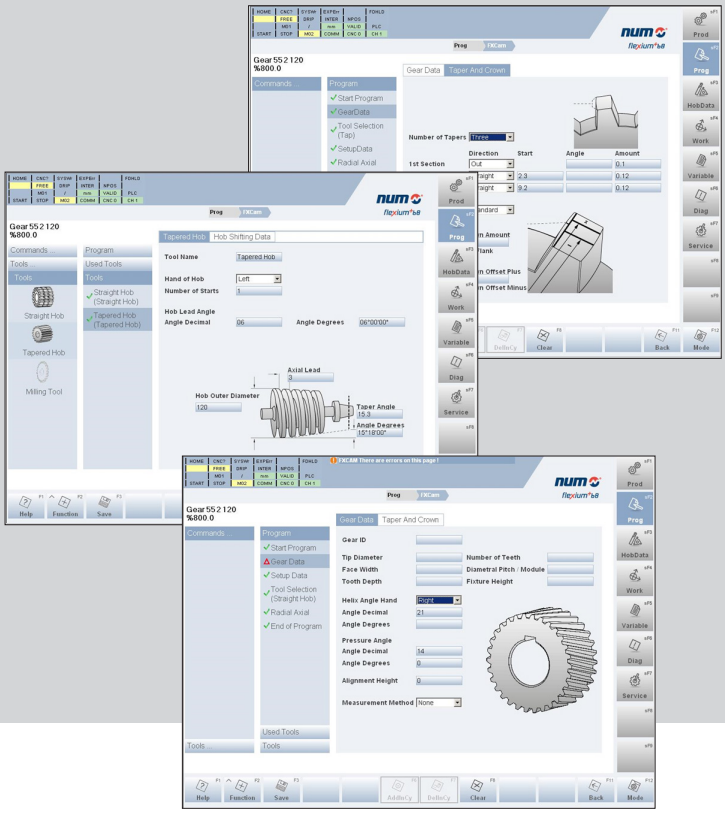
Flexium+
NUMgear Hobbing

Instruction manual

M00048EN-00

Edition 05.2018



flexium+

www.num.com

num®
CNC HighEnd Applications

Summary

Page deliberately empty

Page deliberately empty

Summary

Page deliberately empty

1 Foreword

1.1 Introduction

This software runs on NUM Flexium+.

NUMgear Hobbing allows the operator to create and run gear hobbing programs by entering data into the graphic user interface without programming CNC code.

The *NUMgear Hobbing* is fully integrated into the *Flexium+ HMI* making use of the Flexium+ soft keys. It allows data entry while the machine is running.

Switching between the NUMgear screens and the other Flexium+ screens is possible without data loss.

All data necessary to define the machine setup, the gear setup, the gear itself, the tool, and the actual hobbing process can be set using the entry masks.

Graphics illustrate content and meaning of entry fields.

NUMgear can be used on a variety of hobbing machines. The basic configuration however has the hobbing tool mounted on a spindle or an axis and the workpiece on a C table with the A axis used to set the angle of the hobbing tool.

The X axis moves the tool radially into the gear teeth, the Z axis moves the hobbing tool axially along the gear teeth and the Y axis shifts the tool tangentially along the diameter of the gear.

The *Fast Gear Alignment* feature is included. It records the position of a precut master gear and then automatically adjusts the position of all following gears.

1.2 NUMgear Machining Features

Radial and axial movement of the hobbing tool can be used singly or combined to run a number of processes.

They are ranging from the most common radial / axial approach in X to cutting depth combined with the cutting in Z along the tooth gap to splines or worm gears.

1.2.1 Options for Axial Cycles

- Crowning
- Offset Crowning
- Taper
- Selectable Dwell at Start, Infeed and End
- Diagonal Hobbing (Taper Root)
- Single Indexing
- Radial Infeed
- Automatic Alignment to Previous Gear Tooth
- Automatic Gear Alignment to Master Gear.

1.2.2 Options for Radial Cycles

- Worm Gear Cycle with or without Tangential Feed
- Single Indexing
- Selectable Dwell at Start, Infeed and End
- Automatic Alignment to Previous Gear Tooth
- Automatic Gear Alignment to Master Gear.

1.2.3 Options for Shifting

- Incremental Shifting - a shift step after a defined number of cuts
- Synchronous Shifting - where the gear moves with the shift during cutting.

1.2.4 Options for Machine Control

- Automatic operation of the tailstock
- Automatic clamping and unclamping of the workpiece
- Coolant control
- Tailstock pressure or support .

1.2.5 Options for Program Flow

- Unlimited number of sequential machining cycles allows for hobbing of cluster gears
- Fast Gear Alignment allows for precut gears.

1.3 NUMgear User Interface Features

All the data that pertains to one particular workpiece is called a project in NUMgear.

The project is stored in a file and can be edited with the NUMgear HMI.

You can create projects, delete or save, edit projects, copy projects and finally generate the CNC code that will let the machine hob the part defined.

A project remains in the project directories - only the CNC program generated from the project will be transferred to the NC.

Projects can be saved and reloaded for further editing.

Project files can also be created on a PC connected to a Flexium+ controller. In addition, project files can be copied and used between different machines of the same type.

Changes made to the NC program will not be reflected in the project.

The basic structure of a project is very simple:

- Start Program - general machine parameters
- Setup of the gear shaft
- Gear Data
- Tool Selection
- Hobbing cycle
- End Program.

For the hobbing of cluster gears, the steps between Start and Stop Program can be repeated.

You would simply define another gear and another process, and select another tool.

1.4 Use of the manual

This manual is for use by qualified personnel only.

To reduce the risk due to an improper use of the product, do not perform any servicing other than that contained in this manual unless you are qualified to do so.

To ensure proper use of these products, please always read this User's manual carefully and retain for future reference. Should the unit require maintenance, please contact a NUM authorized service center only.

1.5 How to read the manual

This manual provides information about the NUMgear HMI.

Chapter	Description
1	**Foreword** This chapter introduces: - NUMgear Machining Features. - Options for Axial Cycles, Radial Cycles, Shifting, Machine Control and Program Flow. - Options for Radial Cycles. - Options for Shifting. - Options for Machine Control. - Options for Program Flow. - NUMgear User Interface Features. - Use of the manual.
2	**Get Started with the NUMgear GUI** This chapter introduces: - Create a Project. - How to get around in NUMgear: Input Devices, Mouse and touch panel, Keyboard, Error Messages Predefined Patterns, Calculations. - Put Together a Program: Options Column.
3	**Pages Description** This chapter introduces: - Straight Hob Page. - Tapered Hob. - Milling Tool. - Start Program. - Tool Selection. - Gear Data. - Setup Data. - Hobbing Cycles. - Gear Alignement. - Radial Axial Hobbing Cycle. - Radial Hobbing Cycle. - Radial Tangential Hobbing Cycle. - Diagonal Hobbing Cycle.

1.6 Document revisions

Date	Index	Reason for revision	See chapter
05 - 2018	00	Document creation	-

1.7 NUM Agencies

The list of NUM agencies is given click here http://www.num.com

Page deliberately empty

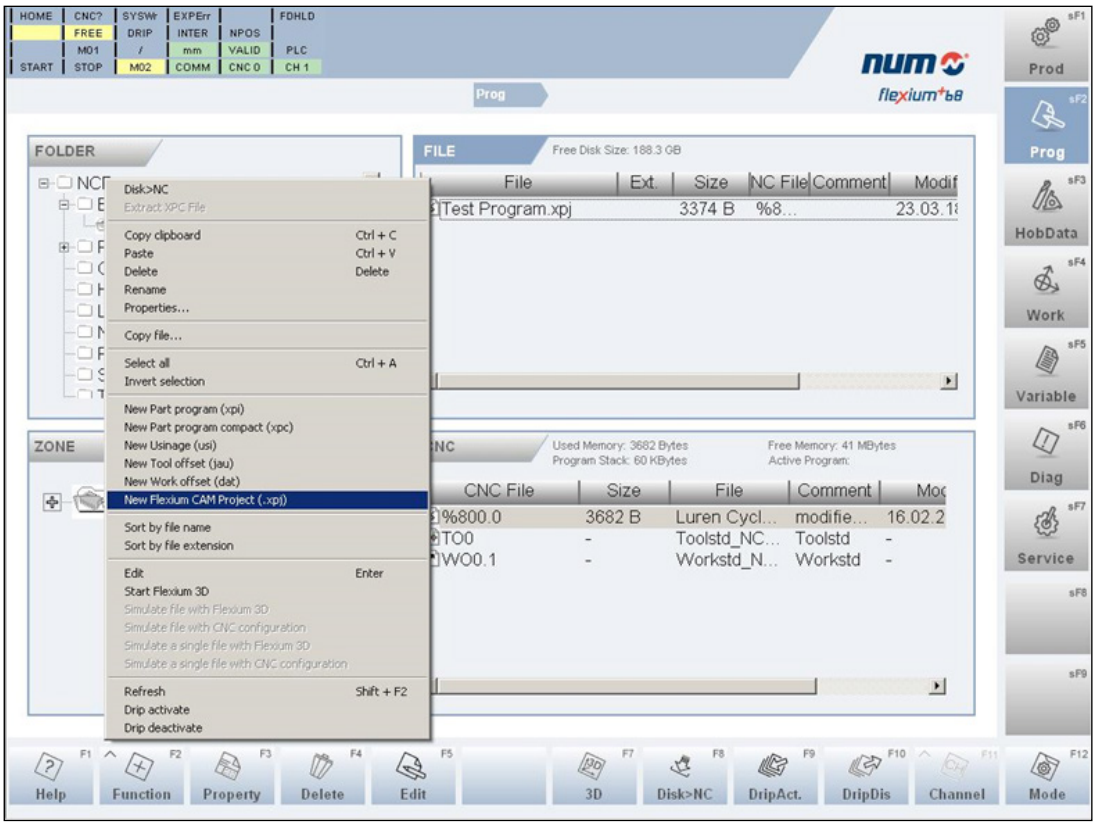
Summary

Page deliberately empty

2 Get Started with the NUMgear HMI

2.1 Create a Project

- Go to the programming page of the Flexium⁺ by using the *soft key sF2 Prog*.
- You create a new project by selecting the *soft key F2 Function*, create new *Flexium CAM project*.
- Enter the required data and save. Setting of the NC program number can be by default or set individually for each project on the service page.



A new project file will be generated into the NC Data folder selected.

To edit a project, select the project file.

2.2 How to get around in NUMgear

2.2.1 Input Devices

This program can be used with a variety of input devices such as keyboard, mouse, or touch panel as well as a combination of them.

2.2.2 Mouse and touch panel

- With mouse and touch panel, every item on the screen can be reached directly.
- A single tap or a single click will activate the corresponding page or entry control.
- To add items to the program flow, double tap or double click are necessary.

2.2.3 Keyboard

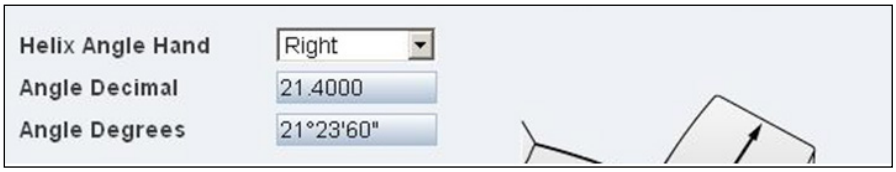
- With a keyboard, you use the tab key to switch between page areas.
- Use the arrow keys to move the focus between items in both left hand columns - the options and the program flow column.
- To get to the controls of a details page, activate that page using the tab keys, and use the arrow down once to set the focus to the first control on the page.
- Then move between the controls using the tab keys.

2.2.4 Error Messages

- Some entry fields have rules specified for them.
- The value for module e.g. cannot be 0 or negative. If you enter an invalid value, the system will display an error message.
- That error message will let you know that there are errors on the specific page as long as the focus is on an entry field with a correct value or without any rule defined.
- If the entry field that causes an error message gets the focus, the message displayed will change to the message defined for that entry field.

2.2.5 Predefined Patterns

- Some entry fields have predefined patterns that will not let you enter invalid keys, like characters for a numerical value.
- Some entry fields provide patterns that make data entry simpler.
- There are fields that require an angle as 00°00’00”.
- The signs **°**, **‘** and **“** are given, so that entering 000000 will result in the value defined.



2.2.6 Calculations

- Also, entry fields have calculations included, that will change the values of other fields.
- Angles can be entered in decimal or with minutes and seconds.
- Only one form needs to be entered, the other will be calculated automatically as soon as the field loses the focus.

2.3 Put together a Program

Here we will go over the process of putting together a program in a project file ready to be used to create the NC code out of it.

For the description of each details page and its controls please see the next chapter. The details page will open automatically once an item is created or selected.

Please note that to add items to the program flow, double tap or double click are necessary.

The NUMgear HMI is made up of three elements.

From left to right they are:

- Options Column
- Program Column
- Details Pages.

2.3.1 Options Column

The Options Column is made up of two major categories:

- Commands
- Tools.

2.3.2 Tools

The Tools category contains:

- Straight Hob
- Tapered Hob
- Milling Tool.

The first step for this project would be to define a tool.

Tools that have already been defined in other projects don't have to be redefined.

Double click a tool and edit the appropriate values in the details page.

2.3.3 Commands

The Commands category consists of three subdivisions

- Basic Data
- Hobbing Cycles
- General.

- Open the *Basic Data* - select *Start Program* and edit the values.
- Go to *Tool Selection* and select the tool you previously created.
- Select *Gear Data* and edit the details pages.
- Select *Setup Data* and enter the values.
- Now open the *Hobbing Cycles* and *select a cycle*. Set up the values for hobbing.
- To end the program flow, open *General* and select *End Program*. No data entry required.

You can always review the data you entered by selecting the entry in the program flow.

This is the program flow:

1. Start Program
2. Tool Selection
3. Gear Data
4. Setup
5. Hobbing cycle
6. End Program.

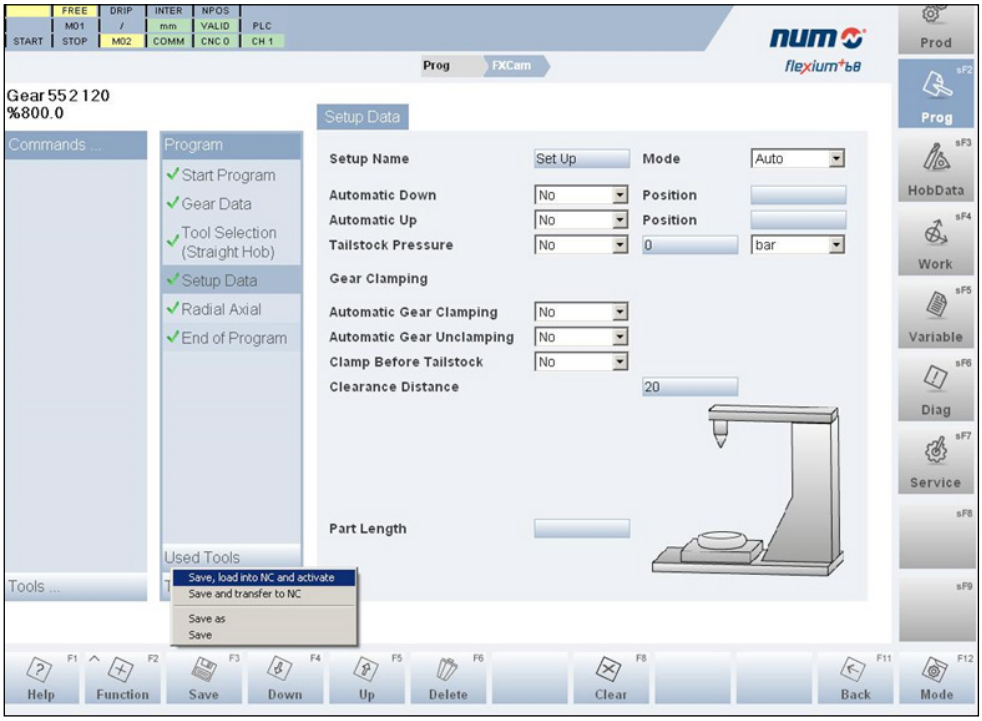
If you have a cluster gear, you can repeat steps 2 - 5 by inserting more entries.

Entries in the program flow can be added and deleted, and the order of entries can be changed.

The program flow reflects the CNC program that will be created and sent to the NC.

And that is all that is required to create a program.

Now select *Save*, *Transfer to NC* and *Activate*.



Page deliberately empty

Page deliberately empty

Summary

Page deliberately empty

3 Pages Description

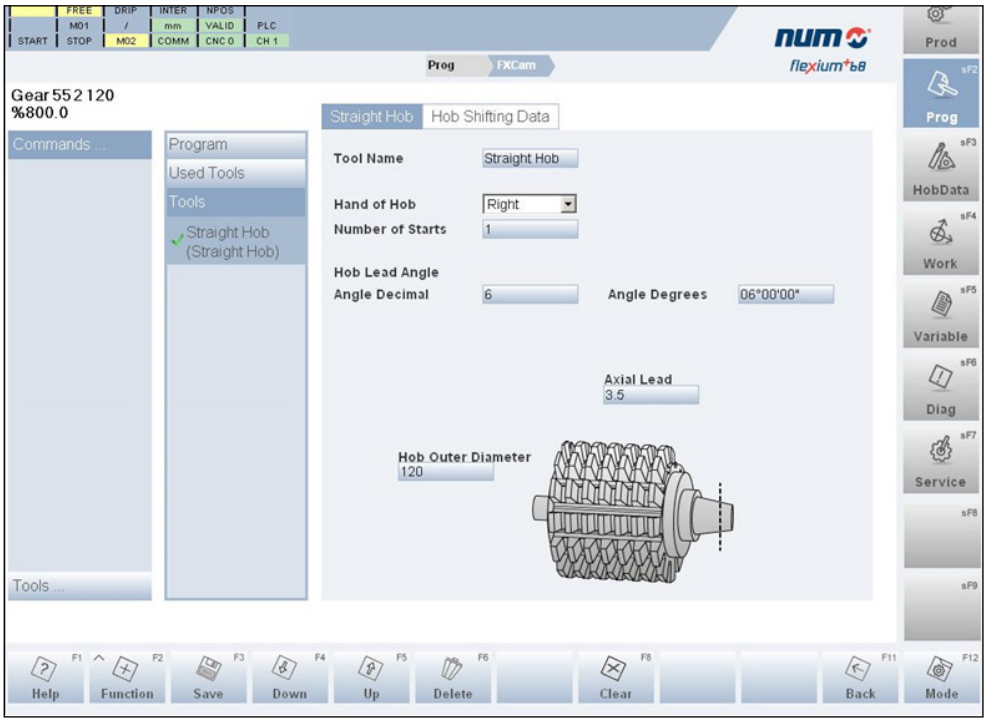
3.1 Straight Hob Page

The Straight Hob Page has two tabbed subpages.

The first is used to enter the common hob data.

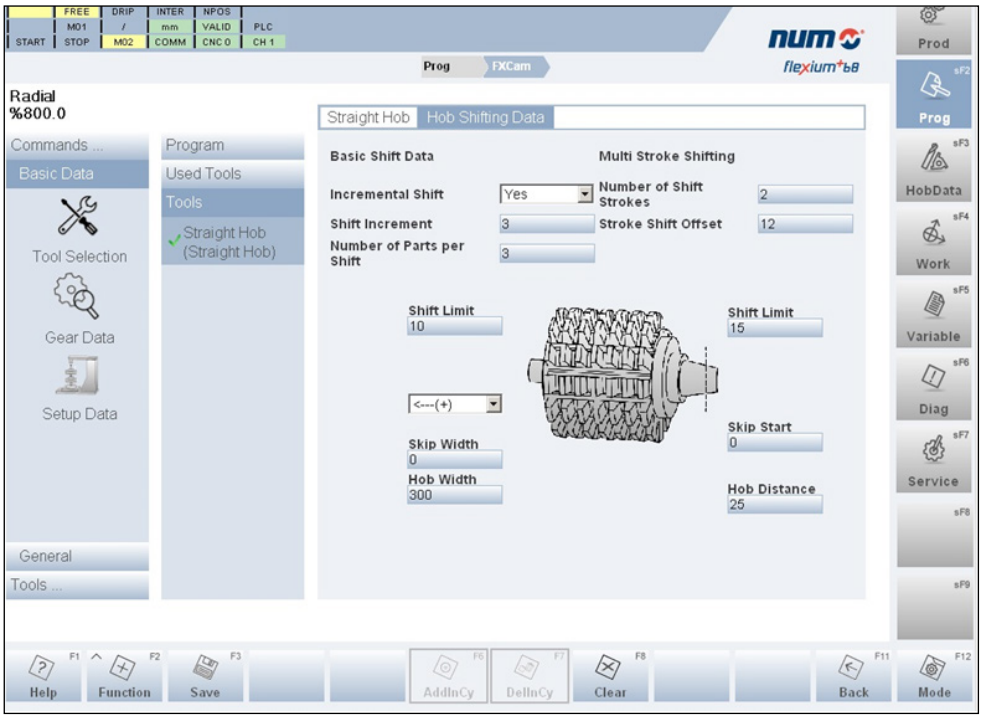
In the second subpage, you can specify the shifting for the hob so that it is used along the whole length available for cutting.

3.1.1 Basic Hob Data



Field Name	Description
Hand of Hob	Can be right or left. If you think of the hob as a screw, a right hand hob when turned clockwise will screw in, and a left hand hob when turned clockwise will screw out. Same applies to the hand of the gear.
Number of Starts	Defines the number of thread starts on the hob and can be a value between 1 and 9.
Lead Angle	Can be entered in either decimal degrees, or in degrees° minutes'seconds".
Axial Lead	This is the distance from thread to thread of one hob start.
Diameter	Outer diameter of the Hob.

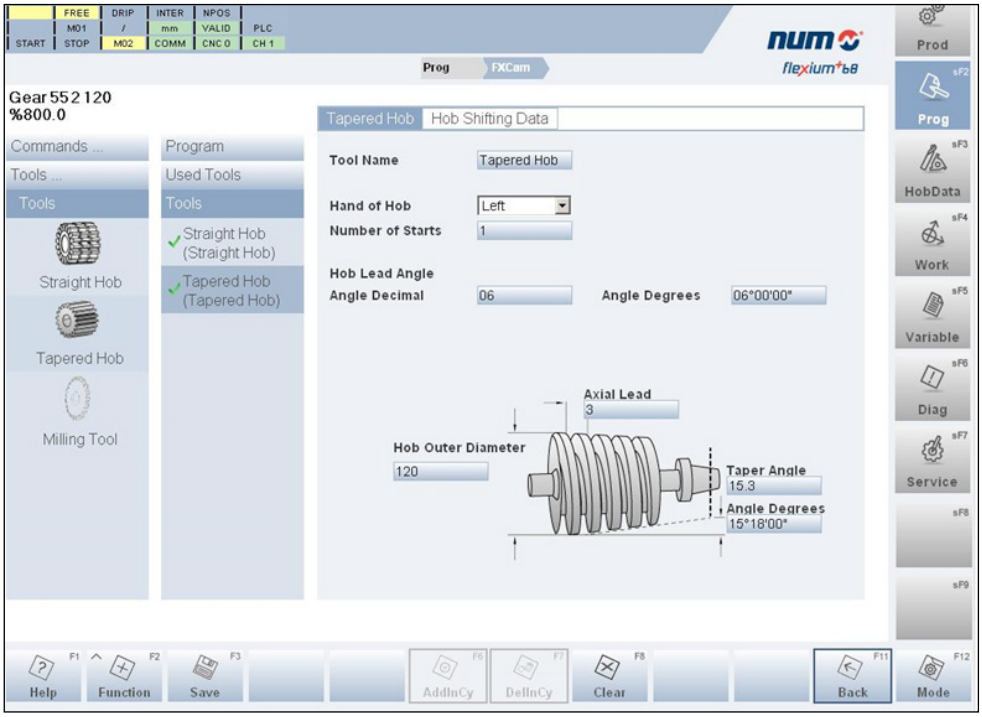
3.1.2 Shifting Data



Field Name	Description
Incremental Hob Shift	Enables or disables the incremental shifting function.
Gear Rotates With Shift	Enables or disables the Y-->C link. For servo controlled Y axes, it is possible to link the Y axis with the C axis so that when the Y axis moves, the C axis rotates along with it, essentially keeping in sync with the hob. This feature can be used to maintain Hob / Table alignment during incremental hob shifting.
Shift Increment	This is the distance the hob will move when the hob is shifted. Must be a positive value.
# of Parts per Shift	This field determines the frequency of the shifting operation. The hob will be shifted every #th part hobbed.
Part Counter	Contains the current part counter for this interval between shifts.
Direction	Direction of the hob shift.
Shift Limit	Defines the beginning and end positions that are possible for the shifting. Distance of the shifting positions both are positive numbers. They both relate to the distance between the spindle nose and the corresponding limit. They do not reflect the actual Y axis positions.
Current Position	Controls the position of the Y axis for the next cycle. It does contain the Y axis position directly, but always as a positive value, never negative. You can use this field to control the start point of the hob on the next cycle: *Current Pos = 0* Causes a hob shift rewind to the starting point, the extreme travel limit, in preparation for a new shifting setup. *Current Pos = Non Zero Value* Forces a hob position change to the new position specified. The position is relative to the centerline of the gear table (C axis) .

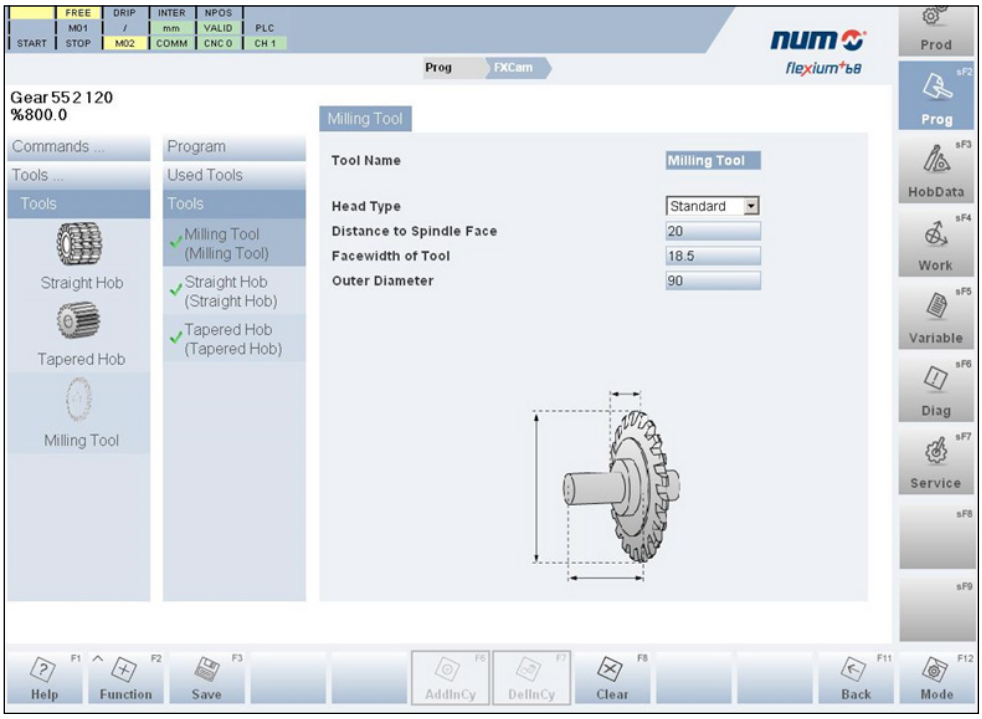
Number of Shift Strokes	Controls the number of complete shifting cycles from one to the other shift limit. After that, the hob must be checkeds.
Current Shift Strokes	Contains the current number of shifting cycles. When this value amounts to the Number of Shift Strokes, a message will be displayed and the cycle will continue only if the value is manually set to a lower value. If it is set to 0, then it will return to the starting limit.
Stroke Shift Offset	The offset from the shift limits after the hob moves to the shift limit when the far limit is reached.

3.2 Tapered Hob



Field Name	Description
Hand of Hob	Can be right or left. If you think of the hob as a screw, a right hand hob when turned clockwise will screw in, and a left hand hob when turned clockwise will screw out. Same applies to the hand of the gear.
Number of Starts	Defines the number of thread starts on the hob and can be a value between 1 and 9.
Lead Angle	Of the hob threads. Can be entered in either decimal degrees, or in degrees°minutes'seconds".
Axial Lead	This is the distance from thread to thread of one hob start.
Diameter	Largest outer diameter of the Hob.
Hob Taper Angle	Amount of taper in the hob. Can be entered in either decimal degrees, or in degrees°minutes'seconds".

3.3 Milling Tool

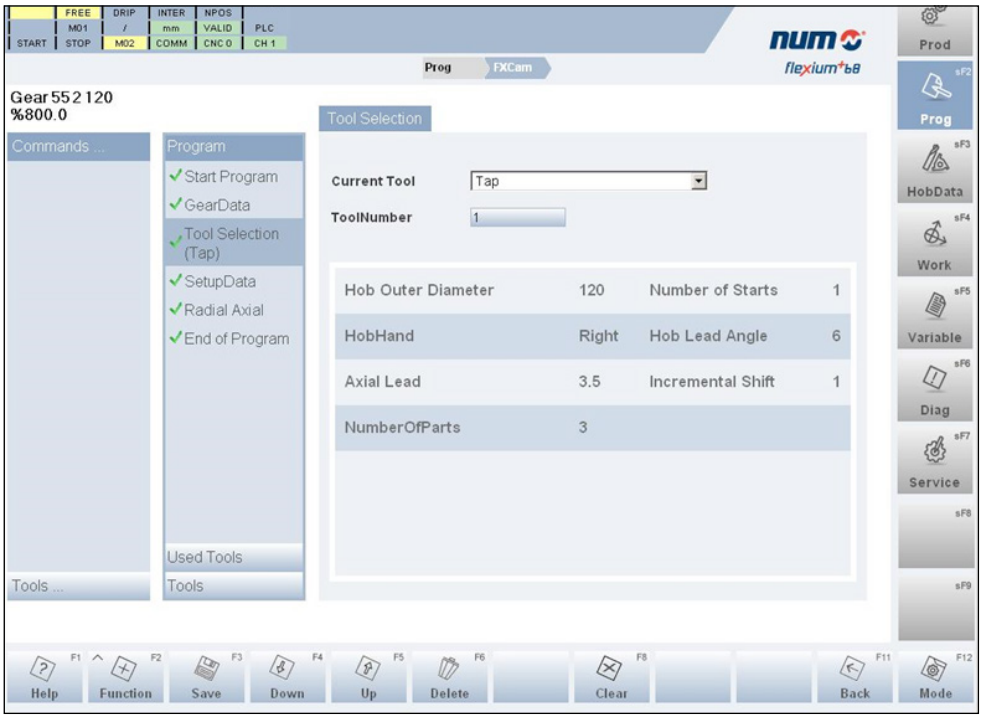


Field Name	Description
Head Type	Selects a standard or a 90 degree type milling head. 90 degree head allows for thread milling. If 90 Degree head type is selected, the tool swivel angle is inverted.
Distance to Spindle Face	Is the distance between the spindle face (nose) and the center of the milling cutter. This value is used to position the cutter on the centerline of the table.
Face Width of Cutter	The width of the milling cutter must be entered when a 90 degree head, a thread milling head is selected. This value is used to determine approach and over travel points of the Z-Axis. This value is not used for a standard head type.
Milling Tool Diameter	Defines the diameter of the milling cutter. The diameter is used as one of the variables in determining the scratch point and the depth of cut.

3.4 Start Program

Field Name	Description
Comment	Free to describe the process and / or the work piece.
Units	Defines the unit of the numbers entered in the data fields of the program as in millimeter or in inches.
Part Loading Mode	Can define a manual or an automatic loading process.
Spindle Range	Can be used to define a spindle range. Range one is used for older machines with only one range. Range two will program an M41.
Error Tolerance	Sets the maximum synchronization error between the table and the hob spindle.
Number of Gear Rotations	Refers to the number of gear rotations done for Fast Gear Alignment. Default value is 1, the maximum is 999 divided by number of teeth.
Tooth Index Error Tolerance	This field is used during the Fast Gear Alignment to determine if the gear is valid. It refers to the maximum deviation allowed. Any gear tooth / gear gap that doesn't match this limit, is considered to be out of range. Default value is 0.5. If the value for deviation of the indexed teeth is above the threshold, a message will be displayed and the cycle will stop as there is the danger of damage to either the gear or the tool.

3.5 Tool Selection



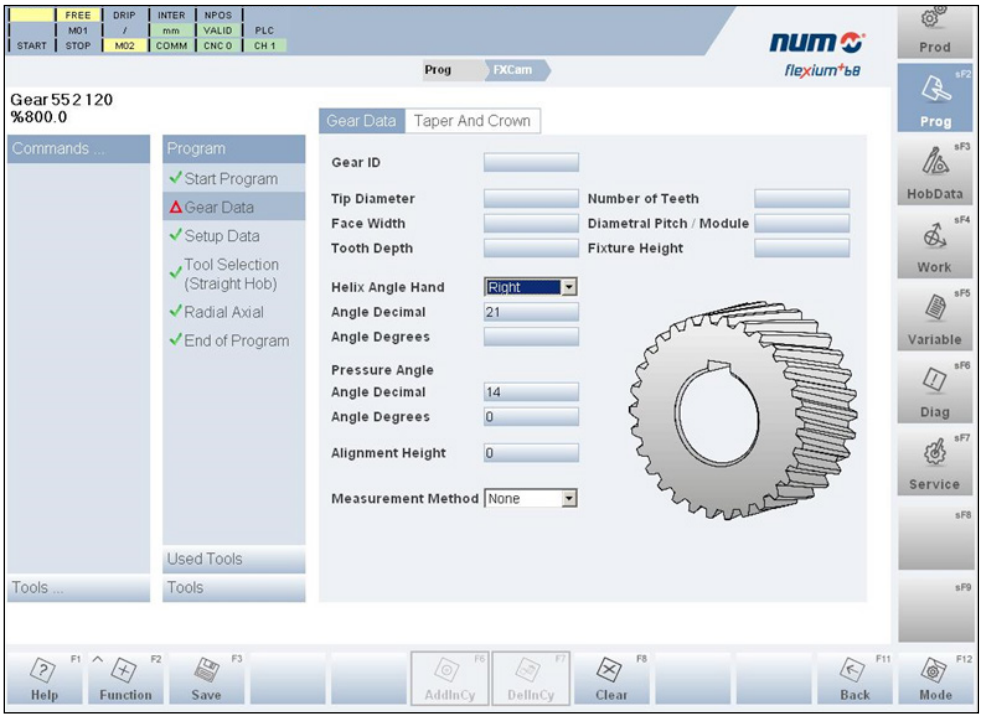
Field Name	Description
Tool Number	The number of the tool in the sF3 Tool Management.
Current Tool	A selection of all tools available. Select the appropriate tool for the hobbing process.

3.6 Gear Data

The gear data page has two tabbed subpages.

The first is used to enter the common gear data, in the second subpage, you can specify crowning and tapering for the gear flanks.

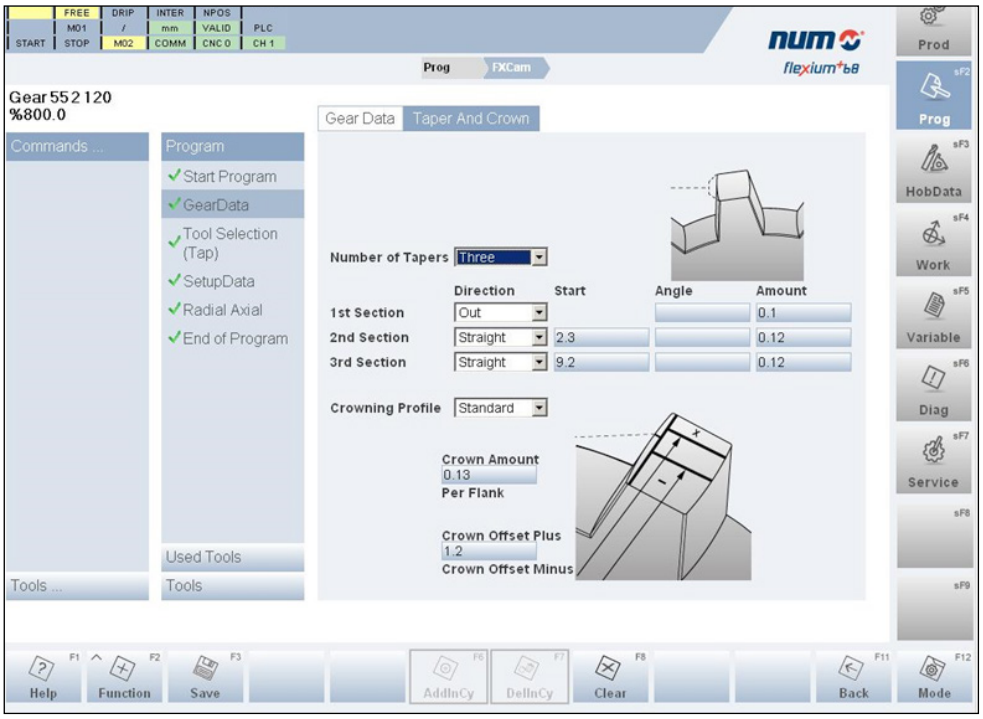
3.6.1 Basic Gear



Field Name	Description
Gear ID	This is a string you can use to identify your gear.
Tip Diameter	This is the outer diameter of the gear to be hobbed.
Face Width	Refers to the length of the gear tooth. You must define exactly how long the teeth on the gear are. If is not a spline or shaft gear, then the value you enter here must be the face width of the gear blank. If you are cutting splines or end of shaft gears then enter the desired tooth length.
Number of Teeth	Number of teeth to be cut on this gear blank. Minimum is three, maximum number is 9990.
Tooth Depth	Controls the distance between the tooth tip and the gap root of the gear. The whole depth of the cut - the maximum - should be entered. The tooth depth determines the travel of the hob in radial direction into the gear blank.
Module or Pitch	Either value can be entered, depending on the value for unit set in the Start Program Page.
Helix Hand	You can select spur gear - no helix angle, right hand or left hand. If you think of the gear as a screw, a right hand gear when turned clockwise will screw in, and a left hand gear when turned clockwise will screw out. Same applies to the hand of the hob.

Helix Angle	This field is displayed only for helical, not for spur gears. The angle can be entered in decimal degrees, or in degrees°minutes'seconds" and is always positive, as the sign is determined by the helix hand - right hand results in a positive and left hand results in a negative value.
Pressure Angle	This is used to enter the angle of the flank at the pitch circle.
Alignment Height	You can specify the position of the Fast Gear Alignment sensor in relation to the bottom position of the gear.
Fixture Height	Defines the distance from the face of the machine's table to the bottom edge of the gear teeth or bottom edge of the gear blank depending on whether you are cutting shafts or plain gears. This controls the position of the cycle in the Z (axial) direction. The location of the entire cycle depends on this data. Changing this data moves the entire cycle plus or minus along the Z axis.
Measuring	You can define measuring over pins or measuring over a span. The corresponding entry fields will be displayed after you have selected a method.
Pin Diameter	Enter the diameter of the pins of your measuring device.
Diameter Over Pins	Enter the diameter that should be the result of the measurement.
Teeth in Span	The number of teeth used for span measurement.
Span	The desired result of the measurement.

3.6.2 Taper and Crowning



The gear flank can be modified by taper sections and crowning.

Both taper and crowning will be combined.

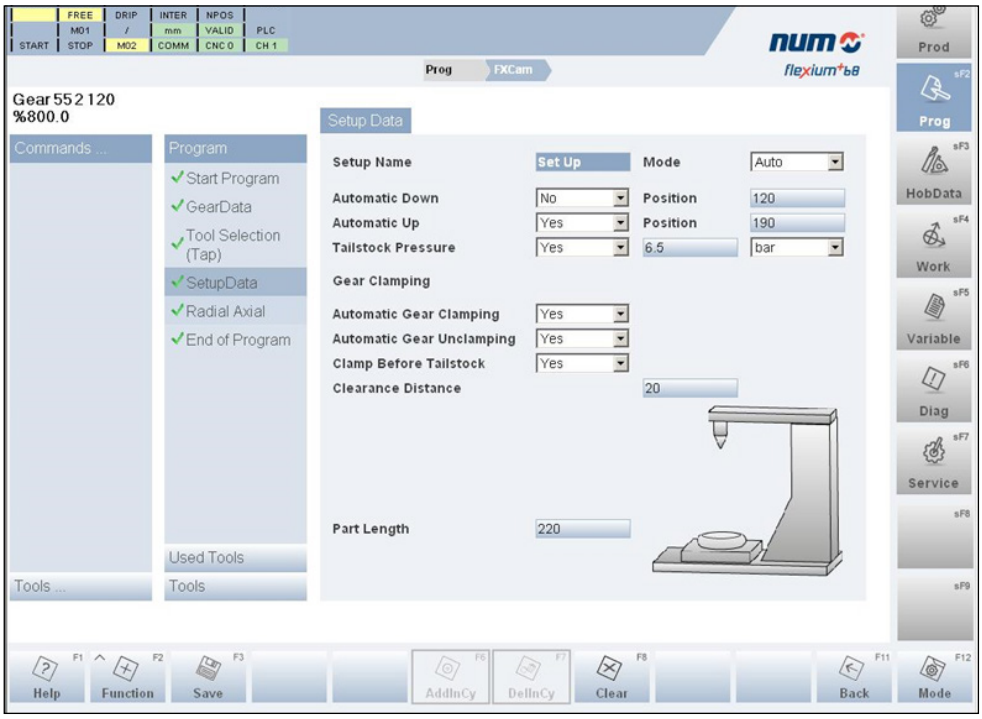
Field Name	Description
Number of Tapers	You can select up to three taper sections.
Start	Defines the position of the start of a taper section. It refers to the edge of the gear where the cutting starts. On a vertical machine during a climb cut, the initial cut edge is the bottom of the gear. In this case, the Start distance is with respect to the bottom corner of the gear. On a vertical machine during a conventional cut, the initial cut edge is the top of the gear. In this case, the Start distance is with respect to the top corner of the gear. When entering a single taper, the start of the taper is always at the initial cut edge of the gear. The taper continues to the opposite gear end. When entering two taper sections, the Start of the second taper is the distance with respect to the initial cut edge of the gear, where taper number one will end and taper number two will start.
Direction	Defines the direction of each taper section as in, out, or straight.
Angle	Defines the amount of taper as an angle. Always enter a positive value, the direction of the taper is entered in the “Direction” field. The angle value is in decimal degrees. Do not enter taper as both an angle and amount.
Amount	Defines the amount of taper but entered as an incremental distance in the X axis direction. Always enter a positive value, the direction of the taper is determined by the direction. Do not enter taper as both an angle and amount.

Two types of crowning are possible, straight crowning with an optional offset, and crowning that leaves straight sections at the top and the bottom of the gear.

The corresponding fields will be displayed after selection of the crowning method.

Field Name	Description
Amount	Define the amount of crowning.
Offset	Changes the position of the high point of the crowning. Enter an incremental value here other than zero which defines amount and direction of the offset. A value of zero will always center the high point of the crown in the middle of the tooth length. This value can have either a plus or minus sign to indicate the offset direction. You can use offset crowning to approximate a gear which has both taper and crown. If you draw a line from the start of the crowning point through the gear along a tooth to the end of the crowning point, you will notice that this line is on an angle.
Straight Distance Plus / Minus	Used to define the portion of flank that is to be left out of crowning. If this is specified, the crowning point will be centered between the two end points of the straight distance parts seen from top and bottom of the gear. Enter positive values as the position is determined by the entry field name.

3.7 Setup Data



Setup allows for the definition of tailstock use, clamping and loading positions.

Field Name	Description
Setup Name	Allows for a description of the setup.
Operating Mode	Defines if it is a mechanical or a servo controlled tailstock.
Automatic Tailstock Down	Used to select, if the tailstock is to lower automatically prior to the cutting of the gear.
Automatic Tailstock Up	Used to select, if the tailstock is to rise automatically prior to the cutting of the gear.
Pressure	Used to select, if the tailstock will automatically apply pressure to the fixture or gear blank during the cutting process.
Automatic Gear Clamping	Used to select, if the clamping system will automatically clamp prior to cutting of the gear.
Automatic Gear Unclamping	Used to select, if the clamping system will automatically unclamp after the cutting of the gear.
Clamp Before Tailstock	Used to select, if the clamping system will automatically clamp before the tail-stock comes down or if the tailstock will come down before the gear is clamped.
Position	Sets the upper position of the tailstock servo axis.
Pressure	The amount of pressure of the tailstock.
PSI or bar	Sets the unit of the value of the pressure amount.
Clearance Distance	The safety distance between tool and gear for loading and unloading.
Part Length	Length of the part, including shaft.
Fixture Height	Height of the Fixture.

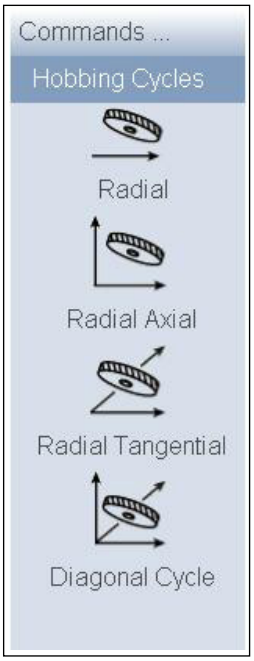
3.8 Hobbing Cycles

There are predefined hobbing cycles available for selection.

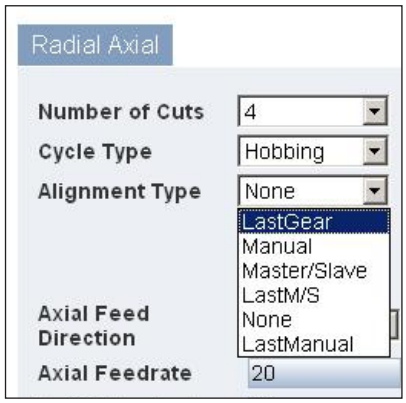
The normal cylindrical gear, spur or helical with involute flanks would be cut with a radial axial cycle.

Several entry fields are common throughout all cycles, will however be explained in each cycle so the user won't have to search across the cycles.

Only the Alignment Type, and the Degree of Correction fields will be explained ahead as they refer to the Fast Gear Alignment that can be used to align precut gears or to align cluster gears on a shaft.



3.9 Gear Alignement



Field Name	Description
Alignment Type	This field allows selection of an alignment function. It allows for the selection of *Last Gear, Manual, Master Gear, Slave Gear, Last M/S, No Alignment*. • *Last Gear* allows for re-cutting gears or for cluster gear tooth alignment between multiple gears on the same shaft. • *Manual* allows for manually aligning the cutting tool to the workpiece, via a touch off routine. • *Master* is part of the *Automatic Gear Alignment* function. The gear is manually aligned with the cutting tool. Then the cutting tool is retracted and the sensor is moved in. The gear will rotate and the position of the teeth is recorded and stored to save the gear location. This field will automatically switch from *Master to Slave* upon completion of the master calibration phase. • After the *Master gear* has been calibrated and stored, *Slave* indicates that automatic gear alignment will be executed prior to the cycle to synchronize the gear location with respect to the tool. • After the *Master gear* has been calibrated and stored, *Last M/S* indicates that automatic gear alignment will be not be executed and that the current gear will be set at the previous sensed gear's location with respect to the tool. This feature allows for re-cut of a previously sensed gear or cluster gear tooth alignment between multiple gears on the same shaft. • *None* will reduce process time at the beginning of the cycle as no alignment will take place. • After the *Master Gear* has been calibrated and stored, adjusting the *Alignment Type* field to *Last Gear, Master Gear* or *No Align*, will invalidate the recorded *Master data*.
Degrees of Correction	• Allows an offset from the aligned position before cutting the gear. For example, if after cutting the first cluster gear shaft you find gear one and two are offset by 1 degree, enter a 1 degree offset to correct this problem. During *Automatic Gear Alignment* this field will be automatically calculated and loaded. • If your setup requires *Automatic Gear Alignment*, while also requiring cluster gear tooth alignment between multiple gears on the same shaft, you must manually copy the *Degrees of Correction* value from the sensed gear to the second gear on the shaft.

3.10 Radial Axial Hobbing Cycle

The Radial Axial Cycle can be used to plunge cut splines, cylindrical helical gears, cylindrical spur gears, and any type of gear cut from a blank or shaft.

In this cycle, the hob first approaches the work in the radial (X) direction. The feed then switches to axial (Z) and the first pass is taken.

The hob then backs out or goes deeper if a second, third or fourth cut is selected. The second pass is then taken using the (Z) axis. No tangential (Y) axis motion is allowed except for shifting at the end of the cycle.

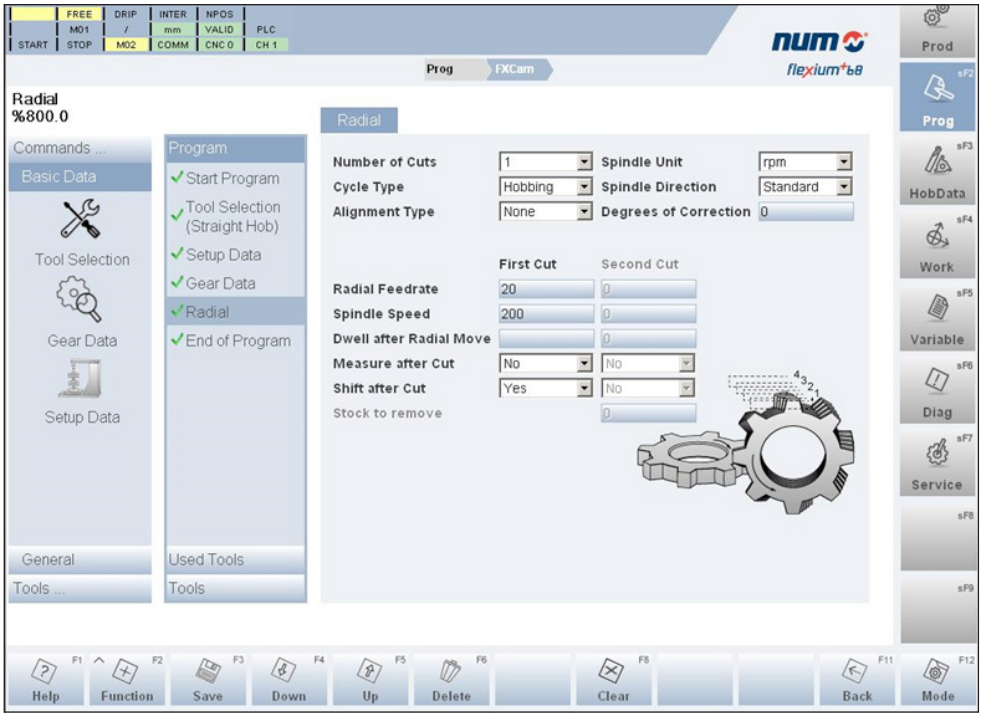
Field Name	Description
Number of Cuts	Selection of the number of radial infeed steps. Maximum number of cuts is four.
Cycle Type	This field allows selection of either a hobbing or single indexing cycle.
Spindle Direction	The direction of the cutting tool is automatically determined by the axial feed direction, climbing or conventional. The spindle rotation can be inverted.
Axial Feed Direction	The direction in which the cutter is to be moved through the gear. Climbing is the positive Z direction and conventional is the negative Z direction.
Radial Feed Rate	The rate at which the cutter is fed into the workpiece (using X axis). Expressed as distance/revolution of the gear when hobbing and distance/minute when milling. The selection of a radial feed rate will cancel all axial approach over travel calculations. The cutter will approach the gear at the start of the teeth, and then feed into the blank from there. If multiple cuts are selected, then this is mandatory only in the subsequent cuts.
Spindle RPM	The speed at which the cutter is turned during the hobbing process.
Dwell After Radial Move	In the hobbing mode, the number of gear revolutions to dwell when the cut is complete. In the milling mode the number of seconds to wait after the cut is complete.

Dwell After Axial Move	In the hobbing mode, the number of gear revolutions to dwell when the cut is complete. In the milling mode the number of seconds to wait after the cut is complete.
Measure After Cut	After this cut, the axis will back out and the hob spindle will stop. The operator is then prompted to measure the part and make the appropriate size adjustment. The value entered as a diameter, will adjust only the X axis position. Positive adjustment results in a larger part. After the adjustment is entered the cycle will continue to the next phase.
Shift After Cut	After the programmed cut pass, the axis will back out and the hob spindle shift the Shift Increment amount pre-defined in the Hob Shifting Data page. The shifting data Part Counter Total will reset to zero after the shift. After the hob was shifted, the cycle will continue to the next phase.
Stock to Remove	Can be input for the second cut only. It defines the amount of stock to remove for the current cut. The first cut will be made at whole depth plus total stock for all cuts - X axis moved in the plus direction. The final cut will be made at exactly the whole depth.

3.11 Radial Hobbing Cycle

The Radial Cycle can be used to cut worm wheels, or undercut sections of a gear.

There is no axial motion except to center the hob in the middle of the tooth length prior to the radial operation in X.



Field Name	Description
Number of Cuts	Selection of the number of radial infeed steps. Maximum number of cuts is two.
Cycle Type	This field allows selection of either a hobbing or single indexing cycle.
Spindle Direction	The direction of the cutting tool is automatically determined by the axial feed direction, climbing or conventional. The spindle rotation can be inverted.
Radial Feed Rate	The rate at which the cutter is fed into the workpiece (using X axis). Expressed as distance/revolution of the gear when hobbing and distance/minute when milling. The selection of a radial feed rate will cancel all axial approach over travel calculations. The cutter will approach the gear at the start of the teeth, and then feed into the blank from there. If multiple cuts are selected, then this is mandatory only in the subsequent cuts.
Spindle RPM	The speed at which the cutter is turned during the hobbing process.
Dwell After Radial Move	In the hobbing mode, the number of gear revolutions to dwell when the cut is complete. In the milling mode the number of seconds to wait after the cut is complete.
Measure After Cut	After this cut number, the axis will back out and the hob spindle will stop. The operator is then prompted to measure the part and make the appropriate size adjustment. The value entered as a diameter, will adjust only the X axis position. Positive adjustment results in a larger part. After the adjustment is entered the cycle will continue to the next phase.

Shift After Cut	After the programmed cut pass, the axis will back out and the hob spindle shifts the Shift Increment amount pre-defined in the Hob Shifting Data page. The shifting data Part Counter Total will reset to zero after the shift. After the hob was shifted, the cycle will continue to the next phase.
Stock to Remove	Can be input for the second cut only. It defines the amount of stock to remove for the current cut. The first cut will be made at whole depth plus total stock for all cuts - X axis moved in the plus direction. The final cut will be made at exactly the whole depth.

3.12 Radial Tangential Hobbing Cycle

The Radial/Tangential Cycle can be used to plunge cut worm gears.

In this cycle, the hob first approaches the work in the radial (X) direction. The feed then switches to tangential (Y) and the first pass is cut.

The hob then backs out or goes deeper if a second cut is selected. The second pass is then cut using the (Y) axis. No axial (Z) axis moves are made except at the beginning of the cycle, to align the centerline of the hob and the centerline of the gear blank.

Field Name	Description
Number of Cuts	Selection of the number of radial infeed steps. Maximum number of cuts is two.
Cycle Type	This field allows selection of either a hobbing or single indexing cycle.
Spindle Direction	The direction of the cutting tool is automatically determined by the axial feed direction, climbing or conventional. The spindle rotation can be inverted.
Tangential Feed Direction	The direction in which the cutter is to be moved across the gear. Plus is the positive Y direction and minus is the negative Y direction.
Tangential Feedrate	The rate at which the cutter is fed across the workpiece (using Y axis). Expressed as distance per revolution of the gear when hobbing and distance per minute when milling.
Radial Feed Rate	The rate at which the cutter is fed into the workpiece (using X axis). Expressed as distance/revolution of the gear when hobbing and distance/minute when milling. The selection of a radial feed rate will cancel all axial approach over travel calculations. The cutter will approach the gear at the start of the teeth, and then feed into the blank from there. If multiple cuts are selected, then this is mandatory only in the subsequent cuts.
Spindle RPM	The speed at which the cutter is turned during the hobbing process.
Dwell After Radial Move	In the hobbing mode, the number of gear revolutions to dwell when the cut is complete. In the milling mode the number of seconds to wait after the cut is complete.

Dwell After Tangential Move	In the hobbing mode, the number of gear revolutions to dwell when the cut is complete. In the milling mode the number of seconds to wait after the cut is complete.
Measure After Cut	After this cut number, the axis will back out and the hob spindle will stop. The operator is then prompted to measure the part and make the appropriate size adjustment. The value entered as a diameter, will adjust only the X axis position. Positive adjustment results in a larger part. After the adjustment is entered the cycle will continue to the next phase.

3.13 Diagonal Hobbing Cycle

The Diagonal Cycle can be used to cut helical, spur or spline gears as well as taper root splines.

In this cycle, the hob first approaches the work in the radial (X) direction.

- If no radial feed is selected, the cutter will approach above or below the gear blank. The feed then switches to axial (Z) until the hob is fully engaged in the blank, then a combination of axial and tangential (Y) feed is made for the first pass.
  The hob then backs out or goes deeper if a second cut is selected.
  The second pass is then cut using the combination of axial and tangential motion.
- If radial feed is selected, the cutter will approach at the bottom edge or the top edge of the gear blank. The feed first switches to radial (X) until the hob is fully engaged in the blank, then a combination of axial and tangential (Y) feed is made for the first pass.
  The hob then backs out or goes deeper if a second cut is selected. The second pass is then cut using the combination of axial and tangential motion.

Field Name	Description
Number of Cuts	Selection of the number of radial infeed steps. Maximum number of cuts is two.
Cycle Type	This field allows selection of either a hobbing or single indexing cycle.
Spindle Direction	The direction of the cutting tool is automatically determined by the axial feed direction, climbing or conventional. The spindle rotation can be inverted.
Axial Feed Direction	The direction in which the cutter is to be moved through the gear. Climbing is the positive Z direction and conventional is the negative Z direction.
Tangential Feed Direction	The direction in which the cutter is to be moved across the gear. Plus is the positive Y direction and minus is the negative Y direction.
Axial Feed Rate	The rate at which the cutter is fed axially along the workpiece in Z as distance/ revolution.

Radial Feed Rate	The rate at which the cutter is fed into the workpiece (using X axis). Expressed as distance/revolution of the gear when hobbing and distance/minute when milling. The selection of a radial feed rate will cancel all axial approach over travel calculations. The cutter will approach the gear at the start of the teeth, and then feed into the blank from there. If multiple cuts are selected, then this is mandatory only in the subsequent cuts.
Spindle RPM	The speed at which the cutter is turned during the hobbing process.
Dwell After Radial Move	In the hobbing mode, the number of gear revolutions to dwell when the cut is complete. In the milling mode the number of seconds to wait after the cut is complete.
Dwell After Axial Move	In the hobbing mode, the number of gear revolutions to dwell when the cut is complete. In the milling mode the number of seconds to wait after the cut is complete.
Measure After Cut	After this cut number, the axis will back out and the hob spindle will stop. The operator is then prompted to measure the part and make the appropriate size adjustment. The value entered as a diameter, will adjust only the X axis position. Positive adjustment results in a larger part. After the adjustment is entered the cycle will continue to the next phase.
Shift After Cut	After the programmed cut pass, the axis will back out and the hob spindle shift the *Shift Increment* amount pre-defined in the *Hob Shifting Data* page. The shifting data *Part Counter Total* will reset to zero after the shift. After the hob was shifted, the cycle will continue to the next phase.
Stock to Remove	Can be input for second cut only. It defines the amount of stock to remove for the current cut. The first cut will be made at whole depth plus total stock for all cuts - X axis moved in the plus direction. The final cut will be made at exactly the whole depth.

Page deliberately empty

Page deliberately empty

Published by NUM Spa

Via F. Somma 62
20012 Cuggiono (MI) (Italy)
Tel. : +39 02 979 691
website: www.num.com

Printed May 2018

CNC HighEnd Applications