6 Program

In the main page, press F2 to Program Page.

In the program page, the users can browse the internal memory of the controller,a nd the USB Stick, or the Net Disk when the enternet is built up.

In the Program Page, the users can Delete a file, Rename a file, Create a file, Copy and Paste a file;

In the Program Page, the users can edit a file with the vitual keyboard;

In the Program Page,the users can copy a file from USB-stick / Net Disk to Local, or Copy a file from Local to USB-stick / Net Disk.

In the Program Page, the users can simulate a G-code file, only to preview toolpath, without sending any pulse.

Figure 6-1 Press F2 To Program Page

In the first Program Page, press F1 (Switch Disks) , System will swtich between the Local and USB Disk / Net Disk. Please note that, if Ethernet build up and controller can communicate the computer, then the system only can swtich between the Local and Net Disk;

Press F2 (Del) , the System will delete the current file;

Press F3 (Rename) , we can rename the file by the panel keyboard or by external USB keyboard.

Press F4 (Copy To U Disk) , the system can copy the current file from Local to USB-Stick;If in the U Disk,this column will be “ Copy To Local ” , then can quickly copy file from USB-Stick to Local.

Press F5 (New) , the system will create a new “ .nc ” file;

Press F6 (Edit) , the system can open the current file, and on the right page, pop up a vitual keyboard to edit. The vitual keyboard usage,please refers to Chapter 5.1.10.

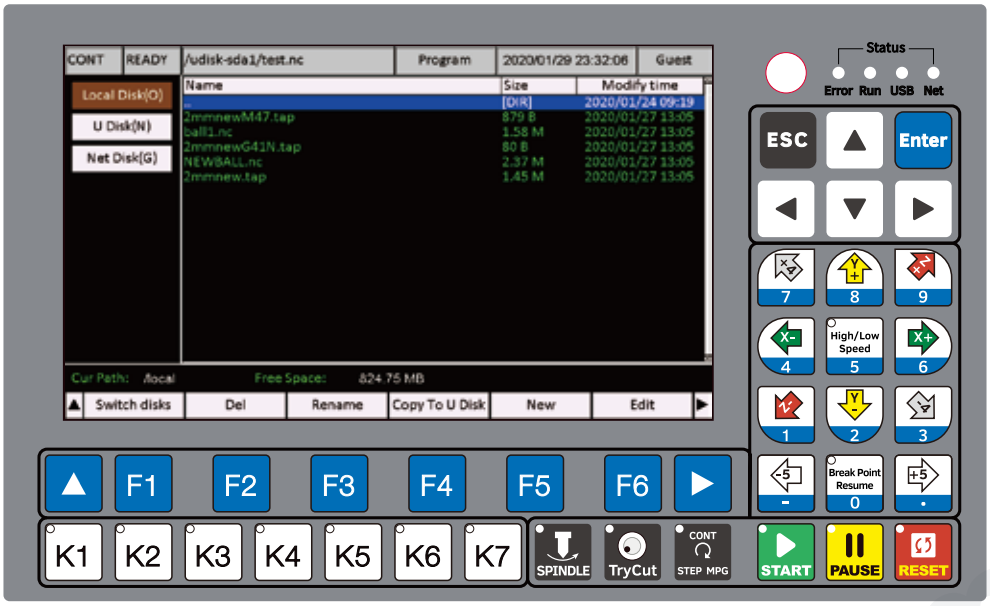


Figure 6-2 First Sub-Page of Program Page

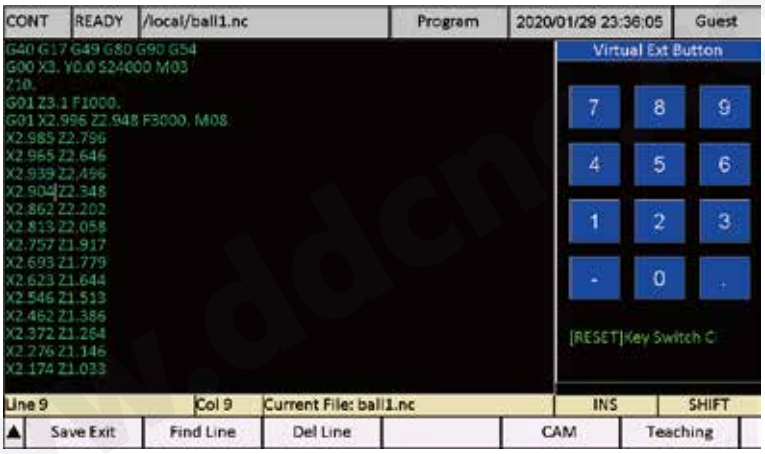


Figure 6-3 Edit a program by the vitual keyboard

Press  to the second sub-page of Program Page.

In the second Program Page, press F1 (Copy) , System copy the current file ;

Press F2 (Paste) , the System will Paste the current file;

Press F3 (Simulate) , the system will simulate the current file, just for users to preview the toolpath, the control system don' t send any commands;

Press F4 (Load NC) , the system load the current file;

Press F5 (Clear Local) , the system will delete all the files or folders in the Local,the Local memory is empty.

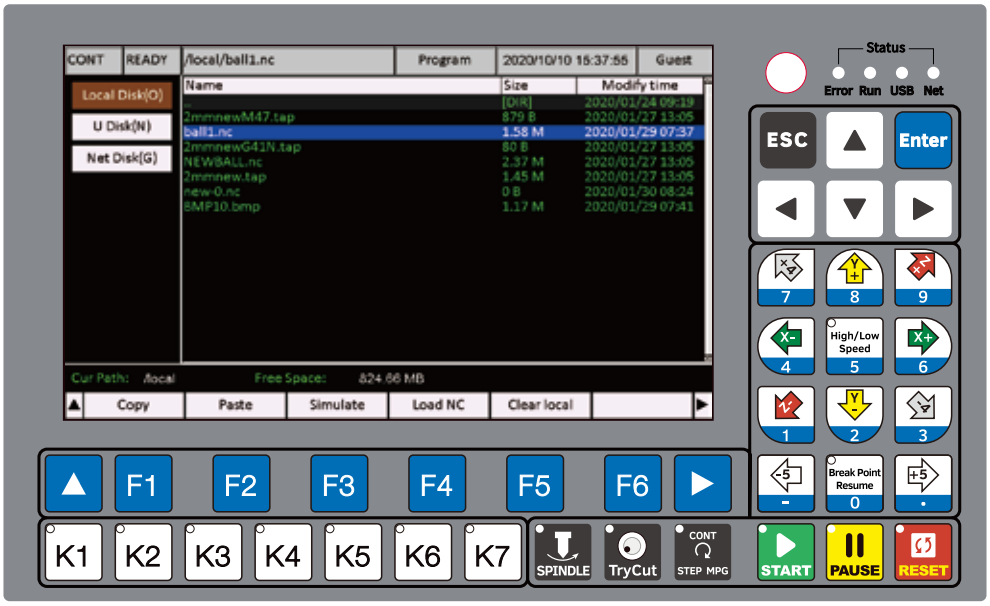


Figure 6-4 Second Sub-Page of Program Page

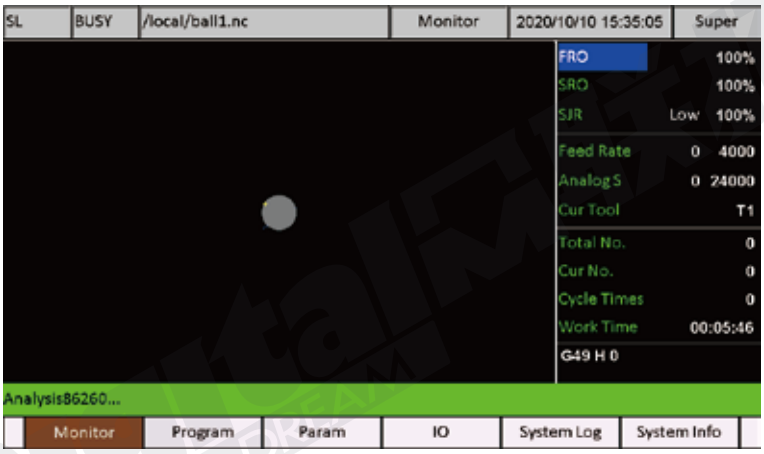


Figure 6-5 Preview the toolpath by Simulating a file

7 Parameters

In the main page, press F3 to Parameters Page.

All the parameters setting are in this page.

Figure 7-1 Press F3 to Parameters Page

7.1 Parameters List and Details

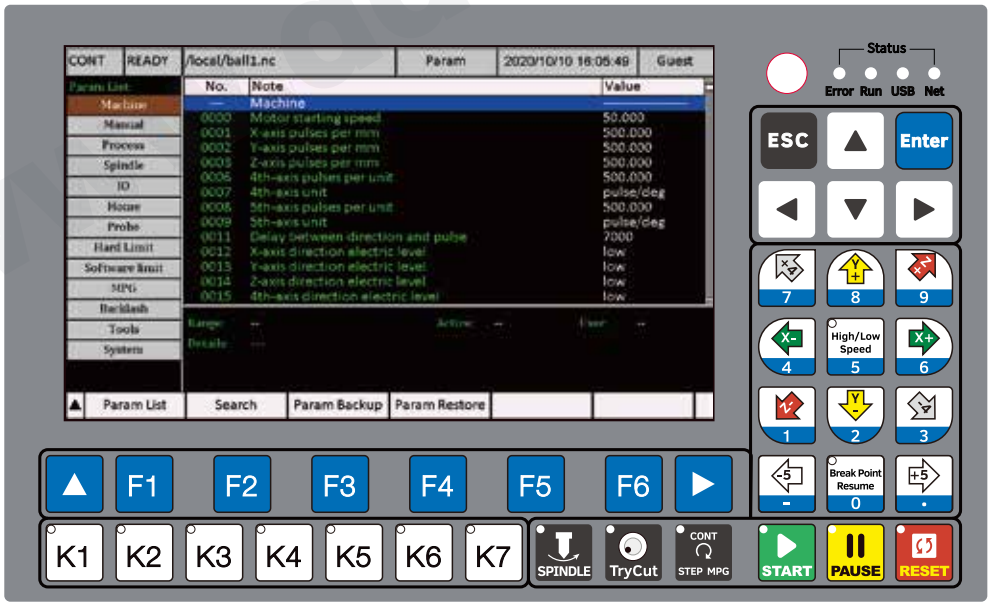


Figure 7-2 Parameters Page

In the Parameters Page,there are 13 kinds parameters. The users can view the parameters one by one very easily by rotary button , up ▲ or ▼ down keys, and the left key ◀ or right key ▶ are for the kinds switching.

1) Machine (Totally 23 items)

No.	Parameter definition	Default value	Range and Unit	User
#0	Motor starting speed	50	1~999, mm/min	Operator
	If the given speed is higher than this speed, the motor will start to accelerate from this speed, or the motor will run at given speed.			
#1	X-axis pulses per mm	500	50~99999.000, pulse/mm	Operator
	When the axis is used to drive the spindle, the unit of this /parameter is the number of pulses per revolution.			
#2	Y-axis pulses per mm	500	50~99999.000, pulse/mm	Operator
	When the axis is used to drive the spindle, the unit of this /parameter is the number of pulses per revolution.			
#3	Z-axis pulses per mm	500	50~99999.000, pulse/mm	Operator
	When the axis is used to drive the spindle, the unit of this /parameter is the number of pulses per revolution.			
#6	4th-axis pulses per unit	500	50~99999.000, pulse/mm	Operator
	When the axis is used to drive the spindle, the unit of this /parameter is the number of pulses per revolution.			
#7	4th-axis unit	pulse/deg	pulse/deg or pulse/circle	Operator
	When this axis is used to drive the spindle motor,set the parameter to " pulse/deg ".			
#8	5th-axis pulses per unit	500	50~99999.000	Operator
	When this axis is used to drive the spindle motor, the unit of this parameter is the number of pulses per revolution.			
#9	4th-axis unit	pulse/deg	pulse/deg or pulse/circle	Operator
	When this axis is used to drive the spindle motor,set the parameter to " pulse/deg ".			
#11	Delay between direction and pulse	7000	0~9999.000, ns	Operator
	The default value is 7000, which is suitable for most drivers.			
#12	X-axis direction electric level	Low	High / Low	Operator
	This parameter is used to set the direction of X-axis.			
#13	Y-axis direction electric level	Low	High / Low	Operator
	This parameter is used to set the direction of Y-axis.			
#14	Z-axis direction electric level	Low	High / Low	Operator
	This parameter is used to set the direction of Z-axis.			
#15	4th-axis direction electric level	Low	High / Low	Operator
	This parameter is used to set the direction of 4th-axis.			
#16	5th-axis direction electric level	Low	High / Low	Operator
	This parameter is used to set the direction of 5th-axis.			
#17	X axis Pulse signal Electric Level	Low	High / Low	Operator
	If the X axis gradually offset during machining, reverse this parameter.			
#18	Y axis Pulse signal Electric Level	Low	High / Low	Operator
	If the Y axis gradually offset during machining, reverse this parameter.			
#19	Z axis Pulse signal Electric Level	Low	High / Low	Operator
	If the Z axis gradually offset during machining, reverse this parameter.			
#20	4th axis Pulse signal Electric Level	Low	High / Low	Operator
	If the 4th axis gradually offset during machining, reverse this parameter.			
#21	5th axis Pulse signal Electric Level	Low	High / Low	Operator
	If the 5th axis gradually offset during machining, reverse this parameter.			
#443	4th-axis name	A	X/Y/Z/A/B/C	Admin
	After restart the controller,the new setting is active.			
#444	5th-axis name	B	X/Y/Z/A/B/C	Admin
	After restart the controller,the new setting is active.			
#449	4th-axis Type	A	Linear/Rotation	Admin
	The parameter define the 4th axis is Linear axis or rotation axis.			
#450	5th-axis name	B	Linear/Rotation	Admin
	The parameter define the 5th axis is Linear axis or rotation axis.			

2) Manual (Totally 30 items)

No.	Parameter definition	Default value	Range and Unit	User
#35	X-axis max. speed in manual mode	20000	99~99999, mm / min	Operator
	The X-axis Max. speed in Manual Mode,even with the effect by SJR. When the X axis is configured to servo spindle, the unit is revolution / min. This Parameter must be bigger than #40.			
#36	Y-axis max. speed in manual mode	20000	99~99999, mm / min	Operator
	The Y-axis Max. speed in Manual Mode,even with the effect by SJR. When the Y axis is configured to servo spindle, the unit is revolution / min. This Parameter must be bigger than #41.			
#37	Z-axis max. speed in manual mode	8000	99~99999, mm / min	Operator
	The Z-axis Max. speed in Manual Mode,even with the effect by SJR. When the Z axis is configured to servo spindle, the unit is revolution / min. This Parameter must be bigger than #42.			
#38	4th-axis max. speed in manual mode	6000	99~99999, deg / min	Operator
	The 4th-axis Max. speed in Manual Mode,even with the effect by SJR. When the 4th-axis is configured to servo spindle, the unit is revolution / min. This Parameter must be bigger than #43.			
#39	5th-axis max. speed in manual mode	6000	99~99999, deg / min	Operator
	The 5th-axis Max. speed in Manual Mode,even with the effect by SJR. When the 4th-axis is configured to servo spindle, the unit is revolution / min. This Parameter must be bigger than #44.			
#40	X-axis manual control HIGH speed	10000	1000~99999, mm / min	Operator
	When the axis is configured to servo spindle, the unit of this /parameter is rpm.			
#41	Y-axis manual control HIGH speed	10000	1000~99999, mm / min	Operator
	When the axis is configured to servo spindle, the unit of this /parameter is rpm.			
#42	Z-axis manual control HIGH speed	5000	1000~99999, mm / min	Operator
	When the axis is configured to servo spindle, the unit of this /parameter is rpm.			
#43	4th-axis manual control HIGH speed	3000	1000~99999, deg / min	Operator
	When the axis is configured to servo spindle, the unit of this parameter is rpm.			
#44	5th-axis manual control HIGH speed	4000	1000~99999, deg / min	Operator
	When the axis is configured to servo spindle, the unit of this /parameter is rpm.			
#45	X-axis manual control LOW speed	1000	1000~99999, mm / min	Operator
	When the axis is configured to servo spindle, the unit of this parameter is rpm.			
#46	Y-axis manual control LOW speed	1000	1000 ~ 99999, mm / min	Operator
	When the axis is configured to servo spindle, the unit of this /parameter is rpm.			
#47	Z-axis manual control LOW speed	1000	1000 ~ 99999, mm / min	Operator
	When the axis is configured to servo spindle, the unit of this parameter is rpm.			
#48	4th-axis manual control LOW speed	1000	1000 ~ 99999, deg / min	Operator
	When the axis is configured to servo spindle, the unit of this parameter is rpm.			
#49	5th-axis manual control LOW speed	2000	1000 ~ 99999, deg / min	Operator
	When the axis is configured to servo spindle, the unit of this /parameter is rpm.			
#50	X-axis start acceleration in manual mode	1000	9 ~ 9999, mm / s2	Operator
	When the axis is configured to servo spindle, the unit of this parameter is the number of revolution per s2.			
#51	Y-axis start acceleration in manual mode	1000	9 ~ 9999, mm / s2	Operator
	When the axis is configured to servo spindle, the unit of this parameter is the number of revolution per s2.			
#52	Z-axis start acceleration in manual mode	1000	9 ~ 9999, mm / s2	Operator
	When the axis is configured to servo spindle, the unit of this parameter is the number of revolution per s2.			
#53	4th-axis start acceleration in manual mode	600	9 ~ 9999, mm / s2	Operator
	When the axis is configured to servo spindle, the unit of this parameter is the number of revolution per s2.			
#54	5th-axis start acceleration in manual mode	600	9 ~ 9999, mm / s2	Operator
	When the axis is configured to servo spindle, the unit of this parameter is the number of revolution per s2.			

No.	Parameter definition	Default value	Range and Unit	User
#55	X-axis stop acceleration in manual mode	1000	9~9999, mm / s2	Operator
	When the axis is configured to servo spindle, the unit of this parameter is the number of revolution per s2.			
#56	Y-axis stop acceleration in manual mode	1000	9~9999, mm / s2	Operator
	When the axis is configured to servo spindle, the unit of this parameter is the number of revolution per s2.			
#57	Z-axis stop acceleration in manual mode	1000	9~9999, mm / s2	Operator
	When the axis is configured to servo spindle, the unit of this parameter is the number of revolution per s2.			
#58	4th-axis stop acceleration in manual mode	600	9~9999, mm / s2	Operator
	When the axis is configured to servo spindle, the unit of this parameter is the number of revolution per s2.			
#59	5th-axis stop acceleration in manual mode	600	9~9999, mm / s2	Operator
	When the axis is configured to servo spindle, the unit of this parameter is the number of revolution per s2.			
#285	X-axis max. ACC G00	1000	9~9999, mm / s2	Operator
	G00 command maximum acceleration.			
#286	Y-axis max. ACC G00	1000	9~9999, mm / s2	Operator
	G00 command maximum acceleration.			
#287	Z-axis max. ACC G00	1000	9~9999, mm / s2	Operator
	G00 command maximum acceleration.			
#288	4th-axis max. ACC G00	1000	9~9999, mm / s2	Operator
	G00 command maximum acceleration.			
#289	5th-axis max. ACC G00	1000	9~9999, mm / s2	Operator
	G00 command maximum acceleration.			

3) Process (Totally 26 items)

No.	Parameter definition	Default value	Range and Unit	User
#60	Speed Selection	Default	G Code / Default ;	Operator
	If #60 is set to Default,system will use the #61 speed value;If #60 set to G Code,system will use F command in the G-code file.This parameter can be quickly set by “Feed Rate” on the Main page.			
#61	default operation speed	3000	10~99999, mm/min	Operator
	If the G code file has no F command or #60 is set to Default, system will cite #61 as the feed rate.This parameter can be quickly set by “Feed Rate” on the Main page.			
#62	G01 ACC	500	9~9999, mm/s2	Operator
	G01 \ G02 \ G03 acceleration. This parameter should be set according to the actual situation of the machine.			
#63	G00 speed	10000	99~99999, mm/min	Operator
	By the Parameter,we can set the speed of G0 Command.			
#64	Maximum speed	12000	99~99999, mm/min	Operator
	Maximum speed of the machine during machining.			
#65	Z-axis lifting protection speed	99999	99~99999, mm/min	Operator
	Z++ maximum speed. G00 is also valid.			
#66	Z-axis dropping protection speed	99999	99~99999, mm/min	Operator
	Z-- maximum speed. G00 is also valid.			
#67	X-axis protection speed	99999	99~99999, mm/min	Operator
	X-axis protection speed. G00 is also valid.			
#68	Y-axis protection speed	99999	99~99999, mm/min	Operator
	Y-axis protection speed. G00 is also valid.			
#69	Z axis safe height	5	0~999 mm	Operator
	When starting or restoring machining and go to work zero, the Z axis will move to Z axis safety height.			

No.	Parameter definition	Default value	Range and Unit	User
#70	Z-axis retraction dist. when paused	3	0~99 mm/min	Operator
	Z lift distance, when paused.			
#72	G0 command motion characteristics	Independent	Interpolation / Independent	Operator
	Interpolation: Synergistic movement of each axis; Independent: each axis independently moves at G0 speed.			
#73	Arc-interpolation algorithm	0	Hard alg / Soft alg	Operator
	Hard alg. : Interpolation accuracy is 0.5 pulses. Soft alg. : Accuracy is set by parameter #74.			
#74	Soft-arc algorithm linear error	0	0.001 ~ 0.1, mm	Operator
	The precision of the Soft-arc Algorithm.			
#75	Circular centrifugal acceleration	0	0~9999, mm/s2	Operator
	Hard alg. : Interpolation accuracy is 0.5 pulses. Soft alg. : Accuracy is set by parameter #74.			
#76	Macro scan switch	0	closed / open	Operator
	Closed:do not scan file before processing; open: Scan files before processing; If the users set it to Open,system will asume a lot of time and calculation memory for scannning, please be careful to set the parameter.			
#77	Macro program file main program No.	0	0~9999	Operator
	In the Macro program,there will be a lot of the program number, so we need to assign a main program number.			
#90	Action selection before starting	No Action	No action / To Safety Z	Operator
	Here we set the Z axis movement when starting or resuming the controller; Safety height set by Param #69.			
#91	Z-axis movement mode during pause	No Action	No action / Z Distance	Operator
	Here we set the Z axis movement when pause the controller; Z-axis lifting distance set by Param #70.			
#220	Go to home before processing?	No	Yes / No	Operator
	A processing cannot be started without Go Home			
#221	Ref speed of arc with radius 5mm	0	0~3600000; mm/min	Operator
	The reference Arc Radius is 5mm; Other Arc speed please refers to this speed; If #221=0,The arc speed is related to parameters #62 and #75.			
#222	4th-axis protection speed	0	99~99999; mm/min	Operator
	4th-axis protection speed. G00 is also valid.			
#223	5th-axis protection speed	0	99~99999, mm/min	Operator
	5th-axis protection speed. G00 is also valid.			
#224	G73/G83 drilling retraction	0	0~20, mm	Operator
	G73 G83 drilling hole retraction distance.			
#230	Execute action after Finished	No Action	No action/Ref Pos/Work Zero	Operator
	Add M30 at the end of the file. Ref Pos: Mach pos of No.122-126.			
#282	G00 ACC	2000	0~9999, mm/s2	Operator
	Here we acceleration in G00 interpolation mode; If we set it to 0, G00 cite linear acceleration; If we set it other value except 0, each axis accleration is limited by " Axis max. ACC G00 " (#285 ~ #289).			

4) Spindle (Totally 9 items)

No.	Parameter definition	Default value	Range and Unit	User
#79	Spindle interface type	Analog	Analog/Plu&Dir/Multi-Speed	Operator
	3 kinds spindle interface mode,users can choose according to the usage.			
#80	Spindle mapping axis	4th Axis	X / Y / Z / 4th / 5th Axis	Operator
	When the Spindle interface type is Pul&Dir, this parameter decide which axis is the servo spindle axis; In the mode of Servo Spindle, all the related parameter Unit is adjusted to "Rotation" .			
#81	Spindle start delay	2	0~9; S	Operator
	Delay time after spindle start command (M03/M04) response.			
#82	Maximum spindle speed	24000	0~99999; rpm	Operator
	When the spindle is in Multi-Speed Spindle,this parameter and #88 decide the spindle output segment.			
#83	Ignore the S command	No	No / Yes	Operator
	Start or resume the controller, spindle speed adopts parameter #85; This parameter also can be quickly set in the Main page.			
#84	Stop spindle when program is paused?	Yes	No / Yes	Operator
	When controller paused, this parameter decide to stop the spindle or not.			
#85	Default spindle speed	24000	0~99999; rpm	Operator
	If there is no S command in the G-code file, or #82 is Yes, the spindle speed can adopt this value . This parameter can be quickly set on the Main page.			
#88	Multi-speed section count	8	2~8, S	Operator
	When the section is 2, please define the "Spindle section Speed" outport 1 ; When the section is 3 or 4, please define the "Spindle section speed" output 1 and 2; When the section is bigger than 4,please define "Spindle section speed" output 1 and 2 and 3.			
#89	Spindle stop delay	0	0~9, S	Operator
	Delay time after spindle stop command (M05) response.			

5) IO (Totally 17 items)

No.	Parameter definition	Default value	Range and Unit	User
#92	Duration of M8/M9 commands	2	Analog/Plu&Dir/Multi-Speed	Operator
	Delay time after cooling command response.			
#94	Duration of M10/M11 commands	2	0~9, S	Operator
	Delay time after lubrication command response.			
#95	IO input filter time width	50	0~1000000, ms	Operator
	This parameter helps the users to filter the electrical interference,to avoid the noise.			
#96	Reset IO Configuration bit 01-16	65535	0~65535	Operator
	We use decimal system to set the value; For Example, If OUT01~ OUT16 assigned to 1, then when reset, the current output port closed.			
#97	Reset IO Configuration bit 17-32	65535	0~65535	Operator
	We use decimal system to set the value; For Example, If OUT17~ OUT21 assigned to 1, then when reset, the current output port closed.			
#98	Alarm output status configuration bit 01-16	0	0~65535	Operator
	We use binary systemto to set the value; For example: 7=0111 / OUT0 OUT1 OUT2 output is Open after Alarm, or closed; By #264, the corresponding bit is configured as 1, then current output port enable status: 1: Enabled; 0: Disabled.			
#99	Alarm output status configuration bit 17-32	0	0~65535	Operator
	We use binary system to to set the value; For example: 7=0111 / OUT0 OUT1 OUT2 output is Open after Alarm, or closed; By #265, the corresponding bit is configured as 1, then current output port current status setting is : 1: Enabled; 0: Disabled.			

No.	Parameter definition	Default value	Range and Unit	User
#210	K1 key Function	1	0~2000	Operator
#211	K2 key Function	1	0~2000	Operator
#212	K3 key Function	1	0~2000	Operator
#213	K4 key Function	1	0~2000	Operator
#214	K5 key Function	1	0~2000	Operator
#215	K6 key Function	1	0~2000	Operator
#216	K7 key Function	1	0~2000	Operator
#217	K8 key Function	1	0~2000	Operator
	Note for K1 - K8: 0=run macro file " key-1.nc" ; 1-32=Close or Open OUT1-OUT32; >1000=define as Function shortcuts key, please contact factory fro details information for the details.			
#264	Alarm output enable configuration bit 01-16	0	0~65535	Operator
	We use binary system to set the value; For Example, If OUT1~ OUT16 assigned to 1, then when Alarm, the current enable status 1: Enable; 0: Disable; Before set parameter #98, we need to enable the corresponding bit first.			
#265	Alarm output enable configuration bit 17-32	0	0~65535	Operator
	We use binary system to set the value; For Example, If OUT17~ OUT21 assigned to 1, then when Alarm, the current enable status 1: Enable; 0: Disable; Before set parameter #99, we need to enable the corresponding bit first.			

6) HOME (Totally 28 items)

No.	Parameter definition	Default value	Range and Unit	User
#100	Home mode	Switch	Switch/Absolute	Admin
	Switch: Wire with Mechanical/Proximity limited Switch; Absolute: Bus absolute servo mode.			
#101	Servo absolute laps at the X-axis Home	0	-99999~99999; r	Operator
	The revolution when servo is in null position, floating type with direction.			
#102	Servo absolute laps at the Y-axis Home	0	-99999~99999; r	Operator
	The revolution when servo is in null position, floating type with direction.			
#103	Servo absolute laps at the Z-axis Home	0	-99999~99999; r	Operator
	The revolution when servo is in null position, floating type with direction.			
#104	Servo absolute laps at the 4th-axis Home	0	-99999~99999; r	Operator
	The revolution when servo is in null position, floating type with direction.			
#105	Servo absolute laps at the 5th-axis Home	0	-99999~99999; r	Operator
	The revolution when servo is in null position, floating type with direction.			
#106	Homing cycle count	3	1 ~ 5	Operator
	The repeated Home detection times.			
#107	X-axis homing speed	500	99~99999, mm/min	Operator
	The Initial speed When the X-axis go home.			
#108	Y-axis homing speed	500	99~99999, mm/min	Operator
	The Initial speed When the Y-axis go home.			
#109	Z-axis homing speed	500	99~99999, mm/min	Operator
	The Initial speed When the Z-axis go home.			
#110	4th-axis homing speed	500	99~99999, mm/min	Operator
	The Initial speed When the 4th-axis go home.			
#111	5th-axis homing speed	500	99~99999, mm/min	Operator
	The Initial speed When the 5th-axis go home.			

No.	Parameter definition	Default value	Range and Unit	User
#112	X-axis homing direction	Negative	Negative / Positive	Operator
	The initial moving direction When Home X-axis.			
#113	Y-axis homing direction	Negative	Negative / Positive	Operator
	The initial moving direction When Home Y-axis.			
#114	Z-axis homing direction	Positive	Negative / Positive	Operator
	The initial moving direction When Home Z-axis.			
#115	4th-axis homing direction	Negative	Negative / Positive	Operator
	The initial moving direction When Home 4th-axis.			
#116	5th-axis homing direction	Negative	Negative / Positive	Operator
	The initial moving direction When Home 5th-axis.			
#122	Mach position after X go home	5	-999~999; mm	Operator
	When X axis homing finished,system will excute G28 command, X axis move to the position this parameter set;			
#123	Mach position after Y go home	5	-999~999; mm	Operator
	When X axis homing finished,system will excute G28 command, Y axis move to the position this parameter set;			
#124	Mach position after Z go home	-5	-999~999; mm	Operator
	When X axis homing finished,system will excute G28 command, Z axis move to the position this parameter set;			
#125	Mach position after 4th go home	5	-999~999; mm	Operator
	When X axis homing finished,system will excute G28 command, 4th axis move to the position this parameter set;			
#126	Mach position after 5th go home	40	-999~999; mm	Operator
	When X axis homing finished,system will excute G28 command, 5th axis move to the position this parameter set;			
#127	Home after booting	Yes	Yes / No	Operator
	Yes: When power on the controller, system pop-up dialog box to ask Home System or not;			
#235	X-axis Mach zero offset	0	-999~999; mm	Operator
	We can reduce the error made by machine struction or any other factors by setting the offset for X axis.			
#236	Y-axis Mach zero offset	0	-999~999; mm	Operator
	We can reduce the error made by machine struction or any other factors by setting the offset for Y axis.			
#237	Z-axis Mach zero offset	0	-999~999; mm	Operator
	We can reduce the error made by machine struction or any other factors by setting the offset for Z axis.			
#238	4th-axis Mach zero offset	0	-999~999; mm	Operator
	We can reduce the error made by machine struction or any other factors by setting the offset for 4th axis.			
#239	5th-axis Mach zero offset	0	-999~999; mm	Operator
	We can reduce the error made by machine struction or any other factors by setting the offset for 5th axis.			

7) Probe (Totally 11 items)

No.	Parameter definition	Default value	Range and Unit	User
#128	Is the Floating tool set valid?	Yes	Yes/No	Operator
	Enable or Disable the Floating Probe			
#129	Floating tool set thickness	Yes	0 ~ 99; mm	Operator
	Before floating probe,we need to meansure out the sensor's thickness and set the #129.			
#130	Is the fixed tool set valid?	Yes	Yes/No	Operator
	Enable or Disable the Fixed Probe.			
#131	Probing cycle count	5	1-5	Operator
	The probe times.When the user active the Probe,the system can probe 1 - 5 times as what the users set. At last system calculate an average value.			
#132	Initial speed of Probing	150	50 - 99999; rpm	Operator
	The initial down speed of the Z axis after starting the tool setting.			
#135	Fixed probe X mach position	10	-9999 ~ 9999; mm	Operator
	The initial Position of X axis before Probe in Mach coordinate			
#136	Fixed probe Y mach position	10	-9999 ~ 9999; mm	Operator
	The initial Position of Y axis before Probe in Mach coordinate			
#137	Fixed probe Z mach position	10	-9999 ~ 9999; mm	Operator
	The initial Position of Z axis before Probe in Mach coordinate			
#138	Fixed probe 4th mach position	10	-9999 ~ 9999; mm	Operator
	The initial Position of 4th axis before Probe in Mach coordinate			
#139	Fixed probe 5th mach position	10	-9999 ~ 9999; mm	Operator
	The initial Position of 5th axis before Probe in Mach coordinate			
#140	Retraction distance after the end of probe	10	0 - 999; mm	Operator
	This parameter is relative.			

8) Hard Limit (Totally 5 items)

No.	Parameter definition	Default value	Range and Unit	User
#150	Stop mode when X-axis hard limit trigger	Emergency	Deceleration / Emergency	Operator
#151	Stop mode when Y-axis hard limit trigger	Emergency	Deceleration / Emergency	Operator
#152	Stop mode when Z-axis hard limit trigger	Emergency	Deceleration / Emergency	Operator
#153	Stop mode when 4th-axis hard limit trigger	Emergency	Deceleration / Emergency	Operator
#154	Stop mode when 5th-axis hard limit trigger	Emergency	Deceleration / Emergency	Operator

9) Software limit (Totally 15 items)

No.	Parameter definition	Default value	Range and Unit	User
#155	Enable software limits	Disable	Disable / Enable	Admin
	Total control switch for soft limit function of all axis; If the users want to disable single axis soft limits, just set the soft limit value of negative direction bigger than the limit value of positive direction.			
#156	Stop mode when X-axis software limit trigger	Emergency	Deceleration / Emergency	Operator
#157	Stop mode when Y-axis software limit trigger	Emergency	Deceleration / Emergency	Operator
#158	Stop mode when Z-axis software limit trigger	Emergency	Deceleration / Emergency	Operator
#159	Stop mode when 4th-axis software limit trigger	Emergency	Deceleration / Emergency	Operator
#160	Stop mode when 5th-axis software limit trigger	Emergency	Deceleration / Emergency	Operator
#161	Negative X-axis software limit	-9999	-9999~9999; mm	Operator
#162	Negative Y-axis software limit	-9999	-9999~9999; mm	Operator
#163	Negative Z-axis software limit	-9999	-9999~9999; mm	Operator
#164	Negative 4th-axis software limit	-9999	-9999~9999; mm	Operator
#165	Negative 5th-axis software limit	-9999	-9999~9999; mm	Operator
#166	Positive X-axis soft limit	9999	-9999~9999; mm	Operator
#167	Positive Y-axis soft limit	9999	-9999~9999; mm	Operator
#168	Positive Z-axis soft limit	9999	-9999~9999; mm	Operator
#169	Positive 4th-axis soft limit	9999	-9999~9999; mm	Operator
#170	Positive 5th-axis soft limit	9999	-9999~9999; mm	Operator

10) MPG (Totally 15 items)

No.	Parameter definition	Default value	Range and Unit	User
#171	Enable MPG Precision Control Mode	Disable	Enable / Disable	Operator
	If #171 = Enable, the system will store the pulses the wheels generated and send every single one out, so sometimes when the user stoped turning the wheel but machine axis will still move. This can lead to a crash; If #171 = Disable, when the user stoped turning the wheel the machine axis just immediately decelerate and stop.			
#172	MPG precision	0.004	0.001~0.01	Operator
	When the handwheel rate is X1, the distance one step of the wheel can move;			
#173	Enable ESTOP signal on MPG	Disable	Enable / Disable	Operator
	Enable or disable the reset function of the MPG.			
#174	Electric level of ESTOP on MPG	Low	Low / High	Operator
	Please set this parameter according to the actual MPG status.			
#175	MPG handwheel direction	Positive	Positive/ Negative	Operator
#176	Handwheel X1 speed	50	50~99999	Operator
	When the MPG speed Mode is on X1, the axis moving speed;			
#177	Handwheel X10 speed	50	50~99999	Operator
	When the MPG speed Mode is on X10, the axis moving speed;			
#178	Handwheel X100 speed	50	50~99999	Operator
	When the MPG speed Mode is on X100, the axis moving speed;			
#179	Handwheel stop adjustment increment value	0.05	0.001~0.5	Operator
	In handwheel guiding mode, stop turnning the wheel, the deceleration adjustable increment value.			
#180	Handwheel change adjustment increment value	0.01	0.001~0.5	Operator
	In handwheel guiding mode, the deceleration or acceleration adjustable increment value when turnning the wheel.			
#181	X-axis hand wheel manual Acc	50	9~9999; mm/s2	Operator
#182	Y-axis hand wheel manual Acc	50	9~9999; mm/s2	Operator
#183	Z-axis hand wheel manual Acc	50	9~9999; mm/s2	Operator
#184	4th-axis hand wheel manual Acc	50	9~9999; mm/s2	Operator
#185	5th-axis hand wheel manual Acc	50	9~9999; mm/s2	Operator
	In MPG mode, the start or stop acceleration of each axis.			

11) Backlash (Totally 15 items)

No.	Parameter definition	Default value	Range and Unit	User
#190	Enable X-axis reverse direction backlash	Disable	Disable / Enable	Operator
	When X axis reverse direction backlash enabled, if X axis change the direction, the system will compensate the backlash distance (#195) automatically.			
#191	Enable Y-axis backlash	Disable	Disable / Enable	Operator
	When Y axis reverse direction backlash enabled, if X axis change the direction, the system will compensate the backlash distance (#196) automatically.			
#192	Enable Z-axis backlash	Disable	Disable / Enable	Operator
	When Z axis reverse direction backlash enabled, if X axis change the direction, the system will compensate the backlash distance (#197) automatically.			
#193	Enable 4th-axis backlash	Disable	Disable / Enable	Operator
	When 4th axis reverse direction backlash enabled, if X axis change the direction, the system will compensate the backlash distance (#198) automatically.			
#194	Enable 5th-axis backlash	Disable	Disable / Enable	Operator
	When 5th axis reverse direction backlash enabled, if X axis change the direction, the system will compensate the backlash distance (#199) automatically.			
#195	X-axis backlash distance	0	0~9.999; mm	Operator
#196	Y-axis backlash distance	0	0~9.999; mm	Operator
#197	Z-axis backlash distance	0	0~9.999; mm	Operator
#198	4th-axis backlash distance	0	0~9.999; mm	Operator
#199	5th-axis backlash distance	0	0~9.999; mm	Operator
#200	Backlash speed	0	0~99999; mm/min	Operator
	If the current speed is less than parameter #0, then the Backlash speed is #0 parameter.			
#400	H01 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 1 (H1), the compensation value; G43\G44 H01.			
#401	H02 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 2 (H2), the compensation value; G43\G44 H02.			
#402	H03 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 3 (H3), the compensation value; G43\G44 H03.			
#403	H04 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 4 (H4), the compensation value; G43\G44 H04.			
#404	H05 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 5 (H5), the compensation value; G43\G44 H05.			
#405	H06 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 6 (H6), the compensation value; G43\G44 H06.			
#406	H07 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 7 (H7), the compensation value; G43\G44 H07.			
#407	H08 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 8 (H8), the compensation value; G43\G44 H08.			
#408	H09 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 9 (H9), the compensation value; G43\G44 H09.			
#409	H10 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 10 (H10), the compensation value; G43\G44 H10.			
#410	H11 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 11 (H11), the compensation value; G43\G44 H11.			
#411	H12 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 12 (H12), the compensation value; G43\G44 H012.			

No.	Parameter definition	Default value	Range and Unit	User
#412	H13 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 13 (H13), the compensation value; G43\G44 H013.			
#413	H14 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 14 (H14), the compensation value; G43\G44 H014.			
#414	H15 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 15 (H15), the compensation value; G43\G44 H015.			
#415	H16 tool length offset	0	-999.999~999.999; mm	Operator
	When the tool length compensation number is 16 (H16), the compensation value; G43\G44 H016.			
#420	D01 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 1 (D1), the compensation value; G41\G42 D01.			
#421	D02 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 2 (D2), the compensation value; G41\G42 D02.			
#422	D03 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 3 (D3), the compensation value; G41\G42 D03.			
#423	D04 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 4 (D4), the compensation value; G41\G42 D04.			
#424	D05 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 5 (D5), the compensation value; G41\G42 D05.			
#425	D06 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 6 (D6), the compensation value; G41\G42 D06.			
#426	D07 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 7 (D7), the compensation value; G41\G42 D07.			
#427	D08 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 8 (D8), the compensation value; G41\G42 D08.			
#428	D09 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 9 (D9), the compensation value; G41\G42 D09.			
#429	D10 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 10 (D10), the compensation value; G41\G42 D10.			
#430	D11 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 11 (D11), the compensation value; G41\G42 D011.			
#431	D11 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 12 (D12), the compensation value; G41\G42 D012.			
#432	D11 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 13 (D13), the compensation value; G41\G42 D013.			
#433	D11 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 14 (D14), the compensation value; G41\G42 D014.			
#434	D11 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 15 (D15), the compensation value; G41\G42 D015.			
#435	D11 tool Radius offset	0	-999.999~999.999; mm	Operator
	When the tool radius compensation number is 16 (D16), the compensation value; G41\G42 D016.			

No.	Parameter definition	Default value	Range and Unit	User
#800	Current tool No.	1	0~20	Operator
	When Tool number greater than 20 then it is the virtual tool number.			
#801	Total number of tools in Magazine	12	0~20	Operator
	The actual magazine capacity should be less than 20.			
#802	Tool magazine type	NULL	NULL/Multiple/Fixed row/Servo disc	Operator
	Tool Magazine type selection: Support multiple、Follow row、Fixed row、Servo disc etc.			
#803	The virtual Tool function turned on?	No	No / Yes	Admin
	Enable Virtul tool. When the tool no. is over 20, the system execute as it is the virtual tool.			
#805	Automatic tool setting after tool change?	No	No / Yes	Operator
	Automatic Probe after tool change or not.			
#806	The highest position when chang Tool	0	-9999.999~9999.999; mm	Operator
	Z-axis Mach position			
#807	The lowest position when chang Tool	0	-9999.999~9999.999; mm	Operator
	Z-axis Mach position			
#808	X-axis tool change front Mach position	0	-9999.999~9999.999; mm	Operator
	X-axis machine position of deceleration position before entering the tool magazine.			
#809	Y-axis tool change front Mach position	0	-9999.999~9999.999; mm	Operator
	Y-axis machine position of deceleration position before entering the tool magazine.			
#810	Z-axis tool change front Mach position	0	-9999.999~9999.999; mm	Operator
	Z-axis machine position of deceleration position before entering the tool magazine.			
#811	Spindle move speed when changing the tool	100	9~99999; mm/min	Operator
	Spindle motor speed when changing the tool			
#812	Z-axis lifting speed when changing the tool	100	9~99999; mm/min	Operator
	Z-axis lifting speed when changing the tool.			
#813	Move the magazine speed horizontally	100	9~99999; mm/min	Operator
	The speed when move the magazine in horizontally.			
#814	Spindle lock output delay	100	9~99999; mm/min	Operator
	The delay time when changing the tool.			
#815	Go to the position before the tool change?	No	No / Yes	Operator
	If Yes, Z go to The highest position when chang Tool, and XYA returns to previous position before the tool change.			
#816	X mach pos when manually changing the tool	0	-9999.999~9999.999; mm	Operator
	X axis position in Mach coordinate when changing the tool manually.			
#817	Y mach pos when manually changing the tool	0	-9999.999~9999.999; mm	Operator
	Y axis position in Mach coordinate when changing the tool manually.			
#818	Z mach pos when manually changing the tool	0	-9999.999~9999.999; mm	Operator
	Z axis position in Mach coordinate when changing the tool manually.			
#819	Z axis downward movement speed	100	9~99999; mm/min	Operator
	"Speed when moving to position of Parameter #807.			
#820	Pushing start X mach pos	0	-9999.999~9999.999; mm	Operator
	The Starting Position of X axis in Mach coordinate when Pushing			
#821	Pushing start Y mach pos	0	-9999.999~9999.999; mm	Operator
	The Starting Position of Y axis in Mach coordinate when Pushing			
#822	Push Delay	1	0~600000; us	Operator
	The delay time before Pushing			

No.	Parameter definition	Default value	Range and Unit	User
#823	Pushing end X mach pos	0	-9999.999~9999.999; mm	Operator
	The Position of X axis when the Pushing finished.			
#824	Pushing end Y mach pos	0	-9999.999~9999.999; mm	Operator
	The Position of Y axis when the Pushing finished.			
#825	Pushing completed X mach pos	0	-9999.999~9999.999; mm	Operator
	The X axis position in Mach Coordinate when pushing finished each axis will back distance;			
#826	Pushing completed Y mach pos	0	-9999.999~9999.999; mm	Operator
	The Y axis position in Mach Coordinate when pushing finished each axis will back distance;			
#827	Push speed	0	9~9999; mm/min	Operator
	Each axis moving speed when Pushing.			
#830	T01 X mach pos	0	-9999.999~9999.999; mm	Operator
#831	T02 X mach pos	0	-9999.999~9999.999; mm	Operator
#832	T03 X mach pos	0	-9999.999~9999.999; mm	Operator
#833	T04 X mach pos	0	-9999.999~9999.999; mm	Operator
#834	T05 X mach pos	0	-9999.999~9999.999; mm	Operator
#835	T06 X mach pos	0	-9999.999~9999.999; mm	Operator
#836	T07 X mach pos	0	-9999.999~9999.999; mm	Operator
#837	T08 X mach pos	0	-9999.999~9999.999; mm	Operator
#838	T09 X mach pos	0	-9999.999~9999.999; mm	Operator
#839	T10 X mach pos	0	-9999.999~9999.999; mm	Operator
#840	T11 X mach pos	0	-9999.999~9999.999; mm	Operator
#841	T12 X mach pos	0	-9999.999~9999.999; mm	Operator
#842	T13 X mach pos	0	-9999.999~9999.999; mm	Operator
#843	T14 X mach pos	0	-9999.999~9999.999; mm	Operator
#844	T15 X mach pos	0	-9999.999~9999.999; mm	Operator
#845	T16 X mach pos	0	-9999.999~9999.999; mm	Operator

13) System (Totally 12 items)

No.	Parameter definition	Default value	Range and Unit	User
#240	Language	Eng	Eng/中文	Operator
#241	Enable buzzer feedback	Yes	Yes/No	Operator
#244	Enable realtime toolpath	No	Yes/No	Operator
	If enabled the reatime toolpath, the system operation can be slow down by the realtime processing.			
#245	Toolpath mode	Statue	Statue/Line/3D	Operator
	The 3D mode consumes minimum memory comparing the 3D or Statues Mode.			
#247	Interpolation period	0.005	0.001~0.010; s	Operator
	The smaller interpolation period, the higher the machining accuracy, but it will cost longer machinning time.			
#248	LOGO display time	0.100	0.1~10; s	Operator
#261	X-axis rotation angle in 3D toolpath mode	0.000	-180~180; deg	Operator
#262	Y-axis rotation angle in 3D toolpath mode	0.000	-180~180; deg	Operator
#263	Z-axis rotation angle in 3D toolpath mode	0.000	-180~180; deg	Operator
#266	Serial 1 baud rate	B2400	B2400/B4800/B9600/B19200/B115200	Admin
#267	Serial 1 baud rate	B2400	B2400/B4800/B9600/B19200/B115200	Admin
#278	USB keyboard type	Closed	Closed/keyboard/Scanner	Admin
#279	Barcode file location	Local	Local/Udisk/NetDisk	Admin
#283	Barcode scanning processing	No	No/Yes/Test	Admin
	Please contact the factory to enable and design the Barcode scanning function.			
#284	Network boot mode	Close	Close/auto-IP/manu-IP	Admin
	In the current version, we only support the Set the IP address Manually.			

7.2 Search the Parameters by the Number

In our Parameters List,there are hundreds parameters, it is very difficuilt for the users if there is no search functions. By the search function, the users can search out the according parameters very fast.

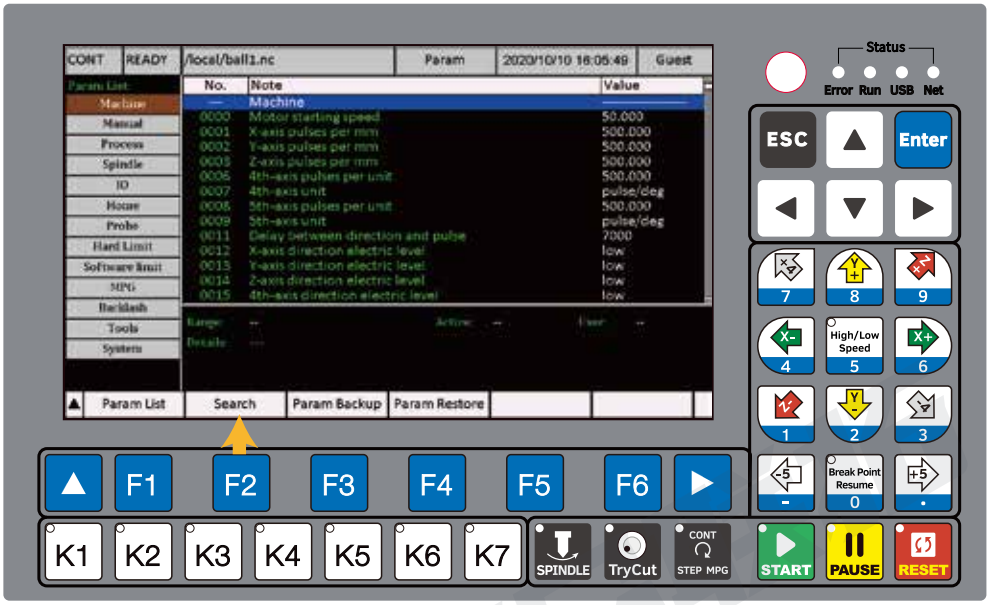


Figure 7-3 Press F3 to Search Page

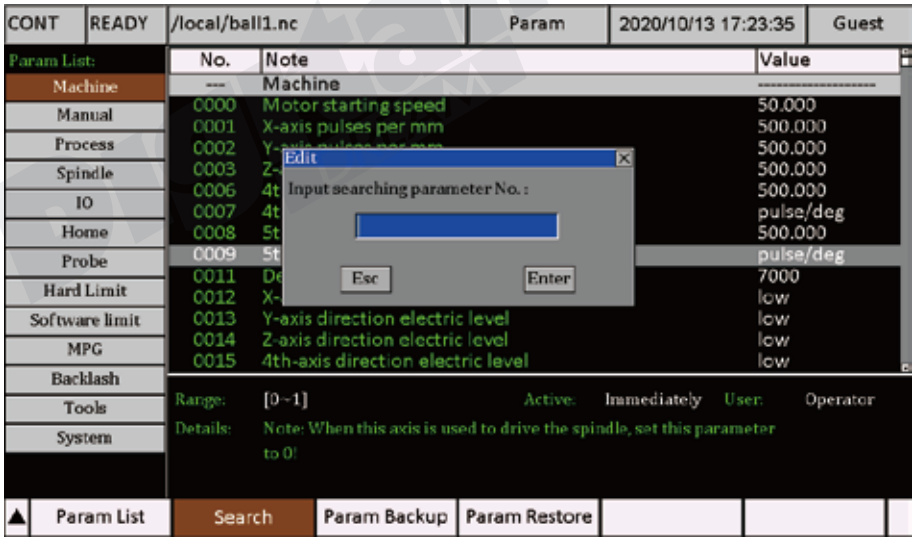


Figure 7-4 Press “Search” dialog box pop up

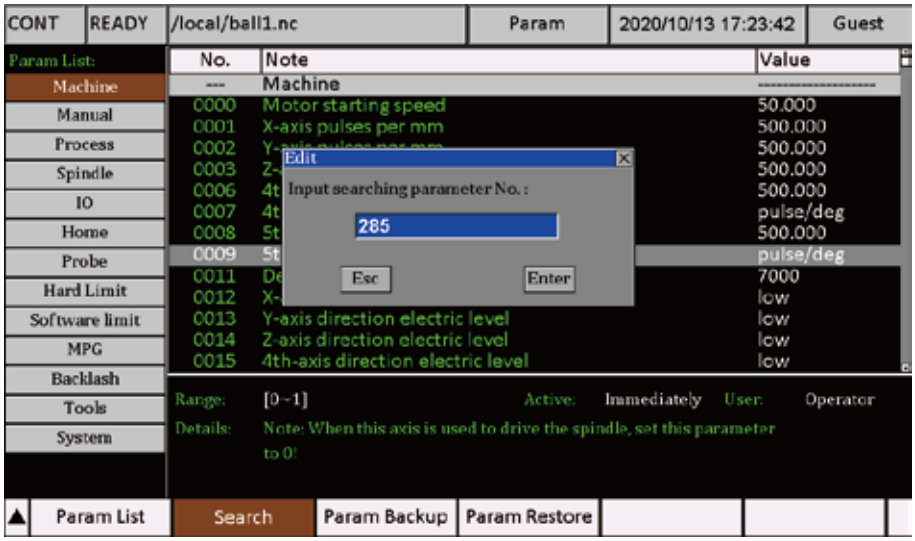


Figure 7-5 We write in Parameter Number and Press Enter

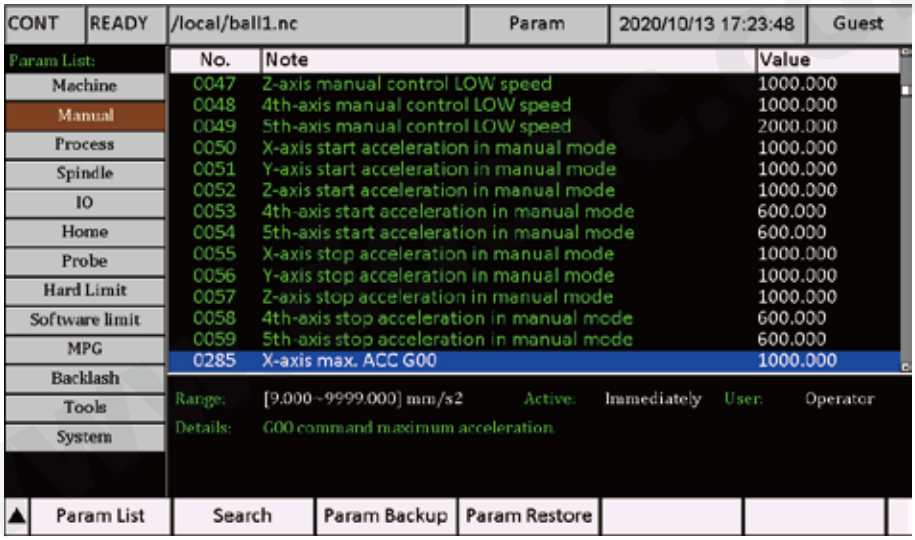


Figure 7-6 Now the according parameters searched out

7.3 Parameter Setting Backup

As the users spend time and engery to configure all the parameters, and want to save all the data, here in DDCS-Expert, we supply One-Key Backup function, convenient and easily.

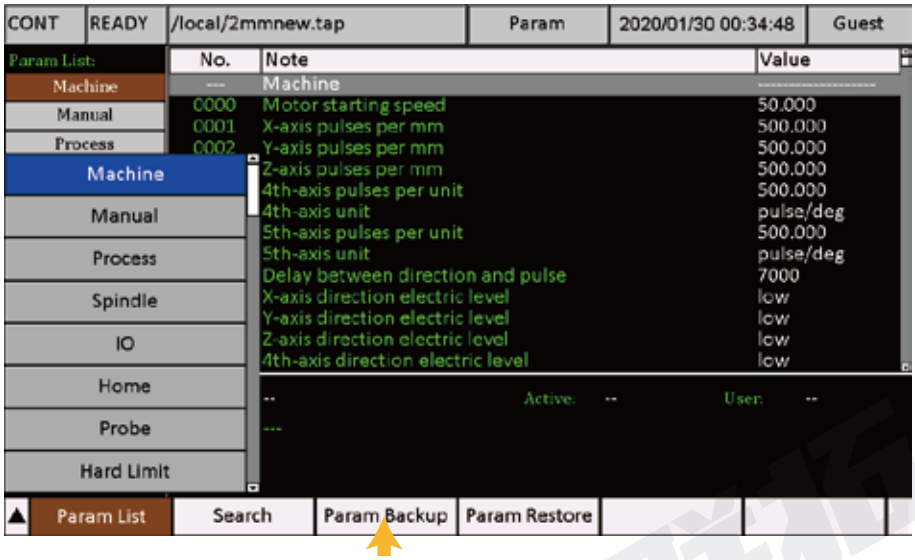


Figure 7-7 Press F3 to Backup the parameters

Please note here, that the system will backup the parameters information in a setting file to the USB-Stick, so we must insert a USB-stick on the controller before the action.

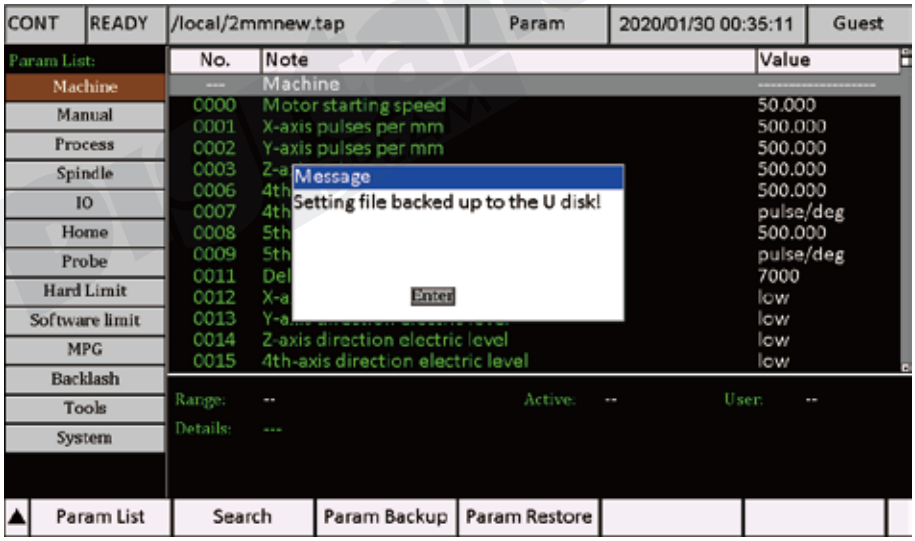


Figure 7-8 Parameters back up successfully

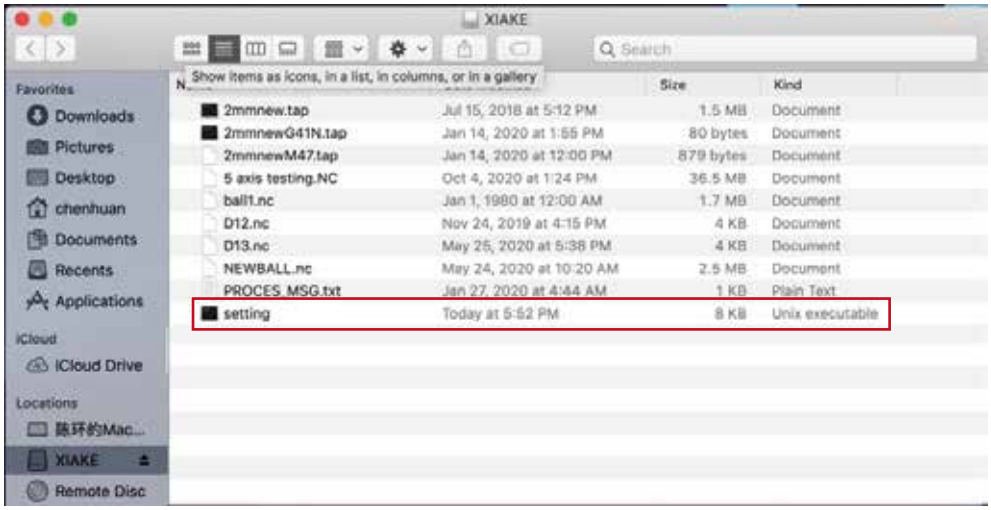


Figure 7-9 Parameters back up in the Setting file

7.4 Parameters Restore

The users can restore the parameters from the USB-stick. Here as we already had the setting file, we can just copy the setting file in the root direction of the USB-stick as the Figure 7-9, and insert it to controller.

We press F4 and turn knob to “Restore from U”, then a dialog box pop up to ask the password of admin or higher user. Input the correct password, and Enter, the system start to restore the parameters, when it finished the system can pop up a dialog box to show it is done.

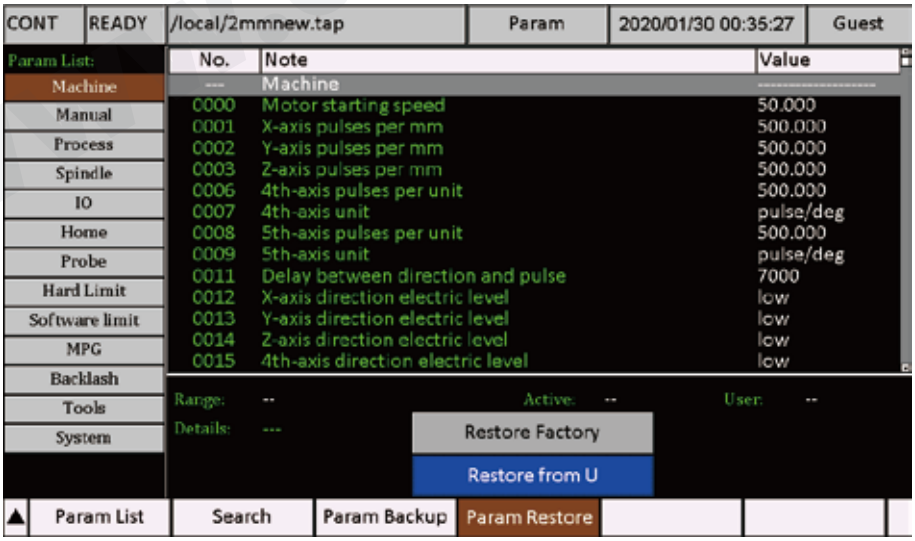


Figure 7-10 Parameters Restore from USB-Stick

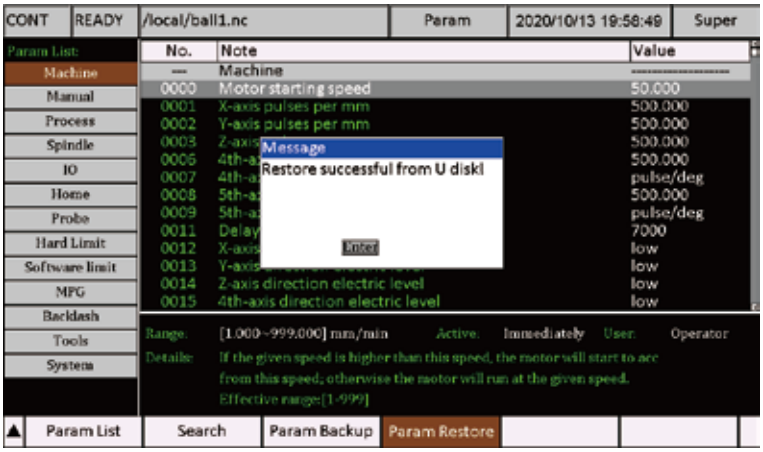


Figure 7-11 Parameters Restore Successful from USB-Stick

8 System Info

In the main page, press F6 to System Info Page.

In the Page, the users can:

1) Registraion: The users can set a system working time;

2) Set the password for Operator, for Admin, and for Super Admin;

3) Can update the system software from the USB-stick;

4) Set the system date and time;

5) Set the IP address.

Figure 8-1 Press F6 to System Info Page

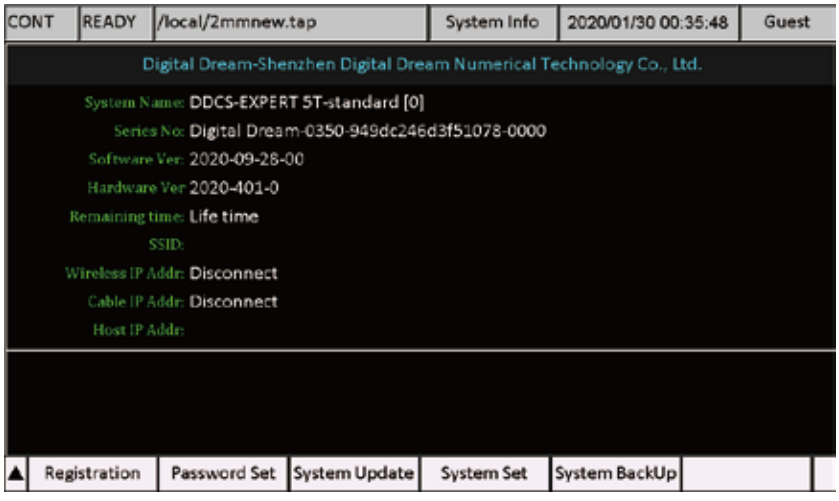


Figure 8-2 System Info Page

8.1 Registration

For the customers who want to control the controller working time, we supply a working time setting software “DDCS-Expert Key Generator” , please visit our website :

www.ddcnc.com

or our Facebook Forum:

https://www.facebook.com/groups/1724999967517167/?ref=group_header

to find the software and download it.

Now your zip program can recognise the file as a compressed file and you can decompress it as the Figure 8-3.

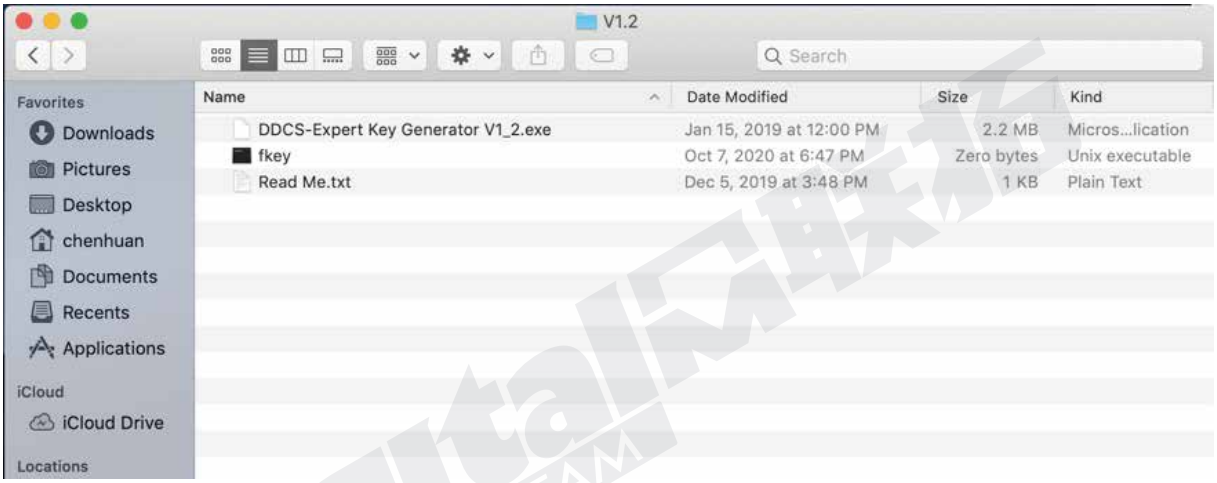


Figure 8-3 DDCS-Expert Key Gernerator Software folder

Double click the “DDCS-Expert key Generator V1_2.exe” , there will be a windows as Figure 8-3 pop on.

1) Series No: Each controller will have a unique series Number, we can input the number to the Series No. in Generator ; It only allow 6 charactors, so only write in “Digita” .

2) Time Setting: “-1” means no limited time; if you put any other numbers (Number range is 1-9999), the system will calculate the power on time, when the time reached to the limit, the controller don’ t work.

3) Super Admin Password, here only the users input right Super Admin Password, the setting can be active.Please note the default Super Admin Password is 888888.

4) When we finished input the numbers and Press button “ Generate ”, the software can update a new “fkey” in the same folder. The Users just copy the “fkey” file to root-directory of the USB-stick ,and insert it to the controller.

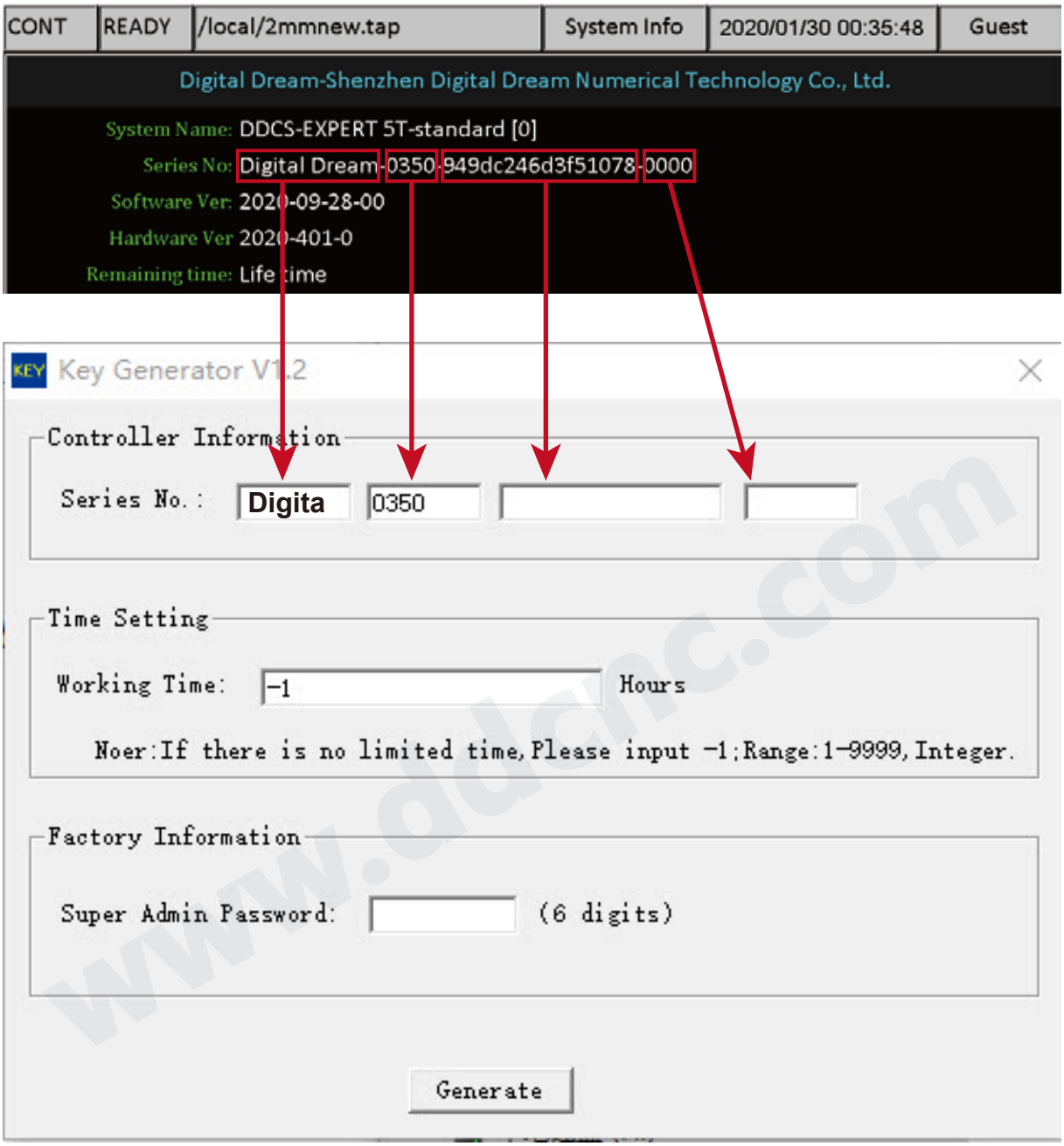


Figure 8-4 DDCS-Expert Key Gernerator Software

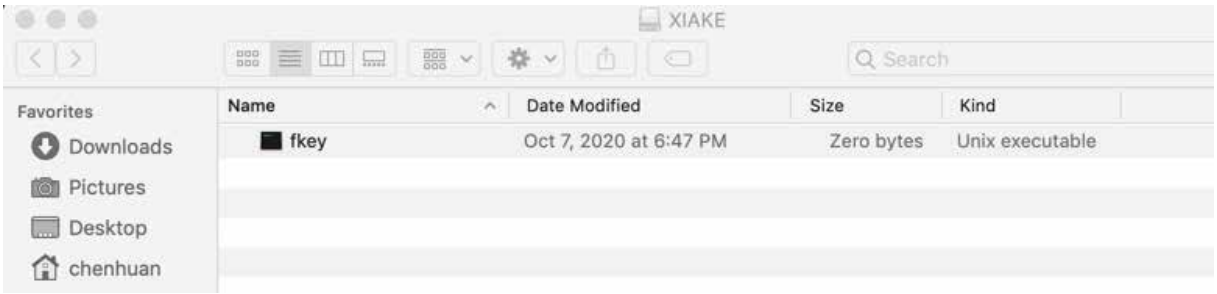


Figure 8-5 “fkey” file is in the root-directory of the USB-stick

5) Then Press F1 to “Registration” , and system will ask if the USB-stick have the “Fkey” file? We Press Enter key and system registrate automatically.

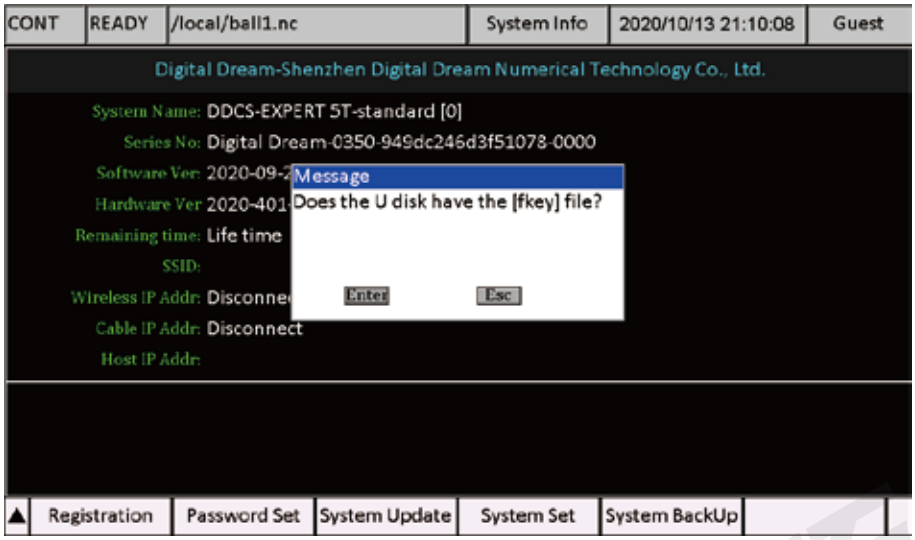


Figure 8-6 System ask if there is “fkey” file is in the root-directory of the USB-stick

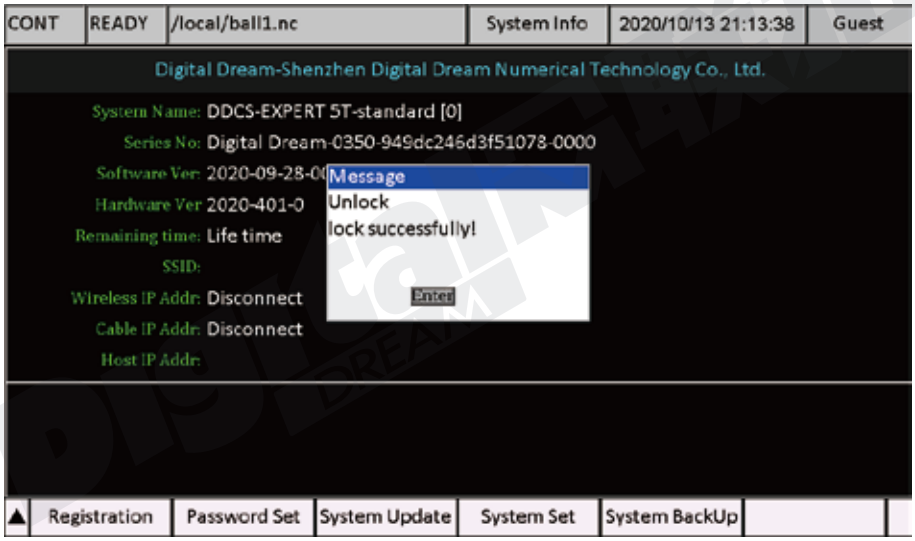


Figure 8-7 System Registrate the working time Successfully

If the left working time is less then 48 hours, when restart the controller, the controller will send a hint;

When the working time updated successfully, system will delete the fkey automatically;

If the updating is not successful, please check the series no. and the super admin password is right or not.

Very Important:

The working time and date caculation, is powered by a lithium battery. Because of the Air delivery control, the products with Battery always in limit. We will take off the battery if delivery by air. So please contact the factory for the informaiton to buy the right battery and install it to the controller.

8.2 Password Setting

The default password for Opeartor ：666666

The default password for Admin: 777777

The default password for Super Admin: 888888

Here in the Password Page, we can reset the passwords.

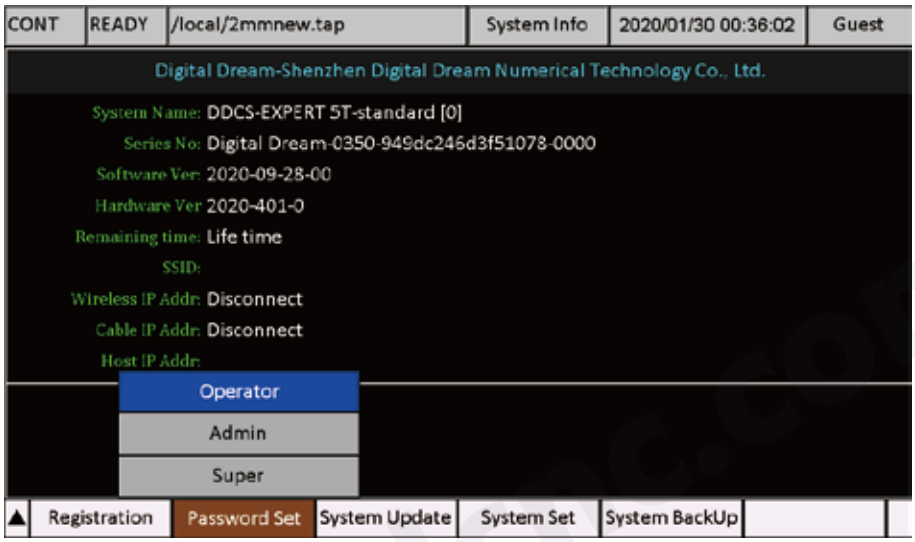


Figure 8-8 Password Reset Page

Press Enter it will ask you to enter the higher rights password. Input the default passwod, and write in the new password two times, the new passwrod is active now.

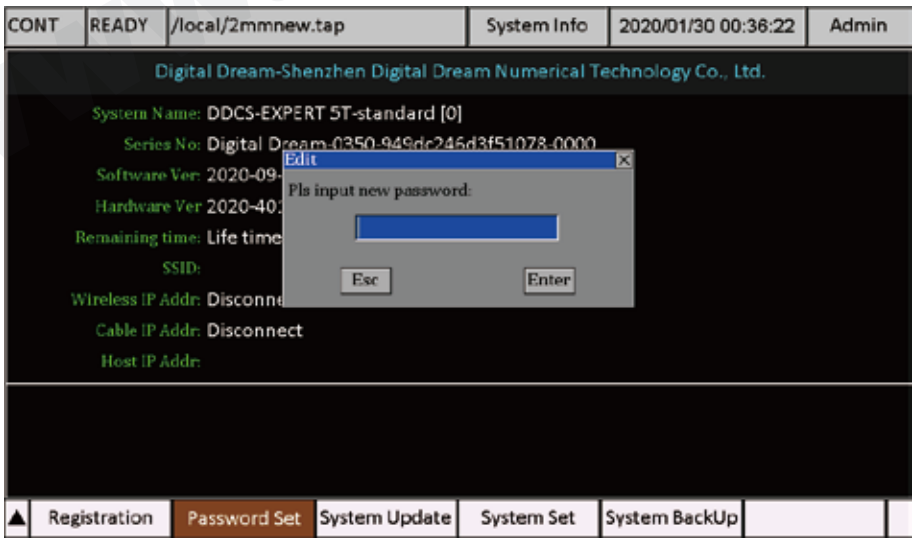


Figure 8-9 Input the new password one time

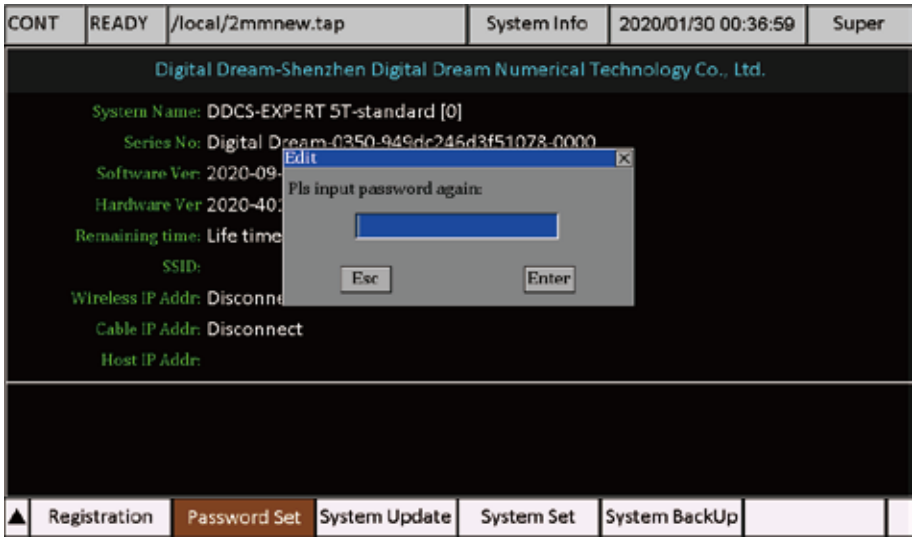


Figure 8-10 Input the new password again

Very Important:

Please be very careful to reset the password of the Super Admin. If the users resest the Super Admin and lost the new password, only send back to the factory, only in factory we can reset the Super admin again.

8.3 System Update (System Software Update)

According to the customer feedback, we will endeavour to update the software for DDCS-Expert to enhance the performance, fix the bugs or add new features always. In order for customer to download the latest firmware, please visit our website :

www.ddcnc.com

or our Facebook Forum:

https://www.facebook.com/groups/1724999967517167/?ref=group_header

or join our forum

http://bbs.ddcnc.com/forum.php

There you can find the latest version software for DDCS-Expert .

In the Main Page of the “System Info”, we can check here the Software version.

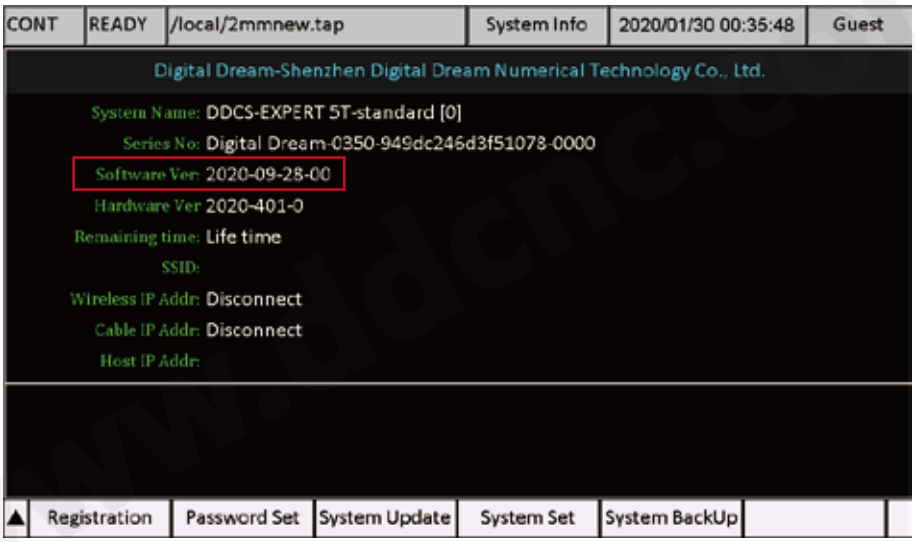


Figure 8-11 Software Version

Download the firmware upgrade file to your computer and prepare a totally empty USB key. Best is to quickly reformat the USB key to MS-DOS FAT32 (right click on the USB key icon and choose Format. Follow the prompts)

After downloading the firmware file check the file name, if download from www.ddcnc.com,it may look like this:

install(2020-09-28-00).rar (example)

If download from the facebook team (https://www.facebook.com/groups/1724999967517167/?ref=group_header) , it may look like this or similar:

install(2020-09-28-00)-rar (example)

This is done to allow the download. Files called “. RAR” are sometimes blocked.

Change the file name to

install(2020-09-28-00).rar (example)

Now your zip program can recognise the file as a compressed file and you can decompress it to the USB key.Please note the upgraded file should be in the Root-directory in the USB Stick and the file name must be“ install”:

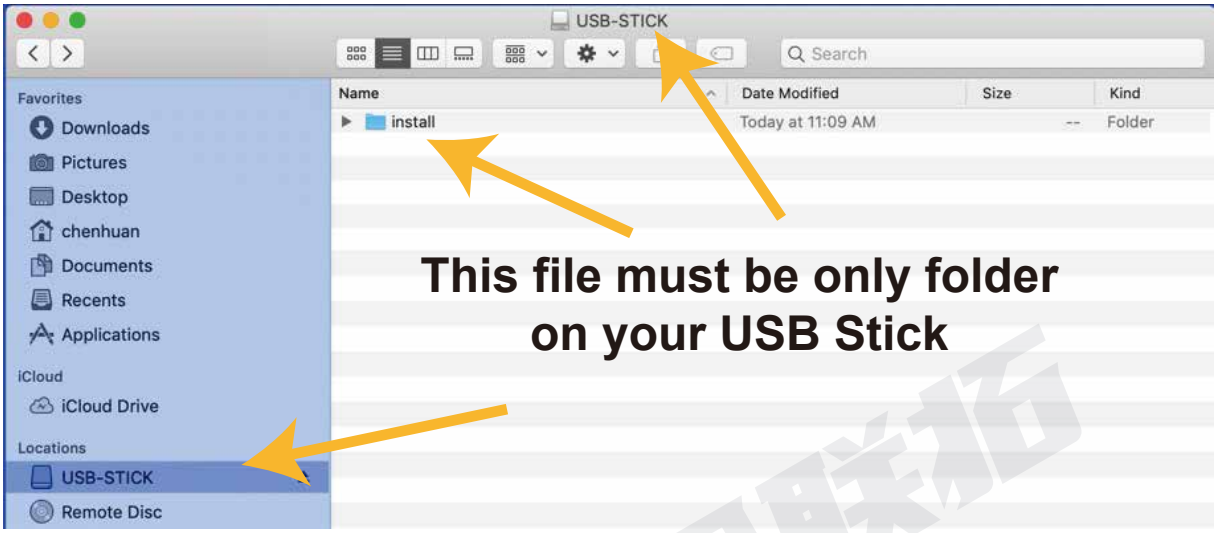


Figure 8-12 Position of Software file

Important:

Because we already delete the setting file from the Install software, When upgrading, the entire configuration will be not replaced. Then you can keep your personal Setting file.

But if you need the Default setting file, you can contact factory ask for it.

Please note the upgraded file should be in the Root-directory in the USB Stick and the file name must be“ install”.

Now your USB key is ready for action.

DDCS-Expert Controller has two way to update the software:

A: Update the software when Power On.

1) Shut down your DDCSE controller for 10 seconds.Insert the USB key into the USB port of your DDCSE controller .

2) Start your DDCSE controller. The controller will read the INSTALL folder on the USB key and upgrade automatically. The screen will be blocked for about 30 seconds, then the controller will start with the new software.

On the Main Screen lower right you can see the firmware version the controller is using.

After upgrading successfully, don’ t forget to remove the Intsall folder from the USB key. If you do not remove the INSTALL folder the controller will update again next time you start the controller.

B: Update the software in the System update page.

1) Go to Main Page of “System Log” as the figure 8-1; And press F3 key of “System update” ;

2) The controller will ask “Does the U disk have the [install] folder” , Press Enter key;

3) The controller will read the INSTALL folder on the USB key and upgrade automatically. The screen will be blocked for about 30 seconds, then the controller will restart with the new software.

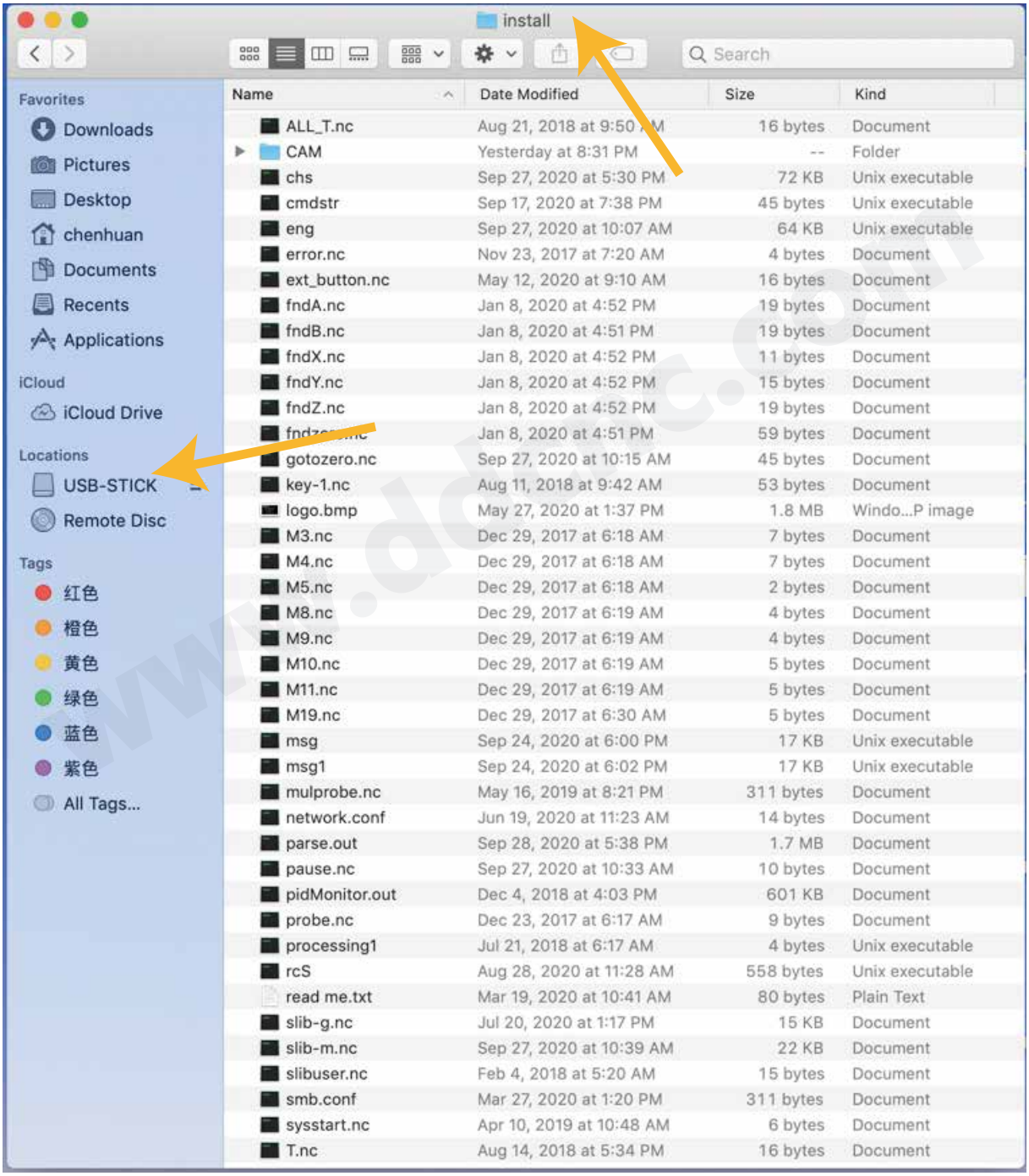


Figure 8-13 The files a INSTALL folder included

8.4 System Set

IN the System Set Page, we can set the system time, build the the network by Ethernet, and build the wireless network.

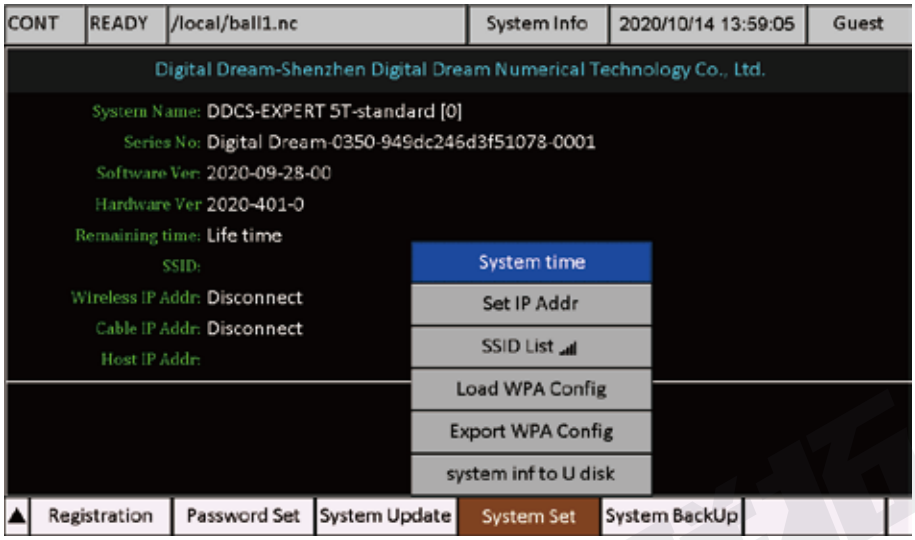


Figure 8-14 System Setting Page

8.4.1 System Time Setting

The system Time setting format is YYYY. MM. dd. HH. mm. ss:

YYYY: 4 digits to show the Year, such as 2020, 2021 ect;

MM: 2 digits to show the month, such as 01, means the January, 12 means the December;

dd: 2 digits to show the date, such as 02, means 2nd of the moth; 30 means the 30th of the month;

HH: 2 digits to show the hour;

mm: 2 digits to show the munits;

ss: 2 digits to show the seconds.

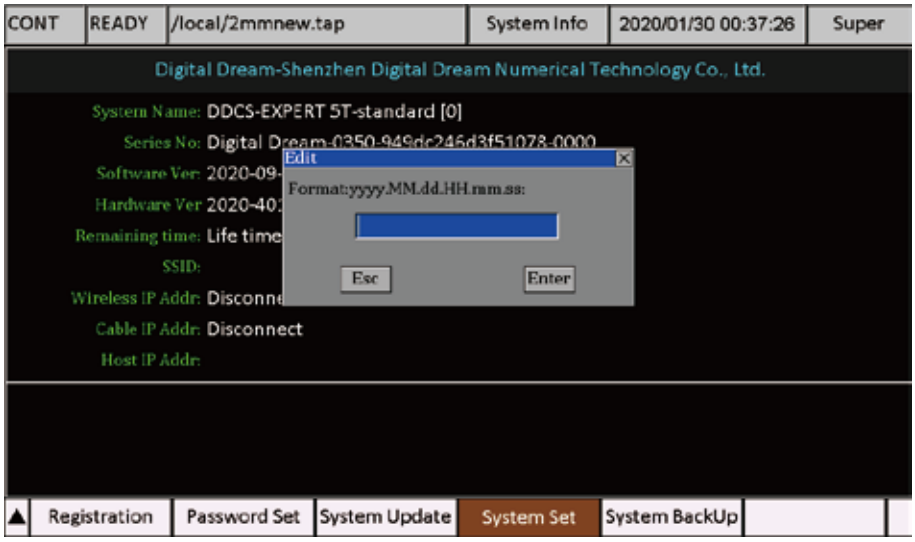


Figure 8-15 Setting the system time

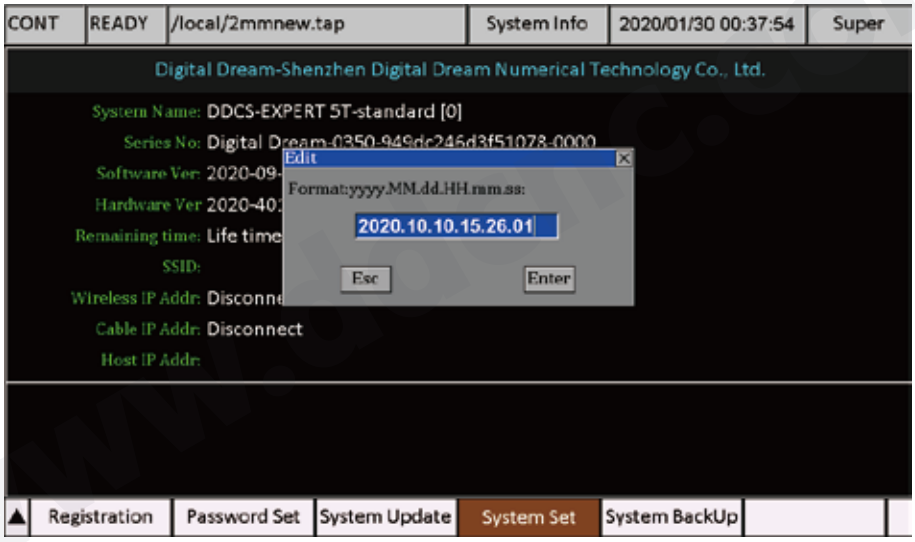


Figure 8-16 Input the settings

Very Important:

The working time and date caculation, is powered by a lithium battery. Because of the Air delivery control, the products with Battery always in limit. We will take off the battery if delivery by air. So please contact the factory for the informaiton to buy the right battery and install it to the controller.

8.4.2 Set IP Address manually by Ethernet Cable

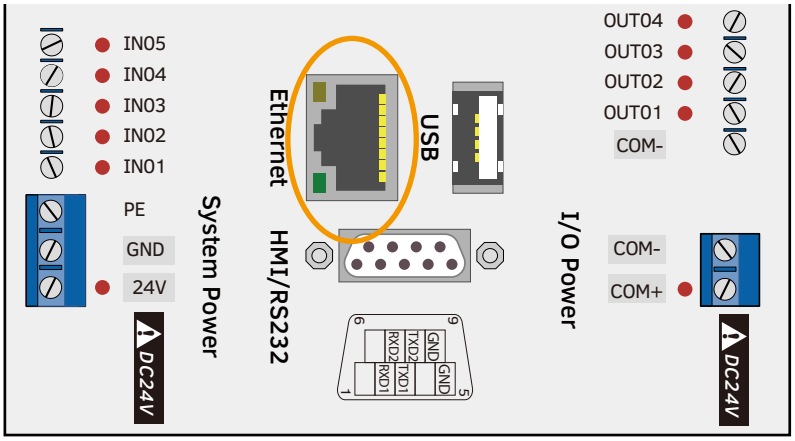


Figure 8-17 Build the Network by Ethernet

Firstly we need a Ethernet cable to connect with the DDCS-Expert and the computer.

If the Network building properly, the Net LED turns red.

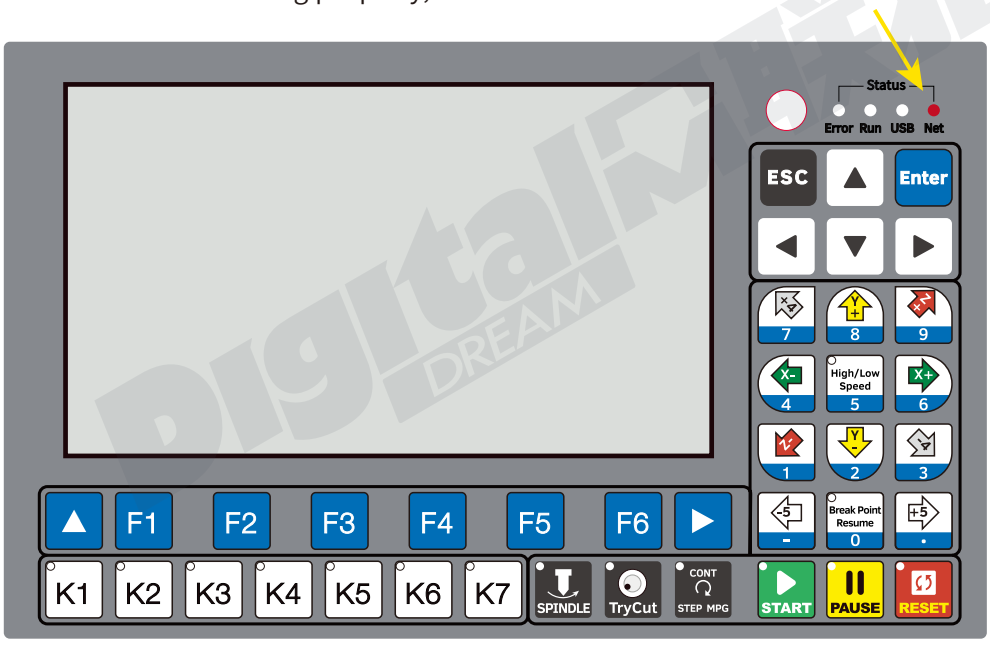


Figure 8-18 Network LED shows the status

1) Computer (Host) Configuration

Step 1: Use an Ethernet Cable to connect the DDCSE controller and the computer;

Step 2: Go to Setting -> Network and Internet -> Network Connections in computer.

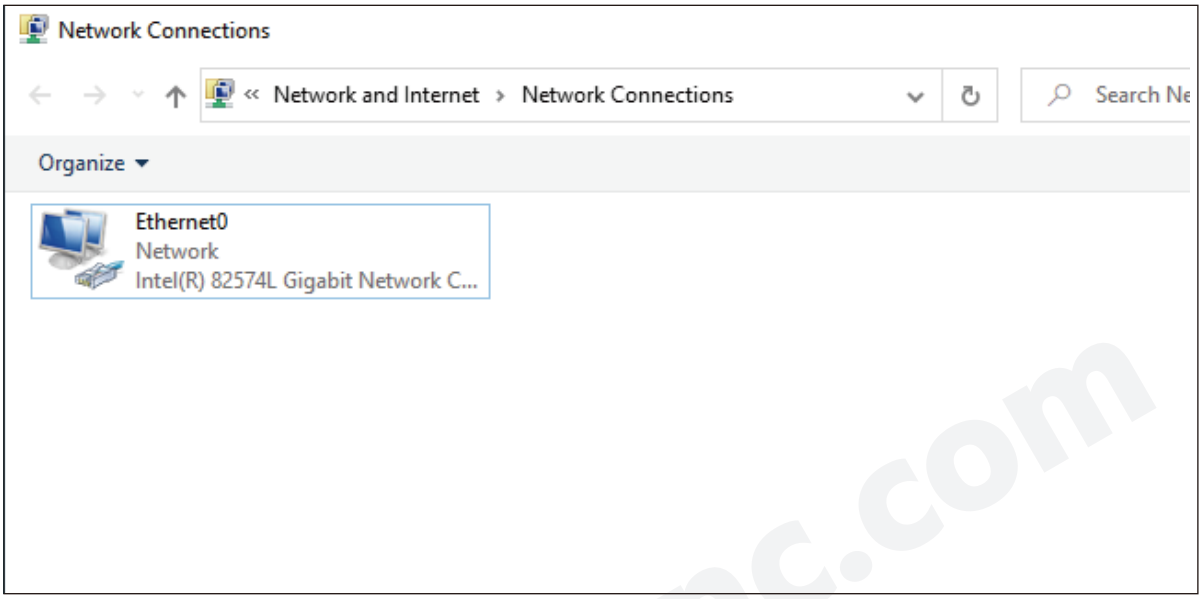


Figure 8-19 Find the Network Connections in Computer

Step 3: Right click and hit “Properties” and it popup the windows and double click “Internet Protocol Version 4 (TCP/IPv4)” as the Figure 8-21:

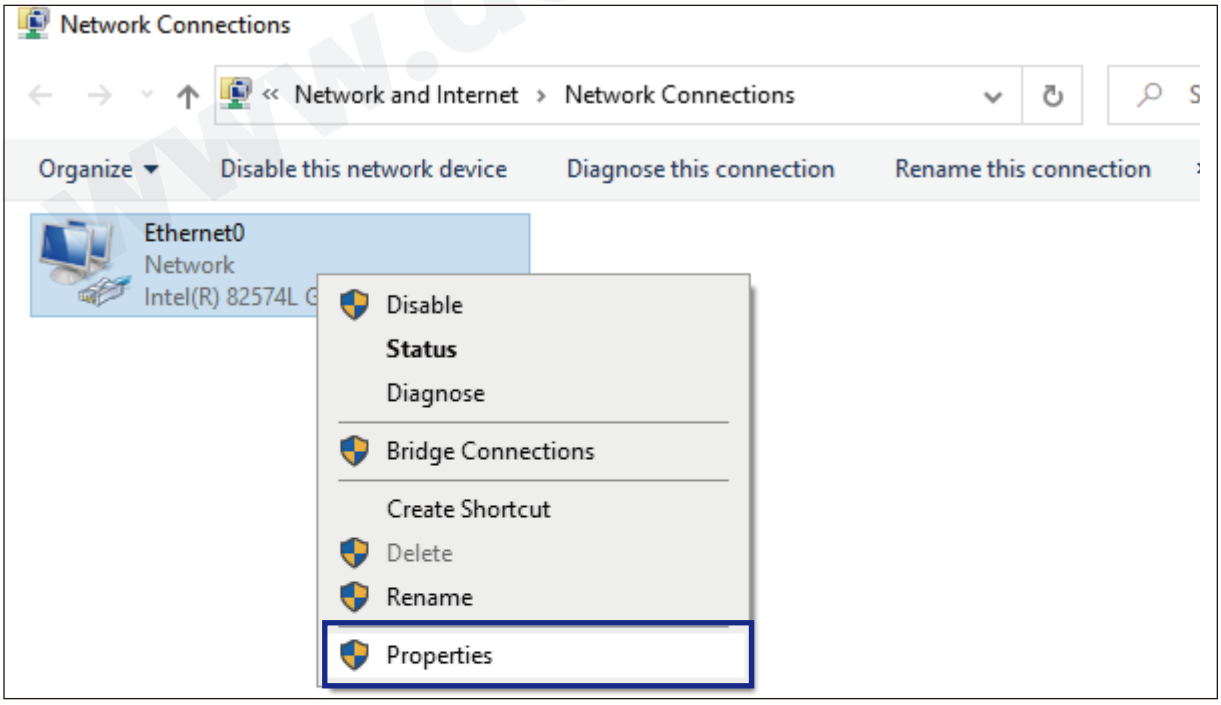


Figure 8-20 Properties

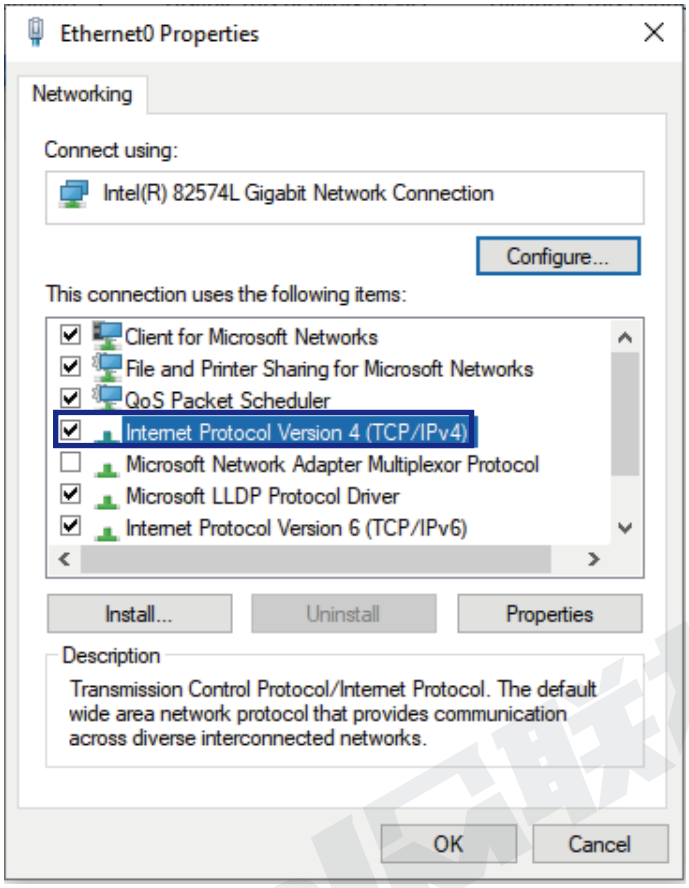


Figure 8-21

Step 4: Set the IP address and Subnet mask as Figure 8-22:

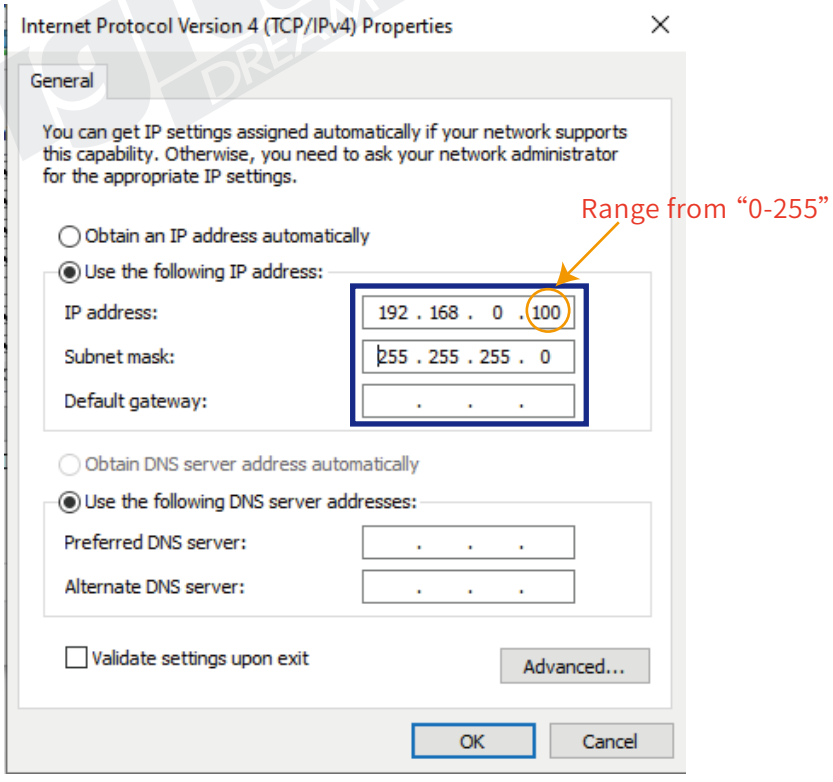


Figure 8-22 Set the IP address

Step 5: Turn Off the FIrewall & network protection:

Go to Setting -> Update & Security -> Windows Security -> Firewall & network protection,and turn off the firewall:

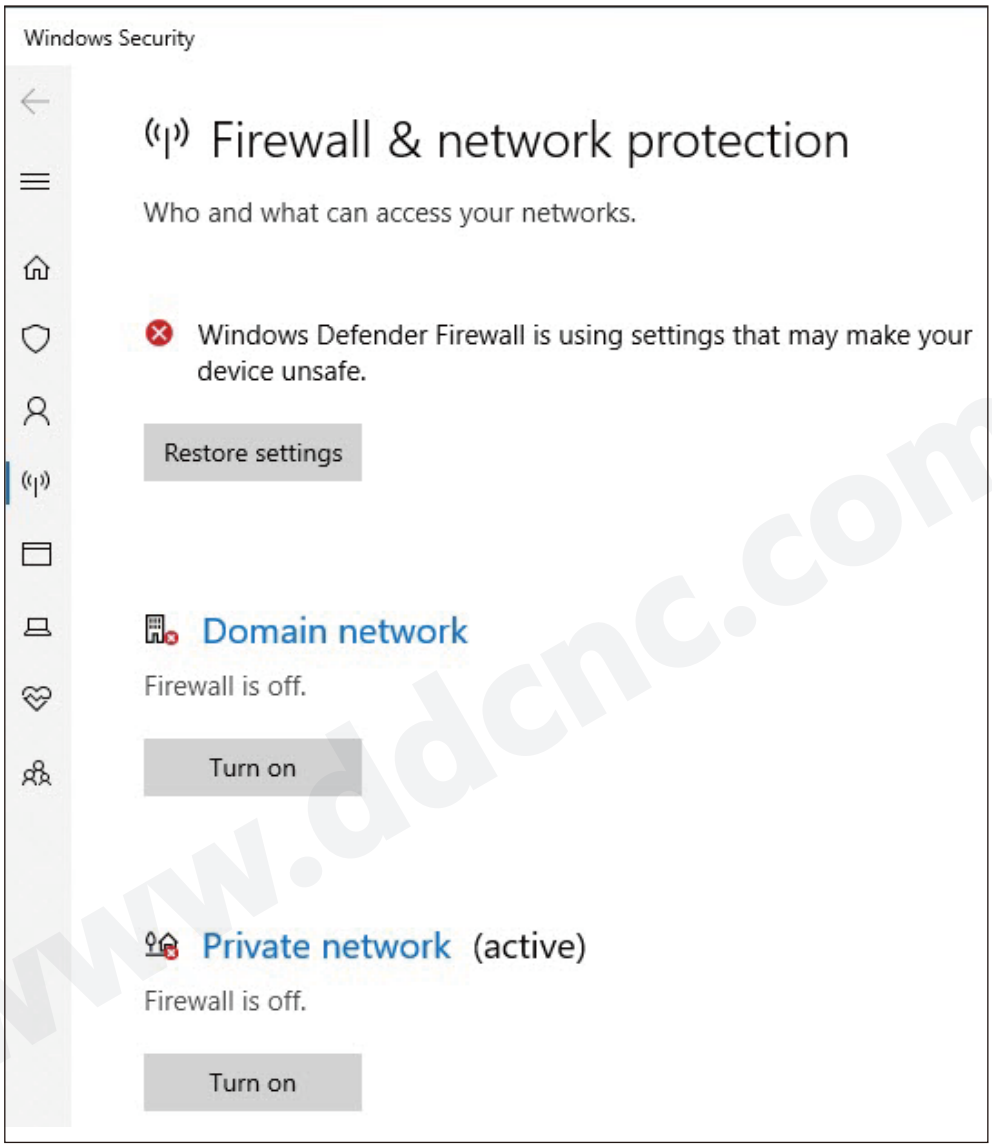


Figure 8-23 Turn off the Firewall of the computer

Step 6: Create a “Share” folder:

Go to Settings -> Network and Internet -> Network and sharing center -> change advanced sharing settings,“Turn on network discovery” and “turn on file and pinter sharing”,as Figure 8-24 and 8-25 shows;

Then in your computer,you can create a forder name it as “share”.

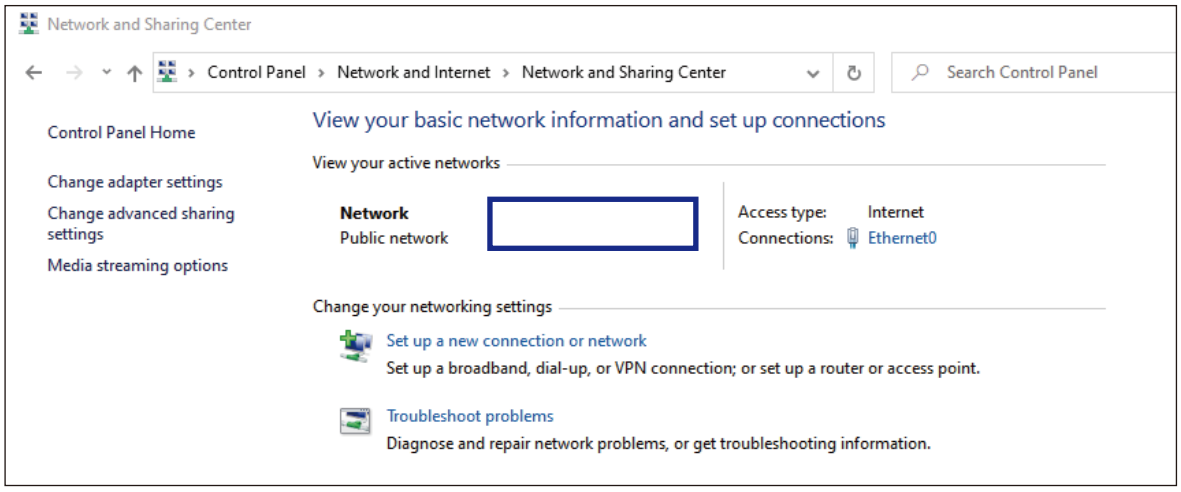


Figure 8-24

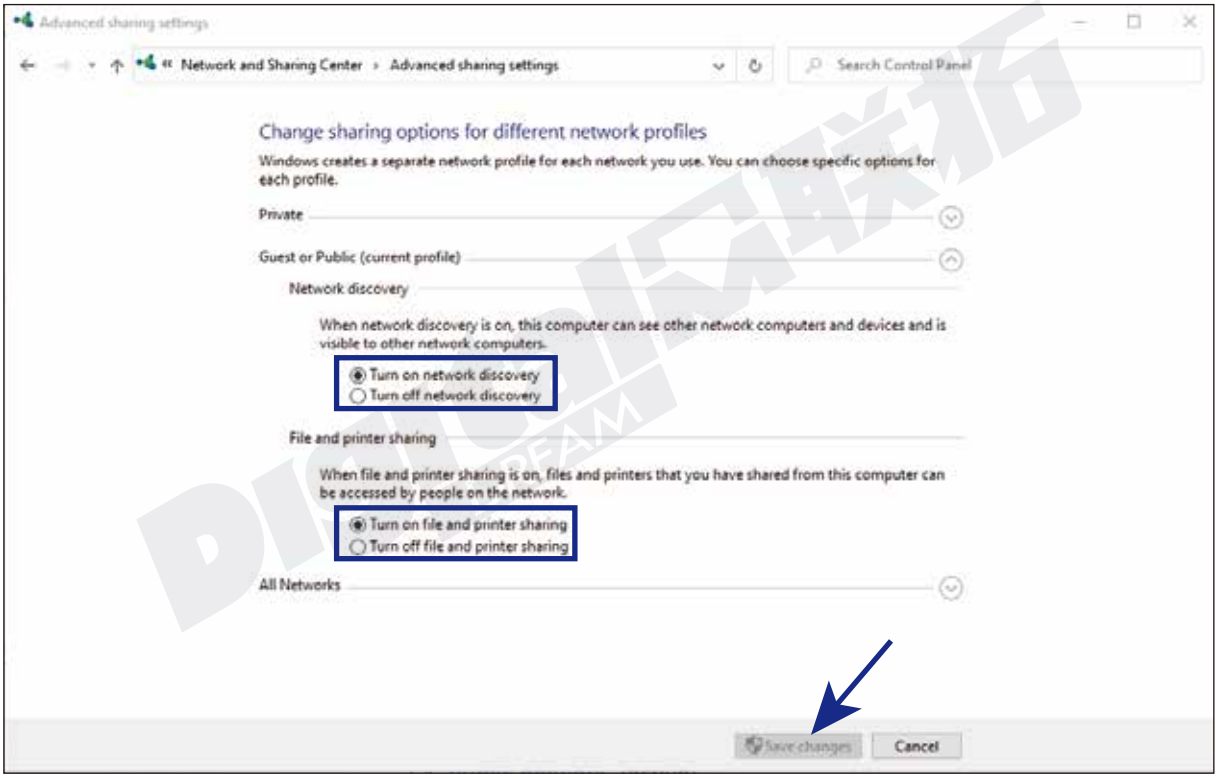


Figure 8-25

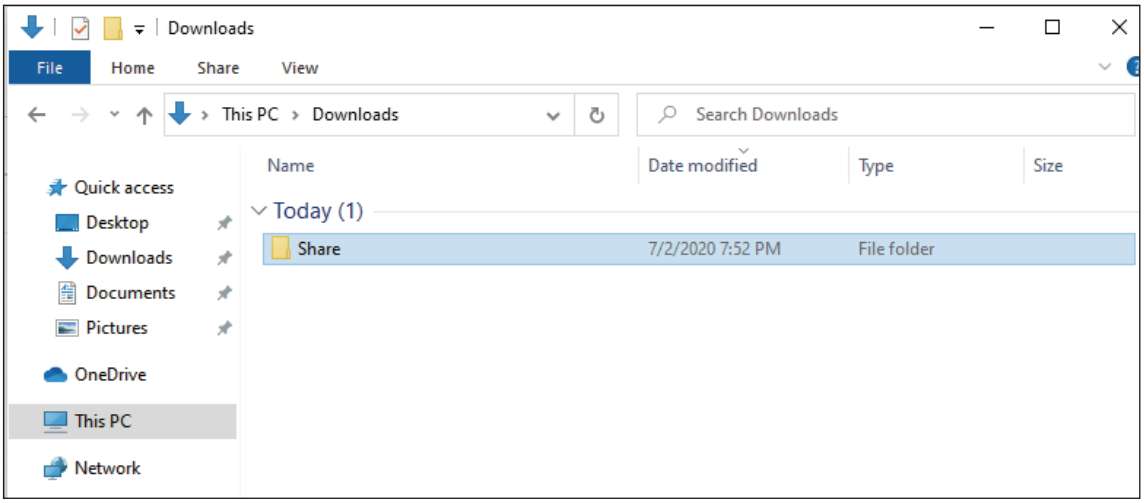


Figure 8-26

Step 7: Set the folder as “share” folder:

1) Right click the folder and click “Properties”,the “Properties” window popup as figure 8-28 shows;

2) click “Share button”,and “Network access” windows popup as figure 8-29 shows;

3) choose “Everyone”,and add it to the list;

4) Change the **Pernission level of “Everyone” to “Read/Write”,and confirm it.**

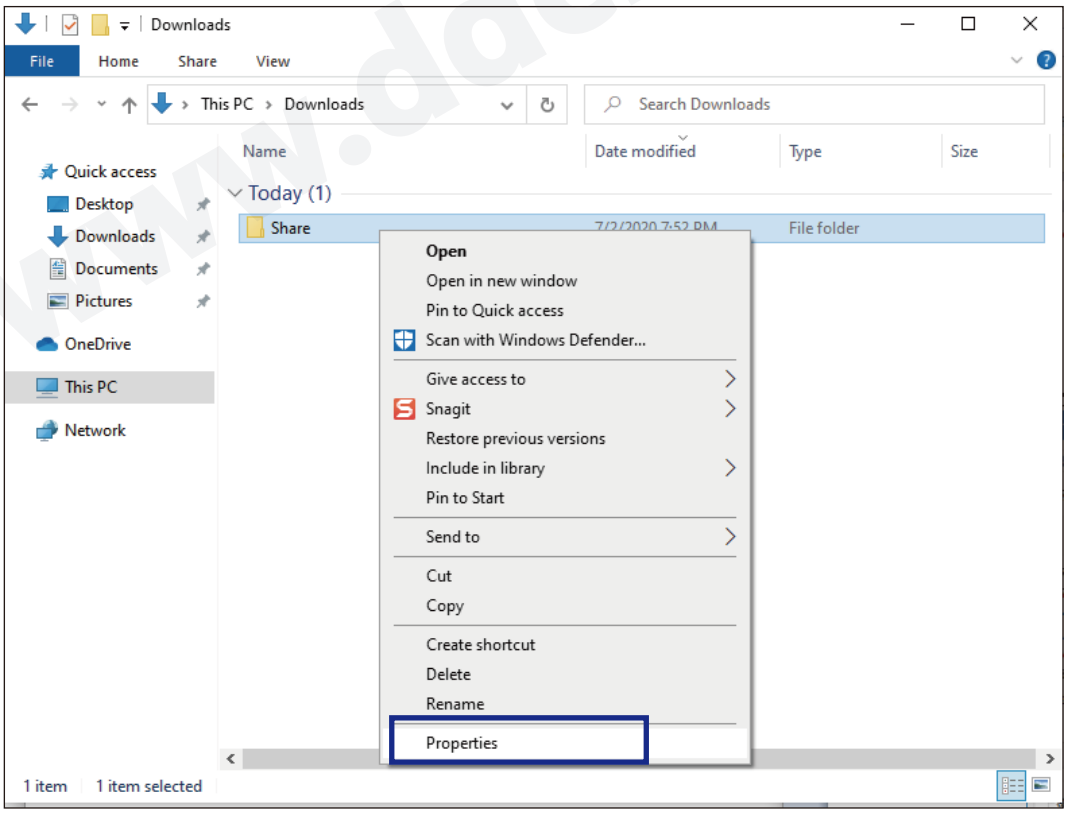


Figure 8-27

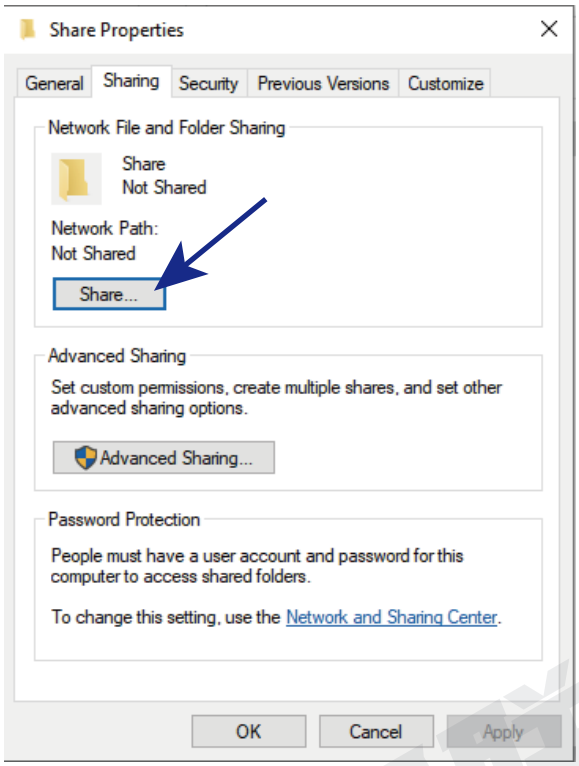


Figure 8-28

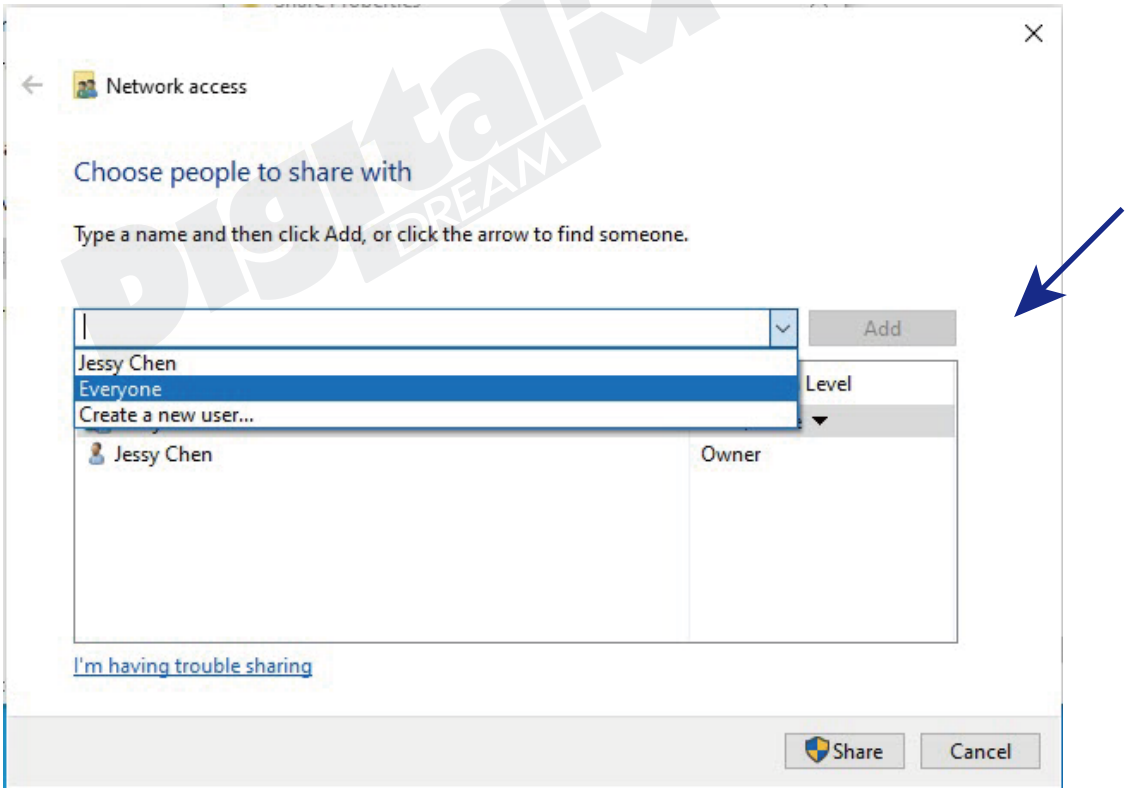


Figure 8-29

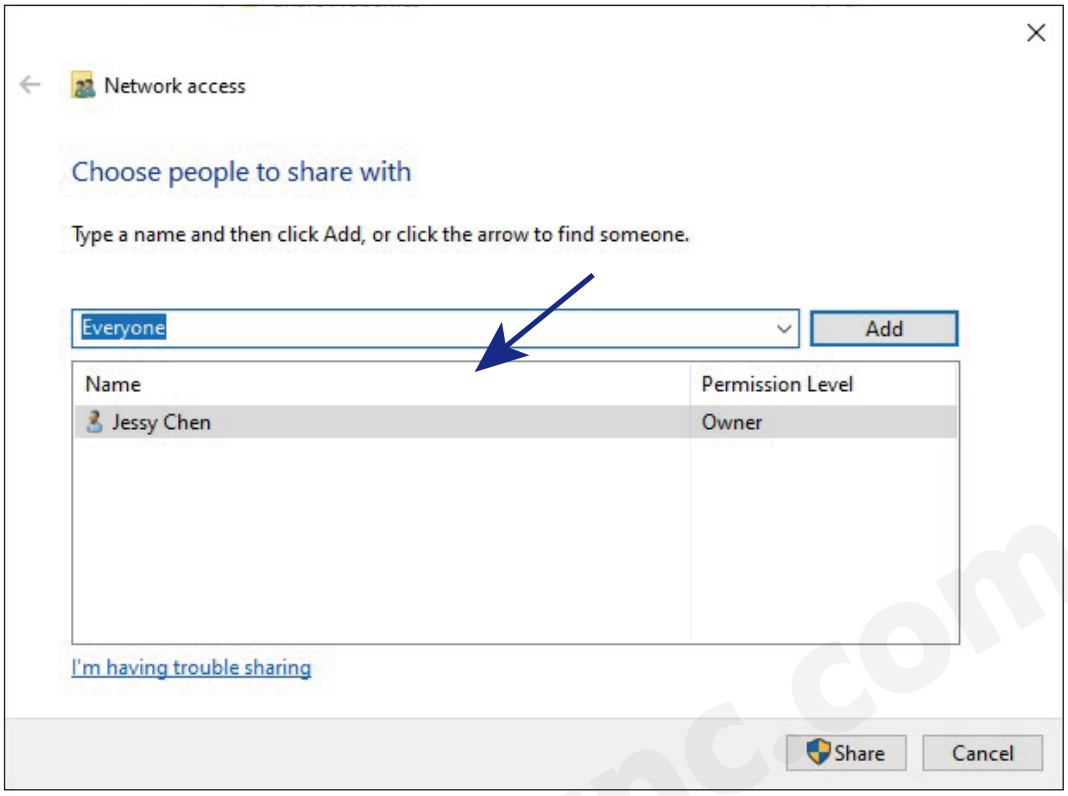


Figure 8-30 Chose “Everyone” and add it to the list

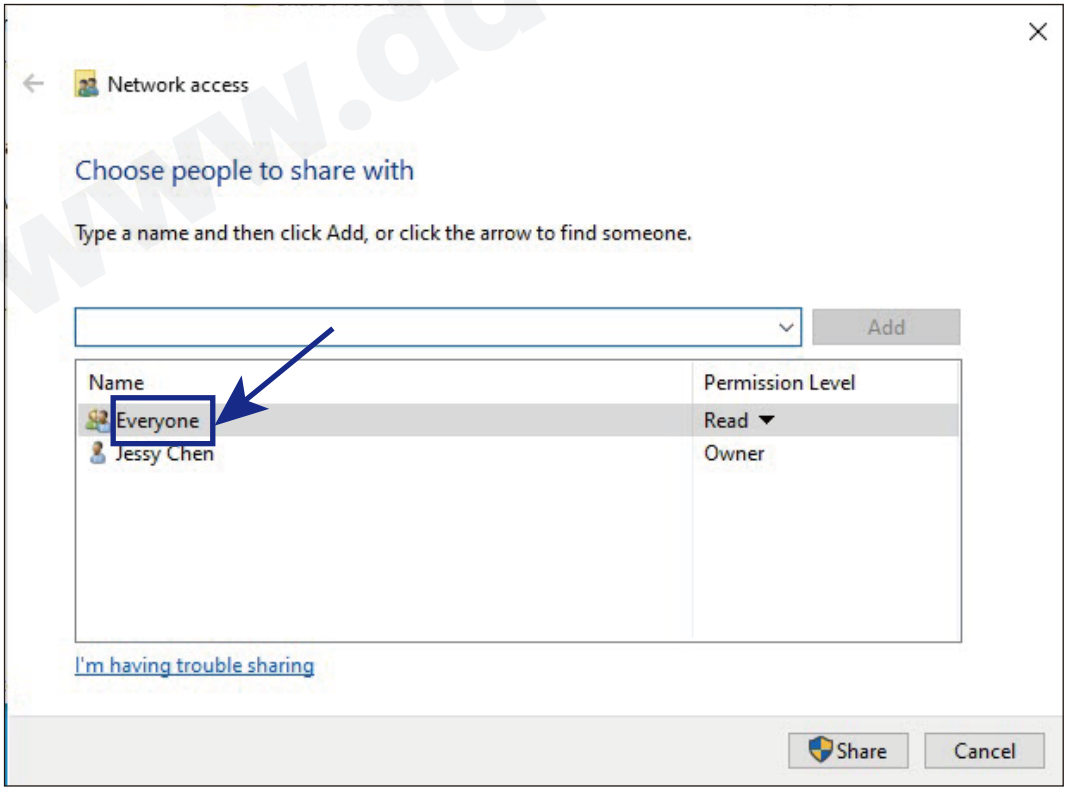


Figure 8-30 Change the Pernission level of “Everyone”

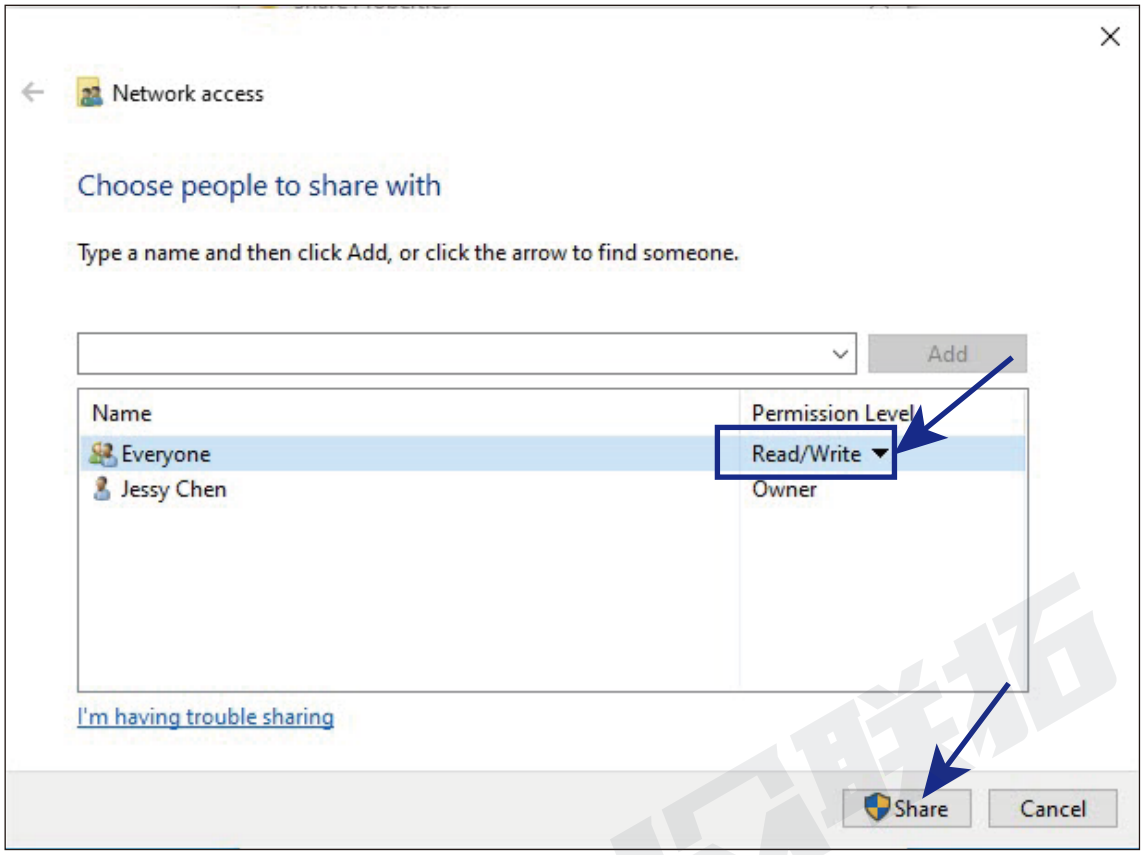


Figure 8-31 No forget to Click “Share”

2) Controller DDCSE Configuration

Step 1: Change the “Network Boot Mode” to “manu-IP”,it means the users can setup the IP settings manually in this mode.

1) Figure 8-32 shows the main page.This is Main Page,Press F3 to Parameter Page;

2) Find Para #284 “Network Boot Mode”,press “Enter”;

3) Password window pop up,please input the Admin password to choose “manu-IP” .

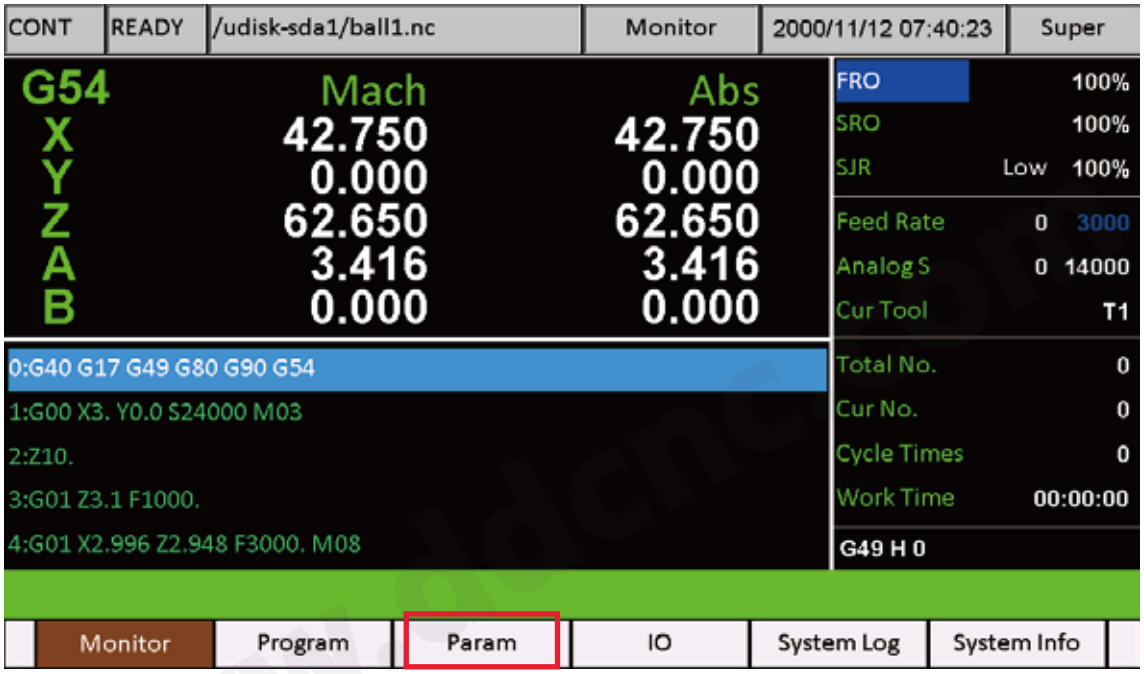


Figure 8-32 Go to Param Page

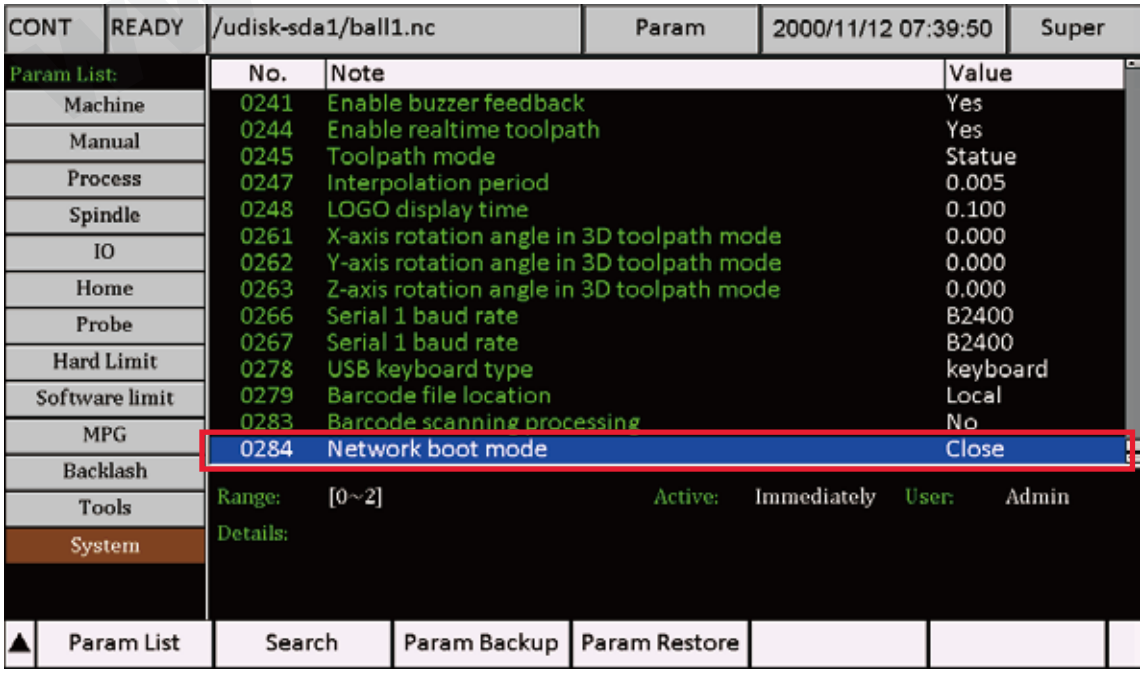


Figure 8-33 Find the Parameter #284

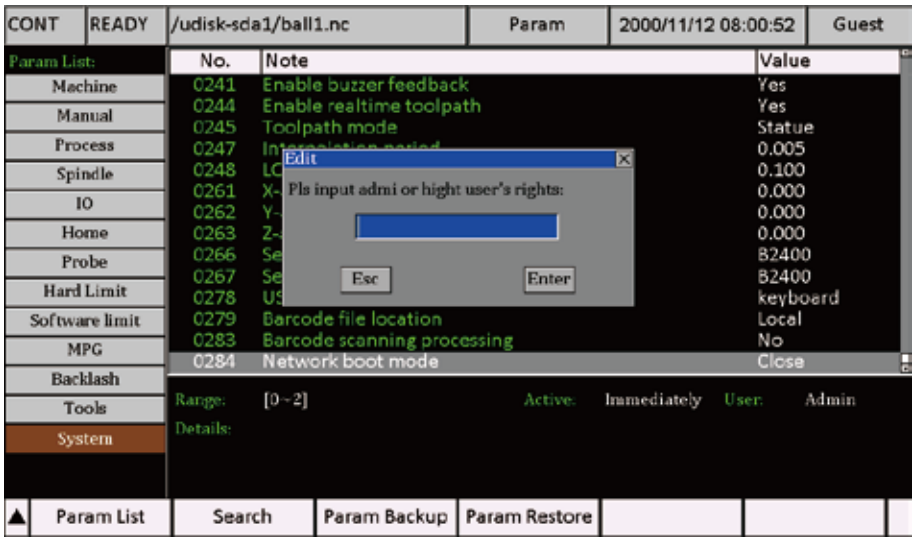


Figure 8-34 Password window pop up

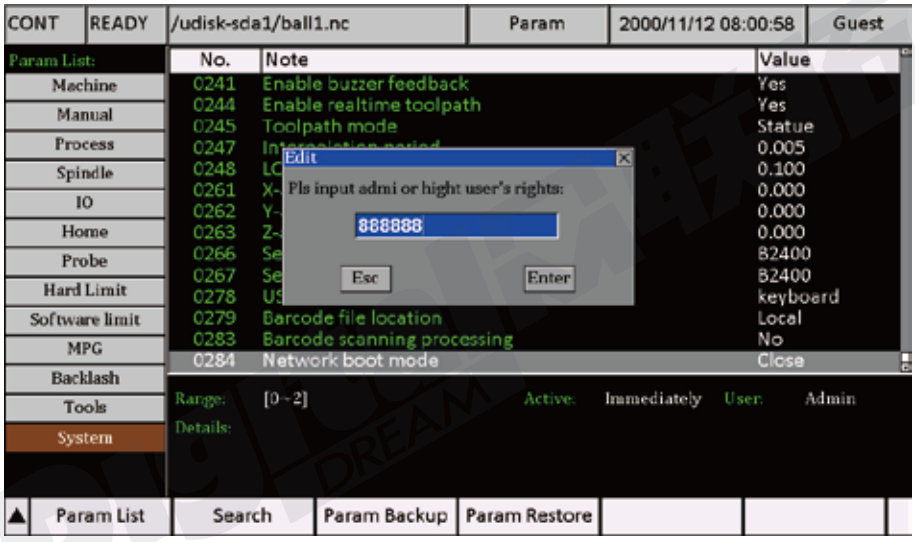


Figure 8-35 Input Admin Password to continue

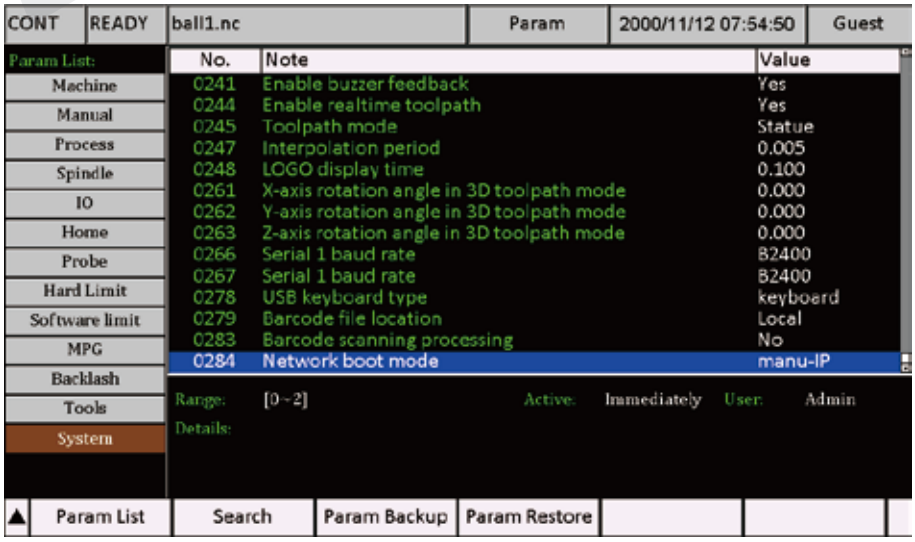


Figure 8-36 Changed the mode to “manu-IP”

Step 2: Set the IP address

1) Go back to Main page and Press F6 To System Info,as figure 8-37shows;

2) Press F4 and choose “Set Ip Addr” as Figure 8-38 shows;

3) Press Enter and move down to “Cable IP Addr”,it shows as Figure 8-39 shows;

4) Press Enter key and input the controller IP address “192.168.0.99”;

5) Press F4 again and Go to “Host IP address” as Figure 24 shows;

6) Press Enter and input Computer (Host) IP address “192.168.0.100” as Figure 8-41 shows;

7) Now,remember,now restart the controller,never forget this step,go to “System info” Page again,it just shows as the Figure 26,that means the IP setting is succesful.

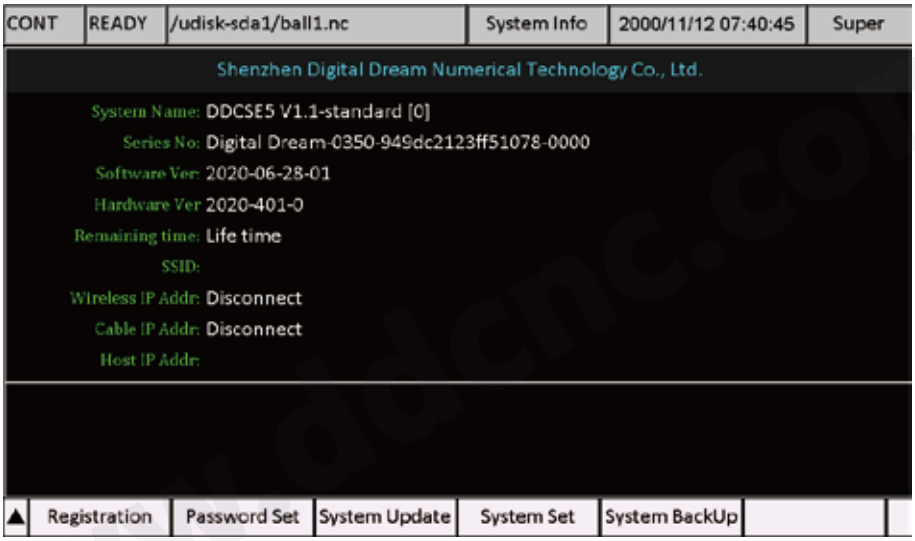


Figure 8-37 “System info” Page

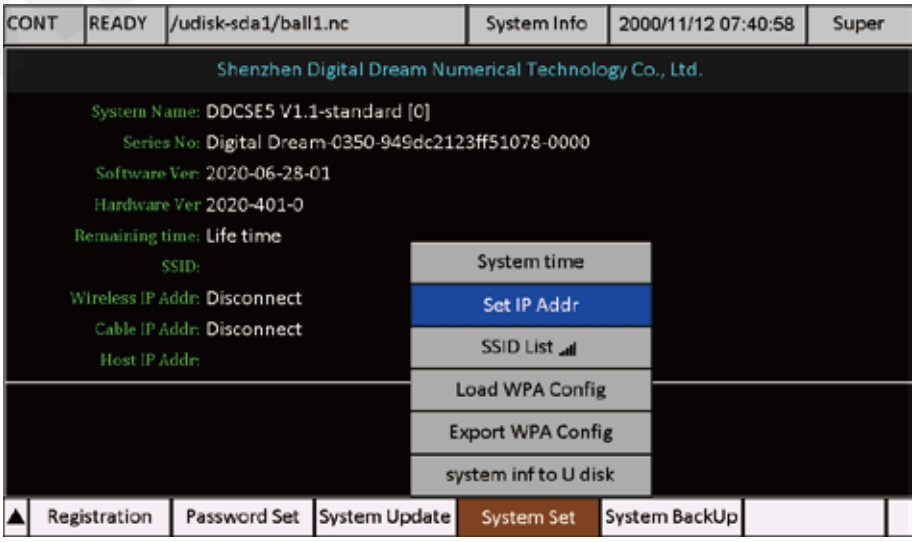


Figure 8-38 Set IP Address

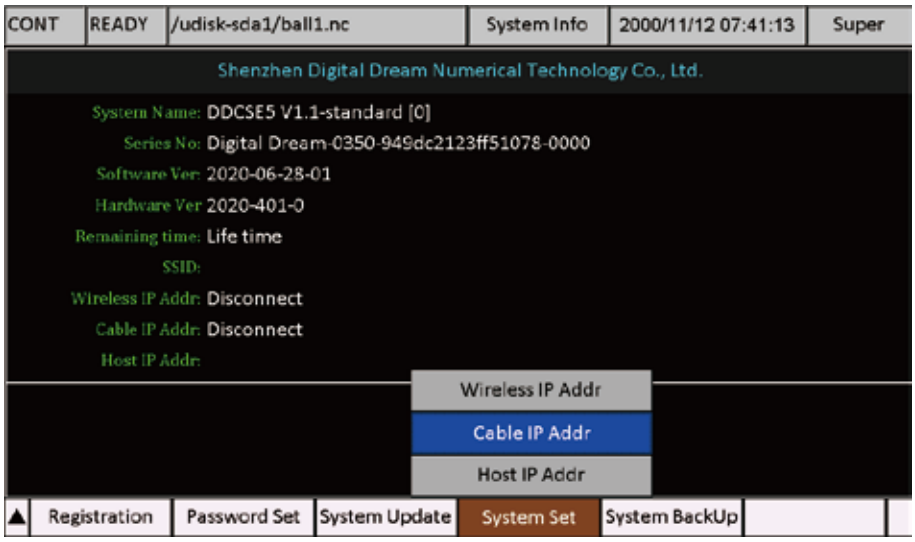


Figure 8-39 Cable IP Address

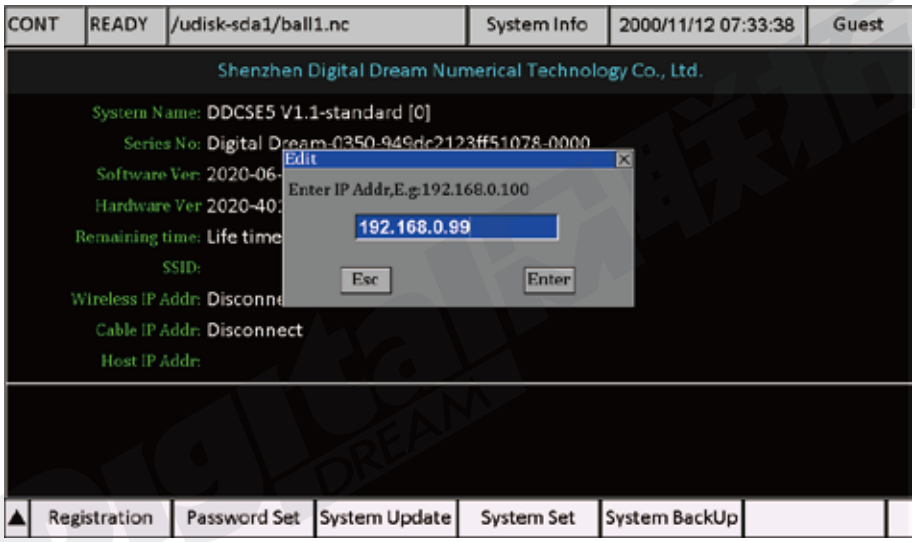


Figure 8-40 Enter controller IP address “192.168.0.99”

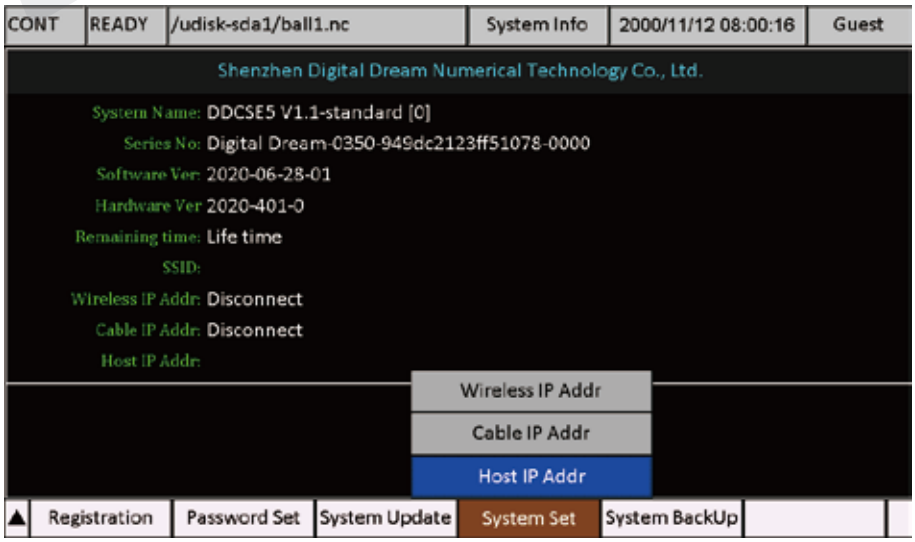


Figure 8-41 Go to Host IP address

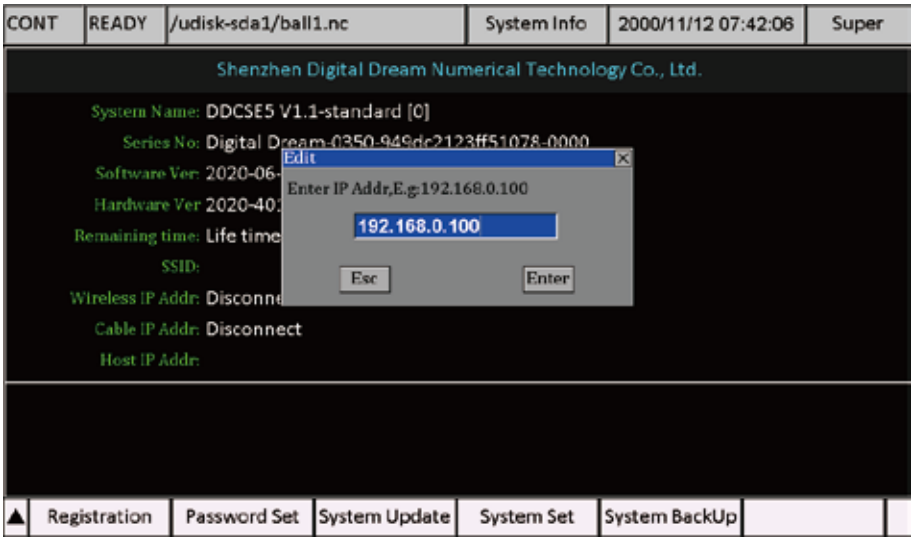


Figure 8-42 Input the Host Computer IP address

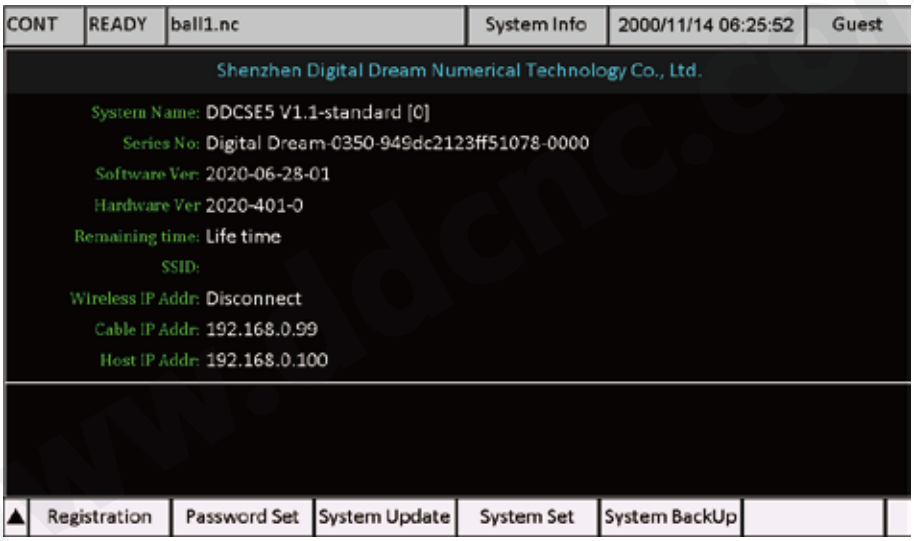


Figure 8-43 After Restart we can see the IP setting is successful

3) Check the files from the Host (computer)

1) Copy the files you need into the folder “share” on the computer as Figure 8-44;

2) In the Page of Program, press the “Swtich disks” button(F1)，switch to “Net Disk”, and we can see the files as Figure 8-45;

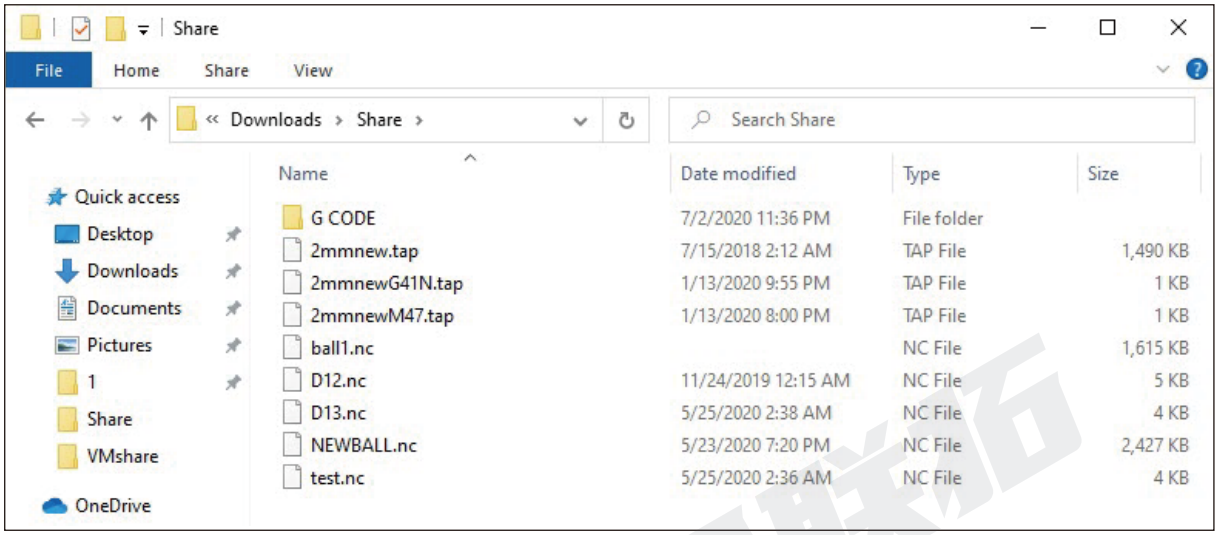


Figure 8-44 The files in the Share folder

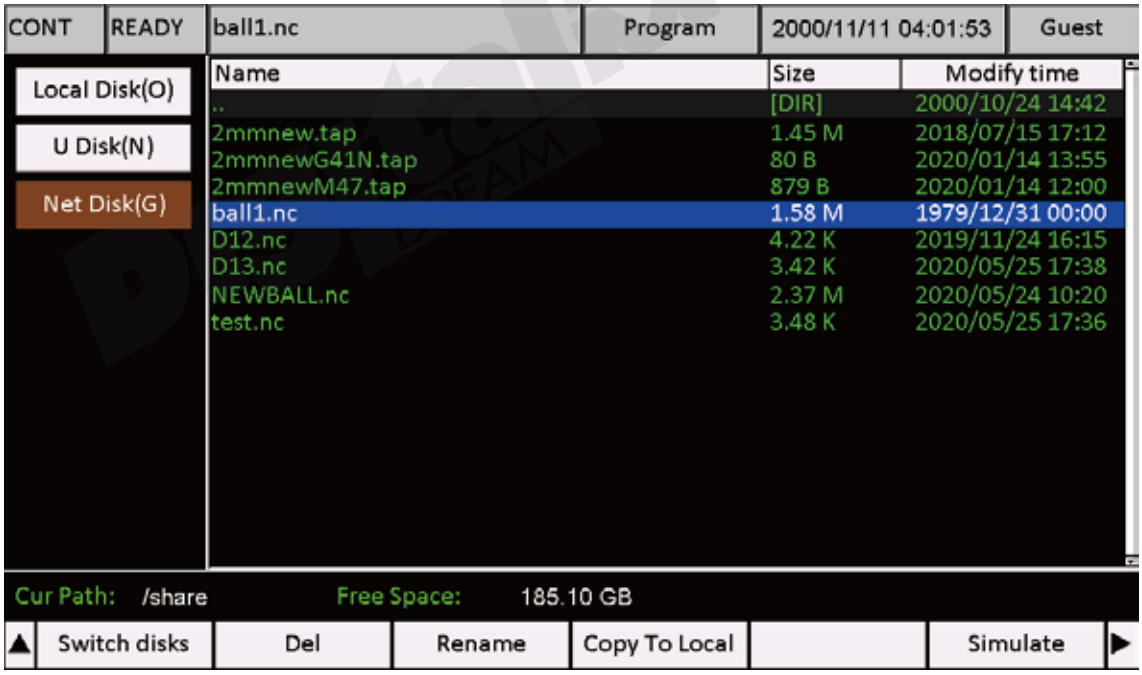


Figure 8-45 Net Disk shows the files from Computer

Please Note: U-disk and Net Disk cannot active at the same time.

8.5 System BackUp

In the System Back Up, there are 3 options:

1) BackUp: It will copy the INSTALL folder of this controller system, to the USB-stick;

2) Clear Cache: System clear the cache, that will make the system running quicker;

3) Clear Local: It will delete all the files in the Local memory.

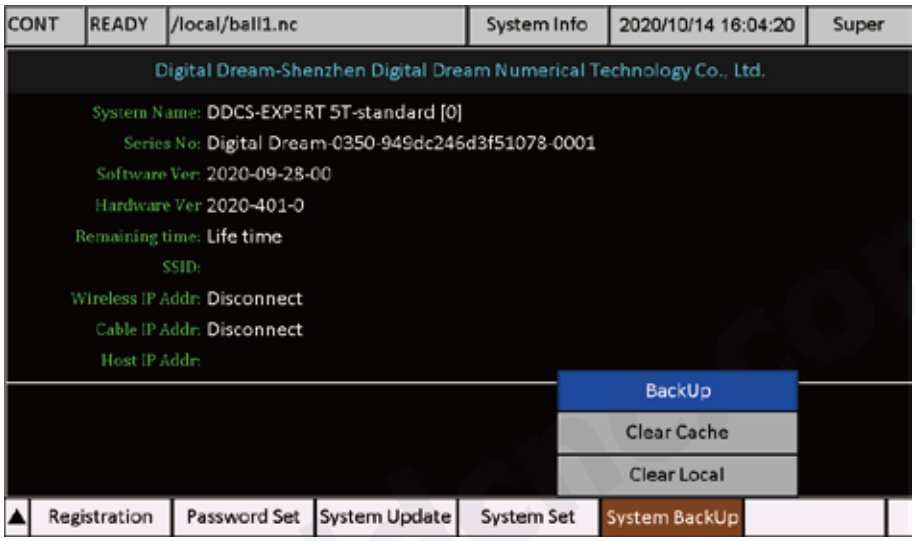


Figure 8-46 The System BackUp Page

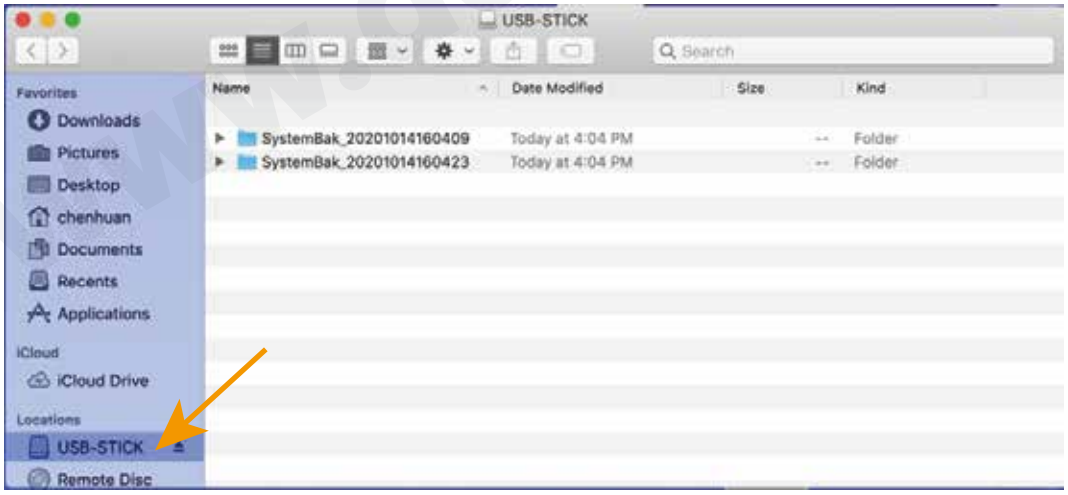
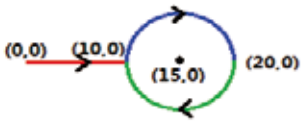
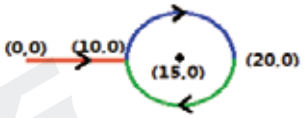
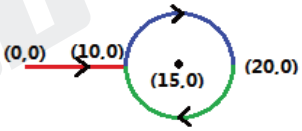
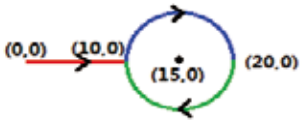


Figure 8-47 After the System BackUp, the install file is saved in the root directory in the USB-Stick

9 G Code and M Code

Command	Options	Description	Example of use	Description of the example
G0, G00	X Y Z A	Moves the axes to the point X Y Z A, at the speed specified in # 80	G0 X10 Y10 Z1	Quickly moves the axes to the point X10 Y10 Z1
G1, G01	X Y Z A	Moves the axes in line to the point X Y Z A, at the speed specified in F. If F is not specified, the speed from parameter # 76 is used.	G0 X10 Y10 Z1 F100	Moves the axes to the point X10 Y10 Z1 at a speed of 100
G2, G02 (mode1)	X Y Z I J K	Moving along the arc, clockwise, specified the center, at the speed specified in F. I, J, K are the coordinates of the arc center (x, y, z), relative to the end point (for G91.1) or in absolute coordinates G90.1), K can be omitted. X, Y is the end point of the arc. Z - for plunging into a spiral (end infeed). The starting point of the arc is given by the preliminary movement of the axes into it.	G0 X0.00 Y-50.00 ---------- - - - - - G2 X100.00 Y-50.00 I50.00 J0 F100	Draws half the circle, D = 100, from 0 to 180 degrees, clockwise, at a speed of 100
G3, G03 (mode1)	X Y Z I J K	Moving along the arc, counterclockwise, specified the center, at the speed specified in F. I, J, K are the coordinates of the arc center (x, y, z), relative to the end point (for G91.1) or in absolute coordinates G90.1), K can be omitted. Z - for plunging into a spiral (end infeed). The starting point of the arc is given by the preliminary movement of the axes into it.	G0 X100.00 Y-50.00 ---------- G3 X0.00 Y-50.00 I-50.00 J0 F100	Draws half the circle, D = 100, 180 to 0 degrees, counter-clockwise, at a speed of 100
G2, G02 (mode2)	X Y Z R	Moving along an arc, clockwise, specified the radius, at the speed specified in F. R is the radius of the arc. X, Y is the end point of the arc. Z - for plunging into a spiral (end infeed). The starting point of the arc is given by the preliminary movement of the axes into it.	G0 X0.00 Y-50.00 ---------- G2 X100.00 Y-50.00 R50 F100	Draws half the circle, D = 100, from 0 to 180 degrees, clockwise, at a speed of 100
G3, G03 (mode2)	X Y Z R	The movement along the arc, counterclockwise, specified the radius, with the speed specified in F. R is the radius of the arc. X, Y is the end point of the arc. Z - for plunging into a spiral (end infeed). The starting point of the arc is given by the preliminary movement of the axes into it.	G0 X100.00 Y-50.00 ---------- G3 X0.00 Y-50.00 R50 F100	Draws half the circle, D = 100, 180 to 0 degrees, counter-clockwise, at a speed of 100
G4, G04	P	Stops processing for the number of milliseconds specified after P. In this case, the machine does not stop the spindle and does not pick up the tool	G4 P10000	Stops processing for 10 seconds
G17		Selecting the working plane X-Y	G17	Select the working plane X-Y
G18		Selection of working plane Z-X	G18	Select of working plane Z-X
G19		Selection of working plane Y-Z	G19	Select of working plane Y-Z
G20		Inch system selection	G20	Inch system selection
G21		Choice of metric system	G21	Metric system selection

Command	Options	Description	Example of use	Description of the example
G28	X Y Z A	Go back to the reference point. Works only with G91. The specified axes, first move to the specified point, then to the machine 0. If 0 is specified, then immediately into the machine zero. The not specified axes do not move.	G91 G28 X10 Y0 Z0	The X axis will first move 10mm to the right, then the XYZ axes will go to the machine axis 0. Axis A does not move.
G40	NO	Cancel tool radius compensation. The function does not work yet.	G40	Cancel tool radius compensation.
G41	D	Compensate the tool radius to the left of the path. D - is the tool number from the table.	G40	Compensates the tool radius 1, to the left of the path.
G42	D	Compensate tool length positively. H - the number of the instrument according to the table. The function does not work yet.	G42 D1	Compensates the length of tool 1 positively.
G43	H	Compensate tool length positively. H - the number of the instrument according to the table.	G43 H1	Compensates the length of tool 1 positively.
G44	H	Compensate for the length of the instrument is negative. H - the number of the instrument according to the table.	G44 H1	Compensates the length of tool 1 negatively.
G49	H	Cancel tool length compensation.	G49	Cancel tool length compensation
G53	H	malfunctioning, working analog G153	G44 H1	Compensates the length of tool 1 negatively.
G54 - G59	X Y Z A	Selecting the coordinate system	G54	Selecting a coordinate system
G73	X Y Z R Q I K	The cycle of step drilling with the full output of the drill, with the speed F. X, Y - the coordinates of the center; Z - is the distance from R to the bottom of the hole; R - drilling depth (usually, 0); Q - is the step size; I - distance of failure to return to G0; K - is the number of repetitions. The drill is retracted and fed at a speed of G0, which can be limited by parameters # 78 and # 79.	G83 X10 Y5 Z-7 R0 Q1,4 I0 K1 F300	Drills the hole at point X10 Y5, from 0 to 7mm, at a speed of 300 mm / min. The step size is 1.4mm, thus 5 steps are done. After each step, the drill is retracted by 1mm.
G81	X Y Z R K	Drilling in 1 pass, with speed F. X, Y - coordinates of the center; Z is the distance from R to the bottom of the hole; R - drilling depth; K is the number of repetitions.	G81 X10 Y5 Z-7 R0 K1 F300	Drills the hole at point X10 Y5, from 0 to 7mm, at a speed of 300 mm / min.
G82	X Y Z R K P	Drilling in 1 pass with a delay at the end (for better processing of the bottom), with speed F. X, Y - coordinates of the center; Z - is the distance from R to the bottom of the hole; R - drilling depth; K - is the number of repetitions, P - is the delay in milliseconds.	G82 X10 Y5 Z-7 R0 K1 P2000 F300	Drills the hole at point X10 Y5, from 0 to 7mm, at a speed of 300 mm / min. At the bottom of the drilling, the pause is 2 seconds.
G83	X Y Z R Q I K	The cycle of step drilling with the full output of the drill, with the speed F. X, Y - the coordinates of the center; Z - is the distance from R to the bottom of the hole; R - drilling depth (usually, 0); Q - is the step size; I - distance of failure to return to G0; K - is the number of repetitions. The drill is retracted and fed at a speed of G0, which can be limited by parameters # 78 and # 79.	G83 X10 Y5 Z-7 R0 Q1,4 I0 K1 F300	Drills the hole at point X10 Y5, from 0 to 7mm, at a speed of 300 mm / min. The step size is 1.4mm, thus 5 steps are done. Failure = 0, in this way the drill is returned on fast feed to the end point of the previous step.G80

Command	Options	Description	Example of use	Description of the example
G74	X Y Z R M	Tapping of right hand threads to be done with M3 spindle rotation.	M03 M8 (Speed & Feedrate) S400 F20 (Tapping) Z1.0 G00 X0.0 Y0.0 G01 M29 G84 Z-0.5 R0.2	we want to tap a 1/4-20 thread 0.500" deep at 0, 0. Here's the code to do that with G84 G Code.
G84	X Y Z R M	Tapping of right hand threads to be done with M3 spindle rotation.		
G90	No	For G0/G1:Specifying absolute coordinates; For G2/G3:The main coordinates are absolute and the centers of the arcs are relative.	G90 G1 X10 Y0 G90 G2 X20 I5	
G91	No	For G0/G1:Specifying relative coordinates; For G2/G3:The main coordinates are relative and the centers of the arcs are relative.	G90 G1 X10 Y0 G91 G2 X10 I5 G2 X-10 I-5	
G90.1	No	For G0/G1:Specifying absolute coordinates; For G2/G3:The main coordinates are absolute and the centers of the arcs are absolute.	G90 G1 X10 Y0 G90.1 G2 X20 I15 G2 X10 I15	
G91.1	No	For G0/G1:Specifying relative coordinates; For G2/G3:The main coordinates are relative and the centers of the arcs are absolute.	G90 G1 X10 Y0 G90.1 G2 X20 I15 G2 X10 I15	
G92	X Y Z A	Setting new current coordinates	G90G92X0Y0Z0A0	Zero all axes
G98	No	After drilling cycles, the tool returns to the Z position, before the start of the cycle. Raises the Z axis to a safe height..	G98 ------- G1 Z1 F1000 ------- G81 X0 Y0 Z-7 R0 K1 F300	After drilling, the tool will be in position 1 to Z
G99	No	After the drilling cycles, the tool returns to the point R (along the Z axis). Raises the Z axis to a safe height.	G99 ------- G1 Z1 F1000 ------- G81 X0 Y0 Z-7 R0 K1 F300	After drilling, the tool will be in position 0 to Z

Command	Options	Description	Example of use	Description of the example
M0, M00	No	Stopping the program, before pressing the "START" button, is completely the same as pressing the "PAUSE" button.	M0	Stops the program, before pressing the "START" button. Raises the Z axis and sets the spindle, if it is set in the settings.
M01	No	Optional Stop: Operator Selected to Enable	M01	Stops the machine unless there is further interaction from the User.
M3, M03	S	Start spindle rotation with speed S	M3 S2000	Starts the spindle at a speed of 2000 rpm
M4, M04	S	Start spindle rotation with speed S in CCW direction	M4 S2000	Starts the spindle at a speed of 2000 rpm in CCW
M5, M05		Stop the spindle	M5	Stops the spindle
M6, M06	T	Plays the contents of the T.nc. file Specifies the tool number for offsets. T specifies the number of the tool (it can be omitted).	M6 T5	Replaces the tool with T5
M8, M08	No	Switch on spindle cooling	M8	Switch on spindle cooling
M9, M09	No	Switch off spindle cooling	M9	Switch off spindle cooling
M10	No	Turn on the coolant pump	M10	Turn on the coolant pump
M11	No	Turn off the coolant pump	M11	Turn off the coolant pump
M30	No	End of the program, cancels all commands and loops. Do not use immediately after M6.	M110	It stops the program, before pressing the "START" button. Peep 3 times with built-in peepal
M47	No	Repeat program from first line.	M47	Restart Program Execution
M50 / M51		Output 01 Open / Close		Control the Output 01
M52 / M53		Output 02 Open / Close		Control the Output 02
M54 / M55		Output 03 Open / Close		Control the Output 03
M56 / M57		Output 04 Open / Close		Control the Output 04
M58 / M59		Output 05 Open / Close		Control the Output 05
M60 / M61		Output 06 Open / Close		Control the Output 06
M62 / M63		Output 07 Open / Close		Control the Output 07
M64 / M65		Output 08 Open / Close		Control the Output 08
M66 / M67		Output 09 Open / Close		Control the Output 09
M68 / M69		Output 10 Open / Close		Control the Output 10
M70 / M71		Output 11 Open / Close		Control the Output 11
M72 / M73		Output 12 Open / Close		Control the Output 12
M744 / M75		Output 13 Open / Close		Control the Output 13
M76 / M77		Output 14 Open / Close		Control the Output 14
M78 / M79		Output 15 Open / Close		Control the Output 15
M80 / M81		Output 16 Open / Close		Control the Output 16
M82 / M83		Output 17 Open / Close		Control the Output 17
M84 / M85		Output 18 Open / Close		Control the Output 18
M86 / M87		Output 19 Open / Close		Control the Output 19
M88 / M89		Output 20 Open / Close		Control the Output 20
M90 / M91		Output 21 Open / Close		Control the Output 21

Command	Options	Description	Example of use	Description of the example
M98		Call a Subprogram with the reference to the separate program created and loaded on the controller.	M98 Pxxxx Ln	xxxx is the line number, nn is the number of repetitions
M99		End Sub-Program or Return or Loop	O01234 ... (Part program) ... M98 P111 (Jumps to program O00111 to run) ... (The M99 at the end of the sub-program will jump back here) ... (Finish part) M30 (End of main program)	This M-code is used to end the sub-program. If M99 is used in the main program, it will cause the program to loop back to the beginning and repeat over and over again without stopping.
F	No Application	Sets the speed of the working feed, for many commands. You can write, as at the end of the line with the command, and a separate line. If F is not specified anywhere, the speed from parameter # 76 is used.	F100 ---------- G1X10.5	Moves the X axis to, at point 10.5, at a speed of 100.
P	No Application	Specifies the pause time, in milliseconds, for the G4 and G82 commands. You can write, as at the end of the line with the command, and a separate line.	P2000 ---------- G4	Pauses the program for 2 seconds
S	No Application	Specifies the spindle speed for the M3 command. You can write, as at the end of the line with the command, and a separate line.	S21000 ---------- M3	Starts the spindle at a speed of 21000 rpm
■	No Application	The symbol for dividing the whole and fractional parts of numbers. Comma - does not work.	G0 X10.5	Moves the X axis on fast feed, to the point 10.5. Option G0 X10.5 - will not work.
SIN	[n]	The sine of the parameter n, in degrees.	#1=SIN[30.0]	
COS	[n]	The cosine of the parameter n, in degrees.	#1=COS[60.0]	
TAN	[n]	The tangent of the parameter n, in degrees.	#1=TAN[45.0]	
SQRT	[n]	The square root of the parameter n.	#1=SQRT[2.0]	
ATAN	[n1,n2]	returns the angle between the ray to the point (n1,n2) and the positive x-axis, confined to (-180, 180].	#1=ATAN[30,10]	
ABS	[n]	returns the absolute value of that parameter n.	#1=ABS[-30.1]	