



AJAN®

Elektronik Servis San. Ve Tic. Ltd. Şti.

DOKÜMAN NO: KTP.09 REVİZYON:00 REVİZYON TARİHİ: YAYIN TARİHİ: 08.04.2009



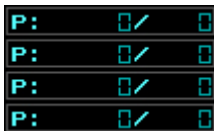
AJAN CNC FREZE MILLING MACHINE DESCRIPTION OF CNC UNIT



AJAN MFREZE SOFTWARE SCREEN CAPTURE

GUIDE FOR THE OPERATION OF THE UNIT

AJAN CNC FREZE MILLING MACHINE is controlled via soft keys shown for the different menus at the bottom of the screen. A hand control unit is also incorporated for the manual control. Status monitoring is achieved by open windows on top of the screen continuously and some other temporary messages are issued whenever needed at the appropriate parts of the screen.



Indicates the subroutine repeat number.



Indicates axis speed and spindle speed.



X axis indication in millimeter



Y axis indication in millimeter



Z axis indication in millimeter



U axis indication in millimeter



TIMER : Indicates the last period with kind of hour, minute and second when machine run or simulate the program with **F7DryRun** function. When press “S” key, indicates time.

DESCRIPTION OF MENUS

Menü 0 (Main Menu)

```
File name: DSL0T2.T0R   Start no: 7   End no: 32   CAMlineNo: 0
F1Run   F2Save   F3File   F4De1LinF5InsLinF6ResumeF7DryRunF8Block F9Quit   F10Menu
ENTER to edit BLOCK                                          Menu:0
```

Menü 1

```
File name: DSL0T2.T0R   Start no: 7   End no: 32   CAMlineNo: 0
F1MDI   F2GoZEROF3M_HandF4   F5Reset F6Accel F7MZR   F8   F9Graph F10Menu
                                          Menu:1
```

Menü 2

```
File name: DSL0T2.T0R   Start no: 7   End no: 32   CAMlineNo: 0
F1Paran.F2T_OffsF3AbsPOsF4G5x   F5   F6PrgGetF7   F8T_Rst F9Turk   F10Menu
                                          Menu:2
```

ANA MENÜ (Menü:0)

```
File name: DSL0T2.TOR  Start no: 7  End no: 32  CAMlineNo: 0
F1Run  F2Save  F3File  F4DelLinF5InsLinF6ResumeF7DryRunF8Block F9Quit  F10Menu
ENTER to edit BLOCK  Menu:0
```

ENTER to edit BLOCK : Edit program lines in the left of screen.

F1

Run: It runs the program written in the screen.

F2

Save: Saves the contents of the edited program.

F3

File: Used to load files from disk.

F4

DelLin: Select the block to be erased and push this function key (**Menu:0**).

F5

InsLin : Used to insert block after chosen block.(**Menu:0**).

F6

Resume: This function is processed only if machine is in “High Speed Machining” mode(**HM** code is used). If our work stops in an unwanted way, such as electricity cut, by this function we can continue our process from the breaking point.

F7

DryRun: It simulates the program graphically without moving the axes.

F8

Block : Sets the start and end block number of the program.

F9

Quit: Used to return back to **DOS**. If **N** key is pushed the operation will be cancelled.

F10

Menu: Used to switch menu bars for **Menu: 0**, **Menu: 1**, **Menu: 2**.

```

File name: DSLDT2.TOR   Start no: 7   End no: 32   CAMlineNo: 0
F1Run   F2Save  F3File  F4DeLlinF5InsLinF6ResumeF7DryRunF8Block F9Quit  F10Menu
ENTER to edit BLOCK                                         Menu: 0

```

PROGRAMMING:

Programming is possible in main menu (**Menu:0**). The reserved part for programming is the left zone of the screen and is activated by pressing **ENTER** key.

```

File name: TS42.TOR   Start no: 0   End no: 7   CAMlineNo: 0
ESC:Exit Editor & Load old data.   ENTER:Next Line.
                                         Menu: 3

```

```

000 G54 G90 F3000 S100
001 X0 Y0 Z10F
002 Z2F
003 HMF
004 G82 Z-10 R2 F300 K
005 X10F
006 X20F
007 X30F
008 X40F
009 X50F
010 G80F
011 HMF
012 G90 Z50F
013 M30F
014 B
015 B
016 B
017 B
018 B
019 B
020 B

```

Total **255** lines can be programmed. **40** characters Per line is possible. Further programming lines can be achieved by calling the names of other subprograms. Programming editor works similar to popular editor programs. In the **menu: 0** bar arrow keys, **Page up** **Page Down** keys are active

ENTER key will register the changes for that block and the next block will be highlighted "**ESC**" key will cause editor to return to menu bar (**Menu: 0**) and if some changes made in that particular block, it will be left unchanged. **F2** key will save the edited file. The reserved right zone of the screen will be used for graphical display for machining and fault messages

FILE NAME: Displays the name of the file.

START NO: Indicates the start line of the program to be executed.

END NO: Indicates the program finish line.

CAMLineNo: Indicates the start line of cam files adding to process with G38 code.

MENU: Pressing **F10** function key will change menu bars under the screen. At the right bottom corner the name of the menu will be displayed.

File name: DSLDT2.TOR Start no: 7 End no: 32 CAMlineNo: 0
F1Run F2Save F3File F4DelLinF5InsLinF6ResumeF7DryRunF8Block F9Quit F10Menu
ENTER to edit BLOCK Menu:0



Run: It runs the program written in the screen. Once this key is pressed and everything is OK. (limit switches, axis drives). Machine starts running, the menu bar will change as show below and the only way to stop the motion momentarily is to press the **PAUSE** button in hand control unit, pressing again the **PAUSE** button will resume the motion pressing the **ABORT** button will cause machine to go in stand by mode(Menu:0)

File name: TS42.TOR <cutting> CAMlineNo: 0
Press PAUSE button to STOP the machine

If **PAUSE** button is pressed while machining the following menu will appear.

P: /	<div>7560 FEED</div> <div>1829 SPINDLE</div>	10.000 MM X	-1.704 MM Z
P: /		0.000 MM Y	0.00 O U
P: /			
P: /			

00:00:04

000 G59 G90 F3000 S300
001 X0 Y0 Z10P
002 Z2P
003 HNP
004 G81 Z-10 R2 F300 K
005 X10P
006 X20P
007 X30P
008 X40P
009 X50P
010 G80P
011 HNP
012 G90 Z50P
013 M30P
014 B
015 B
016 B
017 B
018 B
019 B
020 B

xdiff:0.000
ydiff:0.000
zdiff:0.000
udiff:0.000

PAUSE

File name: TSG81.TOR <cutting> CAMlineNo: 0
[PAUSE:CONTINUE] [ABORT:Menu1] [offset axis from hand control]
Menu:4



Save: Saves the contents of the edited program.

```
File name: TS42.TOR ***** SAVING THE FILE *****
F1Run F2Save F3File F4DelLinF5InsLinF6ResumeF7DryRunF8Block F9Quit F10Menu
ENTER to edit BLOCK Menu:0
```



File: Used to load files from disk. When you press this function key the files with **TOR** extension will be displayed. Page down page up and arrow keys will highlight the files pressing the **ENTER** button will load that particular file. The same key serves to view list of files existing in the current directory with extension. **CAM** and **DXF** by pressing **C** or **X** key. After remembering these files we can link them to our main program in following format **G37 “dxs file“** and **G38 “cam file“** when ever needed. By pressing “S” we can sort files according to their size, “D” according to their creation date and “N” by name. After remembering these files we can link them to our main program in following format **G37 “dxs file“** and **G38 “cam file“** when ever needed.

```

P: 0/0
P: 0/0
P: 0/0
P: 0/0
000 FEED
000 SPINDLE
0.000 X
0.000 Y
0.000 Z
0.000 U
00:00:04
000 G59 G90 F3000 S300
001 X0 Y0 Z10J
002 Z2J
003 HMF
004 G81 Z-10 R2 F300 K
005 X10J
006 X20J
007 X30J
008 X40J
009 X50J
010 G80J
011 HNF
012 G90 Z50J
013 M30J
014 B
015 B
016 B
017 B
018 B
019 B
020 B
12M.TOR
200MM.TOR
25UDRL.TOR
30MAT.TOR
35MAT.TOR
40MAT.TOR
40UDRL.TOR
50MM.TOR
A.TOR
ADA1782.TOR
AJAN.TOR
ALTPROG.TOR
APRT.TOR
ARIF.TOR
BOS.TOR
COKLU.TOR
DEL2.TOR
DELLL.TOR
DENE.TOR
DOGUS.TOR
File name: TS681.TOR Start no: 0 End no: 13 CAMlineNo: 0
F1Run F2Save F3File F4DelLinF5InsLinF6ResumeF7DryRunF8Block F9Quit F10Menu
< press ESC for new file > Menu:0
```

To call the programs stored in the memory, select the file and press “**ENTER**”. In the main menu a sub menu is displayed. For files with extension “**TOR**”;

```
File name: TS42 SAVE THE FILE ? Yes/ ESC
F1Run F2Save F3File F4DelLinF5InsLinF6ResumeF7DryRunF8Block F9Quit F10Menu
Menu:0
```

If we press “**Y**” the last opened file is saved and selected file opens, “**ESC**” will open the selected file without saving the last opened file.

To create a new file; select “file button” by pressing function key **F3**, press **ESC** key and type in 8 character file name to the small window that highlights. This file must have a different name from the list and then the block lines existing already in the editor will be saved as with this new file name

```
File name:   Start no: 0 End no: 7 CAMlineNo: 0
F1Run F2Save F3File F4DeLInF5InsLinF6ResumeF7DryRunF8Block F9Quit F10Menu
ESC: EXIT Menu:0
```

After pressing the “**ENTER**” Key following line appears.

```
File name: test5 TS42 SAVE THE FILE ? Yes/ ESC
F1Run F2Save F3File F4DeLInF5InsLinF6ResumeF7DryRunF8Block F9Quit F10Menu
Menu:0
```

To save the old file press “**Y**” key. To leave the old file unsaved press **ESC** key. Pushing **Y** key saves the old file before creating a new file. Following figure is displayed during saving.

```
File name: test5 ***** SAVING THE FILE *****
F1Run F2Save F3File F4DeLInF5InsLinF6ResumeF7DryRunF8Block F9Quit F10Menu
Menu:0
```

After this operation following menu bar appears.

```
File name: test5 FILE TO BE CREATED! Yes/ ESC/B1k
F1Run F2Save F3File F4DeLInF5InsLinF6ResumeF7DryRunF8Block F9Quit F10Menu
Menu:0
```

If we want to open a new blank file with this name “**B**” letter has to be pushed

“**ESC**” button will cancel operation..

“**Y**” button will create the file with different name but same content as the last opened file.

DELETING FILES: Press **F3** File function key. High light the file using arrow Keys, Page Up and Page Down Keys then press “**DELETE**” key.



Pressing “Y” Key will delete the file., “N” Key will cancel the operation..



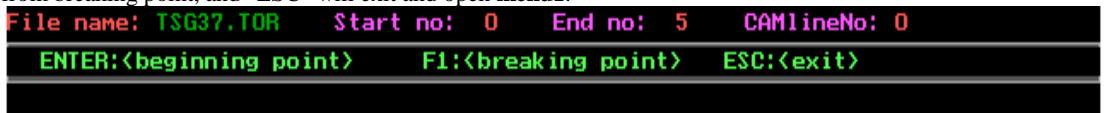
DelLin: Select the block to be erased and push this function key .



InsLin : Used to insert block after chosen block.



Resume : This function is processed only if machine is in “High Speed Machining” mode(HM code is used). If our work stops in an unwanted way, such as electricity cut, by this function we can continue our process from the breaking point. There is no restriction in using this function, can be called in any level of our process. When we press “F6Resume” we have three choices; “ENTER” will start the program from the beginning, “F1” from breaking point, and “ESC” will exit and open menu1.



DryRun : It simulates the program graphically without moving the axes.(menu:0). In dry run pressing “ESC” Key will end scanning the file.

```
File name: TS42.TOR    <calculating>
ESC EXIT
Menu: 0
```



Block : Sets the start and end block number of the program by pushing F8 function Key in **Menu:0**. Enter start number for the program push **ENTER**, write the end number push **ENTER** if the number is more than three digit it will automatically skip to next procedure. At the end it will return back to the main menu.

```
File name: TSG37.TOR   Start no:  End no: 5   CAMlineNo: 0
ENTER PROGRAM START AND END BLOCK
Menu: 0
```



Quit: Used to return back to **DOS**. If N key is pushed the operation will be cancelled.

```
File name: TSG37.TOR   Start no: 0   End no: 5   CAMlineNo: 0
F1Run  F2Save  F3File  F4DelLinF5InsLinF6ResumeF7DryRunF8Block F9Quit F10Menu
Are you sure Y/N
Menu: 0
```



Menu: Used to switch menu bars for **Menu: 0, Menu: 1, Menu: 2**.

MENÜ: 1

File name: DSILOT2.TOR Start no: 7 End no: 32 CAMlineNo: 0
 F1MDI F2GoZEROF3M_HandF4 F5Reset F6Accel F7MZR F8 F9Graph F10Menu
 Menu: 1



MDI: Used to move axes X, Y, Z, U to desired positions when **F1** is pressed. In graphical screen **ENTER** MDI will appear:



F1 MDI Comments;

- X, Y, Z, U** : Used to move axes X, Y, Z, U.
Using : Enter MDI: (Axis) (to desired positions)
Sample : Enter MDI: X100 then press "ENTER". X axis goes to desired position of 100 mm. Each axes (X, Y, Z, U) can be used same manner.
- S** : Use to rotate to SPINDLE for entered value of revolution..
Using: Enter MDI:S (value of revolution)
Sample: Enter MDI: S1000 rotates spindle at 1000 rpms if S0 is given spindle goes to STOP.
- T** : Change the Tool.
Using : Enter MDI: T(tool number)
Örnek: Enter MDI: T5 If the new Tool Number is different from old one, machine changes the Tool.

- **H** : Sets the legal Tool Offset Number.
Using : Enter MDI: **H**(tool offset number)
Sample: Enter MDI: **H5** Sets the legal Tool offset number to 5.
- **G** : Sets the references. (**G54, G55, G56, G57, G58, G59**)
Using : Enter MDI: (reference)
Sample : Enter MDI: **G54** Sets the reference to G54.
- **F** : Sets the Feed.
Using : Enter MDI: **F**(feed value (milimeter/minute))
Sample: Enter MDI: **F2000** Sets the feed to 2000mm/min.

F2 GoZERO: Will pull all axis (**X,Y,Z,U**) to “0” position. It will first move in + **Z** direction until it hits upper **Z** limit switch then moving the other axis to **0** position and than **Z** axis to **0** position. If **ABORT** button is pushed while **Z** axis is going to upper direction it will continue the operation from that **Z** position without need to look for + **Z** limit switch.

F3 M Hand : Plasma is controlled by Hand Control Unit. Independently from the program all axis can be controlled. Only “**ABORT**” button in hand control unit or “**ESC**” button are available to exit and go to the Menu:1.

File name: TSG81.TOR Start no: 0 End no: 10 CAMlineNo: 0 T1 H1 G59
F1 F20fs F30ffsZrF4:2 F5Reset F6Mgz F7T_Elk F8 F9 F10
[Abort/ESC to quit] Menu: 8

F2 F2 T_Ofs : If the machine have tool switch equipments. Press **F2T_Ofs** function key at **M_Hand** menu that take the ofset value to the tool offset number. This ofset must be setted before this operation. User can be set this **H#** with **MDI** by hand. You can check this number before the taking ofset from right bottom side off screen. Automatically assigning tool ofset can be used with any tool and you can see new ofset value in **F2T_Offs** menu.

F3 F3 OffsZr : This function is used for Assigning tool switch equipment location. Before this operation user must go machine zero with **Menu1 F7MZR**. Then user must go to the X,Y position of the tool switch equipment by hand control unit and press “**A**” key for assigning this position. User can be sets **ENABLED** the getting automatic offsets from anywhere after the positioning of the switch equipments. Otherwise user sets the Zero Point **DISABLED**; User must be locate X,Y axis before takign the each tool ofset.

If user set zero point to “**ENABLED**”; When user press **F2T_Ofs** function, firstly the machine goes X,Y position to the tool switch equipment and then moving **Z** axis and take ofset

If user set zero point to “**DISABLED**”; When user press **F2T_Ofs** function, moving **Z** axis and take ofset.

File name: TSG81.TOR Start no: 0 End no: 10 CAMlineNo: 0 T1 H1 G59
F1 F20fs F30ffsZrF4:2 F5Reset F6Mgz F7T_Elk F8 F9 F10
[ESC to Quit] <a>assign zero Enabled/Disabled zero point <PGUP/PGDOWN> Menu: 8

F4

F4 fonksiyon tusu ($\div 2$): Used to divide axis values shown on the display by two while hand control unit is active.

File name: TSG81.TOR Start no: 0 End no: 10 CAMlineNo: 0 T1 H1 G59

F1 F20fs F30ffsZrF4:2 F5Reset F6Mgz F7T_Elk F8 F9 F10

X/2:0.000 Y/2:0.000 Z/2:0.000 U/2:0.000 Menu:8

F5

Reset: X, Y, Z will be reset to “0”. You can also reset each axis separately while in manual positioning (F3M_Hand) by pressing CTRL and related axis. **Example:** Pressing CTRL + X will reset X axis to 0.

F6

Magzn: Used to assigned to tool magazines position. Only use with Tool Changer.

Dosya ismi:TSG81.TOR Baslama No:0 Bitis No:14 CAMsatiNo:0

F1MDI F20_cek F3F_Elk F4 F5Reset F6Accel F7MSF F8 F9GrafikF10Menu

Menu:1

F5

Reset: X, Y, Z will be reset to “0”. You can also reset each axis separately while in manual positioning (F3M_Hand) by pressing CTRL and related axis. **Example:** Pressing CTRL + X will reset X axis to 0.

F6

Accel: Sets the acceleration at the start or while stopping. For example the machine reaches 6 m/min from 0 m/min in 5 milimeters with 1000 acceleration value.

Dosya ismi:TSG81.TOR Baslama No:0 Bitis No:14 CAMsatiNo:0

F1MDI F20_cek F3F_Elk F4 F5Reset F6Accel F7MSF F8 F9GrafikF10Menu

ivme: eski deger: 200 Menu:1

P: /
P: /
P: /
P: /

2520 FEED
0000 SPINDLE

0.000 X
0.000 Y
0.000 Z
0.00 U

00:00:04

```

000 G59 G90 F3000 S300
001 X0 Y0 Z10F
002 Z2F
003 HNF
004 G81 Z-10 R2 F300 K
005 X10F
006 X20F
007 X30F
008 X40F
009 X50F
010 G80F
011 HNF
012 G90 Z50F
013 M30F
014 B
015 B
016 B
017 B
018 B
019 B
020 B

```

g54x	0.000	g55x	480.008
g54y	0.000	g55y	767.792
g54z	0.000 H1	g55z	-352.088 H1
g54u	0.000	g55u	0.000
g56x	480.008	g57x	480.008
g56y	767.792	g57y	767.792
g56z	-352.088 H3	g57z	-352.088 H3
g56u	0.000	g57u	0.000

For Single axis Machine Zero
Enter the letter of the axis (X/Y/Z/U)

File name: TSG81.TOR Start no: 0 End no: 13 CAMlineNo: 0

F1MDI F2GoZEROF3M_HandF4 F5Reset F6Accel F7MZR F8 F9Graph F10Menu

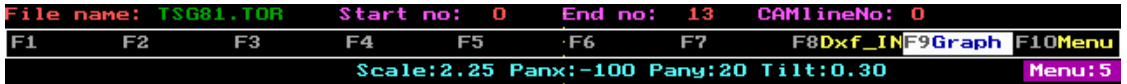
ESC to quit or Press a Key to continue Menu:1

F7

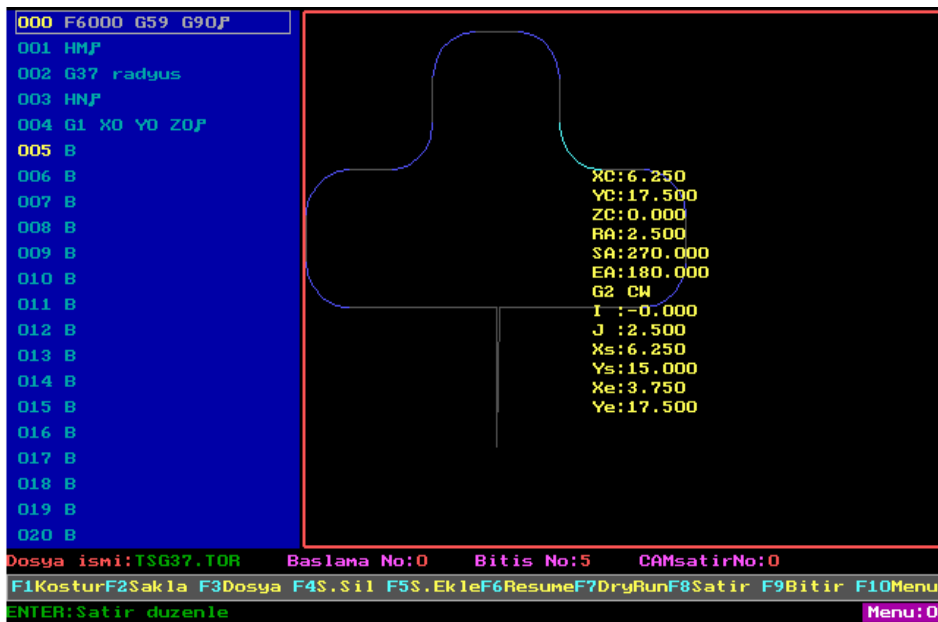
MZR: This must be performed in each start up of the machine to have a reliable positioning after electricity cut out or shut down of the machine. It searches for machine zero point. Single axis machine zero point is also possible by pressing in single axis letter keys when prompted. The (G54, G55, G56, G57,G58) work piece zero point settings are available only if this machine zero operation will be performed.



Graphic: When F9 function Key is Pressed following menu bar will appear.



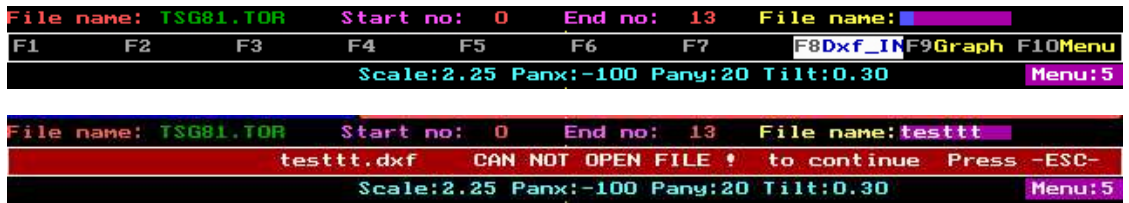
Use to control graphically screen, zoom and pan. Zoom scale and pan value will be displayed on the graphically screen. In case when a **dx**f file is loaded graphically the figure will respond to these changes. Down (↓), Up (↑), right (→), left (←) Keys will shift the dotted line origin and page up and page down will change the zoom scale.



Example: Codes and Co-ordinate explanations for circular interpolation is given below.

XC.....:	Center for “X” axis.
YC.....:	Center for “Y” axis.
ZC.....:	Center for “Z” axis.
RA.....:	Radius
SA.....:	Start angle.
EA.....:	End angle.
G2 CW.:	Indicates arc direction G2 :Clockwise , G3 :Counterclockwise.
I.....:	X axis distance form the center to actual position.
J.....:	Y axis distance forms the center to actual position.
Xs	“X” axis start point.
Ys	“Y” axis start point.
Xe	“X” axis end point.
Ye.....:	“Y” axis end point.

F8 DXF IN: Type in the name of the DXF file without the extension and press “ENTER” Key. And clicking the left mouse button on the entity will give the information about that entity.



If called **DXF** file does not exist a warning message will appear.



Menu: Toggles between **Menu: 0**, **Menu: 1**, **Menu: 2** The button right corner shows the menu number.

MENÜ:2

File name: TSG81.TOR Start no: 0 End no: 13 CAMLineNo: 0
F1Param.F2T_OffsF3AbsP0sF4G5x F5 F6PrgGetF7 F8T_Rst F9Turk F10Menu
Menu: 2



Param.: This key used to opening parameters menu. Optional key is “T”.

PARAMETERS	
TOOLS CHANGER	ON
TOOL	1
STROKE LIMIT	0 mm
Z SAFE DISTANCE	0 mm
MAGAZINE SAFE DISTANCE	0 mm
MAGAZINE AXIS	Y
TOOL OPEN/CLOSE	
MACHINE ZERO X AXIS	+X
MACHINE ZERO Y AXIS	+Y
MAX.SPINDLE RPM.	2400 rpm
EXIT	

TOOL CHANGE: For activating tool changer system. If Machine have tool changer equipment this parameter must be set “ON” otherwise is “OFF”

TOOL: Indicates Legal Tool Number

STROKE LIMIT: Restricts working area with specified magazine axis. This Function is active if tool change parameter is “ON”.

Z SAFE DISTANCE: Specifies the parameter that distance from magazine Tool Z coordinate to top level machine traveling magazine to magazine with this parameter. This Function is active if tool change parameter is “ON”.

MAGAZINE SAFE DISTANCE: Tool changing safe distance for clamping axis for this distance is to the magazine axis. This Function is active if tool change parameter is “ON”.

MAGAZINE AXIS: This indicate which axis that magazines of tool sets. This Function is active if tool change parameter is “ON”.

TOOL OPEN/CLOSE: Manuel Tool clamping function. This function is toggle for unclamping 5 second and than clamping tool. This Function is active if tool change parameter is “**ON**”.

MACHINE ZERO X AXIS: It specifies machine zero in X Axis (+) or (-).

MACHINE ZERO Y AXIS: It specifies machine zero in Y Axis (+) or (-).

AXIS TYPE: X Axis reduction type.

MAX SPINDLE RPM: Maximum spindle revolution per minute that machine have.

EXIT : Aborting from Parameters Page.

File name: DSL0T2.TOR Start no: 7 End no: 32 CAMLineNo: 0
F1Param.F2T_OffsF3AbsP0sF4G5x F5 F6PrgGetF7 F8T_Rst F9Turk F10Menu
Menu: 2



T Offs: Open the Tool Offsets Page. Also the shortcut “O” key opens this page with keyboard.

TOOL OFFSETS	RefNum	Switch Offset	Adds Offset
	# 1	-661.536	-2.000
	# 2	0.000	0.000
	# 3	-696.414	-2.000
	# 4	0.000	-0.298
	# 5	0.000	0.000
	# 6	0.000	0.000
	# 7	0.000	0.000
	# 8	58.000	0.000
	# 9	0.000	0.000
	#10	0.000	553.000
	#11	0.000	0.000
	#12	0.000	0.000
	#13	0.000	0.000
	#14	58.000	0.000
	#15	0.000	0.000
	#16	0.000	0.000
	#17	0.000	0.000
	#18	0.000	0.000
	#19	0.000	0.000
	#20	0.000	0.000
	#21	0.000	0.000
	#22	0.000	0.000
	#23	0.000	0.000
	#24	0.000	0.000
	#25	0.000	0.000

EXIT (ESC)

RefNum: Indicates Tool reference number.

Switch Offset: Tool Offset.

Adds Offset: Tool additional offset.

You can use tool offset numbers between 1 and 99. You can assign, change and list switch offsets and tool additional offset with the PageUp, PageDown keys.

Assign Switch Offset: If the machine have get tool switch equipments. Press **F2T_Offs** function key at **ElKont** menu that take the offset value to the tool offset number. This offset must be settled before this operation. User can be set this **H#** with **MDI** by hand. You can check this number before the taking offset from right bottom side off screen. Automatically assigning tool offset can be used with any tool and you can see new offset value in **F2T_Offs** menu. After this operation if you want assign this offset value another tool offset Numbers; Enter this menu and left click the Mouse **RefNum**. Another

way for this process located the highlighted area to the **RefNum**. cell with keyboard keys **PageUp-PageDown** and press **ENTER** to assign.

TOOL OFFSETS	RefNum	Switch Offset	Adds Offset
	# 1	-661.536	-2.000
	# 2	0.000	0.000
	# 3	-696.414	-2.000
	# 4	0.000	-0.298
	# 5	0.000	0.000
	# 6	0.000	0.000
	# 7	0.000	0.000
	# 8	58.000	0.000
	# 9	0.000	0.000
	#10	0.000	553.000
	#11	0.000	0.000
	#12	0.000	0.000
	#13	0.000	0.000
	#14	58.000	0.000
	#15	0.000	0.000
	#16	0.000	0.000
	#17	0.000	0.000
	#18	0.000	0.000
	#19	0.000	0.000
	#20	0.000	0.000
	#21	0.000	0.000
	#22	0.000	0.000
	#23	0.000	0.000
	#24	0.000	0.000
	#25	0.000	0.000

EXIT (ESC)

Finally if user want to Enter Tool Offset or Additional offset by manually left click the Mouse at correct **Switch Offset** and **Adds Offset** box then user can be enter new values of offset.

You can also see this operation left side #3 numbered. **Switch Offset** chosen and open textbox. User can enter this offset value manually then warning box opened to assigning press **ENTER** and **ESC** to cancel like below.

TOOL OFFSETS	RefNum	Switch Offset	Adds Offset
	# 1	-661.536	-2.000
	# 2	0.000	0.000
	# 3	-696.414	-2.000
	# 4	0.000	0.000

Offset is Assigned? Enter/ESC

File name: DSL0T2.TOR Start no: 7 End no: 32 CAMlineNo: 0
F1Paran.F2T_OffsF3AbsPOsF4G5x F5 F6PrGGetF7 F8T_Rst F9Turk F10Menu
Menu: 2

F3 **AbsPOs:** Shows the actual position of the axis with respect to machine zero point.

```

aposx 0.000
aposy 0.000
aposz 0.000
aposu 0.000
Press a key.

```

F4 **G5x:** For assigning workpiece zeros before using this function user must be use **Menu 1 F7MSF** for getting machine zero point (X, Y, Z). There are 5 workpiece zero to use (54, 55, 56, 57, 58). In this page indicates and assigns the workpiece references by automatically and manually.

OTOMATIK		H	X	Y	Z	U	
MANUEL	G54	1	0.000	0.000	0.000	0.000	ALL
CIKIS	G55	1	480.008	767.792	-152.000	0.000	ALL
	G56	3	240.004	383.896	-628.176	0.000	ALL
	G57	3	480.008	767.792	-628.176	0.000	ALL
	G58	1	480.238	561.640	-628.176	0.000	ALL

For automatic Assigning : Select automatic cell and pres All cell by mouse or keyboard. The messagebox appears "G54 reference Assign ? ENTER / ESC". If pres ENTER assigning complete or ESC to cancel operation.

G54 Reference is Assigned? Enter/ESC							
OTOMATIK		H	X	Y	Z	U	
MANUEL	G54	1	0.000	0.000	0.000	0.000	ALL
CIKIS	G55	1	480.008	767.792	-152.000	0.000	ALL
	G56	3	240.004	383.896	-628.176	0.000	ALL
	G57	3	480.008	767.792	-628.176	0.000	ALL
	G58	1	480.238	561.640	-628.176	0.000	ALL

If user want to assign unique axis select one of the X, Y, Z, U cell.

For Manuel Assigning: Select the MANUEL cell then select one of the X, Y, Z, U then enter the new value of corresponding axis by manual this value entered by absolute or incremental. Incremental means corresponding machine axis simultaneous value + entered new value.

OTOMATİK		H	X	Y	Z	U	
MANUEL	G54	1	0.000	0.000	0.000	0.000	ABSOLUTE
ÇIKIŞ	G55	1	480.008	767.792	250	0.000	ABSOLUTE
	G56	3	240.004	383.896	-628.176	0.000	ABSOLUTE
	G57	3	480.008	767.792	-628.176	0.000	ABSOLUTE
	G58	1	480.238	561.640	-628.176	0.000	ABSOLUTE



GetPrg : When this function key is pressed the following menu appears.

File name: TSG81.TOR	Start no: 0	End no: 13	CAMlineNo: 0
F1Param.F2T_OffsF3AbsPOsF4G5x	F5	F6PrgGetF7	F8T_Rst F9Turk F10Menu
drawing.exe		NEW?	Yes/ENTER/ESC Menu:2

“Y” To call a program with a new path name.

“ENTER” To accept already existing path written in memory.

“ESC” To cancel the operation.

NOTE 1 : Path name up to 30 characters is possible

2 : If you enter “P” shortcut key, you can reach last entered program.



T_Rst: Reset the Timer.

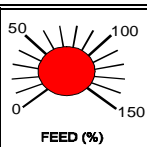
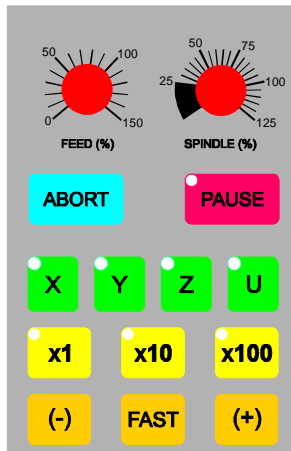


Turk: Changes program language to Turkish.

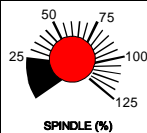


Menü: Toggles the menu.

OPERATION OF HAND CONTROL UNIT



FEED Percent override of the entered feed value by **F** during programming.



SPINDLE : Percent override of the **SPINDLE** entered by **S** during programming.



PAUSE : Serves to stop the machines axis movements temporarily. Pressing this button again will resume the motion.



ABORT : This button serves to stop the operation of machining and motion, when **PAUSE** button is active. It also serves to exit from the manual positioning mode.














It is selected in manual positioning mode to move **X** axis by pressing **(-)** and **(+)** button the axis can be moved to desired position.



It is selected in manual positioning mode to move **Y** axis by pressing **(-)** and **(+)** button the axis can be moved to desired position.



It is selected in manual positioning mode to move **Z** axis by pressing **(-)** and **(+)** button the axis can be moved to desired position.

	It is selected in manual positioning mode to move U axis by pressing  and  button the axis can be moved to desired position.
	It causes axis to be moved by 1 micrometer increments in jogging mode while manual positioning. When active the LED will be illuminated. Pressing again the same button or one of the other two jog buttons will disable the function of this button
	It causes axis to be moved by 10 micrometer increments in jogging mode while manual positioning. When active the LED will be illuminated. Pressing again the same button or one of the other two jog buttons will disable the function of this button
	It causes axis to be moved by 100 micrometer increments in jogging mode while manual positioning. When active the LED will be illuminated. Pressing again the same button or one of the other two jog buttons will disable the function of this button
	Used to position axis in “ - ” direction while machine is in manual control mode or decrement the selected values while machining.
	Used to position axis rapidly while machine is in manual control mode when pressed together with  or  buttons.
	Used to position axis in “+” direction while machine is in manual control mode or increment the selected values while machining.

G PREPARATORY AND AUXILIARY FUNCTIONS USED IN AJAN CNC MILLING MACHINE

NAME	EXPLANATION
G0	Rapid traverse linear move.
G1	Linear move with the programmed speed F .
G2	Clock wise circular interpolation. (CW)
G3	Country clock wise circular interpolation (CCW)
G37	Used to call files with DXF extension. First type G37 then press space then type in the 8 character file name. Do not type extension.
G38	Used to call tool path files with CAM extension. First type G38 then press space then type in the 8 character file name. Do not type extension.
G40	Cancels tool radius compensation
G41	Tool radius compensation to the left
G42	Tool radius compensation to the right.
G43	Indicates Tool Numbers and Tool Offset Numbers Sample. G43 T2 H2
G50	Cancels mirror image effected mode
G51	G51 X_x Y_y I_ip J_jp _x : Center coordinate value of X axis _y : Center coordinate value of Y axis _ip : indicate make mirror image effect on X axis as to sign of the ip _jp : indicate make mirror on Y axis as to sign of the jp.
G54-G55 G56-G57- G58	G54 : First work pieces zero point. G55 : Second work pieces zero point. G56 : Third work pieces zero point. G57 : Forth work piece zero point. G57 : Fifth work piece zero point.
G59	Work piece zero point shown by the actual axis indicator positions. F5 will reset positions.
G62	Cancels threading mode.
G63	Threading mode. When this code is issued spindle axis (if any) will become as rotary axis for threading and rotary axis will behave like spindle controlled by S address in program blocks.
G72	when machining DXF files this code causes electrode retraction when the end point of the first contour does not coincide with the start point of the next contour Example: G72 Z2 causes electrode to lift Z axis to absolute 2 mm position
G75	Disables G72 and discontinuous contours are connected to each other. Care must be taken when original drawing has large discontinuous separations.

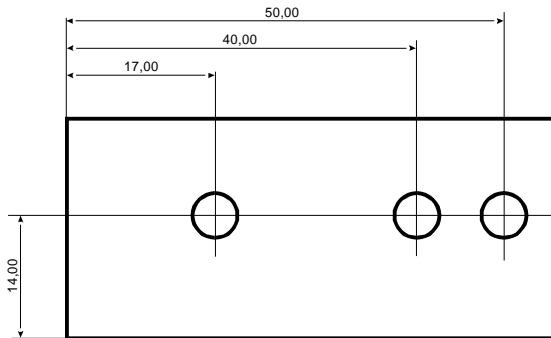
NAME	EXPLANATION
G80	G80 cancels canned cycles.
G81	<p>This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. The tool is then retracted from the bottom of the hole in rapid traverse.</p> <p>G81 X_Y_Z_R_F_K X_Y_ : Hole position data Z_ : The distance from point R to the bottom of the hole R_ : The distance from the initial level to point R level F_ : Cutting feedrate K_ : Number of repeats</p>
G82	<p>This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. At the bottom, a dwell is performed, then the tool is retracted in rapid traverse. This cycle is used to drill holes more accurately with respect to depth.</p> <p>G82 X_Y_Z_R_P_F_K X_Y_ : Hole position data Z_ : The distance from point R to the bottom of the hole R_ : The distance from the initial level to point R level P_ : Dwell time at the bottom of a hole F_ : Cutting feedrate K_ : Number of repeats</p>
G83	<p>This cycle perform peck drilling. It performs intermittent cutting feed to the bottom of a hole while removing shaving from the hole.</p> <p>G83 X_Y_Z_R_Q_F_K X_Y_ : Hole position data Z_ : The distance from point R to the bottom of the hole R_ : The distance from the initial level to point R level Q_ : Depth of cut for each cutting feed F_ : Cutting feedrate K_ : Number of repeats</p>
G90	Absolute coordinate system
G91	Incremental coordinate system
G201	2 dimensional drawings by using the Autocad are used to interpolate in program. The procedure of this code: Dxf extensions file is call by the subprogram with working G91 code. Than repeating this subprogram in z interpolation direction process can be done. G200 code cancels G201 code.
M0	When this code is encountered in a program block the machine will stop until operator hits a key.
M3	Clock wise spindle (CW) orientation
M4	Counter clock wise spindle (CCW) orientation
M5	Spindle stop.
M8	Open coolant system.
M9	Close coolant system
M17	<p>Indicates the end block for the subroutine L: Indicates the start of the block that ha to be repeated. P: Indicates the subroutine repeat number. EXAMPLE: L10 P3 This will call a block starting t line 10 and repeats the part until M17 is written 3 times. IMPORTANT NOTE: Subroutines are written always outside the main program blocks. (i.e they must be written after the program end number.4 nested subroutines are possible.</p>
M30	Program stop.

YARDIMCI FONKSİYON KODLARI:

İSİM	AÇIKLAMA
FD	Speed that is first approach to special drill.
ZF	Approach distance to special drill.
ZD	Total depth of drilled.
P	Number of holes for special drill.
F	Defines the feed rate in mm/min for X, Y, Z, U axis.
HM	Code for High-Speed-Machining.
HN	Cancel High-Speed-Machining
T	Takım numarası girmek için kullanılır.
H	Takım ofset numarası girmek için kullanılır. (G43 komutu ile kullanılmalıdır. Aksi Halde Çalışmaz.) Örnek: G43 H2
U	<p>This is a parameter of threading. There is a formulation of this processing for ease to use.</p> $U = 360 \times \frac{\text{Depth (Z)}}{\text{STEP}}$ <p>Depth : Threading length STEP : Step of threading</p>
SN	It represents the dwell time. End of the programmed contour delay process by writing related program block.
S	When this code is issued the spindle axis (if any) will rotate with the programmed speed. If U axis is used as spindle axis with G63 code the axis motor will rotate with the programmed speed.

SAMPLES PROGRAMS

Three drill and preparing subprogram



```

000 S1000 F2000
001 G59 X0 Y0 Z0
002 G0 X17 Y14
003 L9 P1
004 X40
005 L9 P1
006 X50
007 L9 P1
008 S0 X0 Y0 Z0
009 F500 Z-10
010 G0 Z0
011 M17

```

Start and end lines.

Subprogram.

End of subprogram..

Threading above prepared holes.

Threading to 10 mm depth with M3x0,5 :

Z = 10 mm

STEP = 0,5

$$U = 360 \times \frac{\text{DEPTH (in Z)}}{\text{STEP}}$$

$$U = 360 \times \frac{10}{0.5}$$

$$U = 360 \times 20$$

$$U = 7200$$

```

000 S1000 F2000
001 G59 X0 Y0 Z0
002 G0 X17 Y14
003 L9 P1
004 X40
005 L9 P1
006 X50
007 L9 P1
008 S0 X0 Y0 Z0
009 U7200 Z-10
010 U0 Z0
011 M17

```

Start and end lines.

Threading codes..

End of subprograms..

NOTE:

Above programs can be picking up in one program to process. Explanations of program can be written to Entry of program. First of all start line and end line entered by F8 in main menu (menu 0). End of drilled second program lines entered same manner.

```
000 DRILL PROGRAM
001 S1000 F2000
002 G59 X0 Y0 Z0
003 G0 X17 Y14
004 L9 P1
005 X40
006 L9 P1
007 X50
008 L9 P1
009 S0 X0 Y0 Z0
010 F500 Z-10
011 G0 Z0
012 M17
013 THREADING PROGRAM
014 S1000 F2000
015 G59 X0 Y0 Z0
016 G0 X17 Y14
017 L9 P1
018 X40
019 L9 P1
020 X50
021 L9 P1
022 S0 X0 Y0 Z0
023 U7200 Z-10
024 U0 Z0
025 M17
```

Start and end lines.

Subprogram.

End of subprogram..

Start and end lines.

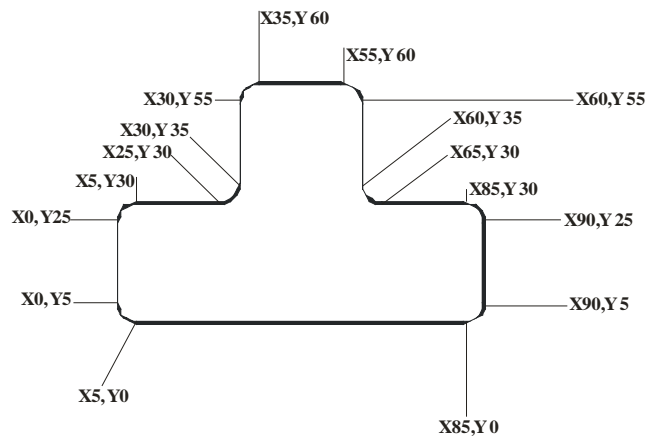
Threading subprogram.

End of subprogram..

```

000 S1000 F2000 G90
001 G59 X5 Y0
002 M8 F100 G1 X85
003 G3 J5 I0 X90 Y5
004 G1 Y25
005 G3 J0 I-5 X85 Y30
006 G1 X65
007 G2 J5 I0 X60 Y35
008 G1 Y55
009 G3 I-5 J0 X55 Y60
010 G1 X35
011 G3 J-5 I0 X30 Y55
012 G1 Y35
013 G2 I-5 J0 X25 Y30
014 G1 X5
015 G3 J-5 I0 X0 Y25
016 G1 Y5
017 G3 I5 J0 X5 Y0
018 M9
019 G1 X0 Y0 S0

```



Sketching figures in above can be done in drawing program and can be called in this way:

```

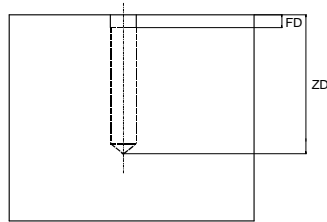
000 RADIUS PROGRAM
001 G90 S1000 F200
002 G59 X0 Y0 X0
003 G37 RADIUS
004 S0

```

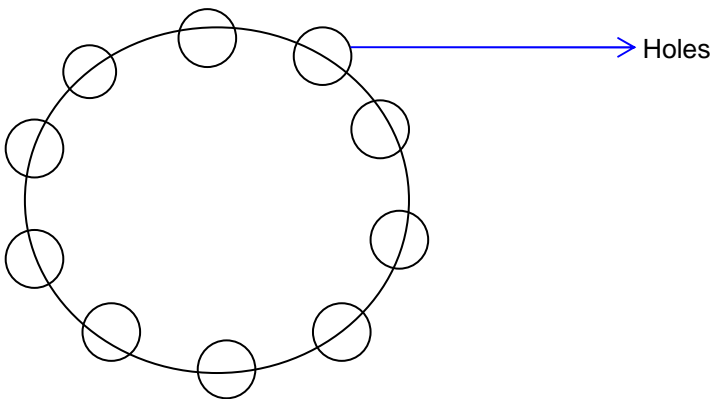
Special drill programs.

One of ease AJAN CNC MILLING is: desired number of drill in radial orbit can be done in one program line.
There is an addition function code:

FD : First approach speed
ZF : First approach distance
ZD : Total depth
P : Total hole number.



Machine in a FD speed (in slow speed) drill to ZF depth the returns to the beginning of the contour. Finally with original F speed drills to the ZD depth this process continue for all holes in a radial orbit.



Example:

```
001 F2000 G59 X0 Y0
002 G0 Z0
003 X20
004 F500 G3 I-20 FD150 ZF-1 P10 ZD-5
005 G0 Z2
006 X0 Y0
007 Z0
```

CANNED CYCLES

G81: This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. The tool is then retracted from the bottom of the hole in rapid traverse.

G81 X_Y_Z_R_F_K

X_Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
F_ : Cutting feedrate
K_ : Number of repeats

G82: This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. At the bottom, a dwell is performed, and then the tool is retracted in rapid traverse. This cycle is used to drill holes more accurately with respect to depth.

G82 X_Y_Z_R_P_F_K

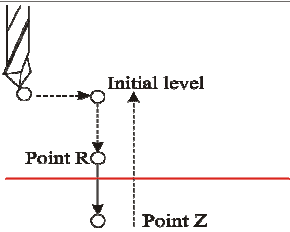
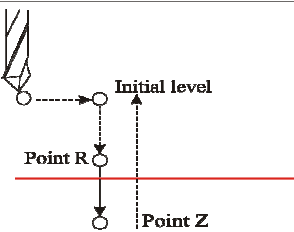
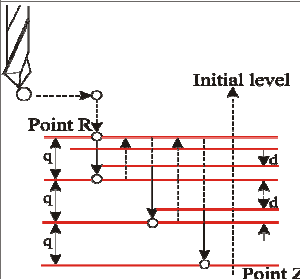
X_Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time at the bottom of a hole
F_ : Cutting feedrate
K_ : Number of repeats

G83: This cycle perform peck drilling. It performs intermittent cutting feed to the bottom of a hole while removing shaving from the hole.

G83 X_Y_Z_R_Q_F_K

X_Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
Q_ : Depth of cut for each cutting feed
F_ : Cutting feedrate
K_ : Number of repeats

Examples are indicates usage of the G81, G82, G83 codes and its application.

G81 code	G82 code	G83 code
	 <p>Dwell time (P) at the bottom of a hole</p>	
<pre> 000 F200 Z50 001 G90 G0 S200 M3 002 G81 Z-25 R5 F500 K1 003 X50 Y50 004 X100 Y50 005 X150 Y50 006 X150 Y100 007 X100 Y100 008 X50 Y100 009 G80 010 S0 </pre>	<pre> 000 F200 Z50 001 G90 G0 S200 M3 002 G82 Z-25 R5 P2000 F500 K1 003 X50 Y50 004 X100 Y50 005 X150 Y50 006 X150 Y100 007 X100 Y100 008 X50 Y100 009 G80 010 S0 </pre>	<pre> 000 F200 Z50 001 G90 G0 S200 M3 002 G83 Z-25 R5 Q1 F500 K1 003 X50 Y50 004 X100 Y50 005 X150 Y50 006 X150 Y100 007 X100 Y100 008 X50 Y100 009 G80 010 S0 </pre>

Figure's shown below output of the above examples in one sketch because of all of codes do same drilling cut. But manner is different in finally. There are 6 holes and depth is 25 mm all of the programs outputs.

