Cycle time optimisation
ISO machines

Edition No. 1
Version 1.6/MC 0609

02.06.2017

Contents

1 General information

1.1 Document contents

This document is the first edition of the Tornos "Tips & Tricks".
We are hoping this document will put our experience in lathe machine operation to good use for your benefit.
This first edition addresses a vital topic in the world of lathe machine operation, namely: cycle time.

1.2 Cycle time

What exactly is cycle time?
The cycle time is the time the machine takes to produce a part.
The cycle time therefore plays a fundamental role in the world of lathe machine operation.
Each part machine means money for the company producing it.
The faster a machine is when machining its part, the more parts will be produced for a given time period, and thus the more money the company will make.

1.3 Every second counts

Let's look at a concrete example:
Imagine:

- a series of parts that requires mass production over one year.
- a fleet of 10 machines intended for machining this part.
- full-time production 24/7.
- a cycle time to produce the part of 65 seconds.
- a price per part of 1.-

The maximum capacity of your workshop is **4,851,692 parts/annum** i.e. a turnover of **4,851,692.-/annum**.

Now imagine you could cut your cycle time by a just 2 tiny seconds, the maximum capacity of your workshop would rise to **5,005,714 parts/annum** in other words a turnover of **5,005,714.-/annum**.

Those two precious 2 tiny seconds of time saved would earn you **154,022.-**.

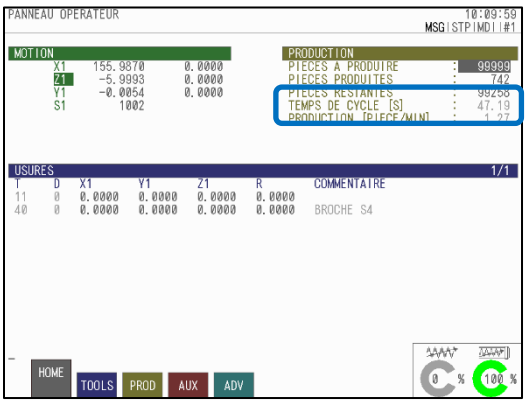
1.4 Part production process

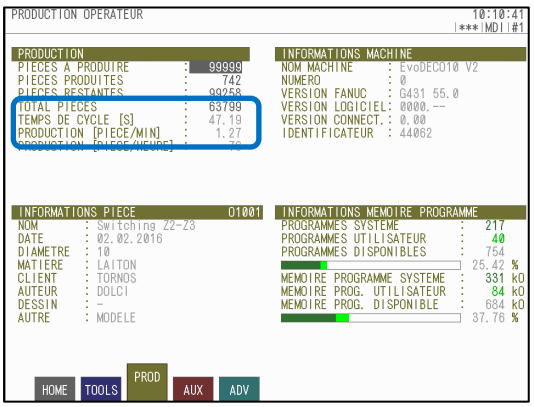
It is important to know that each step in the production of a part is important to achieve an optimal cycle time. Here are the different steps:

1) Defining the operation plan
2) Defining the list of tools
3) Programming the part
4) Performing set-up
5) Fine-tuning the program on the machine (output of correct part)
6) Optimising the cycle time by adapting the program

1.5 Where is the cycle time displayed?

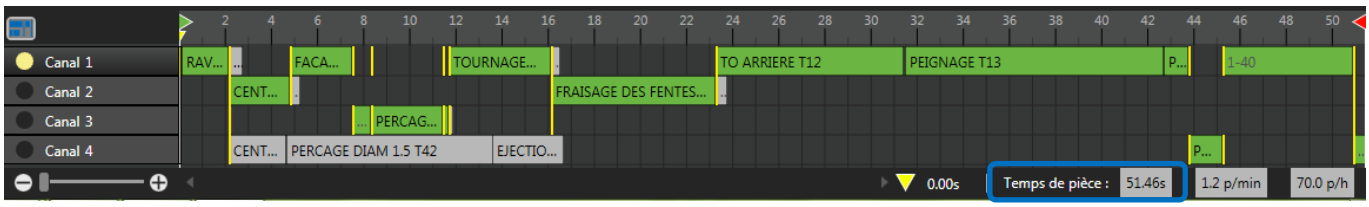
On the latest-generation Tornos ISO machines, you can display the cycle time in the T-MI interface (page of the CNC).
Simply go to the "HOME" or "PROD" page of T-MI.





Remember never to take into consideration the first cycle time, you should always wait for the second round of the program to have a representative time. You also need to know that it is a real timing process so there are some slight fluctuations from one cycle to the next.

Please also note that the TISIS programming software allows you to have an estimated cycle time.



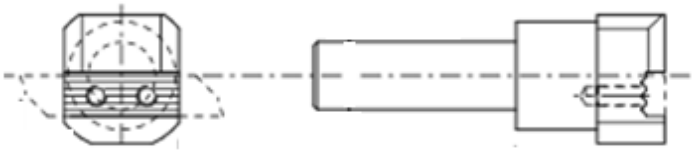
2 Defining the work process

2.1 Defining the operation plan

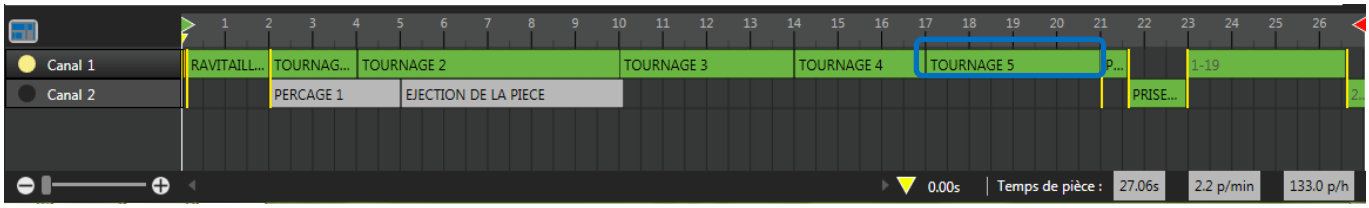
For a cycle time to be optimal, you obviously need to perform as many operations as possible in parallel. You therefore need to organise your operation plan in order to use all the machine channels in the best possible way.

For example, on a simple machine with 2 channels, you can ask yourself whether it might not be advisable to perform turning operations in backwork in order better to balance the machining time in the 2 channels.

It may be interesting to know that certain tool manufacturers offer solutions to have a plate holder to perform turning on end positions. This provides the advantage of being able to perform several turning operations in backwork.



Example:
In the example below, we have moved the "Turning 5" operation in backwork, and thus gained some precious seconds in the cycle time.



2.2 Choosing the tools

To have an optimal cycle time, it is important to have the shortest possible machining times (time during which all the tools spend on the material).

To do this, you need the proper tools according to the part to machine.

When choosing the tools, you should take into account:

- The tool characteristics
- The covering of the tool
- The rigidity of the support
- The number of teeth (for a milling cutter)
- The incorporated coolant spraying (in the centre)

Investing in shaping tools may also prove particularly helpful. This will allow you to machine several elements of a part in a single operation.

Having the right tools is an important investment. Having said this, if your tools allow you to feed further into the material, or to perform more significant drawings strokes, the cycle times will decrease and consequently also the part return costs.

2.3 Choice of machining processes

It is always interesting to ask yourself the question of whether the chosen machining process is the best in terms of cycle times.

Screw pitch:
To perform a screw pitch, is it advisable to do so by maingang work (several strokes)?
Have you thought of thread whirling or rolling, which is often much faster (a single stroke)?

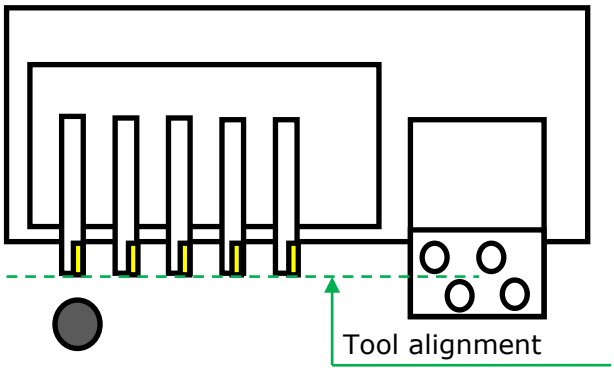
Machining flat parts:
To perform several flat parts on your part, have you considered polygoning, which is much faster than cross milling.

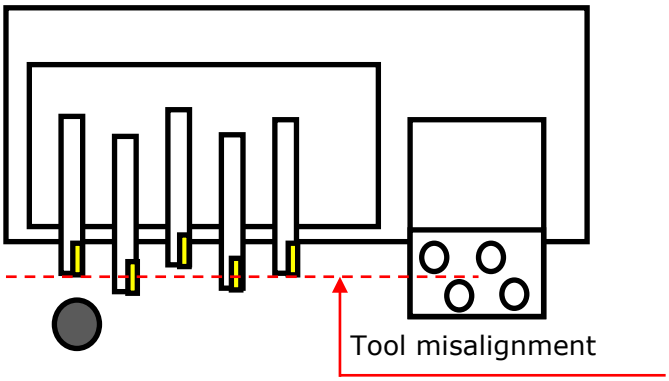
2.4 Fitting the tools

Fitting tools is significantly important in the cycle time.
You should always consider the following:

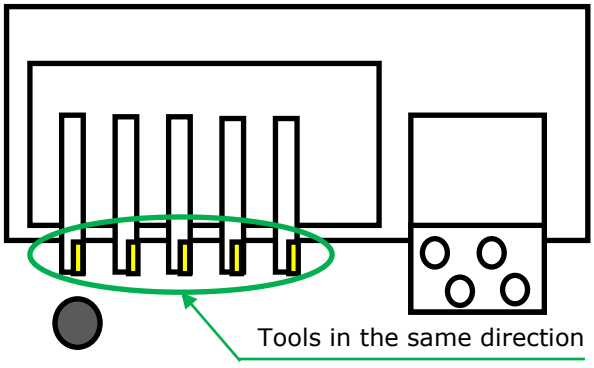
- Tool geometries
- Direction of tools
- Order of tools (according to process)
- Closeness of tools

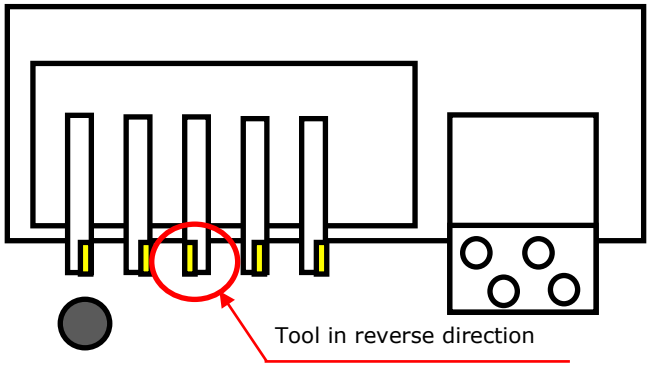
Tool geometries:
It is important to try to have the same geometries (X and Z) on all the tools of any one system. This will allow you to minimise axes movements during tool indexing.





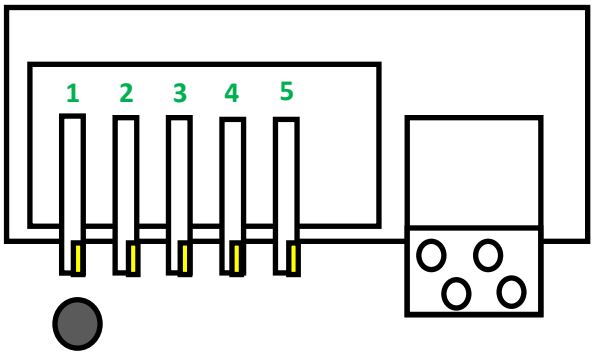
Direction of tools:
It is important to have the tools (plate holder) in the same direction. This avoids material spindle rotation direction reversals, which takes up cycle time.

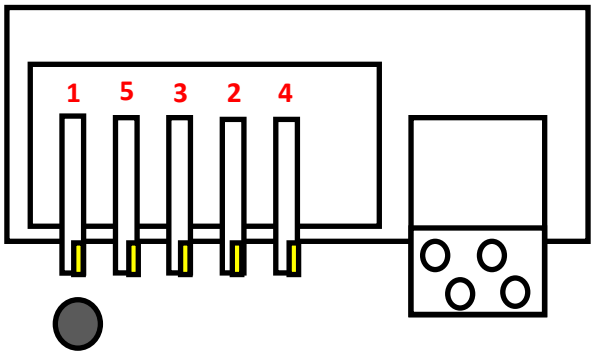




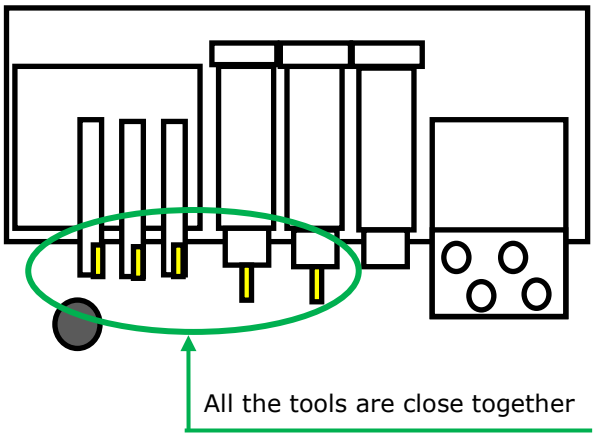
It is interesting to note that during part pickoff, in principle the counter spindle turns in a counter clockwise direction [M404], which is due to the direction of the tools. And very often we use the counter spindle to work on bits in backwork. So in principle the counter spindle should reverse the rotation direction [M403], which may take up cycle time. It may therefore be interesting to use the bits which cut on the left in order to avoid rotation direction reversals.

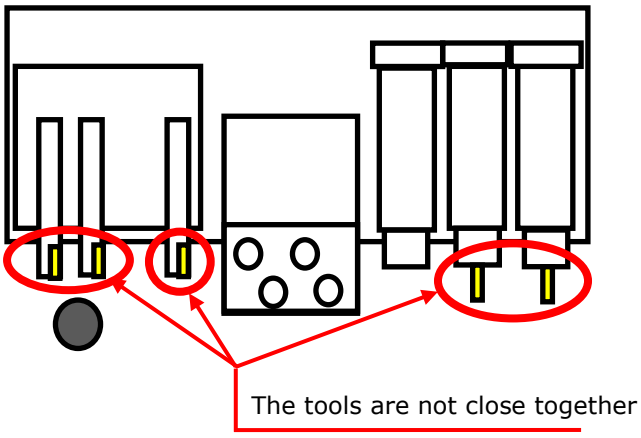
Order of tools:
It is important for the tools to be in the order corresponding to the machining process. This means that the first tool used should be found next to the second, the second tool used next to the third, and so on. This avoids the comings and goings of the tool system during indexing.





Closeness of tools:
It is important to try to put the tools to be used as close together as possible.
Again in order to minimise axes movements during tool indexing.





Also note that certain tool manufacturers offer supports that allow maximum closeness of tools, which not only increases the number of tools you can use on the machine, but also has the added benefit of reducing tool indexing times.

2.5 Recovery of the part on the guidebush

If the part allows it, Tornos offers solutions to recover the part directly from the guidebush. This avoids part pickoff in the counter spindle and obviously saves you time.

2.6 Working without guidebush

Multiple Tornos machines allow working without a guidebush. One of the advantages of working without a guidebush is you can decrease the length of the remnant. By decreasing the length of the remnant, you can obviously save on material, and what's more, you also decrease the number of new bars fed. So we save time.
This may be interesting on long series.

Tornos advises against performing parts of a length above 3 times their diameter in without guidebush mode.

2.7 Feeding several parts per clamping

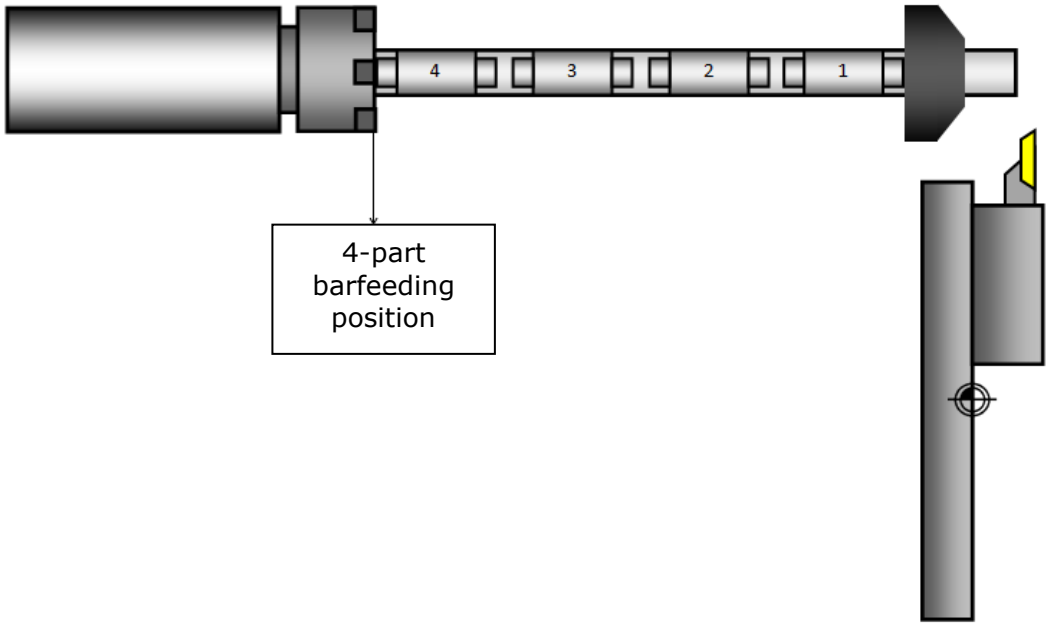
The machine runs one part per clamping by default.

If the main spindle stroke allows it, it may be interesting to feed several parts per clamping to reduce the average machining cycle time of a part.

This phenomenon occurs because the spindle collet opening and closing times, their respective delays and the end of Z movement delay during barfeeding are only taken into consideration once for the number of parts fed.

N.B.: The more parts are machined per clamping, the more the value introduced in the cutter width (G801 B_) has to be precise.

Indicative values	
Number of parts per clamping	Cycle time saver per part [sec]
1	0
2	0.75
3	1
4	1.125
5	1.2
6	1.25
7	1.285
8	1.312
9	1.333
10	1.35



2.8 Bars

The bars used can also be important in terms of cycle time.

Their straightness is important and if it is ensured, the longer the bar is, the less often you will need to feed a new bar, so you will save in productivity.

Profiled bars can also be a good solution to save on cycle time. For example, with a 6-sided bar, this may avoid having to perform time-consuming machining. It is relatively easy these days to find profiled bars as well as shaped collets and guidebushes.

The machining of tubular bars may also be interesting, since it avoids drilling and part cutting is reduced since you do not need to cut all the way to the centre.

2.9 HP pump

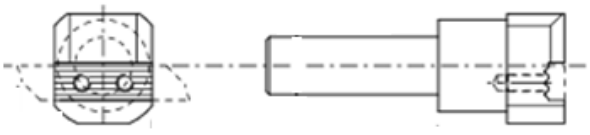
Tornos offers multiple solutions in terms of high pressure (HP) pump. These HP pumps are interesting in terms of cycle times for two reasons:

- They allow better evacuation of chips, and hence of heat. This often allows a slight increase in the machining feeds.
- They allow better evacuation of chips, which may avoid machine stops to remove chips manually.

2.10 Performing the part backwards

Have you thought about performing the part in the reverse direction. In other words, to perform the mainwork-machined part in backwork and vice-versa. It is often interesting to ask yourselves this question. Sometimes, you can save time.

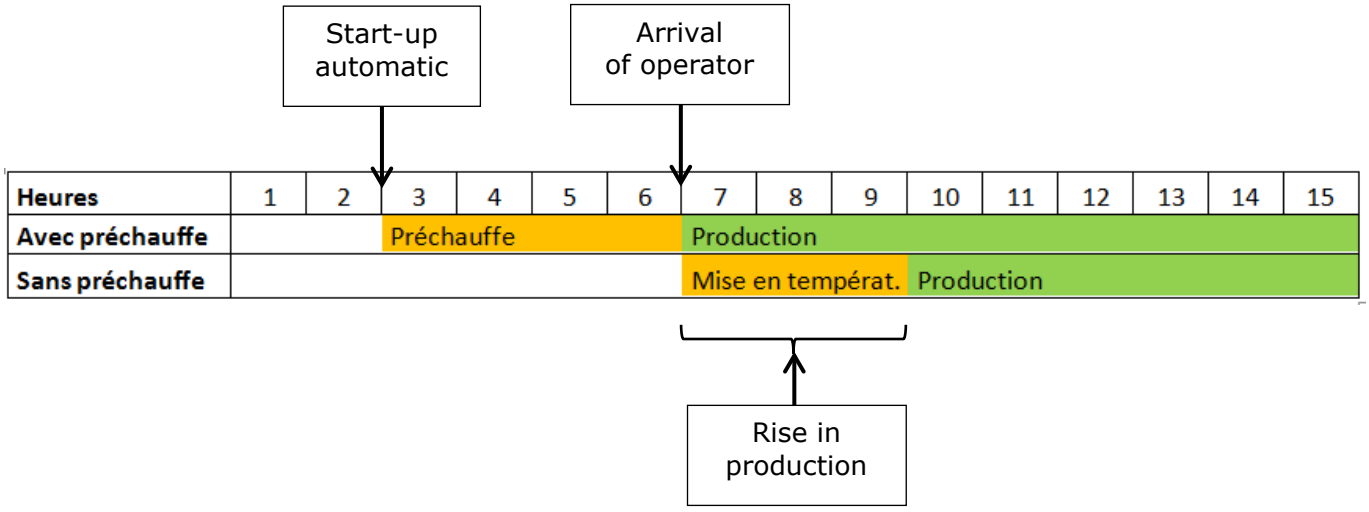
It may be interesting to know that certain tool manufacturers offer solutions to have a plate holder too perform turning on end positions. This provides the advantage of being able to perform several turning operations in backwork.



2.11 Machine preheating option

Tornos offers an optional machine preheating function for very accurate parts. The machine may therefore start automatically, in no material mode, at a time and date defined in advance. The advantage of this function is that you save time waiting for the machine to reach its temperature.

Example:

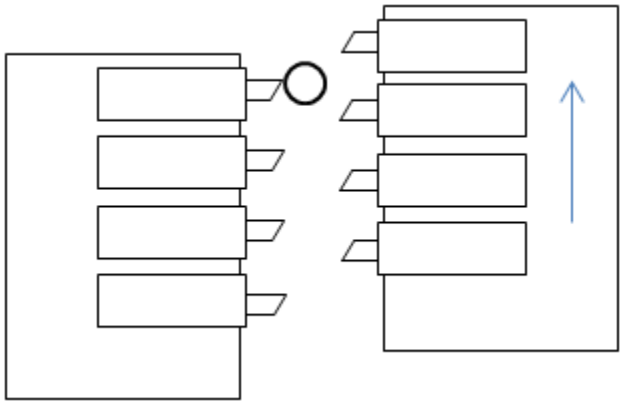


3 Tool indexing

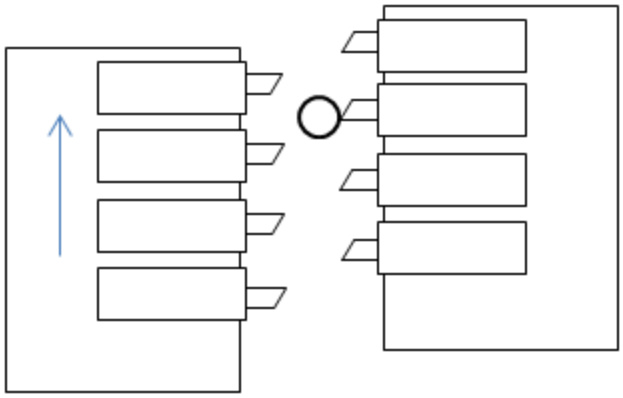
3.1 Calling tools in hidden time

On machines with independent tool systems (EvoDECO, SwissNano) to work on the bar, think about organising your tools smartly, in order to index the tools while the other system is machining, and vice-versa.

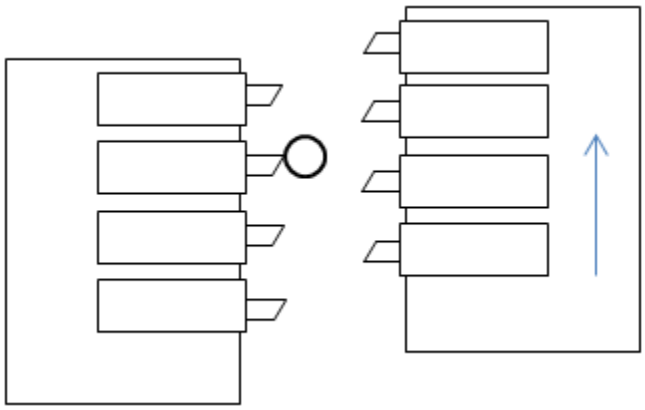
Example:



Machining with a tool in maingang work 2
In parallel preparation of the next tool in mainwork gang 1



Machining with a tool in maingang work 1
In parallel preparation of the next tool in mainwork gang 2



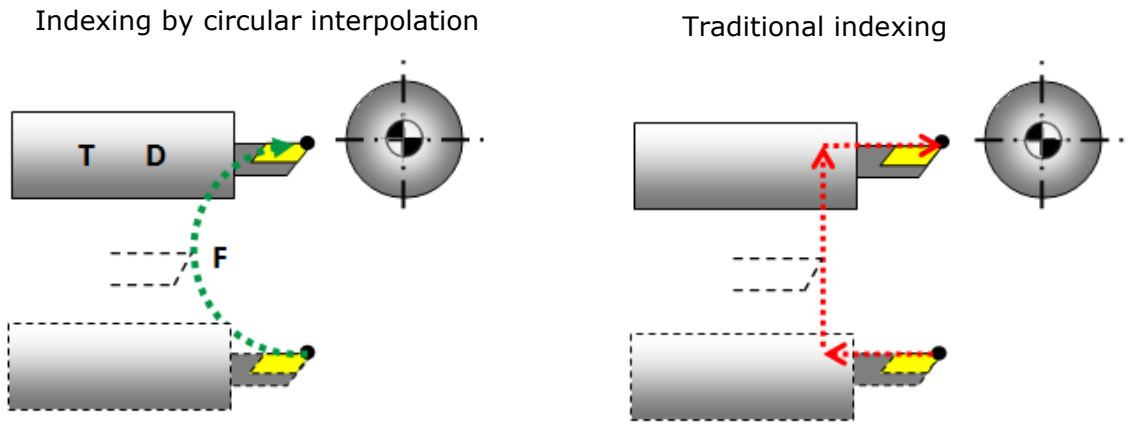
Machining with a tool in maingang work 2
In parallel preparation of the next tool in mainwork gang 1

N.B. : It is also interesting to rotate the turning tools from the other channel in hidden time.

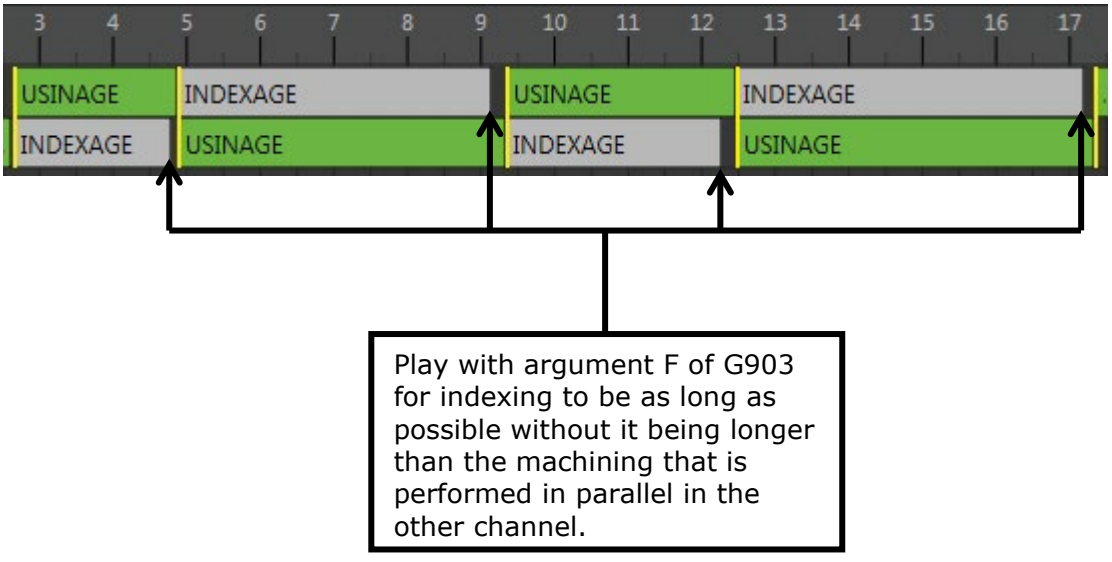
You can index a tool in circular interpolation and set the indexing speed parameter, in order to allow the tool to reach its position at the exact moment the other system has finished machining. This allows you to avoid sudden movements in the machine due to the tool indexing (while the other system works in the material).

Example:

G903 T_ D_ F_
G903: Tool call in circular interpolation
T_ D_: Number of tool and of its desired corrector
F_: Feed during indexing [mm/min]



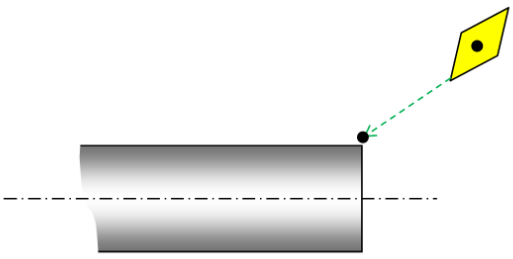
Please note that the Gantt diagram of the TISIS software allows you to determine very easily the indexing feed in order to reach the position at the right moment.



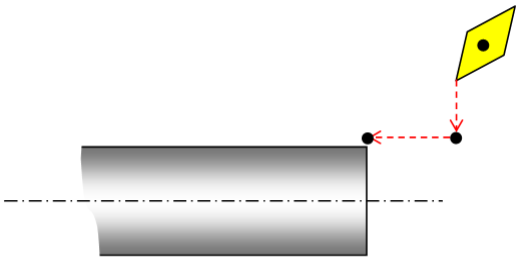
3.2 Tool approach

Wherever possible, try to perform tool approaches in fast feed [G0] on several simultaneous axes.

Example:



Saves time

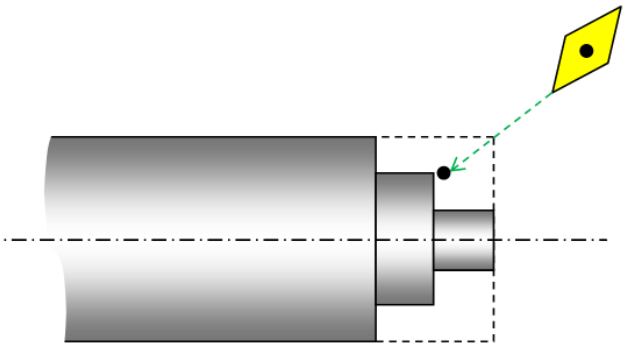


Wastes time

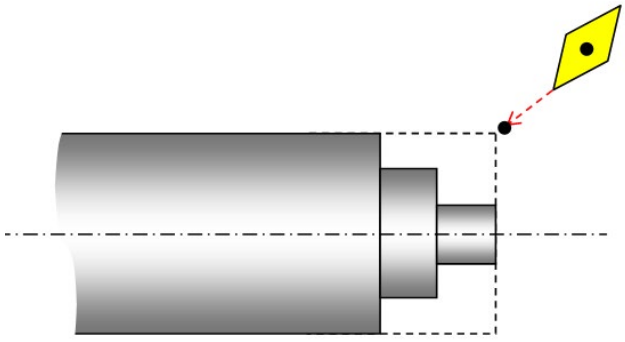
Also do not forget that you can perform a tool approach on linear and rotary axes at the same time (for example Y Z + C).

During a tool approach remember that if the part is already partly machined you can perform these tool approaches closer than the initial unfinished part.

Example:



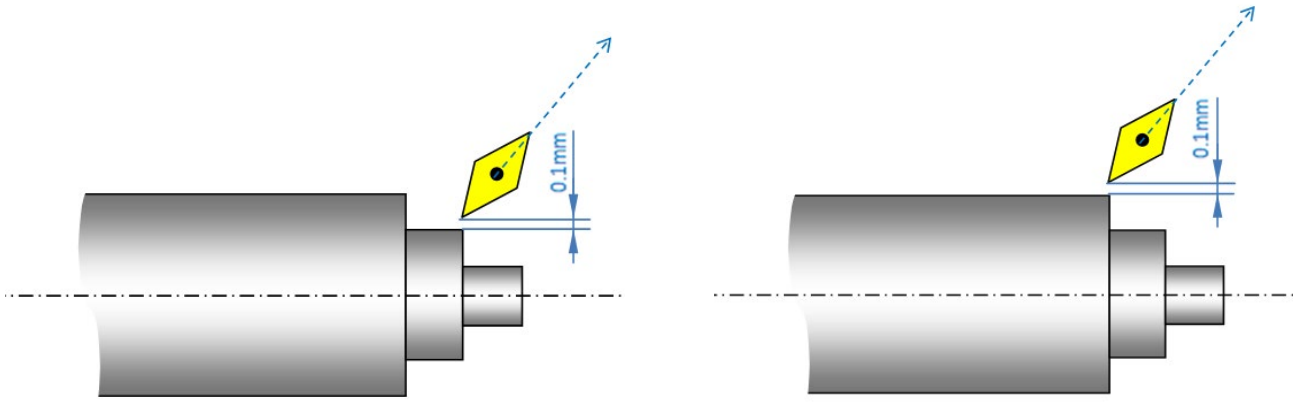
Saves time



Wastes time

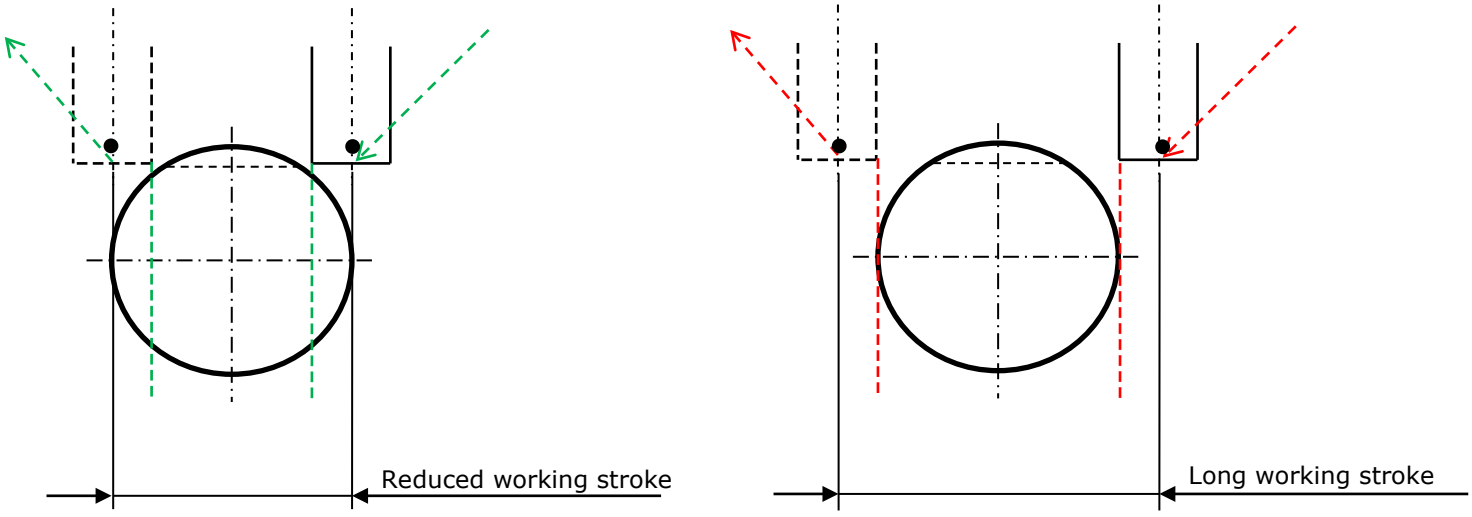
3.3 Tool retraction

When you retract a tool from the material [G1], if the tools are preset precisely, a safety clearance of 0.1 mm is amply sufficient before retraction in fast feed [G0].



3.4 End milling cutter approach/retraction

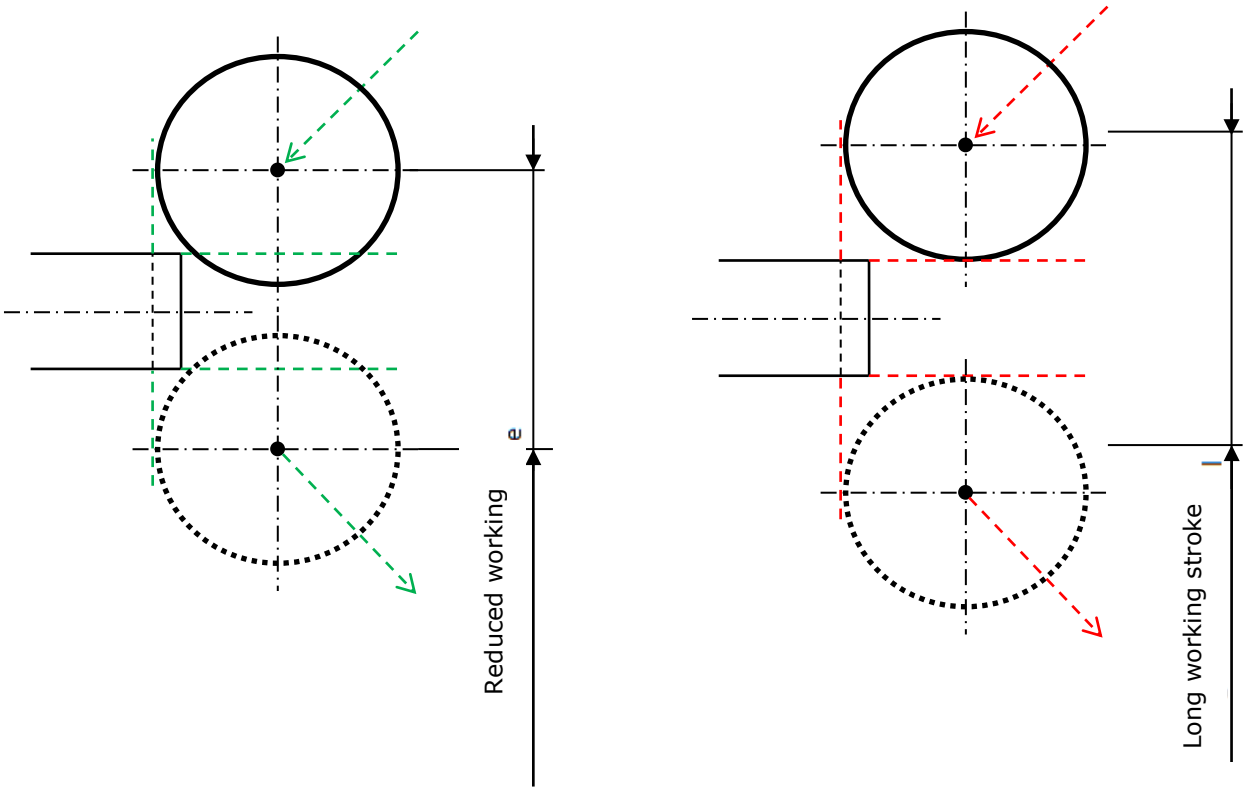
When you perform cross milling, you need to take into account the fact that you can approach in fast feed closer than the material diameter + safety clearance. This also applies for the retraction. The more the working stroke [G1] is reduced, the more cycle time you will save.



3.5 End splitting attachment approach/retraction

When you perform splitting, always remember to optimise your splitting attachment approaches and retractions while taking into account the radius of the milling cutter.
There are two ways to do this:

- Programming using traverses with path correction [G41/G42]
- Optimising approaches by programming approaches in machine positions

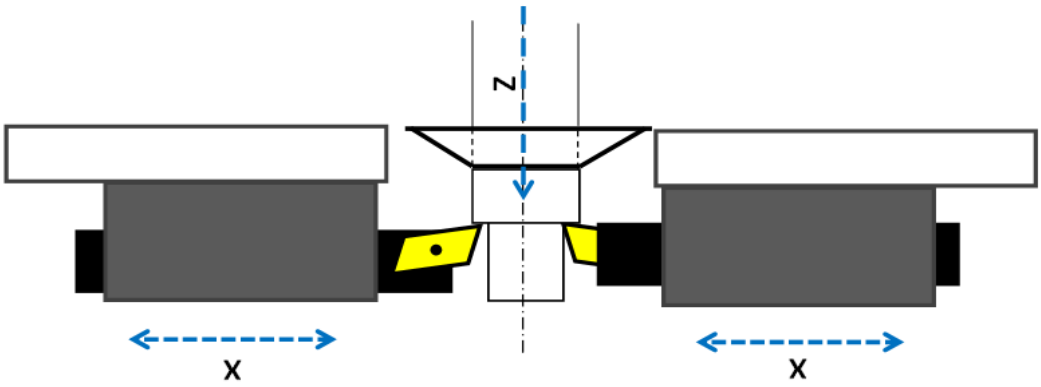


4 Simultaneous machining

4.1 Blank-Simultaneous finishing

On machines fitted with two independent tool systems to work on the material spindle, you have the possibility of performing a blank-stroke finishing simultaneously.
To do this, you need a finishing tool on the first tool system and a blank tool on the second. You then simply need to slightly retract the blank tool in Z (generally 0.1mm) with respect to the finishing tool. For programming there is a synchronisation function of the X axes [M142/M144].

Example:



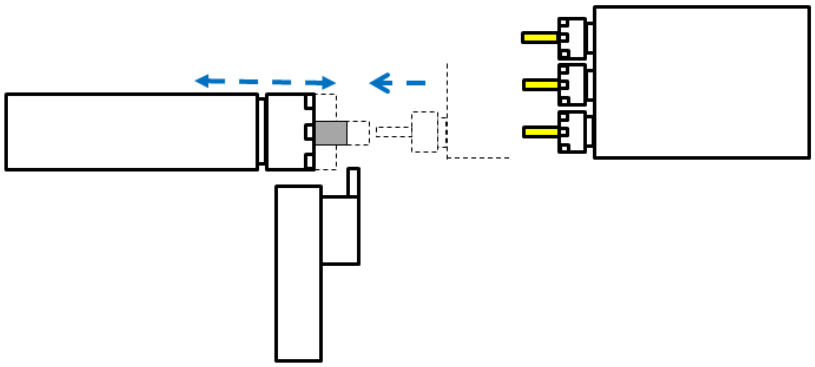
Sample Programming	
Channel 1	**Channel 2**
G54 G0 X5.3 Y0 Z1 T11 D0 M103 S4000 P1	G95 G0 X5 Y0 T21 D0
M9005	**M9005**
M142 G1 Z-12 F0.05 G1 X5.4 G0 X10 Z2 M143	
M9006	**M9006**

4.2 Superimpose

Several Tornos kinematics are useful for the superimpose function [M152]. The superimpose function makes it possible to perform two operations simultaneously, and thus save time. To do so, one of the two tools needs to be on a Z axis independent from the Z axis of the material spindle.

Example:

You can observe traditional turning with the mainwork gang on the spindle and simultaneously a drilling with the frontwork unit. The Z axis of the frontwork unit automatically compensates for the movement of the material spindle instantaneously.

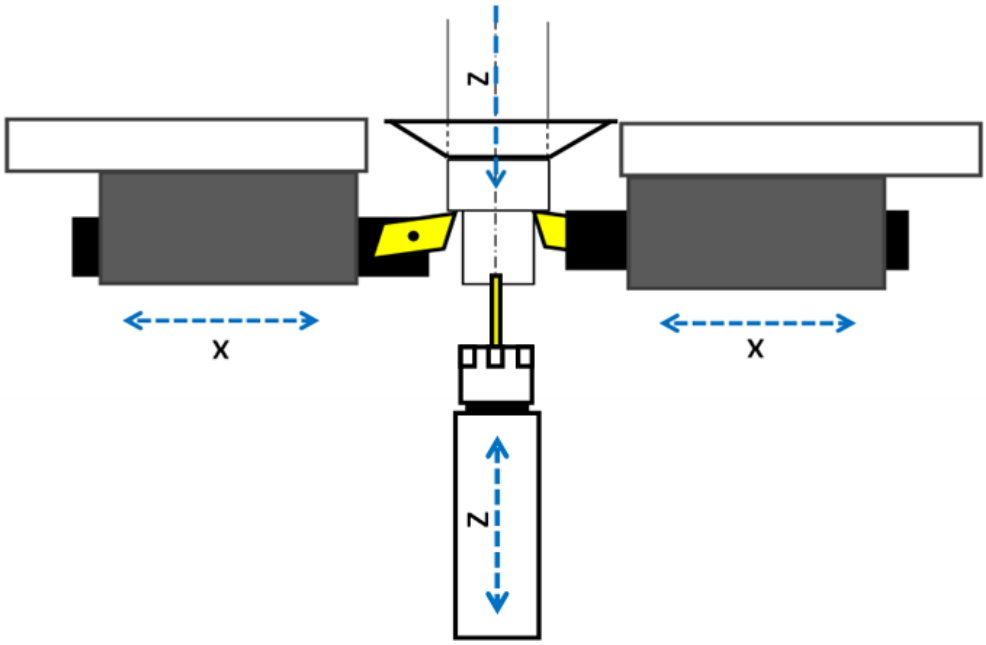


Sample Programming	
Channel 1	**Channel 2**
G54 G95 G0 X3 Y0 Z1 T1 D0 M103 S4000 P1	G95
M9005	**M9005**
	G915 G0 X0 Y0 Z1 T31 D0
M9006	**M9006**
	M152
M9007	**M9007**
G1 Z-4 F0.04 X3.5 Z4.5 F0.03 X5 G0 X10	G1 Z-7 F0.06 G0 Z2
M9008	**M9008**
	M153

4.3 Three tools simultaneously in the material

It is also possible to engage a third tool simultaneously on the bar on EvoDECO.
To do so, use "Blank-Simultaneous finishing" + "Superimpose".

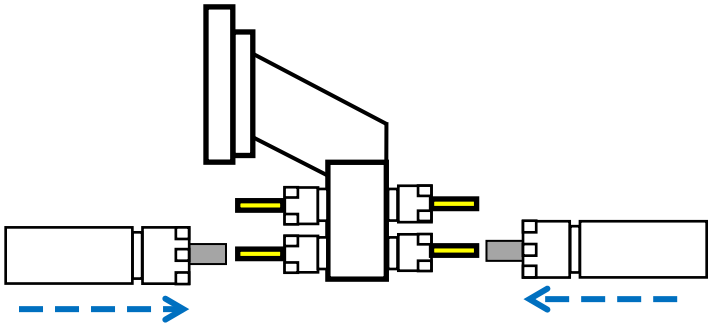
Example:



Sample Programming		
Channel 1	**Channel 2**	**Channel 3**
G54 G95 G0 X5.3 Y0 Z1 T11 D0 M103 S4000 P1	G95 G0 X5 Y0 T21 D0	G95
M9005 P13		**M9005 P13**
		G915 G0 X0 Z1 T31 D0
M9006 P13		**M9006 P13**
		M152
M9007 P123	**M9007 P123**	**M9007 P123**
M142 G1 Z-12 F0.05 G1 X5.4 G0 X10 Z2 M143		G1 Z-7 F0.06 G0 Z2
M9007 P123	**M9007 P123**	**M9007 P123**
		M153

4.4 Simultaneous mainwork & backwork drilling

A large number of Tornos machines can benefit from a two-way frontwork base. To save cycle time, it is interesting to perform drilling in mainwork and backwork simultaneously.



Sample Programming	
Channel 1	**Channel 2**
M9005 G54 G0 G95 Y0 Z1 T354 G97 M103 S4000 P1 G0 X0	**M9005** G55 G0 G95 Z1 T454 G97 M403 S4000 P4
M9006 G1 Z-10 F0.04 G0 Z2	**M9006** G1 Z-12 F0.04 G0 Z2
M9007 …	**M9007** …

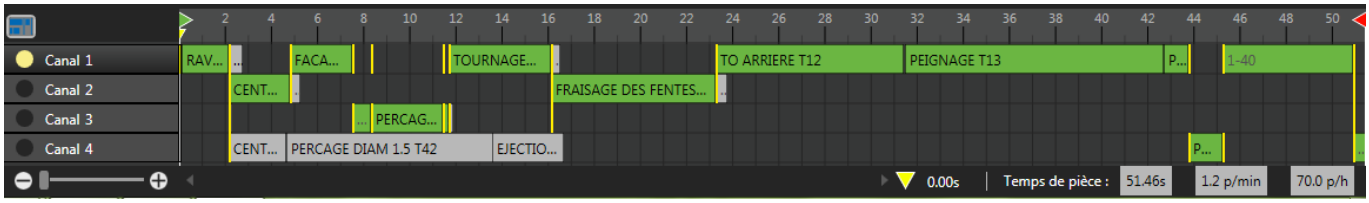
5 Programming

5.1 Where can you save time in a program?

The cycle time is the time that passes between the start of the loop and the end of the loop. The program should therefore only be optimised between the loop start [code M120] and the loop end [code M121].

A part program is split over its various channels. It is important to note that there is no point optimising a program on operations which are not on the critical path.
The "TISIS" programming software provided by Tornos allows you to highlight the critical path.

Example:
In the Gantt diagram below, you can see the operations in green that define the critical path, so it is unnecessary to try to save time on the operations that are not green.



5.2 Model Program

Tornos usually delivers its machines with part program models, allowing the machining of a blank in a completely safe and secure manner. This model can obviously be adapted to suit you in order to save on cycle time.

5.3 Out of loop

You should always perform the maximum operations out of the machining loop, i.e. before code M120.
Sample codes:

- Coolant spraying activation [M8]
- Tool rotation startup [Mxx03]
- Value initialisation
- Preliminary calculations
- Zero Offset
- …

5.4 Comments

The deletion of the comments in the program will save you some time.

Example:

```
(PNEUMATIC FEED);
M124 (DRILL 2) ;
```

```
M124;
```

5.5 Number of code lines

For the cycle time to be optimal, you need to make sure you have as few code lines as possible.
You therefore need to delete unnecessary line breaks

Example:

```
G0 Z0 X10.5;
;
G1 X0 F0.03 T11 D0;
G0 Z0.3 X6;
```

```
G0 Z0 X10.5;
G1 X0 F0.03 T11 D0;
G0 Z0.3 X6;
```

You also need to put as many codes as possible on the same line.

Example:

```
T12 D0;
G54;
M2103 S2000 P21;
G91;
G94;
F3000;
G1 Z12;
```

```
M2103 S2000 P21 T12 D0;
G54 G91 G94 G1 Z12
F3000;
```

N.B.: on SwissNano you cannot start the spindle rotation on the same line as the tool call.

5.6 Code repetitions

Certain codes are modal, which means they remain active until they are cancelled.
You should therefore avoid repeating them.

Here are some examples of modal codes:

- G0/G1
- G40/G41/G42
- G54/G55/G56/G57/G58/G59
- G61/G64
- G90/G91
- G94/G95
- G96/G97
- …

You also need to avoid repeating the call of the same tool T_ D_ multiple times.

5.7 Sub-program call

The sub-program calls [G65] take up cycle time. It is therefore sensible to avoid them if they are not indispensable.

5.8 Synchronisation of channels

Synchronisation of channels [M9xxx] also take up time, so only those that are necessary should be used.
On machines which have more than 2 channels, it is obviously not necessary to synchronise the 4 channels, and you can synchronise just 2 or 3 at a time.

Sample Programming			
Channel 1	**Channel 2**	**Channel 3**	**Channel 4**
M9000 P1234	M9000 P1234	M9000 P1234	M9000 P1234
…	…	…	…
M9001 P12	M9001 P12	…	…
…	…	…	…
…	M9002 P24	…	M9002 P24
	…	…	…
M9003 P134	…	M9003 P134	M9003 P134
…	…	…	…
…	…	…	…

5.9 Related contouring/Exact stop

On our machines, you can work in related contouring mode [G64] or in exact stop mode [G61]. The exact stop is practical for certain finishing operations, but you need to know that once it has been activated, it remains active for as long as it is not cancelled [G64]. Indeed, the exact stop makes you lose time, since it creates a little idle time between each segment.
The default mode is related contouring [G64].

Please note that you could use function G9 which activates an exact stop but only on the block in progress.

5.10Avoiding returns to the reference position

We very often come across axes returns to the reference positions in the programs [G28]. These should under all circumstances be avoided when they are unnecessary, since these lengthy movements obviously take time.

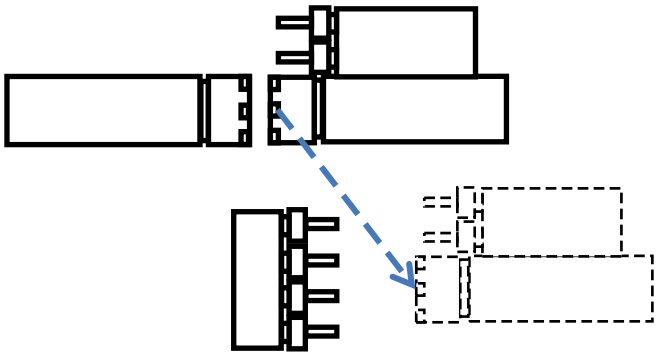
If you simply want programming simplicity, opt for function [G53 G0 *X_Y_Z_*].
[G53] allows you to program a machine position. What's more, it has the advantage of being active only on a single block, so you do not need to reactivate a zero offset.

Example:
G53 G0 X212 ➔ The X axis therefore goes in fast feed to a machine position of 212.

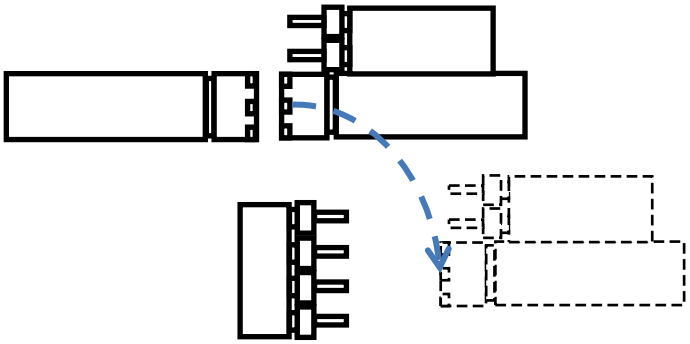
You can also carry out machine positioning on 2 simultaneous axes.

Example:
G53 G0 X212 Z243 ➔ The X axis therefore goes in fast feed to a machine position of 212 and at the same time the Z axis to position 243.

You nevertheless need to know that when you program 2 axes in machine position simultaneously, the movement does not occur in a linear manner.



Linear movement



Non-linear movement

5.11Optimising delays

It is interesting to fine tune the duration of delays.
So carry out a few real tests to fine tune delays as much as possible.
What's more do not forget that some of your pneumatic elements include adjustable flow control valves.

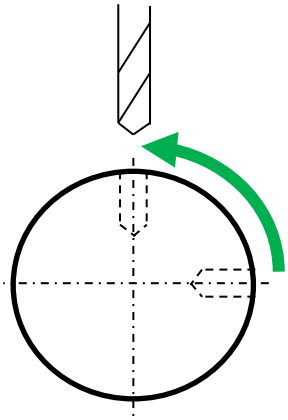
In the model programs delivered by Tornos, there is always function G802 in the initialisation part. This function G802 allows you to set the parameters of the delay that are not accessible by the user, which are inside the Tornos macros (G9xx). By default, Tornos configures them so as to guarantee correct machine operation. This being said, they may sometimes be fine tuned (reduced).

Example:
If you take for example a spindle collet closing delay, its role is to ensure the collet is closed before the machine begins another action.
To fine tune this value, you need to know the machine well. Indeed, the collet closing times depend:

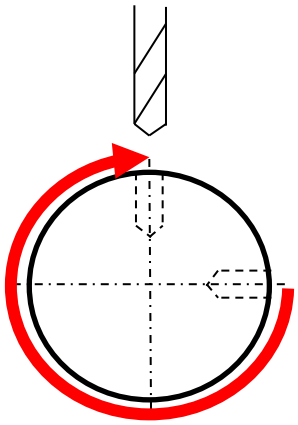
- On the bar diameter
- On the condition of the collet
- On the quality (flow rate) of the pneumatic circuit

5.12Positioning in C axis

When you perform several subsequent orientations of your C axis, always make sure you take the shortest path.



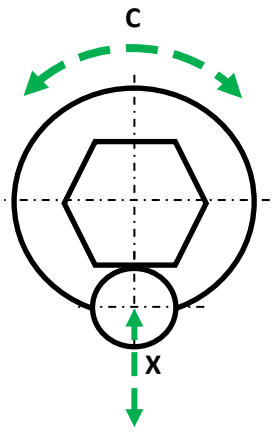
Shortest path



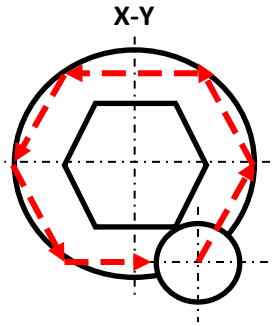
Longest path

5.13Transmit

When you need to perform some milling on a face, opt for milling in Transmit mode [G12.1] (movement in X and C) rather than traditional milling (movement in X and Y). In many cases, this will save you cycle time.



Milling in Transmit



Traditional milling

5.14Part cutting

Generally, part cutting is found on the critical path. It is therefore important for the operation to be carried out as quickly as possible.

Cutting in two steps:

In Tornos models cutting takes place in 2 steps. In step two, the cutting feed is reduced since the cutting effort is more important when the cutter reaches the centre point. Should you decide that it is not necessary, you can cut in one step, to save time.

Example:

```
G1 G95 X0 F0.02 (CUTTING);
G1 X-1 F0.01 (END OF CUTTING);
```

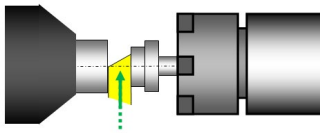
```
G1 G95 X-1 F0.02;
```

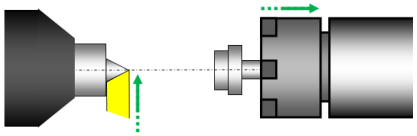
Another possibility is to carry out the end of cutting at the same time as the counter spindle retraction.

Example:

```
G1 G95 X0 F0.02 (CUTTING);
G1 X-1 F0.01 (END OF CUTTING);
M9203;
```

```
G1 G95 X0 F0.02 (CUTTING);
M9203;
G1 X-1 F0.01 (END OF CUTTING);
```

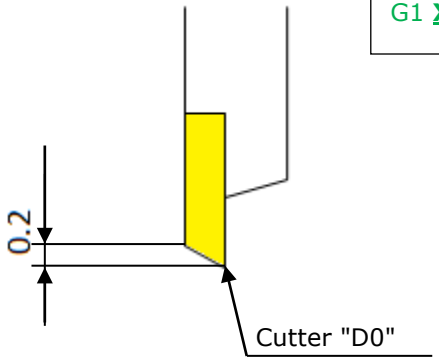




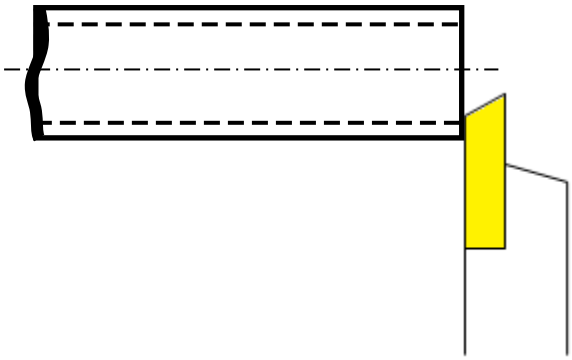
End of cutting position:

In Tornos models, the end of cutting position is at X-1mm. This distance can be fine tuned according to the type of cutter used. For a straight cutter, you can set X0 as the end of cutting position.
For a cutter with an angle, follow the example below:

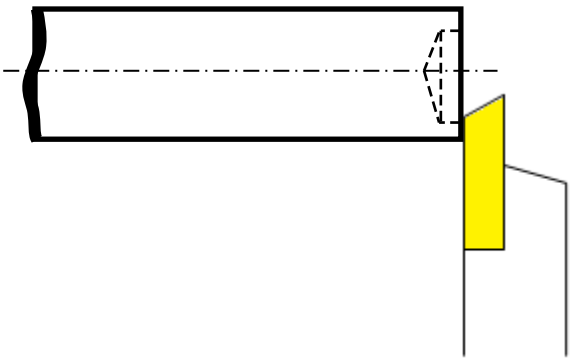
```
G1 G95 X0 F0.02;
G1 X-0.4 F0.01;
```



If you are machining tubes, or parts which are drilled further in mainwork, then you do not need to perform the cut up to the centre. A cut up to the internal diameter of the part will make you save time.



End of cutting position for tube machining



End of cutting position for part drilled beyond

N.B.: for machining tubes, we recommend you fit a plug at the end of the bar, to avoid the cutting oil ending up in the barfeeder.

Cutting at constant feed:

In the Tornos models, the cutting is performed with a feed in mm/rev [G95] and a constant rotation frequency [G97].
However, the closer a tool (insert holder) gets to the centre of the bar, the less material it removes. There is therefore the possibility of performing cutting at constant speed (m/min) [G96], and a feed in mm/rev [G95].
By using[G95] + [G96] the advantage is that the closer you get to the centre of the bar the faster the spindle turns and the more the feed increases.
This is a great benefit especially for cutting large diameters. This will save you cycle time.

Example:

```
G1 G95 X0 F0.02 (CUTTING);
G1 X-1 F0.01 (END OF
CUTTING);
```

```
G92 S_  (MAX SPEED REV/MIN);
G95 G96;
M103 S_ P1 (SPEED M/MIN);
G1 X-1 F0.02 (CUTTING);
```

5.15Checking cutting tool breakage

In the models delivered by Tornos, there is always a built-in broken cutting tool check. This broken cutting tool check works by checking the retraction of counter spindle Z4 after part pickoff (with torque limitation + servo lag error check). This system is practical but evidently it takes up a little cycle time. It is obviously possible to delete it.

Example:

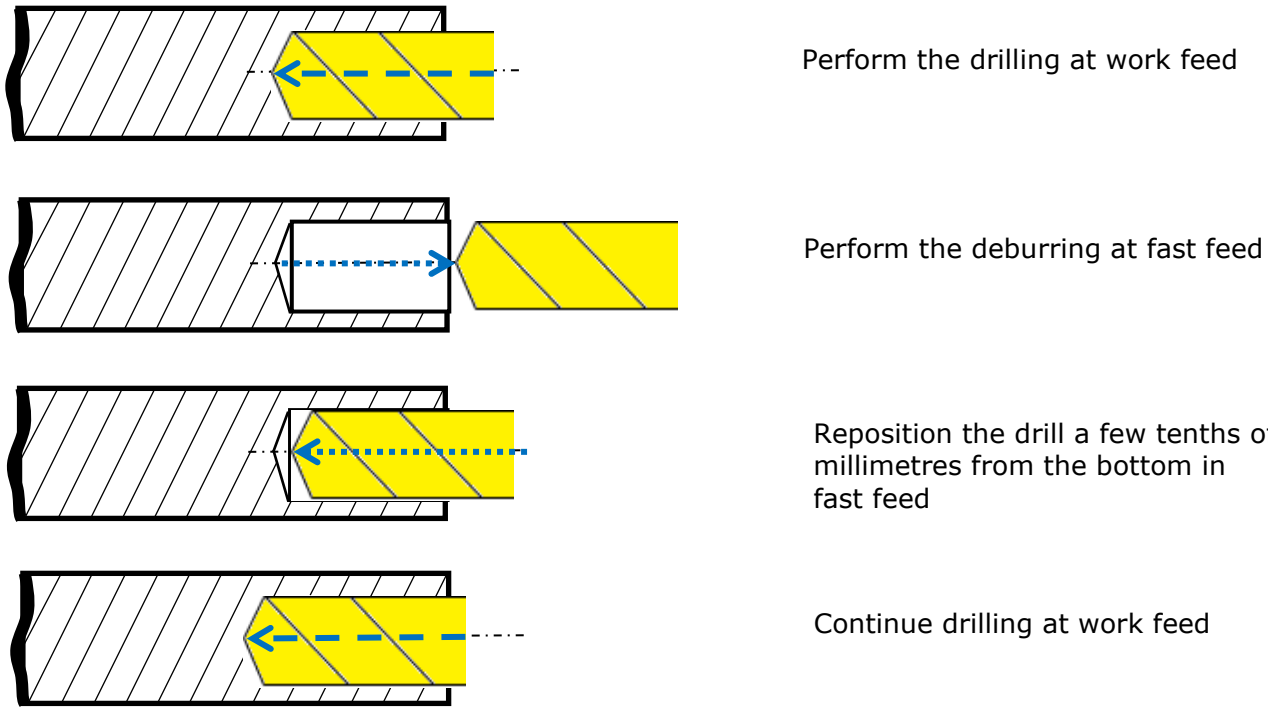
```
M702 M4 M1 (Z4 TORQUE LIMITATION);
M702 M5 M1 (Z4 SERVO LAG ERROR);
G924 (PART PICKOFF);
M9202 ;
M9203 ;
G1 G94 W2 F2000 (CUTTING TOOL BREAKAGE CONTROL);
M702 M5 M0 (TURN OFF SERVO LAG ERROR CONTROL);
M702 M4 M0 (TURN OFF TORQUE LIMITATION);
G53 G0 Z100 (RETRACT COUNTER SPINDLE);
```

```
G924 (PART PICKOFF);
M9202;
M9203;
G53 G0 Z100 (RETRACT COUNTER SPINDLE);
```

5.16Drilling/deburring

When you program a drilling-deburring operation, always remember to perform your movements that do not remove any chips at a faster feed than the work feed.

Example:



5.17Working during material feeding

In the models delivered by Tornos, function G912 allows you to feed a certain quantity of material to allow the machining of a part. Generally, material feeding is found on the critical path. It is sensible to do something in parallel on the other channels.

Example 1:

In the example below, we have moved the retraction of the counter spindle after the part pickoff. Thus, during material feeding, we retract the counter spindle and save cycle time . It is interesting to add this same retraction after M121 so that the counter spindle is in the back position when the cycle stops.

Canal 1	Canal 2
Variables pièce (G800) Variables de coupe (G801) Variables de temporisation (G802) Variables éléments machine (G803)	
M9000	M9000
G900 (Initialisation)	G900 (Initialisation)
M9001	M9001
	Référence des axes Z4, X4 et Y4
M9002	M9002
Référence des axes X1, Y1 G910 (Coupe initiale)	
M9003	M9003
N1 M120 (Début de boucle)	N1 M120 (Début de boucle)
G913 (Test du fin de barre) G930 (Gestion du sans matière) G912 (Ravitaillement matière)	
M9004	M9004
Usinage	Usinage Ejection de la pièce
M9005	M9005
Appel du coupeur	
M9006	M9006
	Prise de pièce
M9007	M9007
Coupe	
M9008	M9008
	Recul de la contre broche
M121 (Fin de boucle)	M121 (Fin de boucle)
Arrêt des broches Arrêt de l'arrosage	Arrêt des broches
M9009	M9009
Arrêt en fin de cycle	Arrêt en fin de cycle
M9010	M9010
M99 P1 M2	M99 P1 M2

Canal 1	Canal 2
Variables pièce (G800) Variables de coupe (G801) Variables de temporisation (G802) Variables éléments machine (G803)	
M9000	M9000
G900 (Initialisation)	G900 (Initialisation)
M9001	M9001
	Référence des axes Z4, X4 et Y4
M9002	M9002
Référence des axes X1, Y1 G910 (Coupe initiale)	
M9003	M9003
N1 M120 (Début de boucle)	N1 M120 (Début de boucle)
G913 (Test du fin de barre) G930 (Gestion du sans matière) G912 (Ravitaillement matière)	**Recul de la contre broche**
M9004	M9004
Usinage	Usinage Ejection de la pièce
M9005	M9005
Appel du coupeur	
M9006	M9006
	Prise de pièce
M9007	M9007
Coupe	
M121 (Fin de boucle)	M121 (Fin de boucle)
Arrêt des broches Arrêt de l'arrosage	**Recul de la contre broche** Arrêt des broches
M9009	M9009
Arrêt en fin de cycle	Arrêt en fin de cycle
M9010	M9010
M99 P1 M2	M99 P1 M2

Example 2:

In the example below, we have deleted the channel synchronisation [M9004] in order to commence machining in backwork at the same time as the material feeding. Beware, since the deletion of this synchronisation is only possible if the Z4 axis works in backwork and not at the bar.

Canal 1	Canal 2
Variables pièce (G800) Variables de coupe (G801) Variables de temporisation (G802) Variables éléments machine (G803)	
M9000	M9000
G900 (Initialisation)	G900 (Initialisation)
M9001	M9001
	Référence des axes Z4, X4 et Y4
M9002	M9002
Référence des axes X1, Y1 G910 (Coupe initiale)	
M9003	M9003
N1 M120 (Début de boucle)	N1 M120 (Début de boucle)
G913 (Test du fin de barre) G930 (Gestion du sans matière) G912 (Ravitaillement matière)	
M9004	M9004
Usinage	Usinage Ejection de la pièce
M9005	M9005
Appel du coupeur	
M9006	M9006
	Prise de pièce
M9007	M9007
Coupe	
M9008	M9008
	Recul de la contre broche
M121 (Fin de boucle)	M121 (Fin de boucle)
Arrêt des broches Arrêt de l'arrosage	Arrêt des broches
M9009	M9009
Arrêt en fin de cycle	Arrêt en fin de cycle
M9010	M9010
M99 P1 M2	M99 P1 M2

Canal 1	Canal 2
Variables pièce (G800) Variables de coupe (G801) Variables de temporisation (G802) Variables éléments machine (G803)	
M9000	M9000
G900 (Initialisation)	G900 (Initialisation)
M9001	M9001
	Référence des axes Z4, X4 et Y4
M9002	M9002
Référence des axes X1, Y1 G910 (Coupe initiale)	
M9003	M9003
N1 M120 (Début de boucle)	N1 M120 (Début de boucle)
G913 (Test du fin de barre) G930 (Gestion du sans matière) G912 (Ravitaillement matière)	**Usinage en contre-opération**
Usinage	Ejection de la pièce
M9005	M9005
Appel du coupeur	
M9006	M9006
	Prise de pièce
M9007	M9007
Coupe	
M9008	M9008
	Recul de la contre broche
M121 (Fin de boucle)	M121 (Fin de boucle)
Arrêt des broches Arrêt de l'arrosage	Arrêt des broches
M9009	M9009
Arrêt en fin de cycle	Arrêt en fin de cycle
M9010	M9010
M99 P1 M2	M99 P1 M2

5.18 Part pickoff by counter spindle

In the models delivered by Tornos, there is always an approach of the counter spindle at 2 mm from the part before performing part pickoff. If you want to save cycle time, you can perform part pickoff directly. You can also increase the feed (which is 4,000 mm/min by default).

If G924 A0 (the approach takes place in fast feed).
If G924 A1000 (the approach takes place at 1,000mm/min).
If G924 (the approach takes place at 4,000mm/min).

Example:

```
G0 X0 T40;
G0 Z2 (COUNTER SPINDLE
APPROACH);
G924 (PART PICKOFF);
```

```
G0 X0 T40;
G924 A0 (PART PICKOFF);
```

In the models delivered by Tornos, there is always a synchronisation between the spindle phases and the counter spindle phases [M418]. The fact that the counter spindle is already turning at the right speed (same speed as the spindle but in the other direction) at the time of M418 will save you cycle time.

The synchronisation of the phases is not indispensable, so if you believe that in your case it is not necessary you can replace it with M417 which will simply synchronise the spindle and counter spindle speeds, which will save you some time.

5.19Managing the ejector

It is sometimes interesting to anticipate the ejector output to just before the counter spindle collet opens. This way, as soon as the collet is physically open, the part will be ejected.

Example:

```
M11 (COLLET OPENING);
G4 X0.5;
M84 (EJECTOR OUTPUT);
G4 X2;
M85;
```

```
M84 (EJECTOR OUTPUT);
M11 (COLLET OPENING);
G4 X_ ;
M85;
```

What's more, it is interesting to retract the ejector [M85] at the same time as something else (for instance, moving an axis).

6 For programming experts

6.1 Deleting macros

Tornos macros [G9xx] provide the huge benefit of managing a very large number of parameters better to accommodate the need of each part and of each machining process. What you still need to know is that multiple tests and calculations are carried out in real time during each cycle, during the interpretation of the program on the machine.
The most advanced programmers can do without certain macros and program directly. This makes it possible to no longer carry out the various tests and calculations within the macros, and thus save a few hundredths of seconds.

The macros which it may be interesting to delete to save cycle time are:

- G911
- G912
- G915
- G924
- *G930*
- *M120/M121*

If these are deleted, you will lose the intelligence tied to the events.
(Work without material, stop at end of cycle, timing, part counting, …)

We recommend never to delete function G913 , since it manages the loading of a new bar.

6.2 Macro B

For B macro programming experts we recommend never to perform a "GOTO_" against the top of the program. This will make you waste time, and will lead to major irregularities in your cycle time. We recommend you use "M99 P_" or a "WHILE".

Example: