity KSI	82
Poisson's Ratio.....	82
Safe Bending Strength PSI.....	82
Shrinkage Rate	82
Tensile Strength Ultimate PSI	82
Hub Data	83
Bore.....	83
Bore Chamfer.....	83
Hub 1st Side Chamfer.....	83
Hub 1st Side Diameter.....	83
Hub 1st Side Projection	83
Hub 2nd Side Chamfer.....	83
Hub 2nd Side Diameter.....	83
Hub 2nd Side Projection	83
Keyway	83
Keyway Position	83
Mounting Bushing Modeling.....	83
Mounting Bushing Side	84
Mounting Split Taper Bushing.....	84
Standard Set Screw	84
Tool Bars.....	85
Insert New Components.....	85
 Insert New Spur or Helical Gear.....	85
 Insert New Bevel Gear.....	85
 Insert New Chain Sprocket	85
 Insert New Gear Belt Pulley	85
 Insert New V-Belt Pulley.....	85
 Insert New Worm Gear Set.....	85
 Insert New External Spline	85
 Insert New Internal Spline	85
 Insert New Planetary Gear Set.....	85
Standard	85
 New	85
 Open.....	85
 Save Active Component	85
 Save All.....	85
 Print the GearTeq Window	85
 Create CAD Models and Assembly.....	85
 Edit CAD Models	85
Standard Views	85

	Front.....	86
	Back	86
	Left.....	86
	Right.....	86
	Top	86
	Bottom.....	86
	Isometric	86
	Normal To Cutter.....	86
	Index Up.....	86
	Index Down.....	86
	Index Left.....	87
	Index Right.....	87
	View.....	87
	Last View	87
	Zoom to Fit	87
	Zoom Area	87
	Zoom In Out.....	87
	Rotate the View.....	87
	Pan.....	87
	Drive	88
	Simulation.....	88
	Stop	88
	Back	88
	Play	88
	Forward.....	88
	Component Selector.....	89
	Part Tree.....	90
	Warnings.....	91
	Mouse controls.....	94
	Current Mouse Position	94
	On Screen Measurements	94
	Mouse wheel	94
	Middle Mouse Button (wheel).....	94
	Creating a Rack.....	95
	Creating an Internal Gear.....	96
	Joined parts, side one or two.....	97
	Adding an internal spline as a bore to another component.....	98
	Addendum Modification and Addendum Modification Coefficient.	99

Creating a Non-Standard Spur Gear	100
Custom Coefficient Method.....	100
Free Form Method	101
Modifying the Involute Profile on a Spur Gear	102
Tutorial – Creating a Spur Gear with an Internal Spline as a Bore	103
Tutorial – Spur Gear Assembly	109
Tutorial – Creating a Spur Gear with two Racks.....	118
Table of Contents.....	122