# GEAR HOBBING MACHINE

CNC 8065

User manual (·G· model).

Ref: 2602

FAGOR AUTOMATION

**TRANSLATION OF THE ORIGINAL MANUAL.**

This manual is a translation of the original manual. This manual, as well as the documents derived from it, have been drafted in Spanish. In the event of any contradictions between the document in Spanish and its translations, the wording in the Spanish version shall prevail. The original manual will be labeled with the text "ORIGINAL MANUAL".

**MACHINE SAFETY.**

It is up to the machine manufacturer to make sure that the safety of the machine is enabled in order to prevent personal injury and damage to the CNC or to the products connected to it. On start-up and while validating CNC parameters, it checks the status of the following safety elements. If any of them is disabled, the CNC shows the following warning message.

- Feedback alarm for analog axes.
- Software limits for analog and sercos linear axes.
- Following error monitoring for analog and sercos axes (except the spindle) both at the CNC and at the drives.
- Tendency test on analog axes.

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC resulting from any of the safety elements being disabled.

**HARDWARE EXPANSIONS.**

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC resulting from any hardware manipulation by personnel unauthorized by Fagor Automation.

If the CNC hardware is modified by personnel unauthorized by Fagor Automation, it will no longer be under warranty.

**COMPUTER VIRUSES.**

FAGOR AUTOMATION guarantees that the software installed contains no computer viruses. It is up to the user to keep the unit virus free in order to guarantee its proper operation. Computer viruses at the CNC may cause it to malfunction.

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC due a computer virus in the system.

If a computer virus is found in the system, the unit will no longer be under warranty.

**DUAL-USE PRODUCTS.**

Products manufactured by FAGOR AUTOMATION since April 1st 2014 will include "-MDU" in their identification if they are included on the list of dual-use products according to regulation UE 428/2009 and require an export license depending on destination.

FAGOR AUTOMATION



The information described in this manual may be subject to changes due to technical modifications. Fagor Automation reserves the right to change the contents of this manual without prior notice.

All the trade marks appearing in the manual belong to the corresponding owners. The use of these marks by third parties for their own purpose could violate the rights of the owners.

It is possible that CNC can execute more functions than those described in its associated documentation; however, Fagor Automation does not guarantee the validity of those applications. Therefore, except under the express permission from Fagor Automation, any CNC application that is not described in the documentation must be considered as "impossible". In any case, Fagor Automation shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC if it is used in any way other than as explained in the related documentation.

The content of this manual and its validity for the product described here has been verified. Even so, involuntary errors are possible, hence no absolute match is guaranteed. However, the contents of this document are regularly checked and updated implementing the necessary corrections in a later edition. We appreciate your suggestions for improvement.

The examples described in this manual are for learning purposes. Before using them in industrial applications, they must be properly adapted making sure that the safety regulations are fully met.

# INDEX


**CAPÍTULO 1 BASIC CONCEPTS.**

1.1 General concepts. ..... 5
1.2 Configuration of the machine axes. ..... 5

**CAPÍTULO 2 BASIC OPERATIONS.**

2.1 Main screen. ..... 7
2.2 HOB Gear. ..... 8
2.2.1 Activate HOB. ..... 10
2.2.2 Add HOB. ..... 10
2.2.3 Delete HOB. ..... 10

**CAPÍTULO 3 CYCLE EDITING.**

3.1 Accessing gear hobbing cycles. ..... 11
3.2 Cycle editing. ..... 12
3.3 Simulation or execution of a gear hobbing cycle. ..... 12
3.4 Inserting a gear hobbing cycle into a program. ..... 13
3.5 Execution or simulation of a program with gear hobbing cycles. ..... 13

**CAPÍTULO 4 GEAR HOBBING CYCLES.**

4.1 Axial. ..... 15
4.2 Radial. ..... 18
4.3 Radial + Axial. ..... 21
4.4 Tooth by tooth Axial. ..... 24
4.5 Tooth by tooth Radial. ..... 27
4.6 Tooth by tooth Radial + Axial. ..... 30

**CAPÍTULO 5 TOOL VARIABLES**

5.1 List of variables from the technological tables. ..... 33
5.1.1 Variable group "COMMON". ..... 33
5.1.2 Variable group "GEOMETRY". ..... 33


FAGOR AUTOMATION

CNC 8065

REF: 2602

FAGOR AUTOMATION

CNC 8065

REF: 2602

# BASIC CONCEPTS.

1

## 1.1 General concepts.

In the world of gear hobbing, there are different models, tools and cycles to be carried out depending on the parts to be machined.

Fagor offers a series of simple cycles to the market that cover a large percentage of the requirements of the parts to be machined.

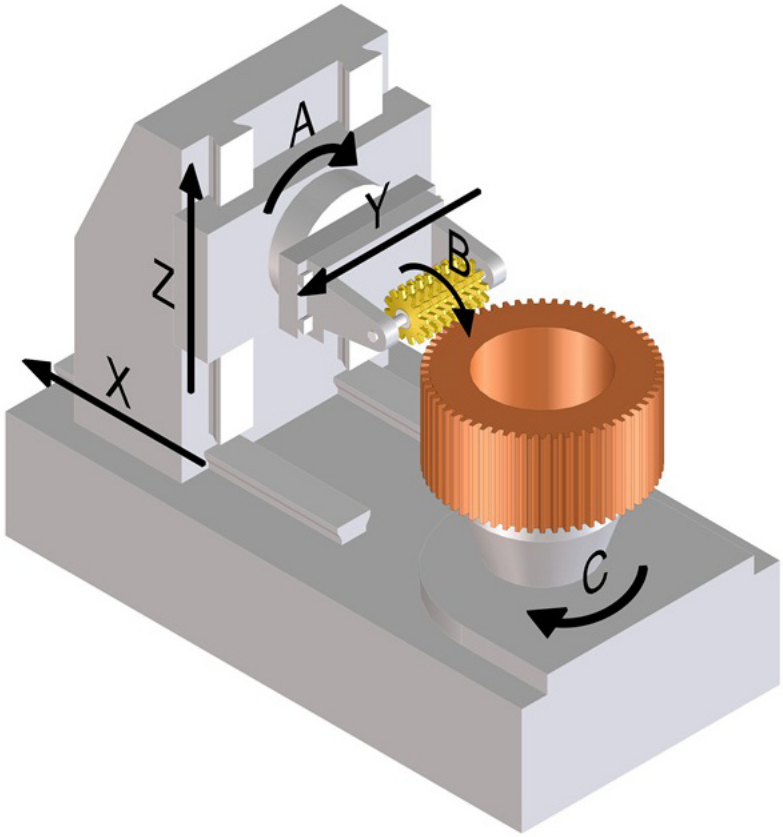
It also includes the option for the manufacturer to develop its own cycles, for better customer service.

The user must take into account the cutting and rotation conditions of the tool to achieve good machining precision.

For the cycles developed and explained in this manual, the specific screens and cycles for the gear hobbing machine must be available.

## 1.2 Configuration of the machine axes.

The following image shows the configuration of the gear hobbing machine axes.



1.

BASIC CONCEPTS.

Configuration of the machine axes.

FAGOR AUTOMATION

CNC 8065

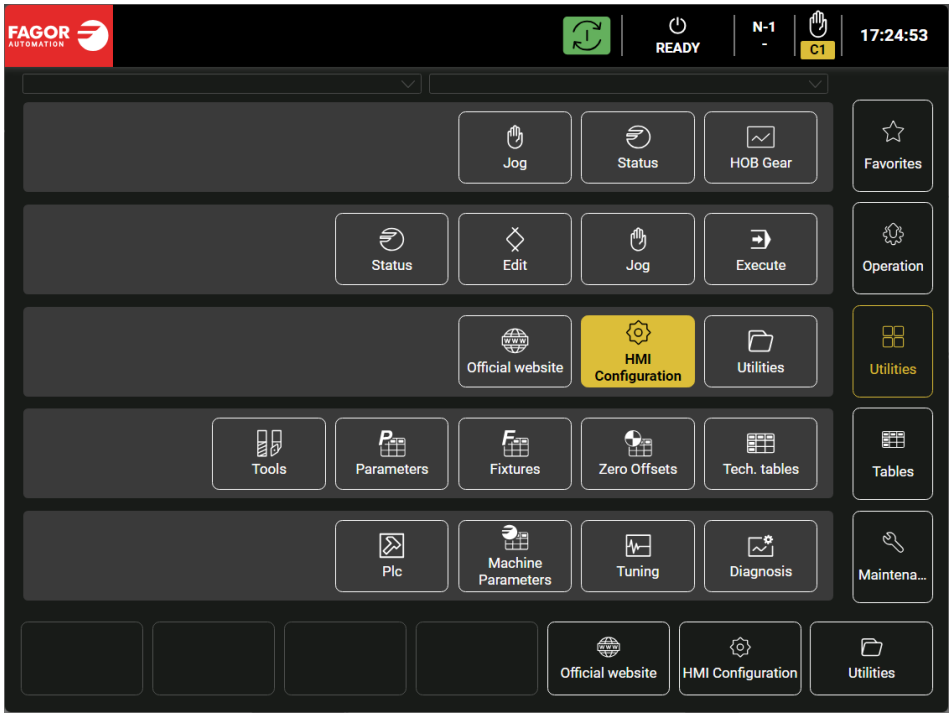
REF: 2602

# BASIC OPERATIONS.

# 2

## 2.1 Main screen.

The main screen can be accessed from any screen by clicking on the red box in the upper left corner. This screen provides access to any of the CNC work screens.



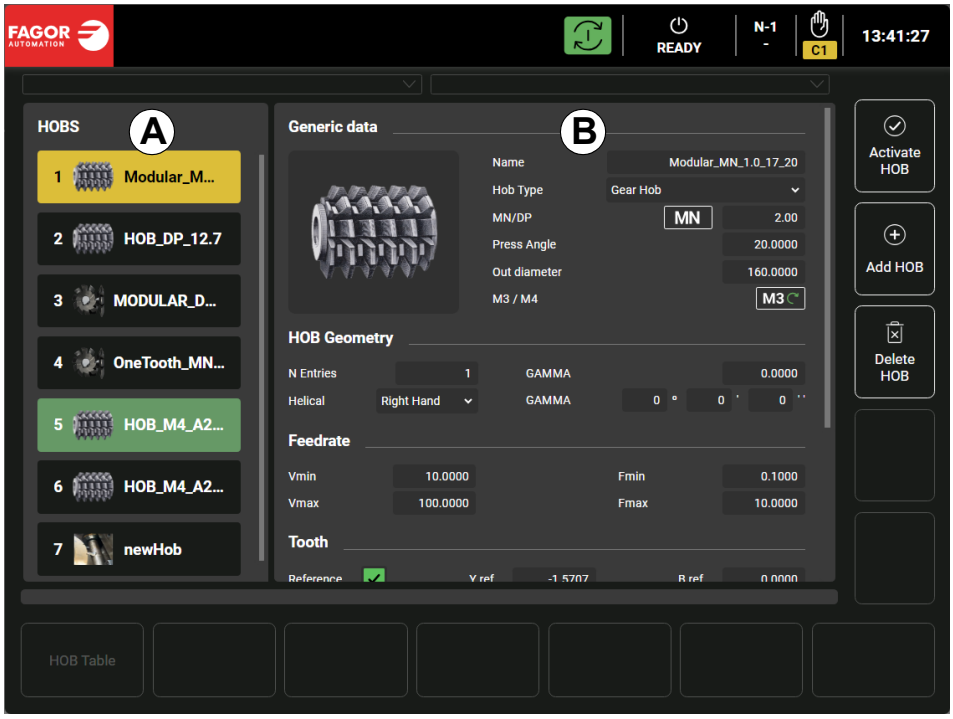

On the main screen, you'll find the screen "HOB Gear". This screen allows you to configure the gear hobbing machine tools.


As well as this screen, the CNC also has a number of cycles specific to the gear hobbing machine.

- Gear hobbing cycles. See chapter *"4 GEAR HOBBING CYCLES."*.

## 2.2 HOB Gear.

This screen displays the available tools together with all of their data and characteristics. This screen also allows you to add or delete tools, as well as activate the desired tool.




### *Description of the data on the "HOB Gear" screen.*

A Tool selection window.

- Allows you to select the desired tool.

  The active tool will be highlighted in green and the tool shown on the screen will be highlighted in yellow.

  Three types of tools are used for machining with gear hobbing machines:

  - Gear hob.
  - Modular (spur).
  - One tooth cutter.

B Data window for the selected tool. Shows the data of the selected tool (highlighted in yellow).

Depending on the type of tool selected, certain data will be displayed.

#### *Generic data.*

- **Name:** Name of the tool.
- **Hob Type:** Tool type (Gear hob, Modular (spur) or One tooth cutter).
- **MN/DP:** Selection of normal module or diametral pitch.

  NM = Normal module.

  DP = Diametral pitch.

  $$MN = \frac{25,4}{DP}$$

- **Pressure Angle:** Pressure angle of the gear.
- **Outside diameter:** Outer diameter of the tool.
- **M3 / M4:** Cutting direction of the tool.
- **Teeth:** Number of teeth on the modular (spur).

### *HOB geometry.*

- **No. inputs:** Number of tool inputs.
- **Helical:** Helical of the tool clockwise or counterclockwise.
- **GAMMA:** Inclination angle of helical pitch of the tool.
- **GAMMA:** Inclination angle of helical pitch of the tool in degrees, minutes and seconds.

### *Feedrate.*

- **Vmin:** Minimum cutting speed of the tool (m/min). With a value of zero there is no limit.
- **Vmax:** Maximum cutting speed of the tool (m/min). With a value of zero there is no limit.
- **Fmin:** Minimum gear feedrate (mm/rev). With a value of zero there is no limit.
- **Fmax:** Maximum gear feedrate (mm/rev). With a value of zero there is no limit.

### *Tooth.*

- **Tooth ref.:** Enable the tool reference tooth.
- **Ref. Y:** Reference position on the Y-axis of the tool tooth in the gear slot. The [RECALL] key enters the current position of the Y axis.
- **Ref. B:** Reference position on the B-axis of the tool tooth in the gear slot. The [RECALL] key enters the current position of the B axis.

### *Shifting.*

- **Initial pos. Y:** Initial position on Y-axis for movement. The [RECALL] key enters the current position of the Y axis.
- **Final pos. Y:** Final position on Y-axis for movement. The [RECALL] key enters the current position of the Y axis.
- **Y increment:** Movement increment on the Y-axis for each "Move parts".
- **Maximum counter:** Number of parts to apply movement increment on the Y-axis of the tool.
- **Current counter:** Current value of the parts counter.
- **Y current:** Current Y-axis position for the movement counter. The [RECALL] key enters the current position of the Y axis.
- **Avoid Zone 1:** Avoid zone 1 in movement on Y-axis.
- **Initial pos. Y:** Initial position on the Y-axis of zone 1 to be avoided. The [RECALL] key enters the current position of the Y axis.
- **Final pos. Y:** Final position on the Y-axis of zone 1 to be avoided. The [RECALL] key enters the current position of the Y axis.
- **Avoid Zone 2:** Avoid zone 2 in movement on Y-axis.
- **Initial pos. Y:** Initial position on the Y-axis of zone 2 to be avoided. The [RECALL] key enters the current position of the Y axis.
- **Final pos. Y:** Final position on the Y-axis of zone 2 to be avoided. The [RECALL] key enters the current position of the Y axis.

### 2.2.1 Activate HOB.

Follow these steps to activate the desired tool:

1 On the "HOB Gear" screen, select the desired tool in the tool selection window (A) and press the vertical softkey [Activate HOB] or [F12].

### 2.2.2 Add HOB.

Follow these steps to add a tool:

1 On the "HOB Gear" screen, press the vertical softkey [Add HOB] or [F11]. The new tool will be highlighted in yellow in the tool selection window (A).

2 Enter the name you want to give the tool along with the tool details.

### 2.2.3 Delete HOB.

To delete a tool, make sure that the tool you want to delete is not the active tool. If you want to delete the active tool, you must first deactivate it. To do this, select another tool to be the active tool. See *"2.2.1 Activate HOB." on page 10.*

Follow these steps to delete a tool:

1 On the "HOB Gear" screen, select the tool you want to delete in the tool selection window (A).

2 Press the [Delete HOB] softkey or [F10].

3 A message will appear indicating that the selected tool is going to be deleted and asking for confirmation to delete said tool.

4 Press "Yes" and the tool will be deleted.

# CYCLE EDITING.

3

All cycles can be simulated or executed individually as a single cycle by pressing the vertical [Simulate cycle] or [Execute cycle] softkeys on the screen for the desired cycle.

They can also be stored in a program by pressing the vertical [Memorize cycle] softkey and inserting the stored cycle in the desired line of the program being edited. Once this is done, the selected program can be simulated or executed.

## 3.1 Accessing gear hobbing cycles.

Follow these steps to access the gear hobbing cycles:

1. On the main screen, press the "Edit" button to access the program editing screen.
2. Once on the "Edit" screen, press the [Cycles] softkey or [F5] and the available cycle options will be displayed.
3. Select option [5. Hobbing cycles]. All available gear hobbing cycles are displayed in the horizontal softkey menu.
4. Press the softkey corresponding to the desired cycle.

## 3.2 Cycle editing.

Once the desired cycle has been accessed, edit the cycle by filling in the fields displayed on the screen and selecting the desired options using the available icons.


***Description of the cycle data:***

A Graphic representation of the cycle.

B Geometry data of the gear.

C Selection of the different machining phases and conditions.

D Data of the tool to be used for the machining of the cycle.

E Machine status information.

This machine status bar can be hidden by pressing the horizontal softkey [+] or [F7], and then selecting the option "TEACH.IN".

## 3.3 Simulation or execution of a gear hobbing cycle.

Once the desired cycle has been accessed, it can be executed or simulated directly from the cycle screen.

Follow these steps to execute or simulate a cycle:

1 Access the cycle you want to execute or simulate.

2 Complete the cycle data.

3 Simulate or execute the cycle.

- If you would like to simulate the cycle, press the [Simulate cycle] softkey or [F12]. The CNC will take you to the simulation screen. To simulate the cycle, press the [Start simulation] softkey or [F12].
- If you would like to execute the cycle, press the [Execute cycle] softkey or [F11]. Press [START] to execute the cycle. The CNC will take you to the execution screen and execute the cycle.

## 3.4 Inserting a gear hobbing cycle into a program.

The CNC allows the user to insert one or more cycles into a program, and then execute or simulate this program.

Follow the steps below to insert a cycle into a program:

1. On the main screen, press the "Edit" button to access the program editing screen.
2. From the "Edit" screen, press the [Select program] softkey or [F1].
3. Press option [1. Explorer] and select the desired program or create a new one.
4. Once inside the desired program, press the [Cycles] softkey or [F5] to display the available cycle options.
5. Select option [5. Hobbing cycles]. All available gear hobbing cycles are displayed in the horizontal softkey menu.
6. Press the softkey corresponding to the desired cycle.
7. Complete the cycle data.
8. After filling in the cycle data, press the [Memorize cycle] softkey or [F10]. The CNC will then display the "Edit" screen.
9. Select the line in which to insert the cycle and press the [F9] softkey. This softkey displays the name of the cycle just stored.

## 3.5 Execution or simulation of a program with gear hobbing cycles.

1. Once the program has been edited with the desired cycles, it will be possible to simulate or execute the program from this screen. The [Simulate program] and [Execute program] softkeys are available for this purpose.
2. Simulate or execute the program.
   - If you would like to simulate the program, press the [Simulate program] softkey or [F12]. The CNC will take you to the simulation screen. To simulate the program, press the [Start simulation] softkey or [F12].
   - If you would like to execute the program, press the [Execute program] softkey or [F11]. Press [START] to execute the program. The CNC will take you to the execution screen and execute the program.

# 3.

## CYCLE EDITING.

Execution or simulation of a program with gear hobbing cycles.

FAGOR AUTOMATION

CNC 8065

REF: 2602

# GEAR HOBBING CYCLES.

4

## 4.1 Axial.

To access this cycle, after entering the gear hobbing cycles, press the [Axial] softkey or [F1].


### *Meaning of the cycle variables.*

A Geometry of the gear.

Helical direction of the gear:

Straight, right or left.

Gear shape type:

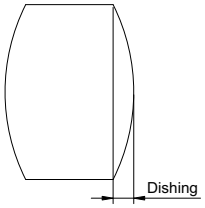
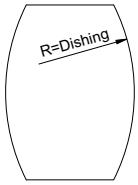
Cylindrical, conical with normal taper, conical with inverted taper or convex dishing.

HELICAL A. = Helical gear angle (not shown if gear geometry is straight).

CONICAL A. = Gear taper angle in degrees (only if the gear shape is conical).
The maximum angle is 4 degrees.

DISHING = Dishing Value (only when the gear shape has convex dishing).

To determine the Dishing value, take into account the following:

| Dishing < 10 mm or Dishing < 0.5 inches | Dishing > 10 mm or Dishing > 0.5 inches |
|---|---|
|   |   |

NM / DP = Selection of normal module or diametral pitch and their value.

Teeth Num. = Number of gear teeth.

DA2 = Outer diameter of gear. The [RECALL] key enters the current position of the X axis.

LA2 = Total gear height.

Z GEAR = Position of gear on Z-axis. The [RECALL] key enters the current position of the Z axis.

- UP (Distance measured from top of workpiece to the table).
- DOWN (Distance measured from bottom of workpiece to the table).

DX = Safety distance on X-axis (radius).

DZ = Safety distance on Z-axis.

Cor. Factor = Gear correction factor (1.0 for standard gears).

Shortening = Shortening of the tooth head (for a shorter head the value must be positive).

C. INIT = Initial position of the gear on the C axis. The [RECALL] key enters the current position of the C axis.

Centering = Activates manual gear centering before cutting.

During manual centering, only the rotation of the B axis is synchronized with the C axis. The tool will rotate on the B axis with the U data from the INPUT column.

For manual centering, with the tool rotating, the X, Y, Z and C axes can be moved manually (with a handwheel).

By touching the two walls of the gear slot with the tool in two steps, the cycle will calculate the resulting C.INIT position at the center position Z of the gear, and publish it in P105 and in "C:\FAGORCNC\USERS\PRG\CENTERING_MANUAL_LAST_RESULT.NC".

Z Home = Withdrawal position on the Z-axis. The [RECALL] key enters the current position of the Z axis.

N ++ = Number of additional gear turns at the end of the cycle.

SHIFT = Activates the movement of the tool.

*Consider the following formulas when entering data.*
*If the data entered is inconsistent with the corresponding formula, an error will be displayed when simulating or executing the cycle, making it impossible to simulate or execute said cycle.*

- *If working in mm:*

$$DA2 = \frac{MN \cdot TeethNum}{\cos(HelicalA)} + CorFactor \cdot MN \cdot 2 - Shortening \cdot 2$$

- *If working in inches:*

$$DA2 = \frac{\left(\frac{1}{DP}\right) \cdot TeethNum}{\cos(HelicalA)} + CorFactor \cdot \frac{1}{DP} \cdot 2 - Shortening \cdot 2$$

*If a value of 0 is entered in one of the parameters "Teeth Num" or "DA2", the CNC will calculate the correct value using the formula given above and a Warning will be displayed showing the result. To validate the data and continue, press [START].*

B Machining parameters.

Depending on the column, parameters LR, U and F refer to machining input, roughing pass or finishing pass.

Roughing and finishing passes can be deactivated, or the machining direction (bottom-top or top-bottom) can be selected.


Machining direction from top to bottom.


Machining direction from bottom to top.

Machining disabled, the specified pass is not carried out.

Meaning of the machining parameters.

| | INPUT | ROUGHING  | FINISHING  |
|---|---|---|---|
| **LR (mm)** | Approach distance | Depth of pass | Depth of pass |
| **U (m/min)** | Cutting speed | Cutting speed | Cutting speed |
| **F (mm/rev)** | Feedrate of axial input | Machining feedrate | Machining feedrate |

*If the roughing pass is disabled, the pass depth (LR) of the finishing will be the sum of the LR data for the roughing and finishing (LR ROUGHING + LR FINISHING).*

C Tool details.

This window allows you to select the tool you want to use for the machining and displays the data for said tool. If you want to modify any of the data of the selected tool, this needs to be done on the "HOB Gear" screen. See *"2.2 HOB Gear."* on page 8.

Hob = Name of the tool selected for the machining. The [RECALL] key opens the “HOB Gear” page.


Hob type.
(Gear hob, modular (spur) or one tooth cutter).

Value of the normal module or diametrical pitch of the selected tool (in DP or NM).

Inclination angle of helical pitch of the selected tool.

Helical of the tool clockwise or counterclockwise (D or L).

## 4.2 Radial.

To access this cycle, after entering the gear hobbing cycles, press the [Radial] softkey or [F2].


### *Meaning of the cycle variables.*

A Geometry of the gear.

Helical direction of the gear:

Straight, right or left.

HELICAL A. = Helical gear angle (not shown if gear geometry is straight).

NM / DP = Selection of normal module or diametral pitch and their value.

Teeth Num. = Number of gear teeth.

DA2 = Outer diameter of gear. The [RECALL] key enters the current position of the X axis.

LA2 = Total gear height.

Z GEAR = Position of gear on Z-axis. The [RECALL] key enters the current position of the Z axis.

- UP (Distance measured from top of workpiece to the table).
- DOWN (Distance measured from bottom of workpiece to the table).

DX = Safety distance on X-axis (radius).

Cor. Factor = Gear correction factor (1.0 for standard gears).

Shortening = Shortening of the tooth head (for a shorter head the value must be positive).

C. INIT = Initial position of the gear on the C axis. The [RECALL] key enters the current position of the C axis.

Centering = Activates manual gear centering before cutting.

During manual centering, only the rotation of the B axis is synchronized with the C axis. The tool will rotate on the B axis with the U data from the INPUT column.

To perform manual centering, with the tool rotating, the X, Y, Z, and C axes can be moved manually (with the handwheel). By touching the two walls of the gear slot with the tool in two steps, the cycle will calculate the resulting C.INIT position at the center position Z of the gear, and publish it in P105 and in "C:\FAGORCNC\USERS\PRG\CENTERING_MANUAL_LAST_RESULT.NC".

Z Home = Withdrawal position on the Z-axis. The [RECALL] key enters the current position of the Z axis.

N ++ = Number of additional gear turns at the end of the cycle.

SHIFT = Activates the movement of the tool.

*Consider the following formulas when entering data.*
*If the data entered is inconsistent with the corresponding formula, an error will be displayed when simulating or executing the cycle, making it impossible to simulate or execute said cycle.*

- *If working in mm:*

$$DA2 = \frac{MN \cdot TeethNum}{\cos(HelicalA)} + CorFactor \cdot MN \cdot 2 - Shortening \cdot 2$$

- *If working in inches:*

$$DA2 = \frac{\left(\frac{1}{DP}\right) \cdot TeethNum}{\cos(HelicalA)} + CorFactor \cdot \frac{1}{DP} \cdot 2 - Shortening \cdot 2$$

*If a value of 0 is entered in one of the parameters "Teeth Num" or "DA2", the CNC will calculate the correct value using the formula given above and a Warning will be displayed showing the result.*
*To validate the data and continue, press [START].*

B Machining parameters.

Depending on the column, parameters LR, U and F refer to machining input, roughing pass or finishing pass.

Roughing and finishing passes can be activated or deactivated.

Roughing or finishing pass activated.

Roughing or finishing pass deactivated.

Meaning of the machining parameters.

| | INPUT | ROUGHING | FINISHING |
|---|---|---|---|
| **LR (mm)** | Approach distance | Depth of pass | Depth of pass |
| **U (m/min)** | Cutting speed | Cutting speed | Cutting speed |
| **F (mm/rev)** | Feedrate of axial input | Machining feedrate | Machining feedrate |

*If the roughing pass is disabled, the pass depth (LR) of the finishing will be the sum of the LR data for the roughing and finishing (LR ROUGHING + LR FINISHING).*

C Tool details.

This window allows you to select the tool you want to use for the machining and displays the data for said tool. If you want to modify any of the data of the selected tool, this needs to be done on the "HOB Gear" screen. See *"2.2 HOB Gear."* on page 8.

Hob = Name of the tool selected for the machining. The [RECALL] key opens the “HOB Gear” page.



Hob type.
(Gear hob, modular (spur) or one tooth cutter).

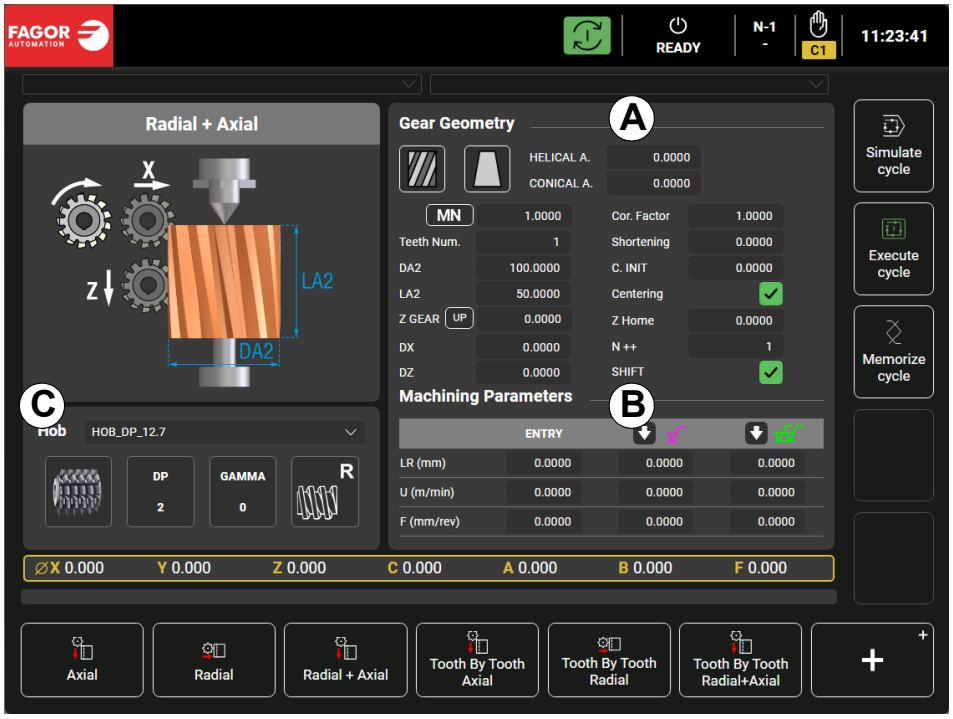

Value of the normal module or diametrical pitch of the selected tool (in DP or NM).


Inclination angle of helical pitch of the selected tool.


Helical of the tool clockwise or counterclockwise (D or L).

## 4.3 Radial + Axial.

To access this cycle, after entering the gear hobbing cycles, press the [Radial + Axial] softkey or [F3].

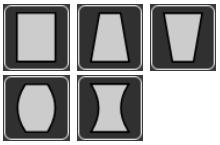



### *Meaning of the cycle variables.*

A Geometry of the gear.

Helical direction of the gear:

Straight, right or left.



Gear shape type:

Cylindrical, conical with normal taper, conical with inverted taper, with convex dishing or concave dishing.

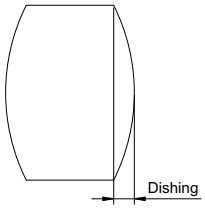
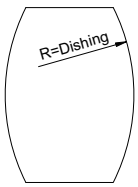
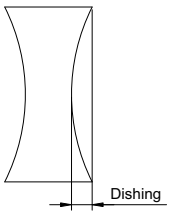
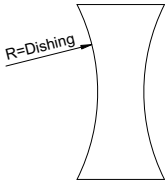
HELICAL A. = Helical gear angle (not shown if gear geometry is straight).

CONICAL A. = Gear taper angle in degrees (only if the gear shape is conical).

The maximum angle is 4 degrees.

DISHING = Dishing Value (only when the gear shape has convex or concave dishing).

To determine the Dishing value, take into account the following:

| Dishing < 10 mm or Dishing < 0.5 inches | Dishing > 10 mm or Dishing > 0.5 inches |
|---|---|
|   |   |
|   |   |

| | | |
|---|---|---|
| NM / DP | = | Selection of normal module or diametral pitch and their value. |
| Teeth Num. | = | Number of gear teeth. |
| DA2 | = | Outer diameter of gear. The [RECALL] key enters the current position of the X axis. |
| LA2 | = | Total gear height. |
| Z GEAR | = | Position of gear on Z-axis. The [RECALL] key enters the current position of the Z axis.<br>• UP (Distance measured from top of workpiece to the table).<br>• DOWN (Distance measured from bottom of workpiece to the table). |
| DX | = | Safety distance on X-axis (radius). |
| DZ | = | Safety distance on Z-axis. |
| Cor. Factor | = | Gear correction factor (1.0 for standard gears). |
| Shortening | = | Shortening of the tooth head (for a shorter head the value must be positive). |
| C. INIT | = | Initial position of the gear on the C axis. The [RECALL] key enters the current position of the C axis. |
| Centering | = | Activates manual gear centering before cutting.<br>During manual centering, only the rotation of the B axis is synchronized with the C axis. The tool will rotate on the B axis with the U data from the INPUT column.<br>For manual centering, with the tool rotating, the X, Y, Z and C axes can be moved manually (with a handwheel).<br>By touching the two walls of the gear slot with the tool in two steps, the cycle will calculate the resulting C.INIT position at the center position Z of the gear, and publish it in P105 and in "C:\FAGORCNC\USERS\PRG\CENTERING_MANUAL_LAST_RESULT.NC". |
| Z Home | = | Withdrawal position on the Z-axis. The [RECALL] key enters the current position of the Z axis. |
| N ++ | = | Number of additional gear turns at the end of the cycle. |
| SHIFT | = | Activates the movement of the tool. |

*Consider the following formulas when entering data.*
*If the data entered is inconsistent with the corresponding formula, an error will be displayed when simulating or executing the cycle, making it impossible to simulate or execute said cycle.*

- *If working in mm:*

$$DA2 = \frac{MN \cdot TeethNum}{\cos(HelicalA)} + CorFactor \cdot MN \cdot 2 - Shortening \cdot 2$$

- *If working in inches:*

$$DA2 = \frac{\left(\frac{1}{DP}\right) \cdot TeethNum}{\cos(HelicalA)} + CorFactor \cdot \frac{1}{DP} \cdot 2 - Shortening \cdot 2$$

*If a value of 0 is entered in one of the parameters "Teeth Num" or "DA2", the CNC will calculate the correct value using the formula given above and a Warning will be displayed showing the result.*
*To validate the data and continue, press [START].*

B Machining parameters.

Depending on the column, parameters LR, U and F refer to machining input, roughing pass or finishing pass.

Roughing and finishing passes can be deactivated, or the machining direction (bottom-top or top-bottom) can be selected.

 Machining direction from top to bottom.

 Machining direction from bottom to top.

 Machining disabled, the specified pass is not carried out.

Meaning of the machining parameters.

| | INPUT | ROUGHING | FINISHING |
|---|---|---|---|
| **LR (mm)** | Approach distance | Depth of pass | Depth of pass |
| **U (m/min)** | Cutting speed | Cutting speed | Cutting speed |
| **F (mm/rev)** | Feedrate of axial input | Machining feedrate | Machining feedrate |

*If the roughing pass is disabled, the pass depth (LR) of the finishing will be the sum of the LR data for the roughing and finishing (LR ROUGHING + LR FINISHING).*

C Tool details.

This window allows you to select the tool you want to use for the machining and displays the data for said tool. If you want to modify any of the data of the selected tool, this needs to be done on the "HOB Gear" screen. See *"2.2 HOB Gear."* on page 8.

Hob = Name of the tool selected for the machining. The [RECALL] key opens the “HOB Gear” page.

 Hob type.
(Gear hob, modular (spur) or one tooth cutter).

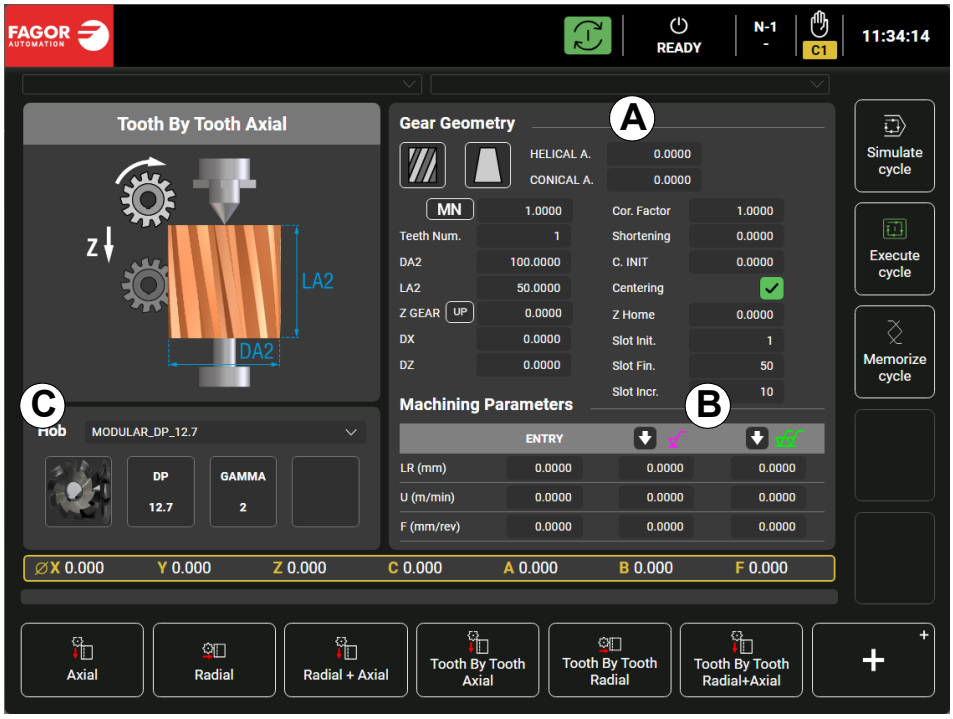
Value of the normal module or diametrical pitch of the selected tool (in DP or NM).

Inclination angle of helical pitch of the selected tool.

Helical of the tool clockwise or counterclockwise (D or L).

## 4.4 Tooth by tooth Axial.

To access this cycle, after entering the gear hobbing cycles, press the [Tooth by tooth Axial] softkey or [F4].




### *Meaning of the cycle variables.*

A Geometry of the gear.

Helical direction of the gear:

Straight, right or left.

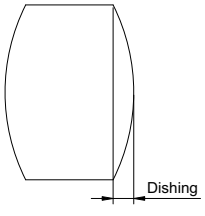
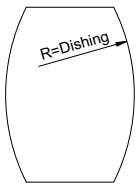
Gear shape type:

Cylindrical, conical with normal taper, conical with inverted taper, with convex dishing.

HELICAL A. = Helical gear angle (not shown if gear geometry is straight).

CONICAL A. = Gear taper angle in degrees (only if the gear shape is conical).
The maximum angle is 4 degrees.

DISHING = Dishing Value (only when the gear shape has convex dishing).
To determine the Dishing value, take into account the following:

| Dishing < 10 mm or Dishing < 0.5 inches | Dishing > 10 mm or Dishing > 0.5 inches |
|---|---|
|  |  |





NM / DP = Selection of normal module or diametral pitch and their value.

Teeth Num. = Number of gear teeth.

DA2 = Outer diameter of gear. The [RECALL] key enters the current position of the X axis.

LA2 = Total gear height.

Z GEAR = Position of gear on Z-axis. The [RECALL] key enters the current position of the Z axis.

- UP (Distance measured from top of workpiece to the table).
- DOWN (Distance measured from bottom of workpiece to the table).

DX = Safety distance on X-axis (radius).

DZ = Safety distance on Z-axis.

Cor. Factor = Gear correction factor (1.0 for standard gears).

Shortening = Shortening of the tooth head (for a shorter head the value must be positive).

C. INIT = Initial position of the gear on the C axis. The [RECALL] key enters the current position of the C axis.

Centering = Activates manual gear centering before cutting.

During manual centering, the B and C axes do not rotate synchronously. The tool will rotate on the B axis with the U data from the INPUT column.

For manual centering, with the tool rotating, the X, Y, Z and C axes can be moved manually (with a handwheel).

Z Home = Withdrawal position on the Z-axis. The [RECALL] key enters the current position of the Z axis.

Slot Init. = Initial tooth to machine from initial C.

- If “Slot Fin = 0” or “Slot Fin = Teeth Num”:
  “Slot Init” is the number of teeth to be machined, taking into account “Slot Incr” from “C. INIT”.
- If “Slot Fin” < “Teeth Num”:
  “Slot Init” is the position of the tooth relative to “C. INIT”.

Slot Fin. = Final tooth to machine from initial C. If “Slot Fin = 0” the value of “Teeth Num” will be used.

Slot Incr. = Tooth increment between machining operations.

*Consider the following formulas when entering data.*
*If the data entered is inconsistent with the corresponding formula, an error will be displayed when simulating or executing the cycle, making it impossible to simulate or execute said cycle.*

- *If working in mm:*

$$DA2 = \frac{MN \cdot TeethNum}{\cos(HelicalA)} + CorFactor \cdot MN \cdot 2 - Shortening \cdot 2$$

- *If working in inches:*

$$DA2 = \frac{\left(\frac{1}{DP}\right) \cdot TeethNum}{\cos(HelicalA)} + CorFactor \cdot \frac{1}{DP} \cdot 2 - Shortening \cdot 2$$

*If a value of 0 is entered in one of the parameters "Teeth Num" or "DA2", the CNC will calculate the correct value using the formula given above and a Warning will be displayed showing the result.*
*To validate the data and continue, press [START].*

B Machining parameters.

Depending on the column, parameters LR, U and F refer to machining input, roughing pass or finishing pass.

Roughing and finishing passes can be deactivated, or the machining direction (bottom-top or top-bottom) can be selected.

 Machining direction from top to bottom.

 Machining direction from bottom to top.

 Machining disabled, the specified pass is not carried out.

Meaning of the machining parameters.

| | INPUT | ROUGHING | FINISHING |
|---|---|---|---|
| **LR (mm)** | Approach distance | Depth of pass | Depth of pass |
| **U (m/min)** | Cutting speed | Cutting speed | Cutting speed |
| **F (mm/rev)** | Feedrate of axial input | Machining feedrate | Machining feedrate |

*If the roughing pass is disabled, the pass depth (LR) of the finishing will be the sum of the LR data for the roughing and finishing (LR ROUGHING + LR FINISHING).*

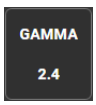
C Tool details.

This window allows you to select the tool you want to use for the machining and displays the data for said tool. If you want to modify any of the data of the selected tool, this needs to be done on the "HOB Gear" screen. See *"2.2 HOB Gear."* on page 8.

Hob = Name of the tool selected for the machining. The [RECALL] key opens the “HOB Gear” page.

 Hob type.
(Gear hob, modular (spur) or one tooth cutter).

Value of the normal module or diametrical pitch of the selected tool (in DP or NM).



Inclination angle of helical pitch of the selected tool.

Helical of the tool clockwise or counterclockwise (D or L).

## 4.5 Tooth by tooth Radial.

To access this cycle, after entering the gear hobbing cycles, press the [Tooth by tooth Radial] softkey or [F5].


### *Meaning of the cycle variables.*

A Geometry of the gear.

Helical direction of the gear:

Straight, right or left.

HELICAL A. = Helical gear angle (not shown if gear geometry is straight).

NM / DP = Selection of normal module or diametral pitch and their value.

Teeth Num. = Number of gear teeth.

DA2 = Outer diameter of gear. The [RECALL] key enters the current position of the X axis.

LA2 = Total gear height.

Z GEAR = Position of gear on Z-axis. The [RECALL] key enters the current position of the Z axis.

- UP (Distance measured from top of workpiece to the table).
- DOWN (Distance measured from bottom of workpiece to the table).

DX = Safety distance on X-axis (radius).

Cor. Factor = Gear correction factor (1.0 for standard gears).

Shortening = Shortening of the tooth head (for a shorter head the value must be positive).

C. INIT = Initial position of the gear on the C axis. The [RECALL] key enters the current position of the C axis.

Centering = Activates manual gear centering before cutting.

During manual centering, the B and C axes do not rotate synchronously. The tool will rotate on the B axis with the U data from the INPUT column.

For manual centering, with the tool rotating, the X, Y, Z and C axes can be moved manually (with a handwheel).

Z Home = Withdrawal position on the Z-axis. The [RECALL] key enters the current position of the Z axis.

Slot Init. = Initial tooth to machine from initial C.

- If “Slot Fin = 0” or “Slot Fin = Teeth Num”:
  “Slot Init” is the number of teeth to be machined, taking into account “Slot Incr” from “C. INIT”.
- If “Slot Fin” < “Teeth Num”:
  “Slot Init” is the position of the tooth relative to “C. INIT”.

Slot Fin. = Final tooth to machine from initial C. If “Slot Fin = 0” the value of “Teeth Num” will be used.

Slot Incr. = Tooth increment between machining operations.

*Consider the following formulas when entering data.*
*If the data entered is inconsistent with the corresponding formula, an error will be displayed when simulating or executing the cycle, making it impossible to simulate or execute said cycle.*

- *If working in mm:*

$$DA2 = \frac{MN \cdot TeethNum}{\cos(HelicalA)} + CorFactor \cdot MN \cdot 2 - Shortening \cdot 2$$

- *If working in inches:*

$$DA2 = \frac{\left(\frac{1}{DP}\right) \cdot TeethNum}{\cos(HelicalA)} + CorFactor \cdot \frac{1}{DP} \cdot 2 - Shortening \cdot 2$$

*If a value of 0 is entered in one of the parameters "Teeth Num" or "DA2", the CNC will calculate the correct value using the formula given above and a Warning will be displayed showing the result.*
*To validate the data and continue, press [START].*

B Machining parameters.

Depending on the column, parameters LR, U and F refer to machining input, roughing pass or finishing pass.

Roughing and finishing passes can be activated or deactivated.

Roughing or finishing pass activated from top to bottom.

Roughing or finishing pass deactivated.

Meaning of the machining parameters.

| | INPUT | ROUGHING | FINISHING |
|---|---|---|---|
| **LR (mm)** | Approach distance | Depth of pass | Depth of pass |
| **U (m/min)** | Cutting speed | Cutting speed | Cutting speed |
| **F (mm/rev)** | Feedrate of axial input | Machining feedrate | Machining feedrate |

*If the roughing pass is disabled, the pass depth (LR) of the finishing will be the sum of the LR data for the roughing and finishing (LR ROUGHING + LR FINISHING).*

C Tool details.

This window allows you to select the tool you want to use for the machining and displays the data for said tool. If you want to modify any of the data of the selected tool, this needs to be done on the "HOB Gear" screen. See *"2.2 HOB Gear."* on page 8.

Hob = Name of the tool selected for the machining. The [RECALL] key opens the “HOB Gear” page.

Hob type.
(Gear hob, modular (spur) or one tooth cutter).

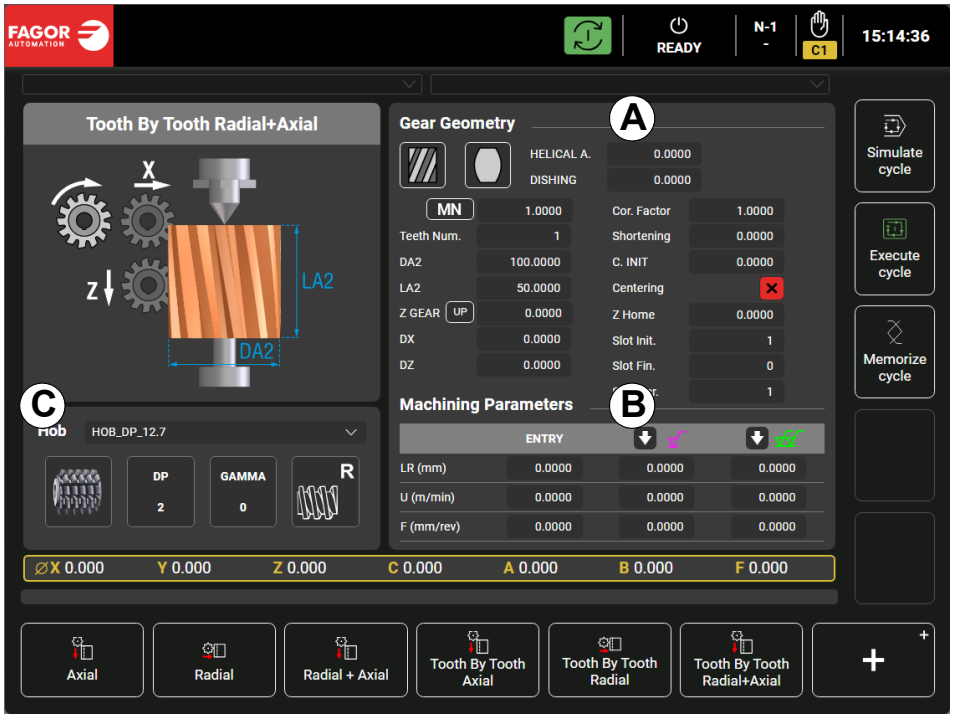

Value of the normal module or diametrical pitch of the selected tool (in DP or NM).


Inclination angle of helical pitch of the selected tool.


Helical of the tool clockwise or counterclockwise (D or L).

## 4.6 Tooth by tooth Radial + Axial.

To access this cycle, after entering the gear hobbing cycles, press the [Tooth by tooth Radial + Axial] softkey or [F6].

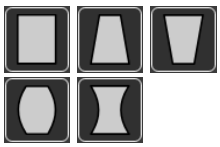



### *Meaning of the cycle variables.*

A Geometry of the gear.

Helical direction of the gear:

Straight, right or left.



Gear shape type:

Cylindrical, conical with normal taper, conical with inverted taper, with convex dishing or concave dishing.

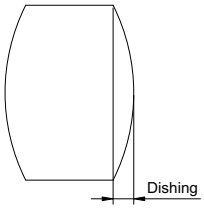
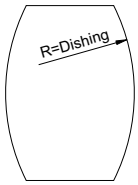
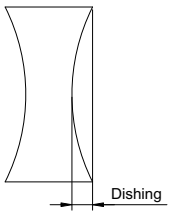
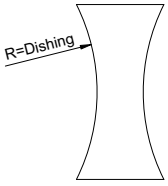
HELICAL A. = Helical gear angle (not shown if gear geometry is straight).

CONICAL A. = Gear taper angle in degrees (only if the gear shape is conical).

The maximum angle is 4 degrees.

DISHING = Dishing Value (only when the gear shape has convex or concave dishing).

To determine the Dishing value, take into account the following:

| Dishing < 10 mm or Dishing < 0.5 inches | Dishing > 10 mm or Dishing > 0.5 inches |
|---|---|
|   |   |
|   |   |

| | | |
|---|---|---|
| NM / DP | = | Selection of normal module or diametral pitch and their value. |
| Teeth Num. | = | Number of gear teeth. |
| DA2 | = | Outer diameter of gear. The [RECALL] key enters the current position of the X axis. |
| LA2 | = | Total gear height. |
| Z GEAR | = | Position of gear on Z-axis. The [RECALL] key enters the current position of the Z axis.<br>• UP (Distance measured from top of workpiece to the table).<br>• DOWN (Distance measured from bottom of workpiece to the table). |
| DX | = | Safety distance on X-axis (radius). |
| DZ | = | Safety distance on Z-axis. |
| Cor. Factor | = | Gear correction factor (1.0 for standard gears). |
| Shortening | = | Shortening of the tooth head (for a shorter head the value must be positive). |
| C. INIT | = | Initial position of the gear on the C axis. The [RECALL] key enters the current position of the C axis. |
| Centering | = | Activates manual gear centering before cutting.<br>During manual centering, the B and C axes do not rotate synchronously. The tool will rotate on the B axis with the U data from the INPUT column.<br>For manual centering, with the tool rotating, the X, Y, Z and C axes can be moved manually (with a handwheel). |
| Z Home | = | Withdrawal position on the Z-axis. The [RECALL] key enters the current position of the Z axis. |
| Slot Init. | = | Initial tooth to machine from initial C.<br>• If “Slot Fin = 0” or “Slot Fin = Teeth Num”:<br>“Slot Init” is the number of teeth to be machined, taking into account “Slot Incr” from “C. INIT”.<br>• If “Slot Fin” < “Teeth Num”:<br>“Slot Init” is the position of the tooth relative to “C. INIT”. |
| Slot Fin. | = | Final tooth to machine from initial C. If “Slot Fin = 0” the value of “Teeth Num” will be used. |
| Slot Incr. | = | Tooth increment between machining operations. |

*Consider the following formulas when entering data.*
*If the data entered is inconsistent with the corresponding formula, an error will be displayed when simulating or executing the cycle, making it impossible to simulate or execute said cycle.*

- *If working in mm:*

$$DA2 = \frac{MN \cdot TeethNum}{\cos(HelicalA)} + CorFactor \cdot MN \cdot 2 - Shortening \cdot 2$$

- *If working in inches:*

$$DA2 = \frac{\left(\frac{1}{DP}\right) \cdot TeethNum}{\cos(HelicalA)} + CorFactor \cdot \frac{1}{DP} \cdot 2 - Shortening \cdot 2$$

*If a value of 0 is entered in one of the parameters "Teeth Num" or "DA2", the CNC will calculate the correct value using the formula given above and a Warning will be displayed showing the result.*
*To validate the data and continue, press [START].*

B Machining parameters.

Depending on the column, parameters LR, U and F refer to machining input, roughing pass or finishing pass.

Roughing and finishing passes can be deactivated, or the machining direction (bottom-top or top-bottom) can be selected.


Machining direction from top to bottom.


Machining direction from bottom to top.

Machining disabled, the specified pass is not carried out.

Meaning of the machining parameters.

| | INPUT | ROUGHING | FINISHING |
|---|---|---|---|
| **LR (mm)** | Approach distance | Depth of pass | Depth of pass |
| **U (m/min)** | Cutting speed | Cutting speed | Cutting speed |
| **F (mm/rev)** | Feedrate of axial input | Machining feedrate | Machining feedrate |

*If the roughing pass is disabled, the pass depth (LR) of the finishing will be the sum of the LR data for the roughing and finishing (LR ROUGHING + LR FINISHING).*

C Tool details.

This window allows you to select the tool you want to use for the machining and displays the data for said tool. If you want to modify any of the data of the selected tool, this needs to be done on the "HOB Gear" screen. See *"2.2 HOB Gear."* on page 8.

Hob = Name of the tool selected for the machining. The [RECALL] key opens the “HOB Gear” page.


Hob type.
(Gear hob, modular (spur) or one tooth cutter).

Value of the normal module or diametrical pitch of the selected tool (in DP or NM).

Inclination angle of helical pitch of the selected tool.

Helical of the tool clockwise or counterclockwise (D or L).

# TOOL VARIABLES

5

## 5.1 List of variables from the technological tables.

The technological table variables for the gear hobbing machine are shown below.

### 5.1.1 Variable group "COMMON".

| Access variable from the part program | Type | Possible values | Description |
|---|---|---|---|
| V.TT.DESC | Numeric | Integer | Descriptor index for internal use by the application. |

### 5.1.2 Variable group "GEOMETRY".

Only column 1 of the technological table is used in these variables.

| Access variable from the part program | Type | Possible values | Description |
|---|---|---|---|
| V.TT.GEOMETRY_TYPE | Enum | 1 (GEAR HOB)<br>2 (MODULAR-SPUR)<br>3 (ONE_TOOTH) | Tool type, gear hob, modular (spur), or one tooth cutter. |
| V.TT.GEOMETRY_MNDPSEL | Enum | 1 (NM Normal module)<br>2 (DP Diametral Pitch) | Select the tool data in Normal Module (MN) or Diametral Pitch (DP). |
| V.TT.GEOMETRY_MNDP | Numeric | | Tool value, Normal Module (MN) or Diametral Pitch (DP). |
| V.TT.GEOMETRY_PRESSAN | Numeric | Degrees | Tool pressure angle. |
| V.TT.GEOMETRY_HELICAL | Enum | 1 (clockwise)<br>2 (counterclockwise) | Master mill with left-hand or right-hand helix. |
| V.TT.GEOMETRY_NENTRIES | Numeric | Integer | Number of starts of the master mill helix. |
| V.TT.GEOMETRY_GAMMA | Numeric | Degrees | Helix angle of the master mill. |
| V.TT.GEOMETRY_TEETH | Numeric | Integer | Modular (spur) mill type code. |
| V.TT.GEOMETRY_DAO | Numeric | mm - inch | Outer diameter of the mill. |
| V.TT.GEOMETRY_SHAN | Numeric | mm - inch | Initial Y-axis position of the master mill for shifting. |
| V.TT.GEOMETRY_SHED | Numeric | mm - inch | Final Y-axis position of the master mill for shifting. |
| V.TT.GEOMETRY_SHINC | Numeric | mm - inch | Y-axis position increment of the master mill for shifting. |
| V.TT.GEOMETRY_SHMAX | Numeric | Integer | Maximum number of workpieces to be machined to perform shifting. |
| V.TT.GEOMETRY_SHCOUNT | Numeric | Integer | Current number of workpieces machined to perform shifting. |
| V.TT.GEOMETRY_SHYACT | Numeric | mm - inch | Current Y-axis position during shifting. |
| V.TT.GEOMETRY_SHIDIR | Enum | 1 (Positive)<br>2 (Negative) | Direction of movement on the Y-axis during shifting. |

| Access variable from the part program | Type | Possible values | Description |
|---|---|---|---|
| V.TT.GEOMETRY_AVOID1ENA | Enum | 1 (No)<br>2 (Yes) | Enable exclusion zone 1 on the Y-axis for shifting. |
| V.TT.GEOMETRY_AVOID1SHAN | Numeric | mm - inch | Initial Y-axis position of exclusion zone 1 for shifting. |
| V.TT.GEOMETRY_AVOID1SHED | Numeric | mm - inch | Final Y-axis position of exclusion zone 1 for shifting. |
| V.TT.GEOMETRY_AVOID2ENA | Enum | 1 (No)<br>2 (Yes) | Enable exclusion zone 2 on the Y-axis for shifting. |
| V.TT.GEOMETRY_AVOID2SHAN | Numeric | mm - inch | Initial Y-axis position of exclusion zone 2 for shifting. |
| V.TT.GEOMETRY_AVOID2SHED | Numeric | mm - inch | Final Y-axis position of exclusion zone 2 for shifting. |
| V.TT.GEOMETRY_VMIN | Numeric | m-feet/min | Minimum allowed cutting speed for the mill. A value of 0 means no limit. |
| V.TT.GEOMETRY_VMAX | Numeric | m-feet/min | Minimum allowed cutting speed for the mill. A value of 0 means no limit. |
| V.TT.GEOMETRY_FMIN | Numeric | mm-inch/rev | Minimum allowed feed rate of the mill per gear revolution. A value of 0 means no limit. |
| V.TT.GEOMETRY_FMAX | Numeric | mm-inch/rev | Maximum allowed feed rate of the mill per gear revolution. A value of 0 means no limit. |
| V.TT.GEOMETRY_M34 | Enum | 1 (M3)<br>2 (M4) | Turning direction of the mill. |
| V.TT.GEOMETRY_TOOTHREF | Enum | 1 (No)<br>2 (Yes) | Enable the use of the mill tooth reference. |
| V.TT.GEOMETRY_YREF | Numeric | mm - inch | Reference position on the Y-axis of the tool tooth in the gear slot. |
| V.TT.GEOMETRY_BREF | Numeric | Degrees | Reference position on the B-axis of the tool tooth in the gear slot. |

## User notes:

## User notes:

FAGOR AUTOMATION

CNC 8065

REF: 2602

## User notes:

FAGOR
AUTOMATION

**Fagor Automation S. Coop.**

Bº San Andrés, 19 - Apdo. 144
E-20500 Arrasate-Mondragón, Spain
Tel: +34 943 039 800
Fax: +34 943 791 712
E-mail: contact@fagorautomation.es
www.fagorautomation.com

AENOR
GESTIÓN DE LA CALIDAD
ISO 9001

IQNET
RECOGNIZED CERTIFICATION