SV2414 / SVL2416
MAINTENANCE MANUAL

Chapter 1

Routine Maintenance

1-1 Routine Maintenance Checklist

Routine maintenance inspection can be classified into daily, weekly, monthly and annually. The maintenance is given when the machine functions normally The actual frequency and the operation frequency are different. When operating the both, pay attention to abnormal noises, oil volume, air pressure and machine abnormalities during the processing.

○：Check ◎：Add oil ⊙：Clean

A：Adjustment F: Function check G: Grease coasting R: Replace when necessary

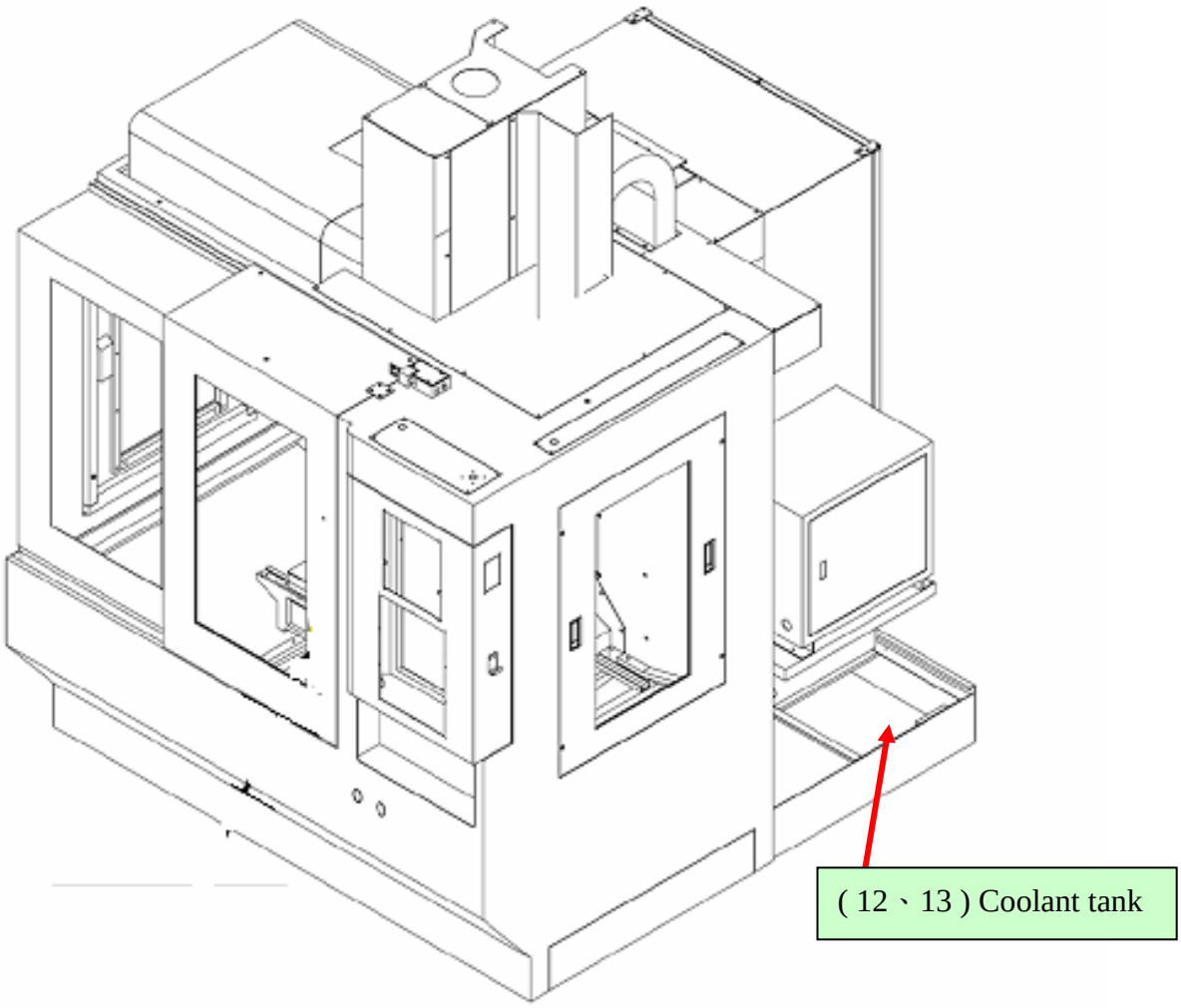
No.	Maintenance		Daily	Weekly	Monthly	Annually	Note
1	Spindle unit	Spindle operation warm-up	○				
2		Spindle nose	⊙				
3	Lubrication unit	Lubrication pump	⊙				Depending on the brand of the lube oil
4		Lubrication pump oil volume		◎			
5	Spindle oil coolant unit	Spindle oil coolant unit			○	⊙	
6		Temperature difference setup	○				-1°C
7		Oil coolant unit filter		⊙			
8	Air Compressor Unit	Air compressor source pressure	○				5.5~7 kgf/cm²
9		Air compressor cylinder and filter drainage	○				
10		Air compressor unit lube oil		◎			
11		Air leakage of the air compressor system	○				
12	Coolant	Coolant capacity	○				
13		Coolant box filter		⊙			

No.	Maintenance		Daily	Weekly	Monthly	Annually	Note
14	Tool magazine automatic tool change (ATC) system	Internal flow of the tool change machine				R/ 1-year	Depending on the brand of the lube oil
15		Tool sleeve and tool change claw	⊙				
16	CNC operation panel	Emergency stop button	○,F				
17		Operation button	○,F				
18	Electric box	Spindle cooling fan	○/⊙				
19		CNC axial battery		○		R	
20	Machine static precision check					○,A	
21	Machine position precision check					○,A	Replace
22	Oil skimmer		○,F		⊙		

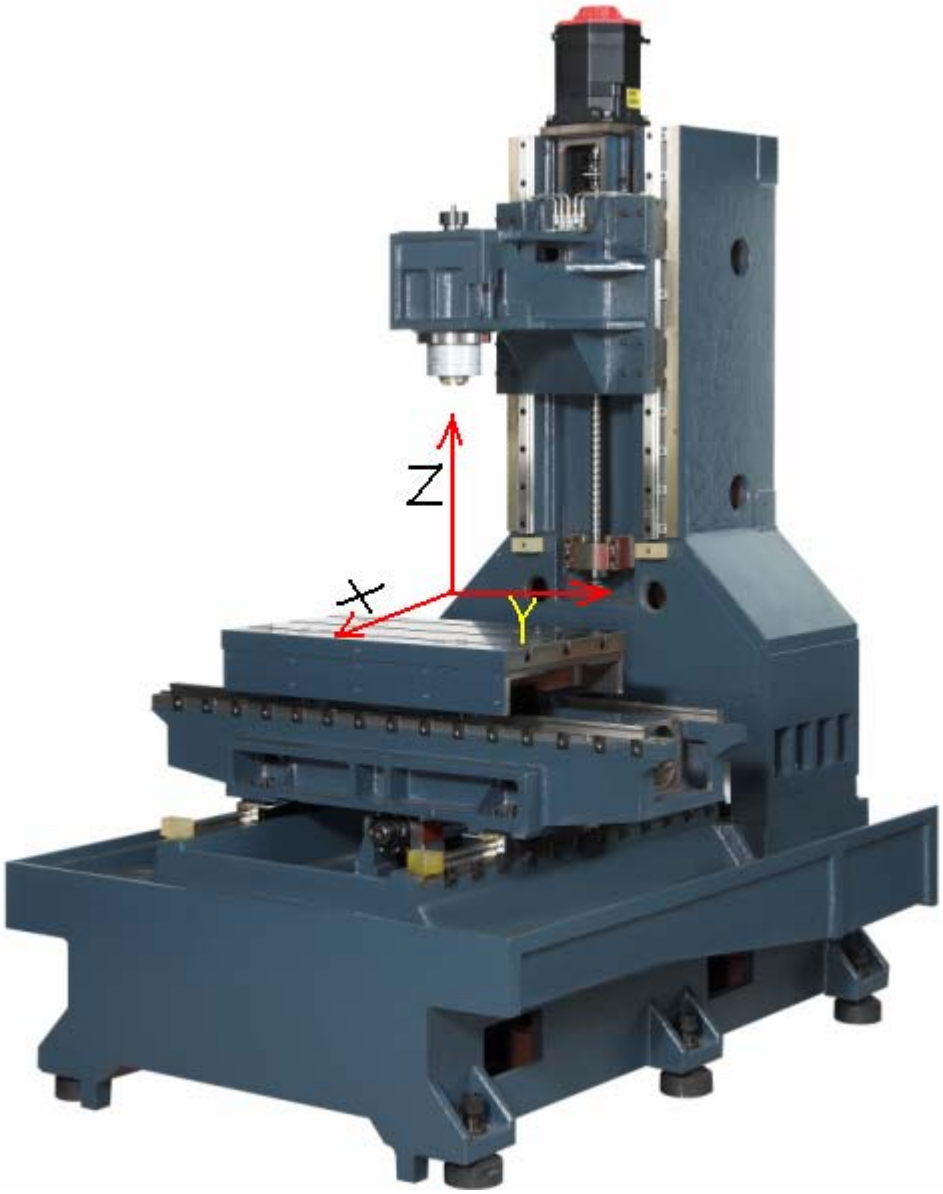
Note: The schedule presented in table is for normal operation only. If the environment is not optimal or if the usage frequency has exceeded the suggested basic standard, you may need to increase the frequency of maintenance

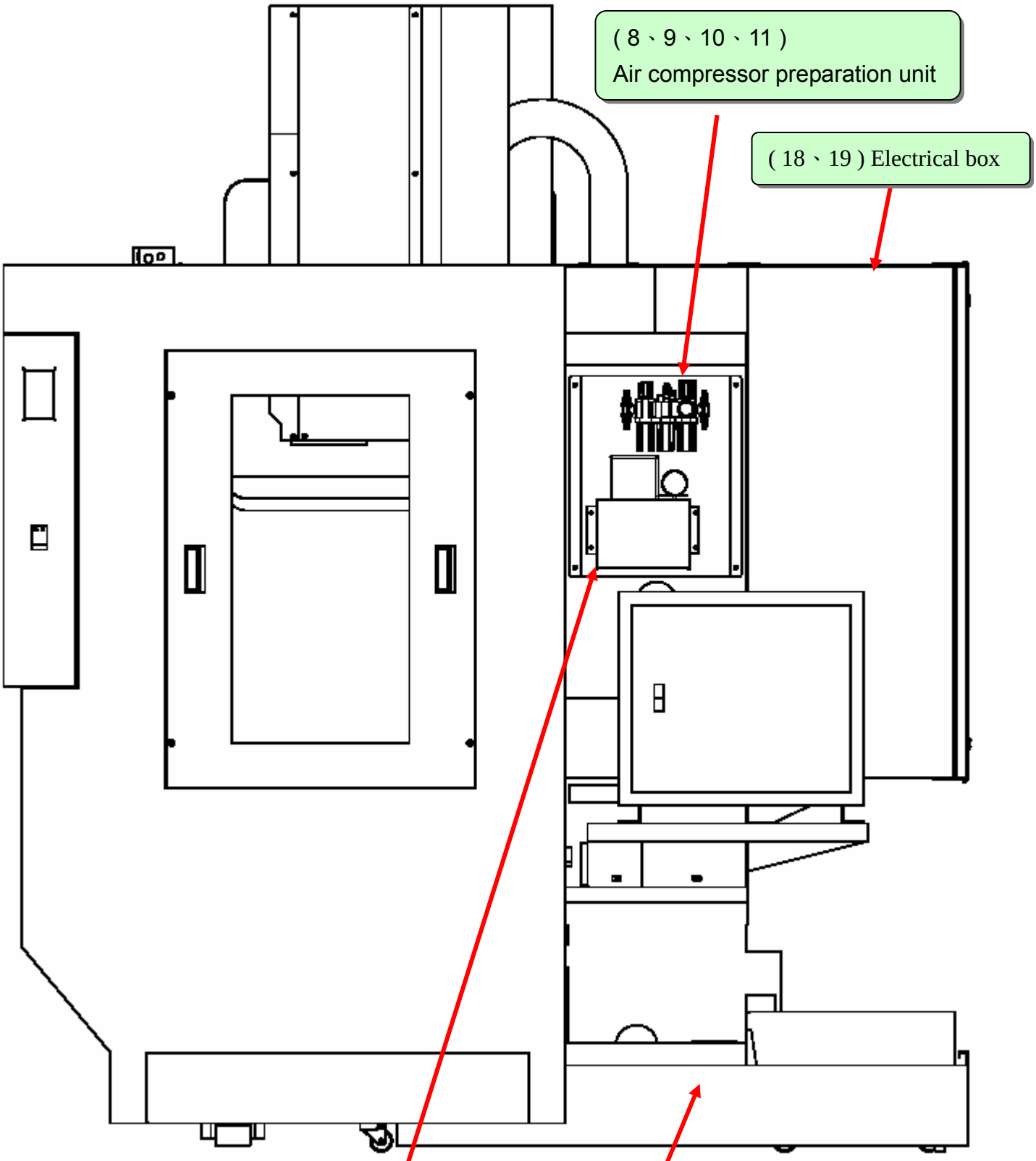
Note 2: If there is any inconsistency between this table and instruction from elsewhere, use the information provided on this table.

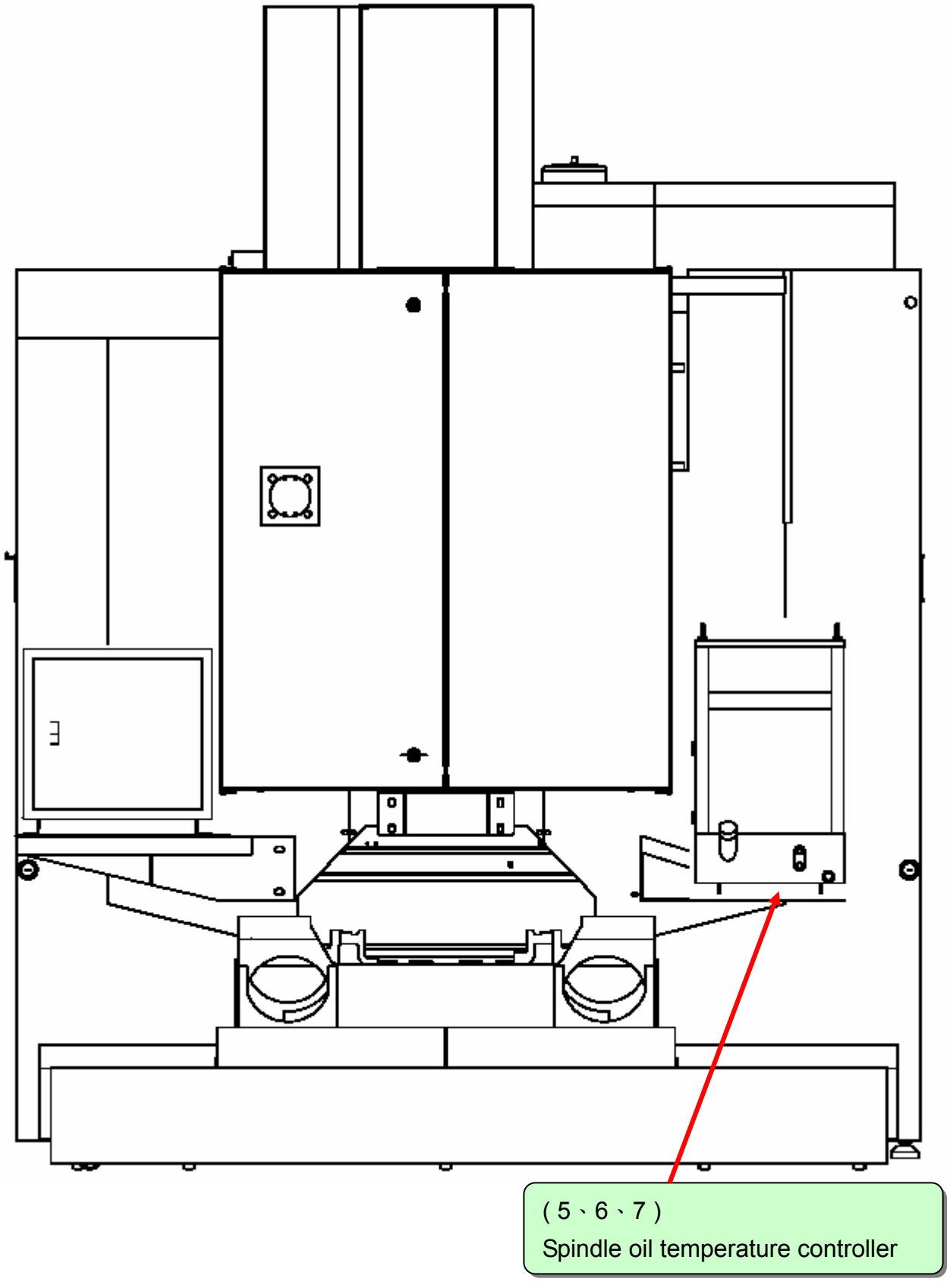
1-2 Routine Maintenance Instruction Diagram


(12、13) Coolant tank

Machine precision check (20 – 21)




(8、9、10、11)
Air compressor preparation unit
(18、19) Electrical box
(3、4)
Central lubrication system
(23) Oil skimmer


(5、6、7)
Spindle oil temperature controller

1-3 Cautions for Machine Maintenance

It is important to perform maintenance on each listed items in order to maintain machine precision and extend machine lifespan.
Erroneous operation may damage the machine or lead to injury of the operator. Be cautious about the following matters：

Cautions	Prevention
Spindle operation warm-up	To extend the lifespan, follow the instructions given in Section 2-3 for machine warm-up before turning on the machine. If the machine has been stopped for a long period, follow the instructions given in Section 2-3 for machine warm-up before turning on the machine in order to extend the life span.
Dynamic balance of the cutter	To extend the lifespan as well as to increase the precision, please follow instructions given in Section 2-1 for usage as well as achieving a dynamic balance for the cutter.
Standard broach bolts	Broach bolt is an important component jointing the spindle and the tool shank. Using a wrong broach bolt is hazardous for the operation. Make sure that only the standard one is used.
Spindle pre-operation check	Check the machine before spindle operation according to instructions given in Section 2-4 in order to extend the lifespan of the spindle.
Manually loosening the spindle's cutter	When loosening the cutter of the spindle, make sure that each step is executed correctly or may lead to personnel injury.
Manual installing the cutter onto the spindle	When changing the cutter, be cautious not to drop the cutter, which may damage the workbench or tools. Manually change the cutter only if necessary, and the procedures are written on the operation panel at the front. Press and let go of the spindle cutter loosening button to loosen up or to grasp the cutter.
Do not touch the spinning tool magazine	The tool magazine is controlled by CNC and motor. After turning on the machine, do not touch the tool magazine or may be injured. Turn off the power for changing the cutter.

Cautions	Prevention
Spindle nose cleaning	Keep the taper of spindle nose and cutter shank clean at all time to prevent dust or iron filings affecting the precision. The spindle has an automatic dust removal function, but the operator should wipe the parts with standard air-laid paper for maintaining spindle precision.
Air quality inside the air compressor	The function pressure of this air compressor is 5kgf/cm²(71 psi). Therefore the air pressure source has to be at least 5.5 kgf/cm²(78 psi) and stabilized. This machine has an air filter that eliminates impurities and water vapor from the air. Air for the air pressure source has to be clean and dry. Check the filter constantly to extend the lifespan of the filter. To effectively eliminates water vapor by the filter, drain the water from the air tank of the air compressor. (Water drainage is more effective in the morning).
Lube oil from the lubricating pump (automatic)	Lube oil is released by the lubricating pump to lubricate the guide ways and the lead screws. Insufficient lubrication may speed up the abrasion and affect machine precision. Always make sure that there is enough oil in the tank. If the amount of oil is below the lower limit of the oil-level gauge, the operation panel as well as the screen will issue a warning message. Please add more oil as early as possible.
Adding oil to the spindle oil cooler and setting up the temperature difference	Oil cooling circulation system was adopted for the spindle to effectively suppress spindle temperature increase and thus extend the lifespan of the spindle. Use the correct type of circulation and add more oil when the level is too low. The machine adopts synchronous machine temperature adjustment Temperature difference is set to 0 to prevent big temperature difference, which can damage spindle bearing. Do not change the value arbitrarily.

Cautions	Prevention
Guide way check	Constantly check the guide ways of the three axes (X, Y & Z). Check whether there are iron filings or other types of grains attached to the guide ways. To extend the lifespan, remove the dust or they may scratch the slide.
Correctly locking the work pieces	When placing work pieces onto the workbench, make sure that they are well locked or they may spin out and cause personnel injury.
Correctly locking the doors	Make sure the operation door and the two side doors are well locked to prevent the cutter from bursting apart or iron filings from flying out, which is dangerous.

Chapter 2

The Spindle Unit

2-1 Cautions for High Speed Processing

During high-speed processing (S = 8000 rpm or above; F = 300 mm/min or above), the shank and the style of the cutter has a critical impact on spindle lifespan and processing precision.
Matters that need more attention are:

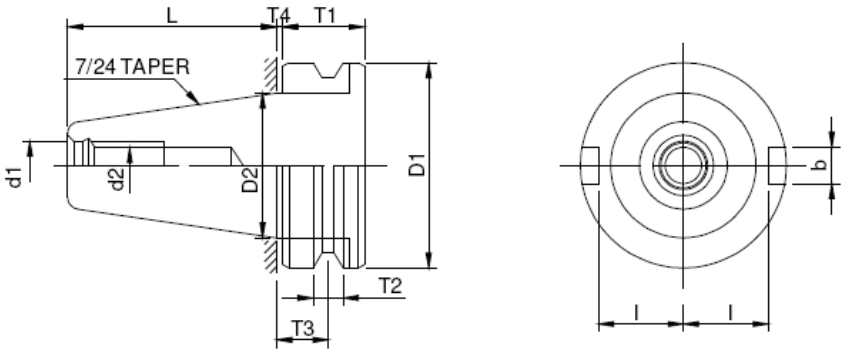
1). Grasp the cutter before running the spindle to avoid damaging the spindle.
2). During high-speeding chipping (8000rpm or above), use only G2.5 level cutters and shanks that have been calibrated for dynamic balance. The reason is that vibration generated from centrifugal force may damage the spindle bearing and rapidly wear out the cutter.
3). Balance tolerance for the shank and the cutter combination is affected by the following three factors：rotation speed of the cutter, balance tolerance of the spindle, and weight of the shank. Therefore, using a shorter cutter with a smaller diameter for high-speed chipping is advantageous in terms of spindle vibration, temperature increase, thermal deformation, and processing precision.
4). The geometric shape of the cutter blade has an effect on cutter abrasion. Increase the angular backlash of the blade can reduce abrasion due to scratches. Suitable cutter materials can also minimize abrasion during high-speed processing; for example, cermets, tin or tic coated carbide cutters have longer lifespan than conventional cutters. Ceramic (Si_3N_4) cutters are better than cermets cutters.
5). Perform another dynamic balance calibration after combining the cutter and the shank together The dynamic balance standard should be G2.5 or above.

Balance level	50 – 6000 rpm	Level G6.3
	6000 – 18000 rpm	Level G2.5

Spindle Rotation Speed (rpm)	Cutter Diameter (mm)	Cutter Length (mm)
6000 – 8000	125	250
8000 – 10000	100	250
10000 – 12000	80	250
12000 – 15000	65	200
15000 – 18000	50	200

2-2 Tool Shank and Broach Bolts

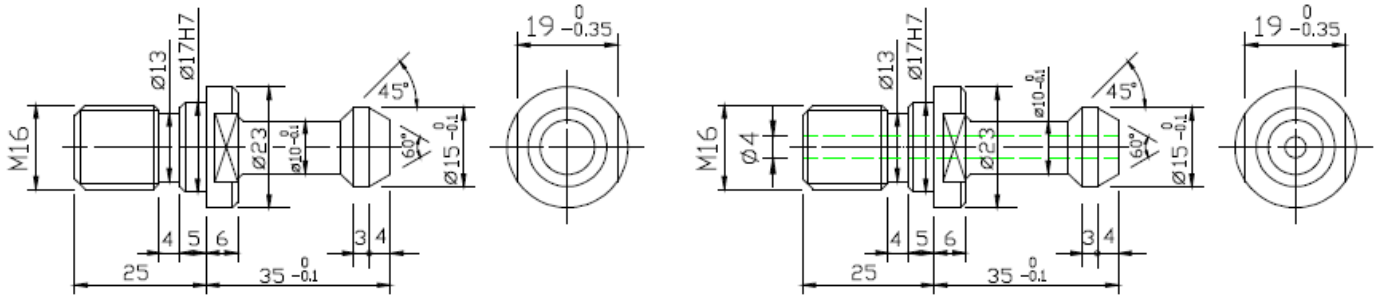
BT SHANK



Unit ：mm

Model No.	D1	D2	d1	d2	L	T1	T2	T3	T4	b	l
BT 40	63	44.45	17	M16	65.4	25	10	16.6	2	16.1	22.6

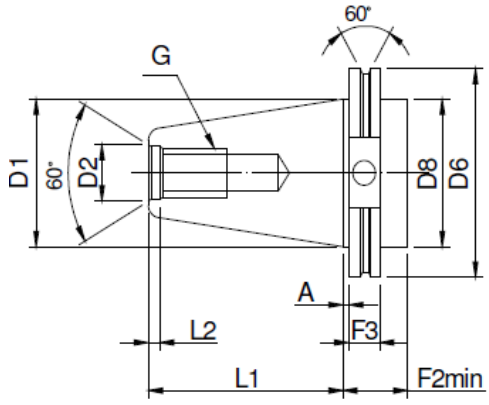
BT40 STUD



Cooling though spindle

CAT SHANK

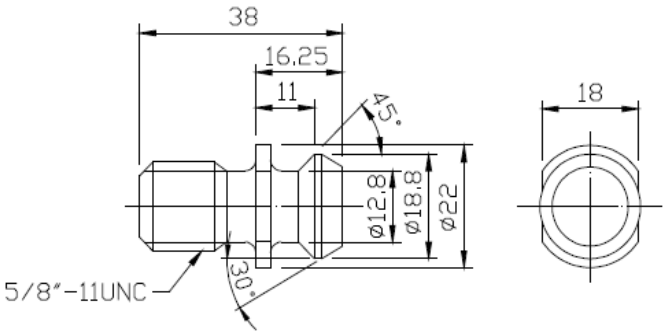
(ANSI B5.50-78)

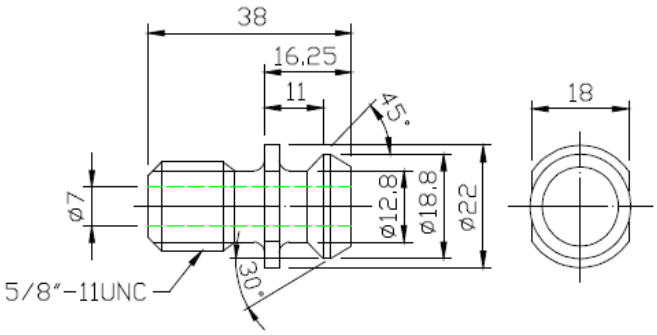


Unit：mm

Model No.	D1	D2	D6	D8	L1	L2	F2	F3	A	G
CAT 40	1.750 (44.45)	0.641 (16.28)	2.5 (63.05)	1.75 (44.45)	2.687 (68.25)	0.188 (4.78)	1.375 (35.00)	0.625 (15.88)	0.125 (3.18)	5/8-11 thread

CAT 40 STUD





Cooling though spindle

2-3 Spindle Warm-up

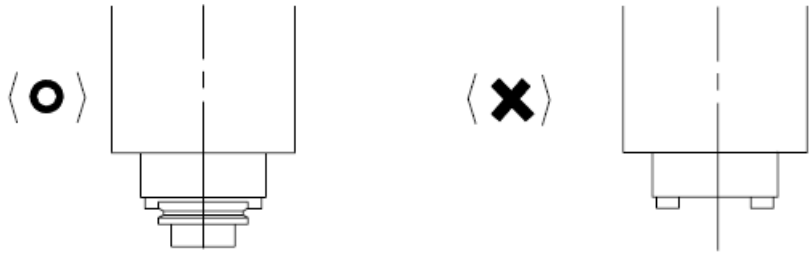
This is an aerosol lubrication design, and so before turning on the machine, follow the warm-up steps below to extend the lifespan of the spindle and to avoid damaging the bearing by letting the oil aerosol getting into the bearing ahead.

Spindle warm-up time

Item	Condition	Spindle RPM (Maximum RPM %)	Rotation Time (min)	Check Contents
1	Routine operation	Max Rpm 20%	10	1. Less than 10°C increase of temperature 2. Shocks 3. Noises
2	If the spindle has been stopped for more than two hours, warm-up is compulsive.	20%	10	1. Less than 10°C increase of temperature Shocks 2. Noises
3	Spindle stopped for more than 72 hours	1. 20% 2. 50%	10 10	1. Less than 10°C increase of temperature 2. Shocks 3. Noises
4	Spindle stopped for more than 2 weeks	1. 500 RPM 2. 20% 3. 40% 4. 60% 5. 80% 6. Maximum RPM	60 10 10 *10 *10 *10	Less than 10°C increase of temperature Shocks Noises 4. Do not go onto the next step until the temperature has become stable.

Cautions

1) If the shank is not installed onto the spindle, do not operate the spindle. See the figure below.



If the temperature of the bearing has increased for more than 10%, reduce the rotation speed of the spindle to 500 and do not carry out the warm-up until the temperature is about 5°C higher than the room temperature.

2-4 Spindle Pre-Operation Check

1) Check whether the air pressure source is normal.
2) Check No.7 electric oil pump tank (Refer to the diagram in Section 3-1) for whether there is enough oil.
3) Run the spindle at 100 rpm to check whether the oil pump is functioning normally.
4) If everything is okay, carry out the warm-up according to the spindle warm-up procedures presented in Section 2-3.

 Caution

1. The oil volume and air supply volume of this aerosol lubrication unit has already been set at the factory. Do not adjust the values arbitrarily.
2. During the maintenance, do not disassemble, bend, or flatten the output oil pipe, which may damage the oil pipe.
3. Timely fill up the oil tank.
4. If the spindle has been stopped for more than 2 hours, run the spindle system, including the aerosol lubrication system, at a speed that is 20% of the maximum speed.
   The machine has to have a forced warm-up for 10 minutes before starting the actual work.
5. Do not adjust the oil/air mixing valve. Use the factory-set default value for lubrication interval. Do not change the values arbitrarily or the bearing may be damaged.
6. For this unit, if the screen displays “ALARM 1002, 1056, 2004,” check and turn off the alarm before continuing the operation.
7. Do not turn on and off the spindle repeatedly in a short period of time during the processing. The inside of the spindle may be over-lubricated, which can cause the spindle overheated.
8. If the spindle alarm went off when starting the operation, it is due to overheated bearing. Stop the operation. Wait until the temperature returns to normal. Check if there is any problem before starting the warm-up and the operation.
9. It is normal to have oil dropped onto the workbench or the processing item when running the machine. This comes from the exhausted oil aerosol. Do not block the outlet or the back pressure may lead to temperature increase, thereby damaging the spindle bearing.

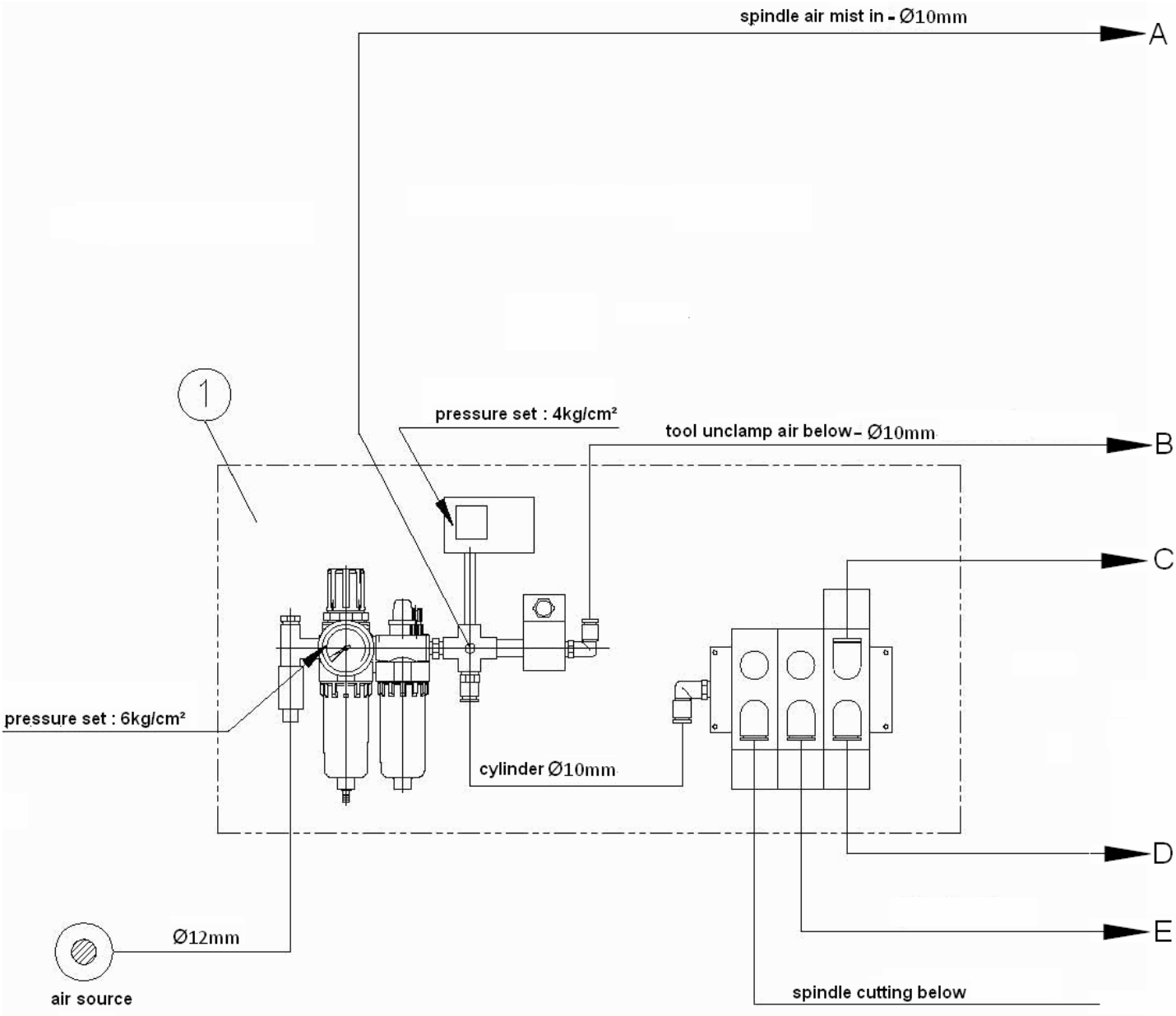
2-5 Spindle Alarm Handling

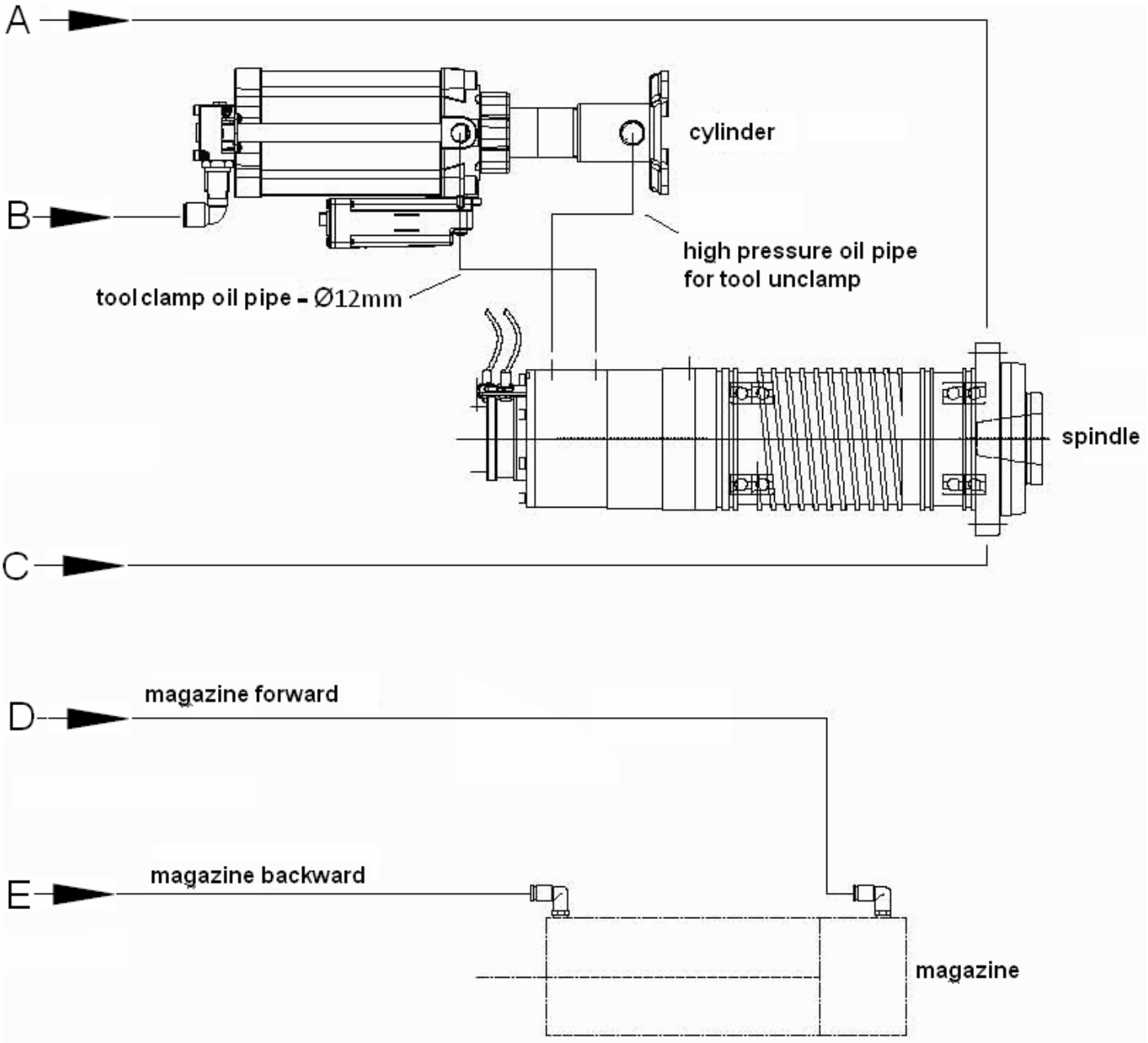
1) For this unit, if the screen displays “ALARM 1002, 1056, 2004,” check whether the temperature of the protective device of the bearing is 50°C. After checking the temperature, check whether the pipelines are abnormal, squeezed or broken. Check whether the oil cooler and the air compressor are normal. Wait for the temperature to drop before turning on the spindle for warm-up. This is the first line of protection for the spindle.
2) The second spindle protection is the oil pump. The oil pump has liquid level detection, and when the level is too low, ALARM will be issued. If the oil pipeline is blocked, the screen will display ALARM 1002.
3) The third spindle protection is the pressure switch for air pressure supply. If there is insufficient air pressure, the mixing valve will be incapable of emitting the oil aerosol. If this ALARM goes off, check whether the oil pipeline is broken and whether air pressure supply is sufficient.
4) If the clients still cannot turn off the alarm after checking relevant pipelines, contact the service department of the Company. Do not modify the setting arbitrarily.
5) Messages of the ALARM signals:
1. 1002 LUBE PRESSURE FAULT：Insufficient pressure from the automatic lubrication pump
2. 1056 SPINDLE COOLER ALARM：Abnormal spindle oil cooling function
3. 2004 LUBE LEVEL ALARM：Under the standard liquid level of the oil tank of the automatic lubrication pump
6) For relevant settings, please refer to the electrical instructions.

Chapter 3

The Air Compressor Unit

3-1 Air Compressor System Layout




A
B
C
cylinder
high pressure oil pipe
for tool unclamp
tool clamp oil pipe - Ø12mm
spindle
D
magazine forward
E
magazine backward
magazine

3-2 Air Compressor Unit Usage Instruction

1) The pressure source of this machine's pressure pipeline goes through the air pressure preparation unit, and the pressure is set to 6kg/cm^2. When the pressure source is less than 5kg/cm^2, and the duration lasts more than 2 – 3 sec, NC will turn on the alarm, and CRT will display 1011 AIR PRESSURE LOW. In this case, check the pressure source and then press the RESET button to continue the operation.
2) Maximum pressure: 10kgf/cm²
3) Adjustment range: 0.5 ~8.7 Kg f/ cm²
4) Filter size：5μ
5) Recommended lube oil：ISO VG32
6) The pressure source should not be lower than: 5.5kgf/cm²
7) Although the air compressor preparation unit of the water filter has an automatically water drainage, the machine operator should check whether there is any water accumulation at the shift or at the end of the day. If the water drainage function is abnormal, water vapor may get into the air compressor component, thereby reducing the lifespan. For the water drainage function of the filter to be more effective, water in the air tank of the air compressor should fully drained out. It is also recommended to install a dehumidifier at the pressure source to make sure that the air is dry.
8) The machine operator should pay attention to whether the pressure pipeline is making a bz kind of noise. This may be due to gas leakage at the connector. Check the machine by tracing the noise. The pipeline of this machine is connected by connectors, and the disassembling method is described below: The nylon tube can be directly inserted into the connector. Use the finger or a flat screw driver to secure the connector when pulling out the nylon tube.

3-3 Air Compressor Unit Troubleshooting

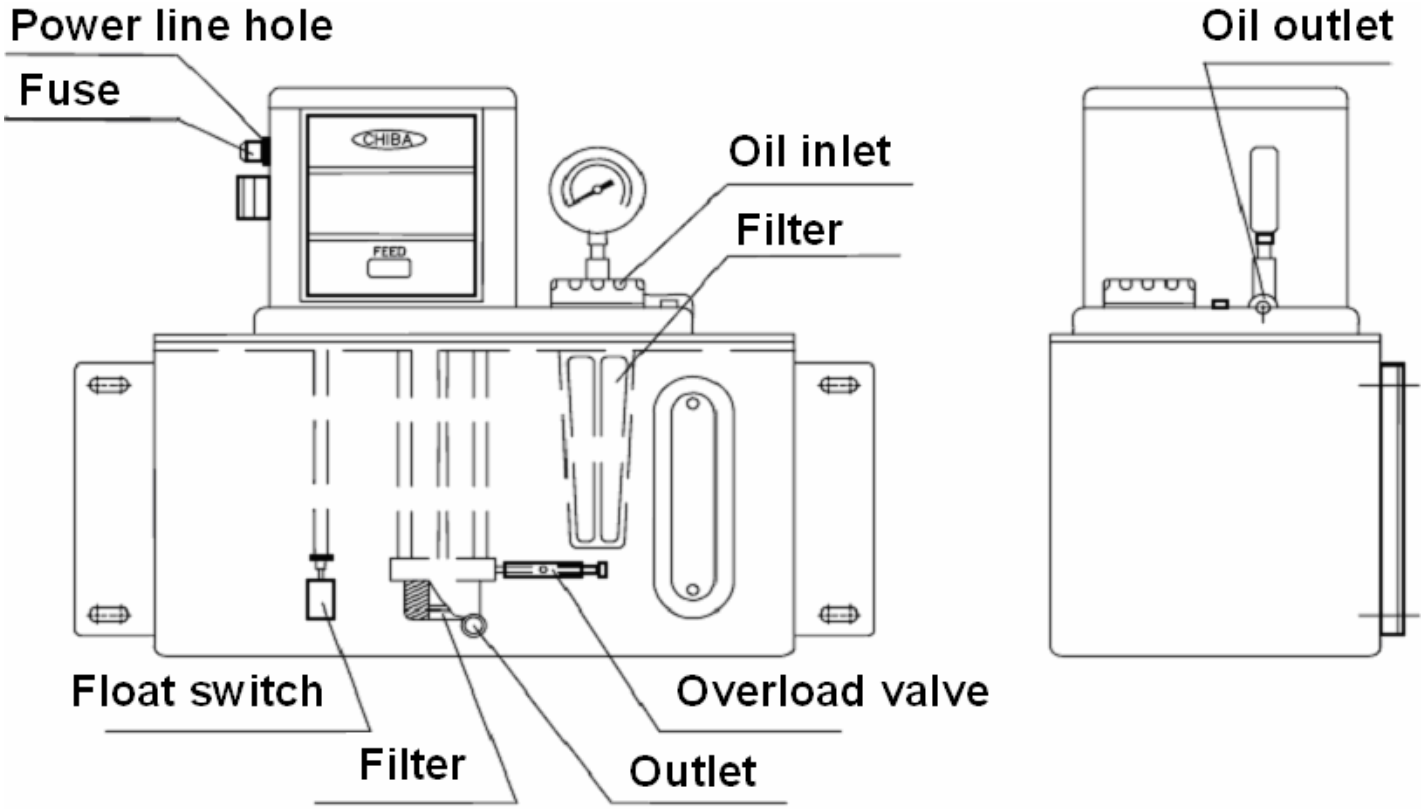
Problem	Cause	Correction
Reduced pressure source output	Blocked filter	Clean the filter
Gas leakage	1. Loosened nut 2. Broken O-ring 3. Broken tube	1. Tighten the nut 2. Replace the o-ring 3. Replace the tube
Disfunctional pressure regulator	Broken pressure regulator 1. Broken spring of the pressure regulator 2. Broken spring of the valve 3. Dust or dirts on the base of the valve 4. Broken lining of the valve 5. Broken film	1. Replace the spring 2. Replace the spring 3. Clean the valve and the base 4. Replace the valve 5. Replace the film
Output side air pressure low	1. Dust on the valve base 2. Broken inner lining rubber 3. Broken valve spring	1. Clean the valve and the valve base 2. Replace the valve 3. Replace the spring
Gas leakage at the screw cap	1. Loosened cap nut 2. Broken film	1. Tighten the cap nut 2. Replace the film
No oil dripped from the oil cup	1. The adjusting needle being too tight 2. The adjusting needle cannot be closed or there are dirts on the needle. The adjustment needle cannot bear the pressure 3. Broken or damaged adjusting needle or needle base	1. Loosen up the needle 2. Reduce the oil volume until the level reaches the maximum oil level 3. Add more oil until the level reaches between the maximum and the minimum level 4. Clean up the blockage
The oil volume cannot be adjusted	1. The adjustment needle is too loose 2. The adjustment needle cannot be closed or dirt on the needle. The adjusting needle cannot bear the pressure 3. Broken or damaged adjustment needle or needle base	1. Tighteen the needle and gradually loosen it for adjustment 2. Remove the dirt 3. Partial replacement
Oil leadkage at the adjustment needle	1. The needle is too loose 2. Broken O-ring	1. Tighten up the adjustment needle 2. Replace the O-ring

Chapter 4

The Lubrication Unit

4-1 Centralized Lubrication System Diagram

1) Components of the lubrication oil pump



(a) Flow rate：300cc / min
(b) Voltage：220V (50 / 60 HZ)
(c) Power：150W(d) Maximum output pressure：15kg/cm²
(e) Oil tank capacity：6L

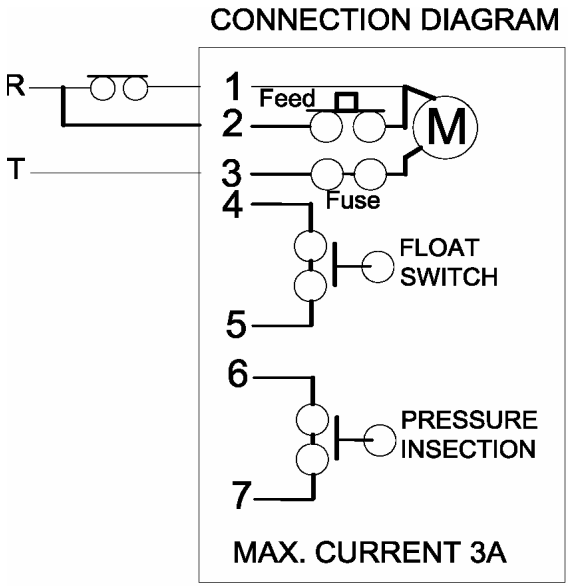
Connections：

1、3：Electric source

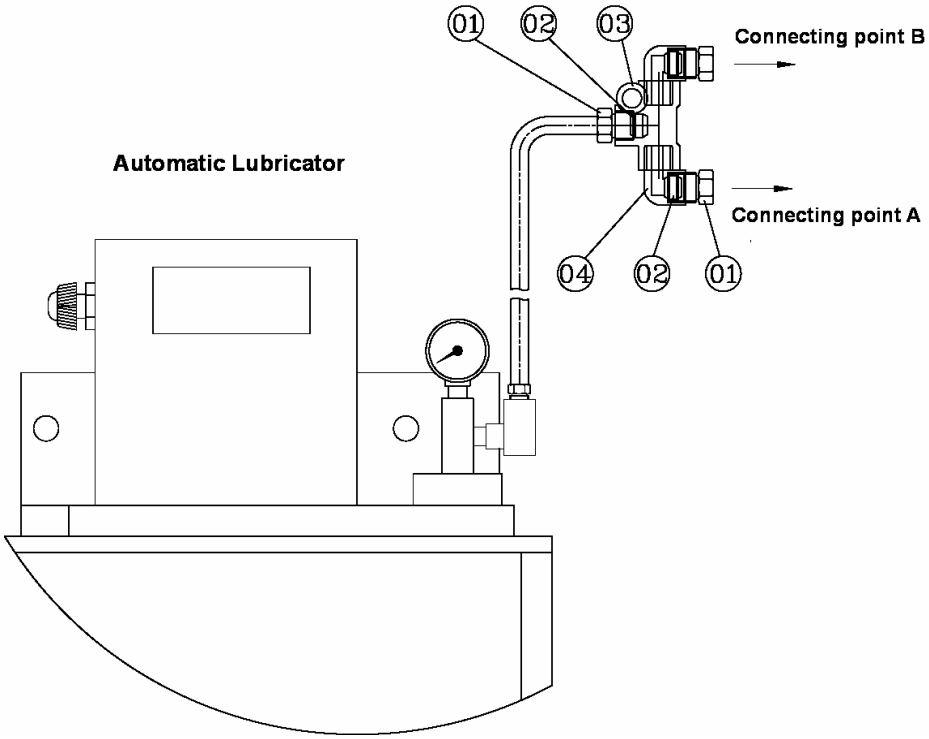
2、3：Manual oil injection

4、5：Floating switch

6、7：Pressure in section



2) Lubrication System Diagram



Number	Model	Name	Quantity
1.	PA6	Pipe cape	3
2.	PB6	Pipe	3
3.	PKD6	3-way pipe	1
4.	PH601	Right angle connector	2

caution：

1) Value of the input source current has to be the same as the one on the plague.
2) The wiring has to follow the instructions on the plaque to avoid damaging the oil pump.
3) Ground the ground wire.
4) A filter is installed on the inlet of the oil tank. Do not contaminate the manifold block components.
5) Wash the filter whenever it is contaminated to maintain its function.
6) Do not drop oil, water or other onto the control circuit, or the control circuit may be damaged.
7) Pay attention to the oil level when filling up the oil.

4-2 Spindle Bearing Lubrication

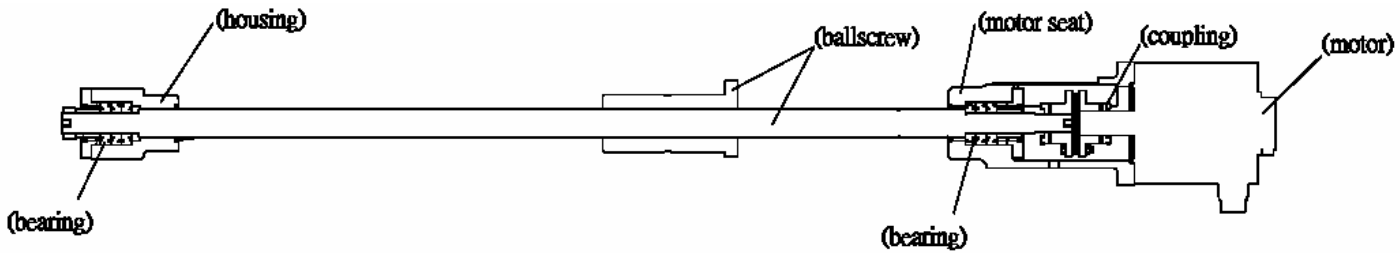
High quality lube oil is used for the spindle bearing, and it is good for high temperature and high speed conditions. Its lubricating effect on the bearing is long-lasting without being too oily, and therefore there is no need to add or to change the lube oil for a while. It is therefore cost-saving.

4-3 Guide way and Ball Screw Lubrication

The linear guide way of X, Y and Z axes and the ball screws are lubricated by a centralized lubrication system which lubricates according to the set schedule. Machine operators must ensure that the oiling machine has enough oil, as well as whether there is any abnormality.

1) Lead Screw Lubrication

The bearing of the lead screw can be divided into the motor end and the housing end. Lube oil is applied onto both ends for oil sealing. If the oil goes bad, dissemble these parts to wash the bearing and reassembling them. Please contact the Company to have professionals checkup and modification for you.



2) Guide way Lubrication

Guide ways of the workbench (X-axis), the saddle (Y-axis), the head (Z-axis), ball screws of the three axes (X, Y and Z axes) and the nuts are lubricated by the automatic centralized lubrication system in which an oil pump is located at the back of the machine.

If the machine has not been operated for a long time, less lube oil will be available in the oil tube. For the following cases:

1) A machine installed for the first time;
2) A machine has not been operated for a long time;
3) Before operating a machine (daily);

You should manually lubricate to ensure that sufficient lube oil has circulated the entire guide way. Press the red button on the lube oil pump for about 40 sec before releasing the button. Wait for 10 sec to repeat the above procedure. Repeat this process until lube oil has leaked out from the guide way.

Cautions：

The oil tank has a 6-L capacity. Add oil from the oil inlet each week (or when necessary). If the oil level reaches the minimum oil level, the operation panel will issue an alarm message informing the risk.

1) Add clean lube oil specified for the guide way through the oil inlet. Do not use other types of lube oil.

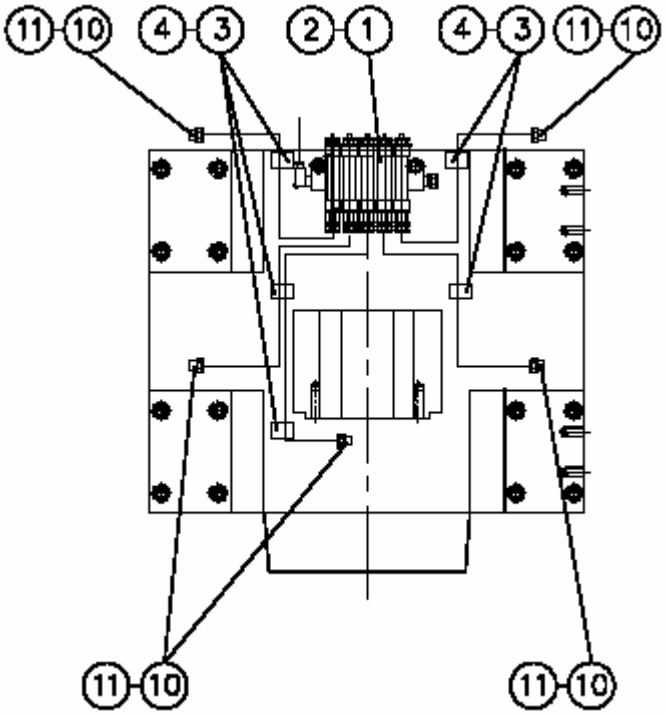
The recommended oils are:

Mobil	VACTRA 1
Shell	TONNA T32

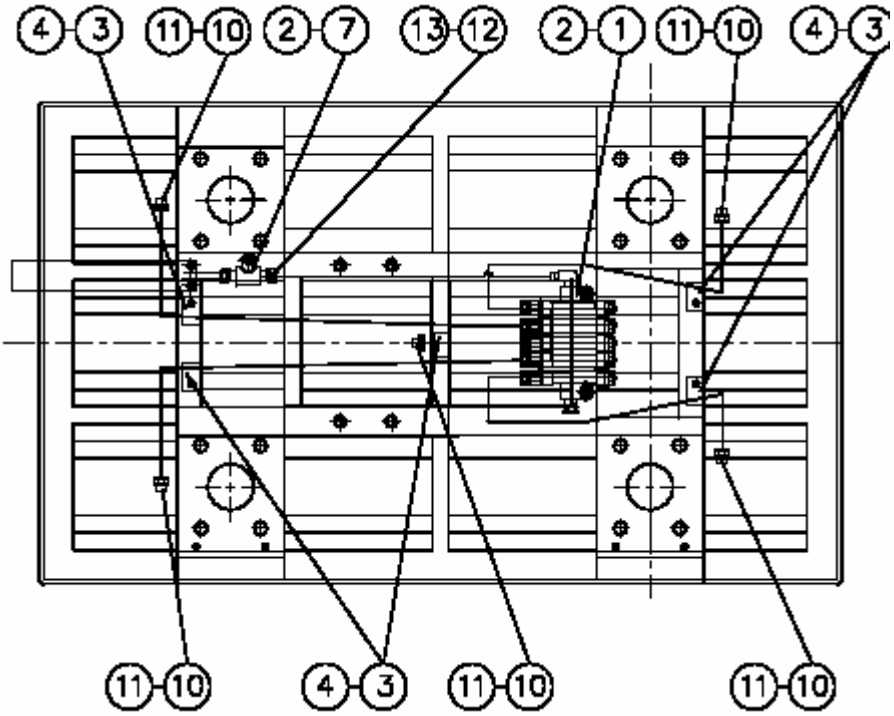
2) Lube oil level:

The level should be maintained between the maximum and the minimum oil level marked on the oil tank. Once the level drops below the midpoint, add oil as soon as possible. Do not add oil only after the alarm goes off, which can affect the operation of the machine.

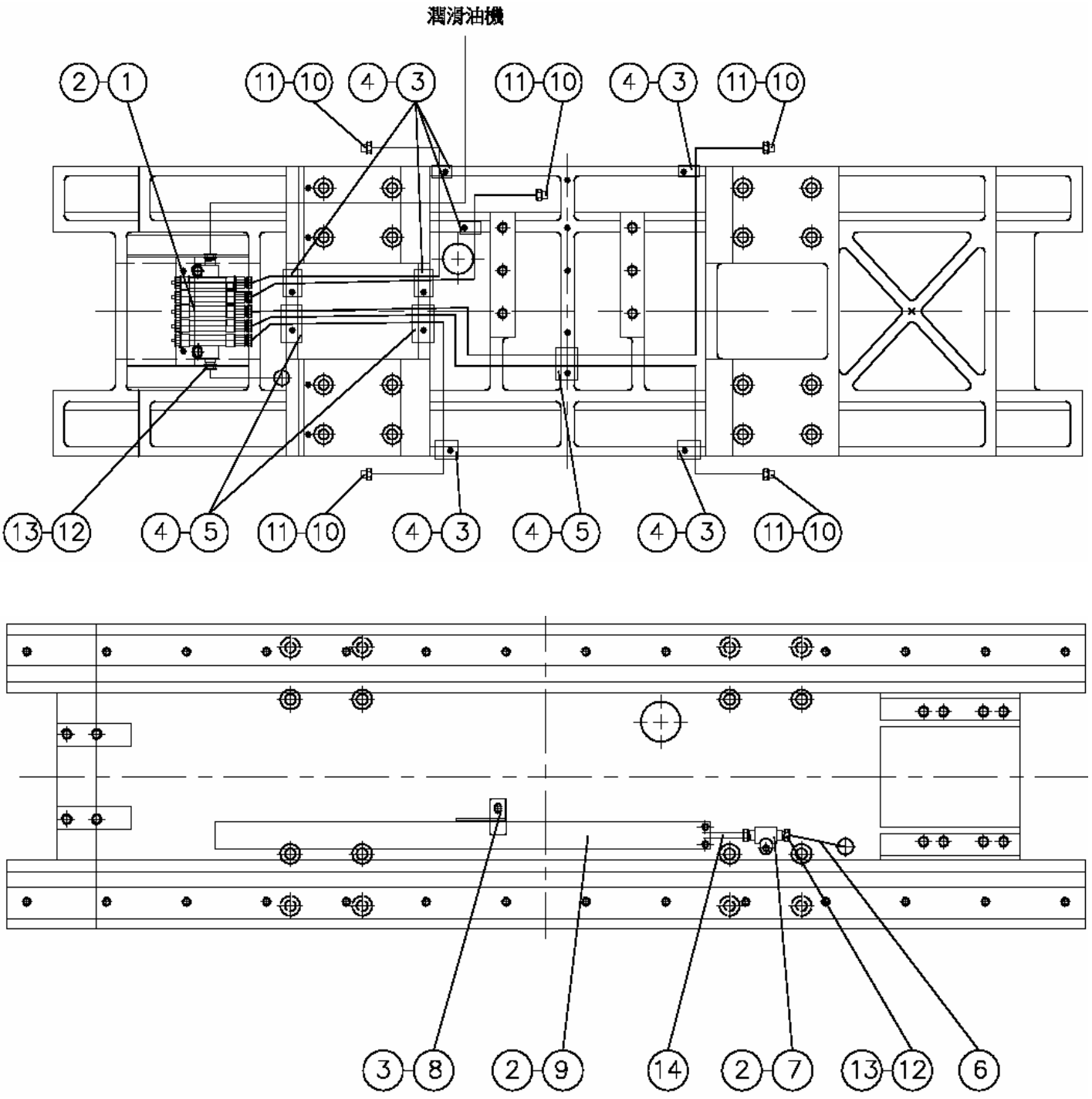
SPINDLE HEAD UNIT



WORK TABLE UNIT



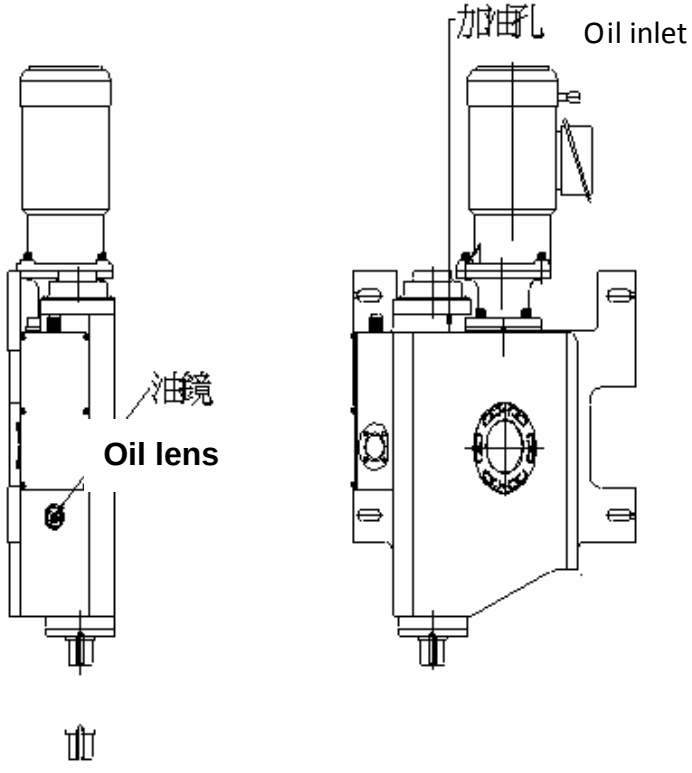
Saddle unit


潤滑油機
2 1 11 10 4 3 11 10 4 3 11 10
13 12 4 5 11 10 4 3 4 5 4 3 11 10
3 8 2 9 14 2 7 13 12 6

Number	Model	Name	Quantity
1.	PKD6	3-way pipe	2
2.	PB6	Pipe	8
3.	PA6	Pipe cape	8
4.	HBL-4	Volume-based distributor	1
5.	HBL-5	Volume-based distributor	2
6.	PAN4	Nut	17
7.	PB4	Pipe	34
8.	PA4	Pipe cape	17
9.	PH4-1	Flat and straight-angle connector	17
10.	PH601	Straight-angle connector	2
11.	HBL-3	Volume-based distributor	1
12.	PQ101	Connector	2
13	PG01	Sealing plug	1

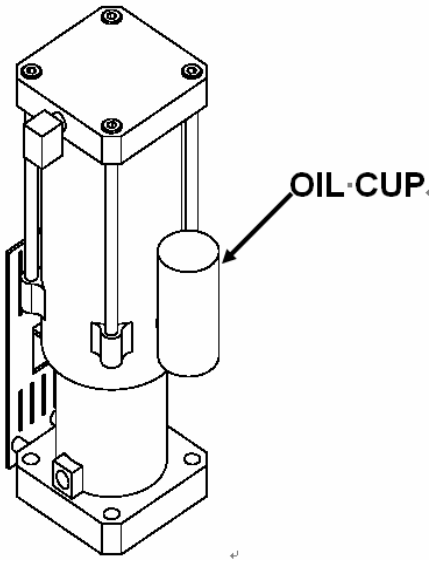
4-4 Lubrication of the Cam Box of Tool Magazine

Tool magazine's rotation is driven by ACT cam box. To ensure a smooth rotation of the cam box, it has to be routinely checked on whether there is enough oil for the oil lens. If not, add oil. After 2,400 hour of operation, replace the circulating oil in the cam box.

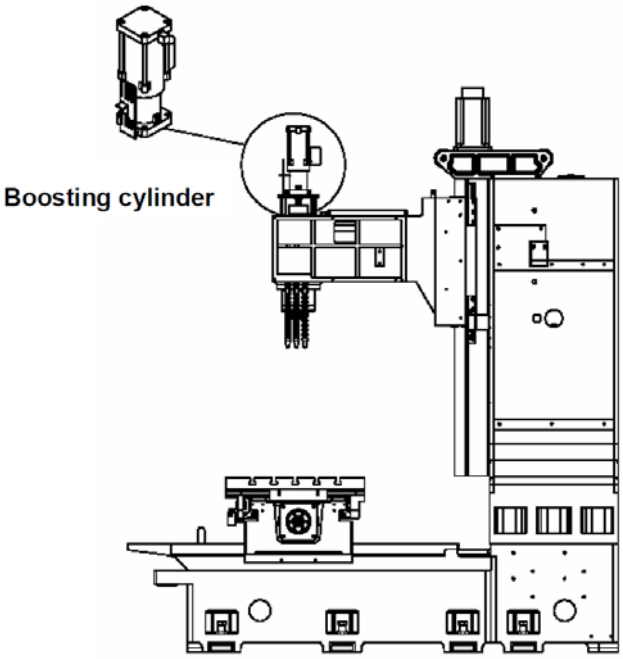


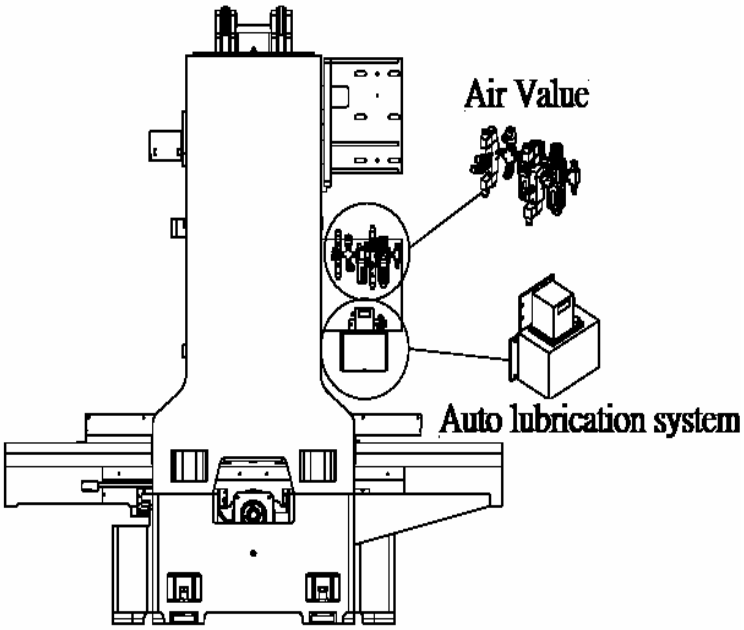
4-5 Cylinder Lubrication

1. Normally, oil in the oil cup will be depleted eventually. The oil will be used up after about 500,000 times.
2. Normally, it is recommended to remove the remaining oil from the oil cup each year and add new one when doing the machine maintenance (dot not disassemble the oil cup, or it may be damaged).
3. The color of the oil will be darkened if it has not been replaced for a long time. It will affect the air pressure and lifespan of the machine. Check it routinely or monthly.
4. It is abnormal is the oil is depleted within 2 to 3 days. Check whether there is leakage at the oil cup or the cup is broken. If so, replace the oil cup. If the depletion is abnormal, the boosting cylinder may need to be replaced immediately to protect the spindle.



4-6 Lubrication Location

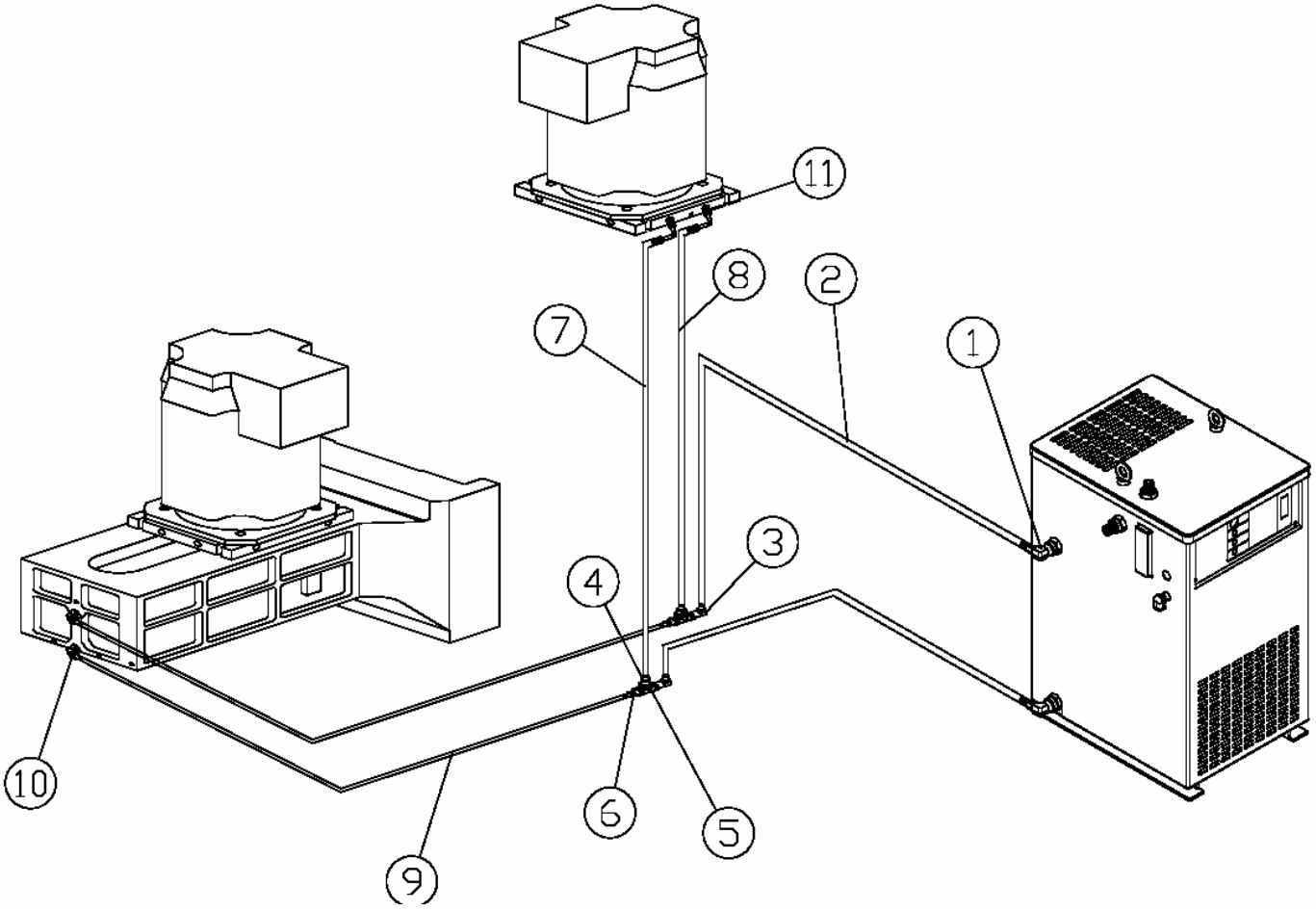




Chapter 5

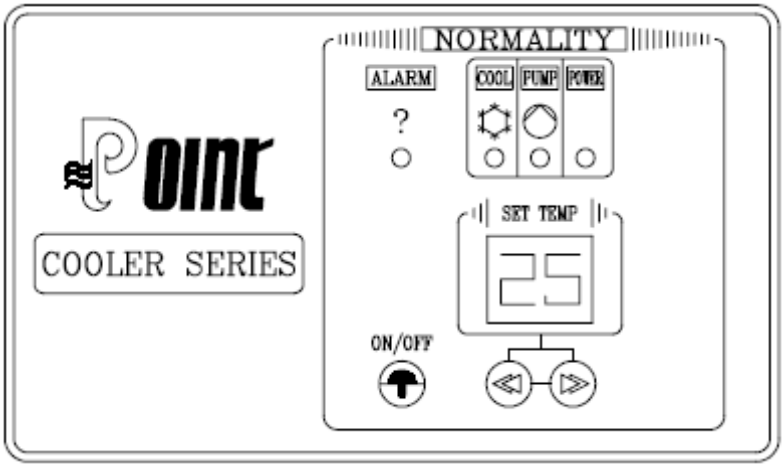
The Spindle Oil Cooling Unit

5-1 Oil Cooling Pipeline Diagram



5-2 Spindle Oil Cooling Unit

a) Add the correct type of oil according to the instructions presented on the plaque.
b) The air filter of the cooling unit should be kept clean. It can be taken out for cleaning.
c) The cooling unit also has a oil filter, which has to be cleaned routinely.



Names and operation of functions on the small panel

NO.	ITEM		OUTLINE	OPERATION&FUNCTION
1	Set keys	ON/OFF SWITCH		Run/Stop Switch (touch type)
2		SET TEMPERATURE		Usually display liquid temp,after pressing the temp setting key, monitor will twinkle and display the setting temp.In the mean time,use setting key to set temp.
3	Working keys	POWER ---green lamp		Power Source lamp Show the cooler is with electricity
4		PUMP ----green lamp		Pump is running normally
5		COOL ----green lamp		Compressor is running normally
6	ALARM	BREAKDOWN ALARM LAMP	?	Cooler is unusual

The spindle has already been appropriately adjusted prior to the installation. It is normal to have a slight increase of temperature. Spindle oil temperature is supplementary only; do not set the temperature too high or too low. The temperature control mechanism for the cooler is base on tracking temperature differences between the oil temperature and the room temperature (or the machine temperature) to make sure that the difference is well maintained.

The cooling unit is turn on and off automatically based on the temperature of the machine. The control key of the oil temperature is on the control panel of the cooling unit. When the SV is set to 0℃, the oil temperature and the machine temperature are the same. When the SV is set to +℃, temperature of the oil will be greater than of machine. When the SV is set to -℃, temperature of the oil will be less than of the machine. In general, these two temperatures are set to be equal. If the oil temperature is set to be lower than the machine temperature, temperature of the oil tank will be smaller than that of the machine. In this case, condensation may occur, which could contaminate the oil. Moreover, the external guide way of the bearing may shrink during the operation because the level of fraction and temperature will be increased, which can damage the spindle.
Do not arbitrarily modify the default value.

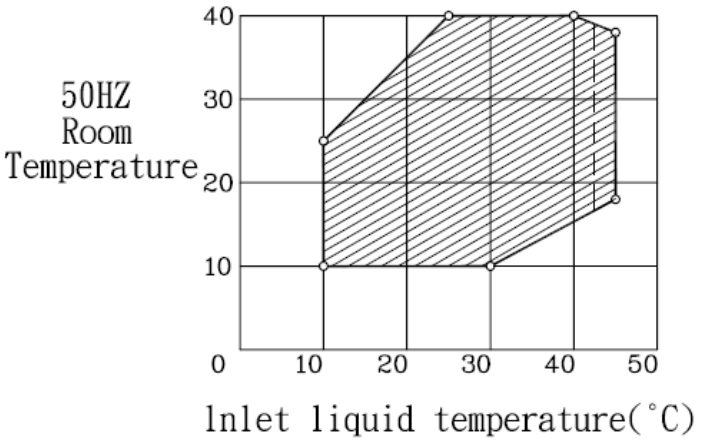
Note: This machine has automatic fault detection. When any fault detected, the panel will flash and display the fault signal. Troubleshooting is required. Then, turn off the machine for at least 10 sec before turning it on again. For operation quality, double-check that all the problems have been resolved and the system is at a good condition.

Application

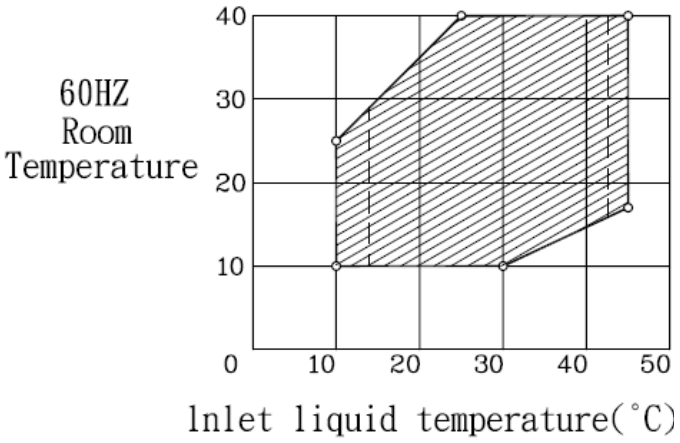
This oil cooler series is designed specifically to remove heat generated in the spindle oil of CNC machining centers.

The following charts indicate the operating oil/liquid temperatures for various ambient room temperatures in order that precision of the finished work piece is obtained.

(figure 1)



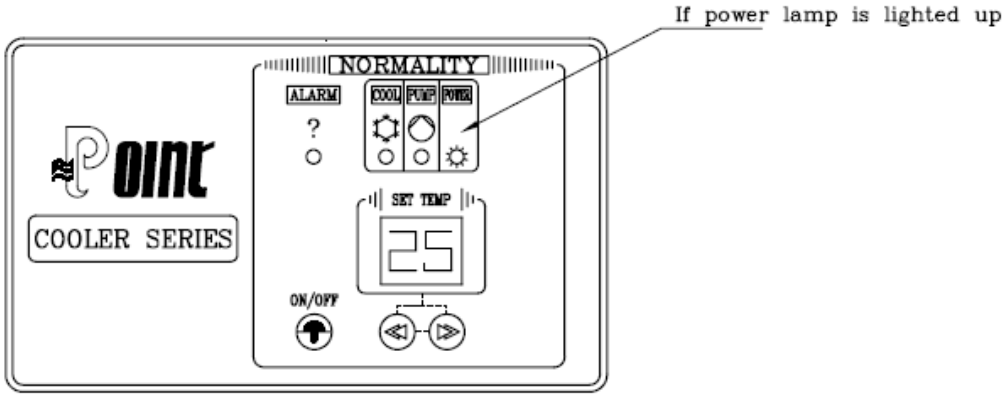
(figure 2)



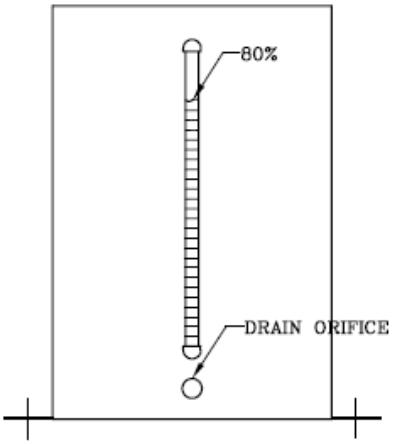
The oil filter shall be installed at the return oil (inlet) of the cooler to filter impurity efficiently and protect the spindle.
Oil cooler uses hydraulic oil and lubrication oil. The directed oil for WEXTEN cooler is 2-300 CST.
The oil below are prohibited.
(1) Hydraulic oil of phosphoric ester and chlorinated hydrocarbon types, water, water/ glycol hydraulic oil
(2) Cutting oil, grinding oil and water-soluble liquid.
(3) Medicine and corrosion liquid.
(4) Gasoline , kerosene and EDM oil.

Check points before operation

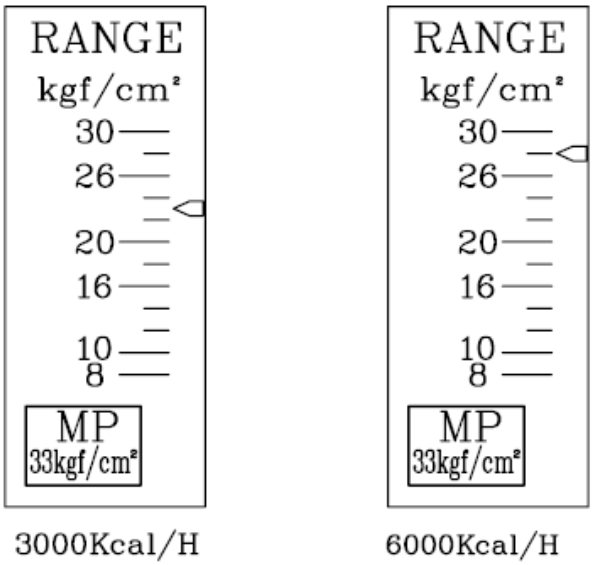
1. When the power source is turned on, check to see if power lamp is lighted up.(see figure 1)
2. Check that electrical resistance of compressor and pump is above 500 ohms.
3. Check that the oil level is at least at 80% level of the oil tank (see figure 2)
4. Check that the outlet of the oil tank is tightly secured.
5. Check that the overload switch is at “on “position.
6. Check that value is set at 5 kg/cm² for the pump pressure (5kg/cm² indicates 12 L/min, 20L/min, 30l/min pump adjust pressure; .5 kg / cm² indicates 4.5L/min, 7.5L/min, pump adjust pressure)
7. Check the high pressure switch of cooler of capacity 3000 kcal/hr is adjusted at 23 kg/cm² and 28 kg/ cm² for cooler of capacity above 6000 kcal/hr. (see figure 3)
8. Check if the oil inlet is fitted with an oil filter. (see figure 4)
9. Check cooling medium high/low pressure, if the indication needle position is as same as the testing report



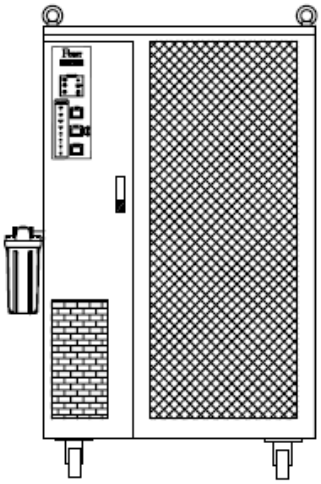
(figure 1)



(figure 2)



(figure 3)



(figure 4)

Troubleshooting

Possible Causes and Remedial Action

1. For any alarm signals, please refer to the following recommended remedial action.
   If problems cannot be solved, please contact us or our nearest agent. Checking the cooler model, sear number and the particular alarm sign show on control panel.

Lack of cooling medium

The following condition is caused by lack of cooling medium:
No alarm information but the motors keep running, cooler can not reach to the setting temperature and working machine's spindle is hot.
If the above situation occurred, please call professional technical staff to dispose or contact us directly.

Oil tank and filter

1. Oil level in tank should be ay least at the 80% level mark to prevent the air into the pump. At the same time, maintain the oil is clean.
2. The oil filter must be replaced or cleaned periodically, in order to prevent accumulating iron powder to reduce the discharge rate of pump and cause noisy.

Note: Machine damaged due to not on a regular basis cleaned filter or unrelated return oil filter is caused by human error, which is not covered by the warranty.

Condition	Cause	Remedy
Power lamp is off	1. LED burnt 2. PC board fuse blown 3. Transformer burnt 4. Wire 18,19 loose connection	1. Replace PC board 2. Replace fuse 3. Replace transformer 4. Reconnect wire 18,19
REV is lighted	1. 3 phase power reverse value 3 phase is above ±10% 2. Pressure reduction and differential	1. Switch any 2 of the R.S.T wires 2. Stability power source
OPS is lighted	1. Inlet oil pipe is clogged or loosened 2. Inlet & outlet are reversed 3. Pump motor runs reverse 4. Pump can not run 5. Circulation oil is not enough 6. Oil filter is dirty 7. Oil pressure switch breakdown	1. Check, clean and lock pipe 2. Correct position of in/outlet 3. Check over-relay of 51p red and white lines. 4. Replace oil pump 5. Supply circulation oil 6. Replace new filter 7. Adjust oil pressure switch DIFF, 0.3 kgf/cm^2 range 0.8~0.8 cmHg
PUMP is lighted	1. Overload relay is off 2. Pump switch is off 3. Inlet pipe is clogged	1. Reset switch 2. Reset pump switch Clean pipe and replace oil filter
COM is lighted	1. Overload relay off 2. Compressor breakdown	1. Reset switch 2. Replace compressor
HP is lighted	1. Condenser is too dirty 2. Air filter is not clean 3. Cooling fan doesn't run or blades fall. 4. Cooling medium pressure switch breakdown. 5. Cooling medium is leaking	1. Use compress air to clean fins and restart cooler. 2. Clean filter. 3. Lock fan blades tightly or replace fan motor. 4. Replace cooling medium pressure switch. 5. Irrigate cooling medium.
RA is lighted	1. Room temp. sensor breakdown	1. Replace RA sensor
RO is lighted	1. Oil temp. sensor breakdown	1. Replace RO sensor
OT is lighted	1. Oil temp. is to high 2. Temp. sensor blown 3. Check cooling medium is enough or not.	1. Stop running cooler until oil temp. returns to normal range start again 2. Replace temp sensor. 3. Supply cooling.

Chapter 6
The Electrical Unit

6-1 The Electrical Unit

1. Check the connectors of the electrical lines to prevent from detachment.
2. Make sure that the battery is replaced once a year to prevent parameter deletion, unstable parameters or unstable three-axis positioning precision.
3. The dust cover of the electrical box and the filter of the thermal heat exchanger have to be kept clean.
4. Operation environment conditions:
   - Low temperature
   - Good air circulation
   - Dry floor
   - Remote from electromagnetic disturbance
   - Stable electricity
5. Basic safe operation conditions: (See the following page)

Please follow the five basic rules below:

1. Only people know about this machine and have qualification as well as permission can operate the machine or performance the maintenance. Those competent individuals have to receive appropriate training to understand the safety and protective measures as well as machine maintenance work. These individuals have to be approved for their safety control ability. Especially those performing the electrical maintenance, they have to be experienced and can recognize the safety standard and official regulations.
2. Before operating the machine, make sure that the safety instruction part as well as descriptions on the operation, programs and maintenance in the user's manual has been well read.
3. Individuals who need to operate the machine or give maintenance should know the emergency stop button and the function.

1) Operation box

Note:

(a) Please well guard the relevant keys.

(b) The dust pan has two security keys.

To prevent the memory to be modified arbitrarily, use the function showed on the right figure.

Emergency stop:
Press this button to halt the operation immediately.

Memory edit lock:
This key is used to prevent unauthorized or arbitrarily changing of the processing procedure.

2) Electrical box

(a) Please well guard the relevant keys.
(b) The machine has two security keys to prevent the electrical box from being opened arbitrarily. The functions are presented on the right figure.
(c) Keep the user's manual at an easily accessed location. If the manual word can't read, please call the agent and confirm the model as well as the model number of the machine.

Chapter 7

Appendix

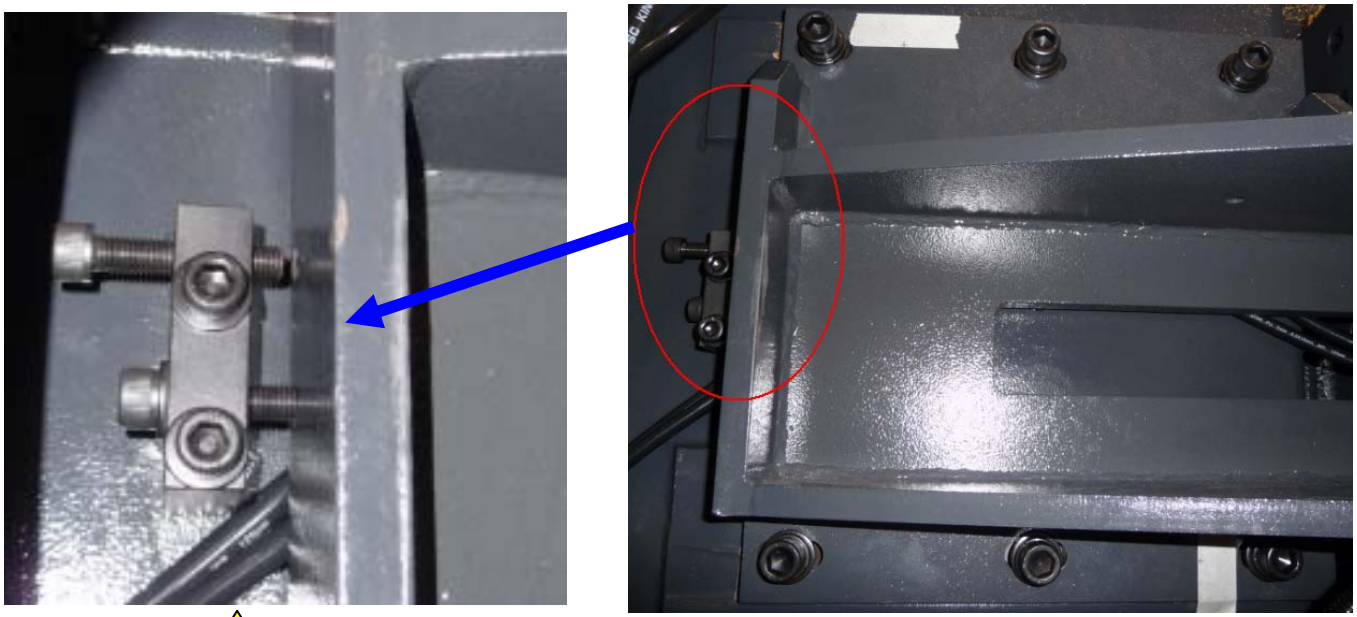
7-1 Oil Selection

Lubrication Location	Air Compressor Preparation Unit	Guide ways and balls screw	Cylinder	Coolant
Characteristics of the oil	(1) Viscosity ISO VG32 (2) Anti rust, foam, oxidation, etc. (3) Good stability; less deterioration	(1) ISO VG32 (2) High oil film intensity (3) Low abrasion (4) High abrasion resistance (5) High thermal stability (6) High corrosion resistance (7) High rust resistance	(1) Viscosity ISO VG68 (2) Anti rust, foam, oxidation, and emulsification	(1) High thermal conduction 2) Good lubrication
Lubrication Methods	Oil feeder	Automatic centralized lubrication		Circulation
Oil replacement interval	When needed	When needed	Add more when needed; replace the oil every year.	When needed
Tank capacity	55CC	3L		270L
Recommended Brands	(1)MOBIL RARUS 424 (2)ESSO TERESSO S32 (3)CASTROL HYSPIN VG32 (4)SHELL TONNA S32	(1)MOBIL VACTRA 1 (2)ESSO FEBIS K32 (3)CASTROL MAGNA GC32 (4)Shell TONNA T32	(1)MOBIL RARUS 424 (2)ESSO TERESSO S32 (3)CASTROL HYSPIN VG32 (4)SHELL TONNA S32	(1)MOBIL RARUS 424 (2)ESSO TERESSO S32 (3)CASTROL HYSPIN VG32 (4)SHELL TONNA S32

7-2 Spindle Tool Change Position Calibration

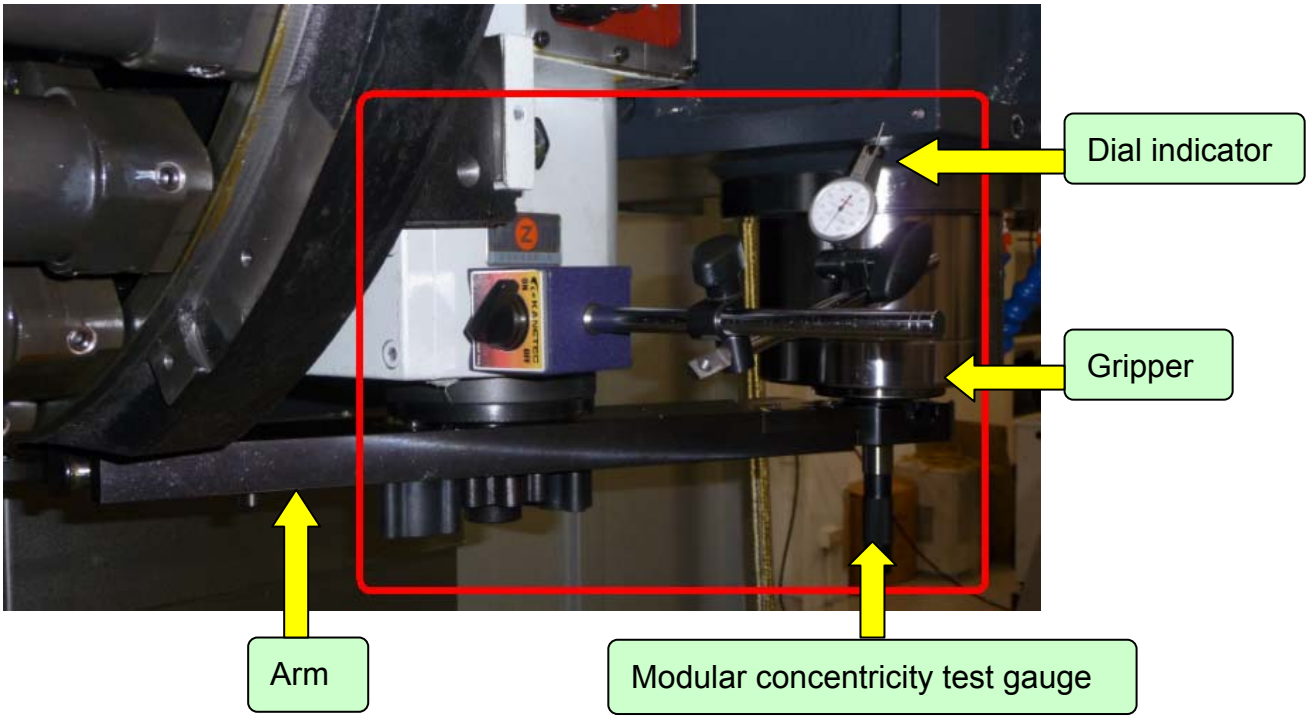
1) Check whether the origin offset of the Z-axis is zero. Change the positive and negative move distance of the Z-axis to 99999999 and -99999999. Afterward, return the Z-axis to zero position . Put a cutter on the spindle. Measure 0.6~ 0.8 mm the distance of tool unclamp .
Then, remove the cutter.

2) Lock the tool calibration magazine module onto the body of the tool magazine



Tool magazine calibration module
(Adjustment block)

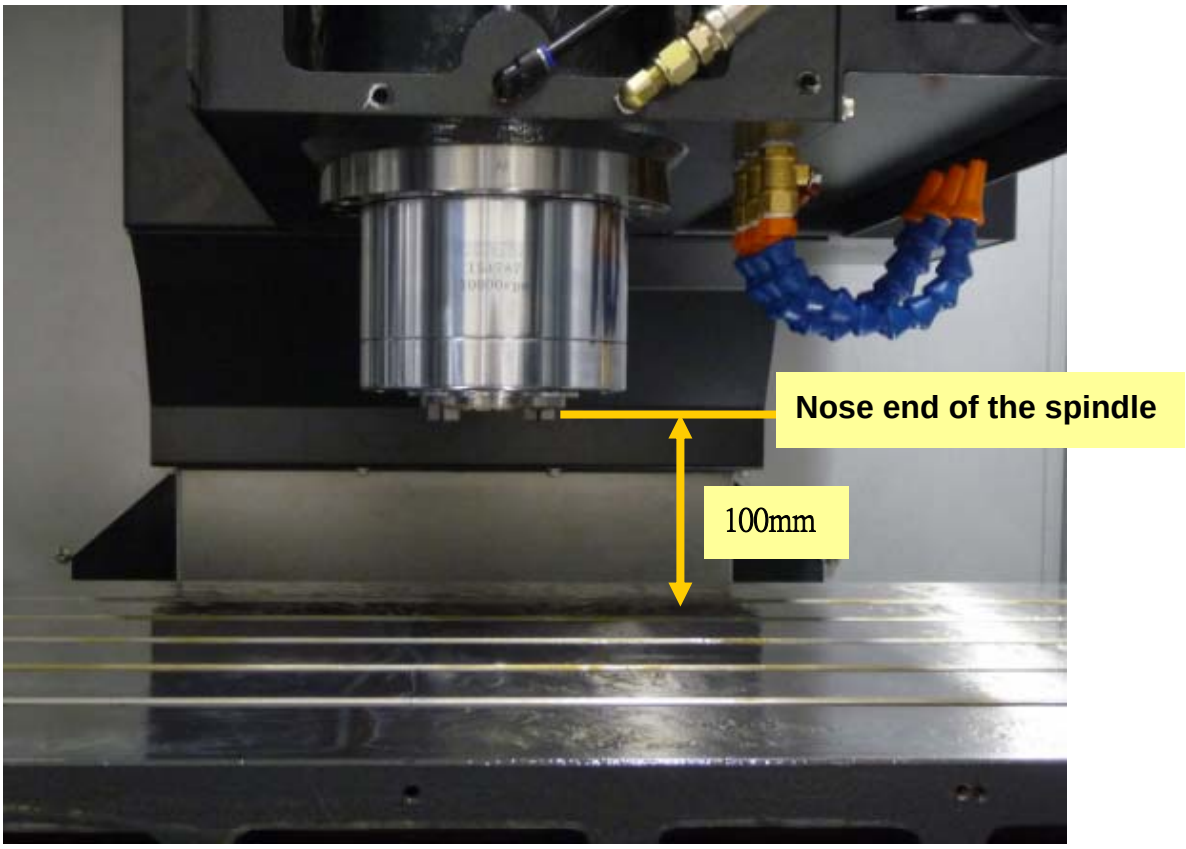
3) Take out a modular concentricity test gauge and a dial indicator. Separate the three sections and place them on the inner hole of the spindle and gripper.



4) Use the dial indicator to measure the position of the arm of the cutter.
The X-axis deviation should not exceed 0.1mm.
5) Fasten the screws (circled by a red line) according to the ISO standard.
6) Hammer the tool magazine into the fixing pin (circled by a yellow line

7-3 Determination of the Z-Axis Coordinate Origin

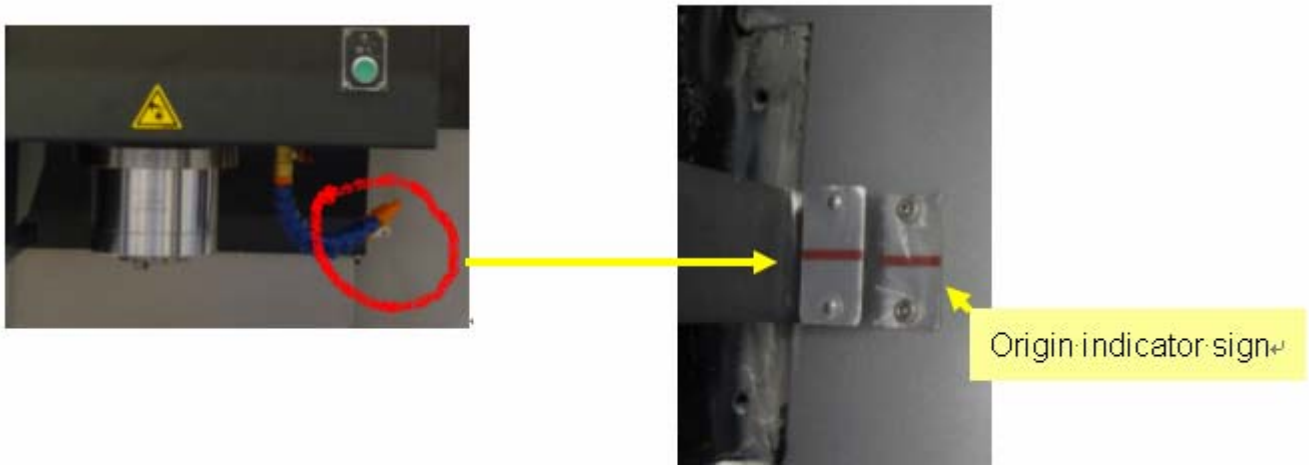
Shift the Z-axis from the nose end of the spindle to a position that is 100mm above the workbench. See the figure below.



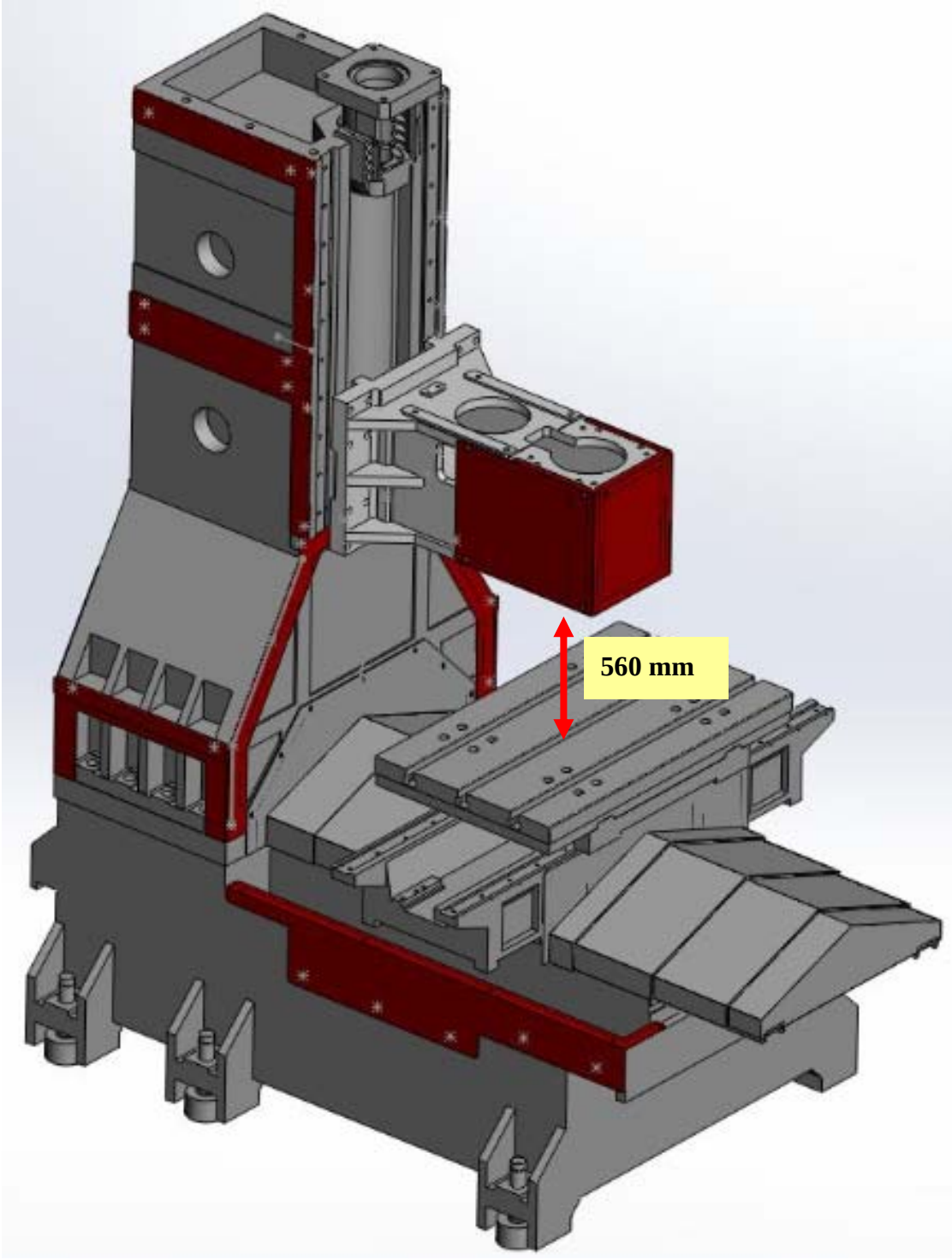
2.

1) Treat this position as the origin. Zero the value corresponded to the Z-axis on the screen.

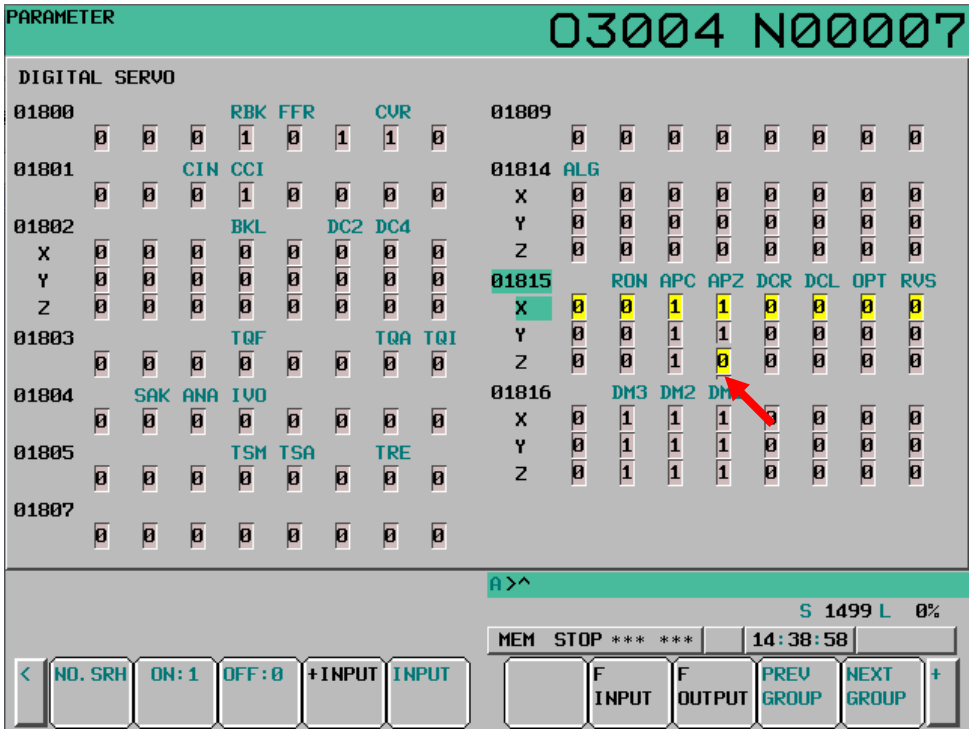
2) Shift the Z-axis to a position where the corresponding position on the screen is 560mm.



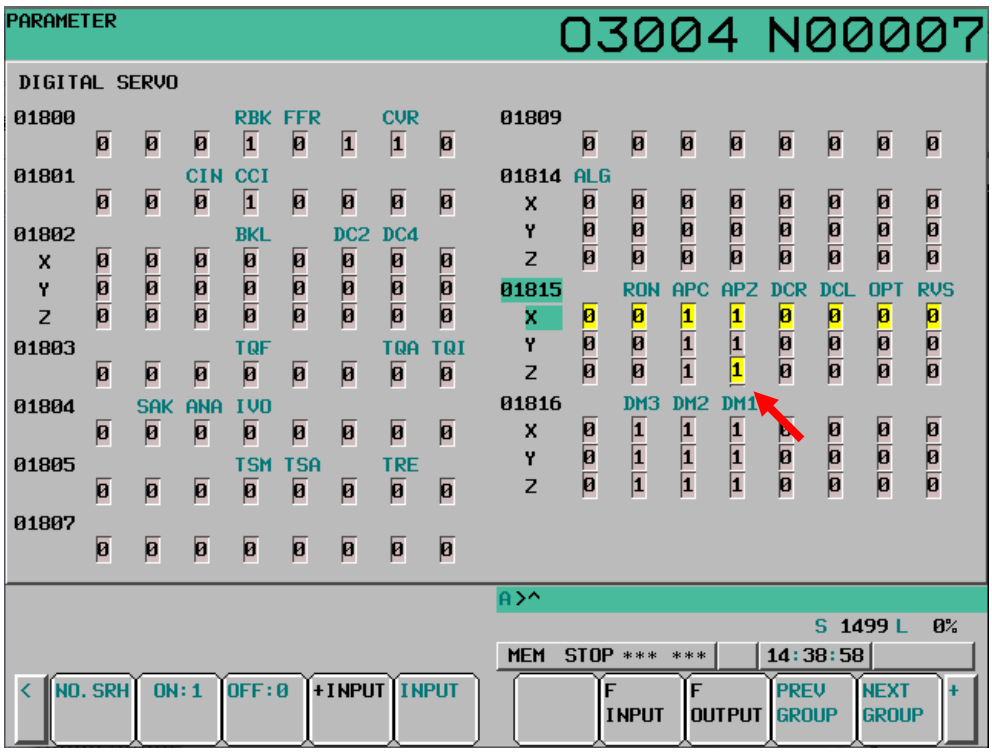
4. At this point, the relative distance between the spindle nose end and the workbench is 560mm. See the figure below.



4. Enter the parameter setup mode and look for 1815. Change the APZ of the Z-axis to 0 before turning on the machine.



5. After turning on the machine, enter Parameter 1815 to change APZ of Z-axis to 1. Turn on the machine again (At this point, the position of the Z-axis will be treated as the machine reference point).



7-4 Descriptions on Program and Servo Alarms

(1) Program errors /Alarms on program and operation (P/S alarm)

Number	Message	Contents
000	PLEASE TURN OFF POWER	A parameter which requires the power off was input, turn off power.
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective.
003	TOO MANY DIGITS	Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.)
004	ADDRESS NOT FOUND	A numeral or the sign " – " was input without an address at the beginning of a block. Modify the program .
005	NO DATA AFTER ADDRESS	The address was not followed by the appropriate data but was followed by another address or EOB code. Modify the program.
006	ILLEGAL USE OF NEGATIVE SIGN	Sign " – " input error (Sign " – " was input after an address with which it cannot be used. Or two or more " – " signs were input.) Modify the program.
007	ILLEGAL USE OF DECIMAL POINT	Decimal point " . " input error (A decimal point was input after an address with which it can not be used. Or two decimal points were input.) Modify the program.
009	ILLEGAL ADDRESS INPUT	Unusable character was input in significant area. Modify the program.
010	IMPROPER G–CODE	An unusable G code or G code corresponding to the function not provided is specified. Modify the program.
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.
014	CAN NOT COMMAND G95 (M series)	A synchronous feed is specified without the option for threading / synchronous feed.
	ILLEGAL LEAD COMMAND (T series)	In variable lead threading, the lead incremental and decremental outputted by address K exceed the maximum command value or a command such that the lead becomes a negative value is given. Modify the program.
015	TOO MANY AXES COMMANDED (M series)	An attempt was made to move the machine along the axes, but the number of the axes exceeded the specified number of axes controlled simultaneously. Modify the program.
	TOO MANY AXES COMMANDED (T series)	An attempt has been made to move the tool along more than the maximum number of simultaneously controlled axes. Alternatively, no axis movement command or an axis movement command for two or more axes has been specified in the block containing the command for skip using the torque limit signal (G31 P99/98). The command must be accompanied with an axis movement command for a single axis, in the same block.
020	OVER TOLERANCE OF RADIUS	In circular interpolation (G02 or G03), difference of the distance between the start point and the center of an arc and that between the end point and the center of the arc exceeded the value specified in parameter No. 3410.
021	ILLEGAL PLANE AXIS COMMANDED	An axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation. Modify the program.
022	NO CIRCLE RADIUS	The command for circular interpolation lacks arc radius R or coordinate I, J, or K of the distance between the start point to the center of the arc.

Number	Message	Contents
023	ILLEGAL RADIUS COMMAND (T series)	In circular interpolation by radius designation, negative value was commanded for address R. Modify the program.
025	CANNOT COMMAND F0 IN G02/G03 (M series)	F0 (fast feed) was instructed by F1 –digit column feed in circular interpolation. Modify the program.
027	NO AXES COMMANDED IN G43/G44 (M series)	No axis is specified in G43 and G44 blocks for the tool length offset type C. Offset is not canceled but another axis is offset for the tool length offset type C. Modify the program.
028	ILLEGAL PLANE SELECT	In the plane selection command, two or more axes in the same direction are commanded. Modify the program.
029	ILLEGAL OFFSET VALUE (M series)	The offset values specified by H code is too large. Modify the program.
	ILLEGAL OFFSET VALUE (T series)	The offset values specified by T code is too large. Modify the program.
030	ILLEGAL OFFSET NUMBER (M series)	The offset number specified by D/H code for tool length offset, cutter compensation, or three–dimensional tool offset is too large. Alternatively, the number of an additional workpiece coordinate system specified with the P code is too large. Modify the program.
	ILLEGAL OFFSET NUMBER (T series)	The offset number in T function specified for tool offset is tool large. Modify the program.
031	ILLEGAL P COMMAND IN G10	In setting an offset amount by G10, the offset number following address P was excessive or it was not specified. Modify the program.
032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.
033	NO SOLUTION AT CRC (M series)	A point of intersection cannot be determined for cutter compensation. Modify the program.
	NO SOLUTION AT CRC (T series)	A point of intersection cannot be determined for tool nose radius compensation. Modify the program.
034	NO CIRC ALLOWED IN ST–UP /EXT BLK (M series)	The start up or cancel was going to be performed in the G02 or G03 mode in cutter compensation C. Modify the program.
	NO CIRC ALLOWED IN ST–UP /EXT BLK (T series)	The start up or cancel was going to be performed in the G02 or G03 mode in tool nose radius compensation. Modify the program.
035	CAN NOT COMMANDED G39 (M series)	G39 is commanded in cutter compensation B cancel mode or on the plane other than offset plane. Modify the program.
	CAN NOT COMMANDED G31 (T series)	Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program.
036	CAN NOT COMMANDED G31 (M series)	Skip cutting (G31) was specified in cutter compensation mode. Modify the program.
037	CAN NOT CHANGE PLANE IN CRC (M seires)	G40 is commanded on the plane other than offset plane in cutter compensation B. The plane selected by using G17, G18 or G19 is changed in cutter compensation C mode. Modify the program.
	CAN NOT CHANGE PLANE IN NRC (T seires)	The offset plane is switched in tool nose radius compensation. Modify the program.
038	INTERFERENCE IN CIRCULAR BLOCK (M seires)	Overcutting will occur in cutter compensation C because the arc start point or end point coincides with the arc center. Modify the program.
	INTERFERENCE IN CIRCULAR BLOCK (T series)	Overcutting will occur in tool nose radius compensation because the arc start point or end point coincides with the arc center. Modify the program.

Number	Message	Contents
039	CHF/CNR NOT ALLOWED IN NRC (T series)	Chamfering or corner R was specified with a start–up, a cancel, or switching between G41 and G42 in tool nose radius compensation. The program may cause overcutting to occur in chamfering or corner R. Modify the program.
040	INTERFERENCE IN G90/G94 BLOCK (T series)	Overcutting will occur in tool nose radius compensation in canned cycle G90 or G94. Modify the program.
041	INTERFERENCE IN CRC (M seires)	Overcutting will occur in cutter compensation C. Two or more blocks are consecutively specified in which functions such as the auxiliary function and dwell functions are performed without movement in the cutter compensation mode. Modify the program.
	INTERFERENCE IN NRC (T seires)	Overcutting will occur in tool nose radius compensation. Modify the program.
042	G45/G48 NOT ALLOWED IN CRC (M series)	Tool offset (G45 to G48) is commanded in cutter compensation. Modify the program.
044	G27–G30 NOT ALLOWED IN FIXED CYC (M series)	One of G27 to G30 is commanded in canned cycle mode. Modify the program.
045	ADDRESS Q NOT FOUND (G73/G83) (M series)	In canned cycle G73/G83, the depth of each cut (Q) is not specified. Alternatively, Q0 is specified. Correct the program.
046	ILLEGAL REFERENCE RETURN COMMAND	Other than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference position return command.
047	ILLEGAL AXIS SELECT	Two or more parallel axes (in parallel with a basic axis) have been specified upon start–up of three–dimensional tool compensation or three–dimensional coordinate conversion.
048	BASIC 3 AXIS NOT FOUND	Start–up of three–dimensional tool compensation or three–dimensional coordinate conversion has been attempted, but the three basic axes used when Xp, Yp, or Zp is omitted are not set in parameter No. 1022.
049	ILLEGAL OPERATION (G68/G69) (M series)	The commands for three–dimensional coordinate conversion (G68, G69) and tool length compensation (G43, G44, G45) are not nested. Modify the program.
050	CHF/CNR NOT ALLOWED IN THRD BLK (M series)	Optional chamfering or corner R is commanded in the thread cutting block. Modify the program.
	CHF/CNR NOT ALLOWED IN THRD BLK(T series)	Chamfering or corner R is commanded in the thread cutting block. Modify the program.
051	MISSING MOVE AFTER CHF/CNR (M series)	Improper movement or the move distance was specified in the block next to the optional chamfering or corner R block. Modify the program.
	MISSING MOVE AFTER CHF/CNR (T series)	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
052	CODE IS NOT G01 AFTER CHF/CNR (M series)	The block next to the chamfering or corner R block is not G01,G02 or G03. Modify the program.
	CODE IS NOT G01 AFTER CHF/CNR (T series)	The block next to the chamfering or corner R block is not G01. Modify the program.
053	TOO MANY ADDRESS COMMANDS (M series)	For systems without the arbitary angle chamfering or corner R cutting, a comma was specified. For systems with this feature, a comma was followed by something other than R or C Correct the program.
	TOO MANY ADDRESS COMMANDS (T seires)	In the chamfering and corner R commands, two or more of I, K and R are specified. Otherwise, the character after a comma(",") is not C or R in direct drawing dimensions programming. Modify the program.
054	NO TAPER ALLOWED AFTER CHF/CNR (T series)	A block in which chamfering in the specified angle or the corner R was specified includes a taper command. Modify the program.

Number	Message	Contents
055	MISSING MOVE VALUE IN CHF/CNR (M series)	In the arbitrary angle chamfering or corner R block, the move distance is less than chamfer or corner R amount.
	MISSING MOVE VALUE IN CHF/CNR (T series)	In chamfering or corner R block, the move distance is less than chamfer or corner R amount.
056	NO END POINT & ANGLE IN CHF/ CNR (T series)	Neither the end point nor angle is specified in the command for the block next to that for which only the angle is specified (A). In the chamfering comman, I(K) is commanded for the X(Z) axis.
057	NO SOLUTION OF BLOCK END (T series)	Block end point is not calculated correctly in direct dimension drawing programming.
058	END POINT NOT FOUND (M series)	In a arbitrary angle chamfering or corner R cutting block, a specified axis is not in the selected plane. Correct the program.
	END POINT NOT FOUND (T series)	Block end point is not found in direct dimension drawing programming.
059	PROGRAM NUMBER NOT FOUND	In an external program number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Alternatively, the program with the program number specified in a one–touch macro call is not found in memory. Check the program number and external signal. Or discontin-ue the background eiting.
060	SEQUENCE NUMBER NOT FOUND	Commanded sequence number was not found in the sequence number search. Check the sequence number.
061	ADDRESS P/Q NOT FOUND IN G70–G73 (T series)	Address P or Q is not specified in G70, G71, G72, or G73 command. Modify the program.
062	ILLEGAL COMMAND IN G71–G76 (T series)	1. The depth of cut in G71 or G72 is zero or negative value. 2. The repetitive count in G73 is zero or negative value. 3. the negative value is specified to Δi or Δk is zero in G74 or G75. 4. A value other than zero is specified to address U or W though Δi or Δk is zero in G74 or G75. 5. A negative value is specified to Δd, thoughthe relief direction in G74 or G75 is determined. 6. Zero or a negative value is specified to the height of thread or depth of cut of first time in G76. 7. The specified minimum depth of cut in G76 is greater than the height of thread. 8. An unusable angle of tool tip is specified in G76. Modify the program.
063	SEQUENCE NUMBER NOT FOUND (T series)	The sequence number specified by address P in G70, G71, G72, or G73 command cannot be searched. Modify the program.
064	SHAPE PROGRAM NOT MONOTO-NOUSLY (T series)	A target shape which cannot be made by monotonic machining was specified in a repetitive canned cycle (G71 or G72).
065	ILLEGAL COMMAND IN G71–G73 (T series)	1. G00 or G01 is not commanded at the block with the sequence num-ber which is specified by address P in G71, G72, or G73 command. 2. Address Z(W) or X(U) was commanded in the block with a sequence number which is specified by address P in G71 or G72, respectively. Modify the program.
066	IMPROPER G–CODE IN G71–G73 (T series)	An unallowable G code was commanded beween two blocks specified by address P in G71, G72, or G73. Modify the program.
067	CAN NOT ERROR IN MDI MODE (T series)	G70, G71, G72, or G73 command with address P and Q. Modify the program.

Number	Message	Contents
069	FORMAT ERROR IN G70–G73 (T series)	The final move command in the blocks specified by P and Q of G70, G71, G72, and G73 ended with chamfering or corner R. Modify the program.
070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.
071	DATA NOT FOUND	The address to be searched was not found. Or the program with specified program number was not found in program number search. Check the data.
072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63 (basic), 125 (option), 200 (option), 400 (option) or 1000 (option). Delete unnecessary programs and execute program registeration again.
073	PROGRAM NUMBER ALREADY IN USE	The commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registeration again.
074	ILLEGAL PROGRAM NUMBER	The program number is other than 1 to 9999. Modify the program number.
075	PROTECT	An attempt was made to register a program whose number was protected.
076	ADDRESS P NOT DEFINED	Address P (program number) was not commanded in the block which includes an M98, G65, or G66 command. Modify the program.
077	SUB PROGRAM NESTING ERROR	The subprogram was called in five folds. Modify the program.
078	NUMBER NOT FOUND	A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found. The sequence number specified by a GOTO statement was not found. Otherwise, a called program is being edited in background processing. Correct the program, or discontinue the background editing.
079	PROGRAM VERIFY ERROR	In memory or program collation,a program in memory does not agree with that read from an external I/O device. Check both the programs in memory and those from the external device.
080	G37 ARRIVAL SIGNAL NOT ASSERTED (M series)	In the automatic tool length measurement function (G37), the measurement position reach signal (XAE, YAE, or ZAE) is not turned on within an area specified in parameter 6254 6255 (value ε). This is due to a setting or operator error.
	G37 ARRIVAL SIGNAL NOT ASSERTED (T series)	In the automatic tool compensation function (G36, G37), the measurement position reach signal (XAE or ZAE) is not turned on within an area specified in parameter 6254 (value ε). This is due to a setting or operator error.
081	OFFSET NUMBER NOT FOUND IN G37 (M series)	Tool length automatic measurement (G37) was specified without a H code. (Automatic tool length measurement function) Modify the program.
	OFFSET NUMBER NOT FOUND IN G37 (T series)	Automatic tool compensation (G36, G37) was specified without a T code. (Automatic tool compensation function) Modify the program.
082	H–CODE NOT ALLOWED IN G37 (M series)	H code and automatic tool compensation (G37) were specified in the same block. (Automatic tool length measurement function) Modify the program.
	T–CODE NOT ALLOWED IN G37 (T series)	T code and automatic tool compensation (G36, G37) were specified in the same block. (Automatic tool compensation function) Modify the program.
083	ILLEGAL AXIS COMMAND IN G37 (M series)	In automatic tool length measurement, an invalid axis was specified or the command is incremental. Modify the program.
	ILLEGAL AXIS COMMAND IN G37 (T series)	In automatic tool compensation (G36, G37), an invalid axis was specified or the command is incremental. Modify the program.

Number	Message	Contents
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective.
088	LAN FILE TRANS ERROR (CHANNEL–1)	File data transfer via OSI–ETHERNET has been stopped due to a transfer error.
089	LAN FILE TRANS ERROR (CHANNEL–2)	File data transfer via OSI–ETHERNET has been stopped due to a transfer error.
090	REFERENCE RETURN INCOMPLETE	1. The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. 2. During reference position return with the absolute–position detector, if this alarm occurs even though condition 1 is satisfied, do the following: After turning the servo motor for the axis at least one turn, turn the power off and then on again. Then perform reference position return.
091	REFERENCE RETURN INCOMPLETE	Manual reference position return cannot be performed when automatic operation is halted.
092	AXES NOT ON THE REFERENCE POINT	The commanded axis by G27 (Reference position return check) did not return to the reference position.
094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to th operator's manual.
095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece offset amount changed.) Perform the correct operation according to th operator's manual.
096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.) Perform the correct operation according to the operator's manual.
097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or P / S 94 to 97 reset, no automatic operation is performed.) Perform automatic operation.
098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search. Perform the reference position return.
099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI. Move axis before a move command or don't interrupt MDI operation.
100	PARAMETER WRITE ENABLE	On the PARAMETER(SETTING) screen, PWE(parameter writing enabled) is set to 1. Set it to 0, then reset the system.

Number	Message	Contents
101	PLEASE CLEAR MEMORY	The power turned off while rewriting the memory by program edit operation. If this alarm has occurred, press <RESET> while pressing <PROG>, and only the program being edited will be deleted. Register the deleted program.
109	FORMAT ERROR IN G08	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.
110	DATA OVERFLOW	The absolute value of fixed decimal point display data exceeds the allowable range. Modify the program.
111	CALCULATED DATA OVERFLOW	The result of calculation turns out to be invalid, an alarm No.111 is issued. -10^{47} to -10^{-29}, 0, 10^{-29} to 10^{47} Modify the program.
112	DIVIDED BY ZERO	Division by zero was specified. (including tan 90°) Modify the program.
113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
114	FORMAT ERROR IN MACRO	There is an error in other formats than <Formula>. Modify the program.
115	ILLEGAL VARIABLE NUMBER	A value not defined as a variable number is designated in the custom macro or in high–speed cycle machining. The header contents are improper. This alarm is given in the following cases: High speed cycle machining 1. The header corresponding to the specified machining cycle number called is not found. 2. The cycle connection data value is out of the allowable range (0 – 999). 3. The number of data in the header is out of the allowable range (0 – 32767). 4. The start data variable number of executable format data is out of the allowable range (#20000 – #85535). 5. The last storing data variable number of executable format data is out of the allowable range (#85535). 6. The storing start data variable number of executable format data is overlapped with the variable number used in the header. Modify the program.
116	WRITE PROTECTED VARIABLE	The left side of substitution statement is a variable whose substitution is inhibited. Modify the program.
118	PARENTHESIS NESTING ERROR	The nesting of bracket exceeds the upper limit (quintuple). Modify the program.
119	ILLEGAL ARGUMENT	The SQRT argument is negative. Or BCD argument is negative, and other values than 0 to 9 are present on each line of BIN argument. Modify the program.
122	FOUR FOLD MACRO MODAL–CALL	The macro modal call is specified four fold. Modify the program.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
124	MISSING END STATEMENT	DO – END does not correspond to 1 : 1. Modify the program.
125	FORMAT ERROR IN MACRO	<Formula> format is erroneous. Modify the program.
126	ILLEGAL LOOP NUMBER	In DOn, 1≦ n ≦3 is not established. Modify the program.

Number	Message	Contents
127	NC, MACRO STATEMENT IN SAME BLOCK	NC and custom macro commands coexist. Modify the program.
128	ILLEGAL MACRO SEQUENCE NUMBER	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched. Modify the program.
129	ILLEGAL ARGUMENT ADDRESS	An address which is not allowed in <Argument Designation > is used. Modify the program.
130	ILLEGAL AXIS OPERATION	An axis control command was given by PMC to an axis controlled by CNC. Or an axis control command was given by CNC to an axis controlled by PMC. Modify the program.
131	TOO MANY EXTERNAL ALARM MESSAGES	Five or more alarms have generated in external alarm message. Consult the PMC ladder diagram to find the cause.
132	ALARM NUMBER NOT FOUND	No alarm No. concerned exists in external alarm message clear. Check the PMC ladder diagram.
133	ILLEGAL DATA IN EXT. ALARM MSG	Small section data is erroneous in external alarm message or external operator message. Check the PMC ladder diagram.
135	ILLEGAL ANGLE COMMAND (M series)	The index table indexing positioning angle was instructed in other than an integral multiple of the value of the minimum angle. Modify the program.
	SPINDLE ORIENTATION PLEASE (T series)	Without any spindle orientation , an attept was made for spindle indexing. Perform spindle orientation.
136	ILLEGAL AXIS COMMAND (M series)	In index table indexing.Another control axis was instructed together with the B axis. Modify the program.
	C/H–CODE & MOVE CMD IN SAME BLK. (T series)	A move command of other axes was specified to the same block as spindle indexing addresses C, H. Modify the program.
137	M–CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as M–code related to spindle indexing. Modify the program.
138	SUPERIMPOSED DATA OVERFLOW	The total distribution amount of the CNC and PMC is too large during superimposed control of the extended functions for PMC axis control.
139	CAN NOT CHANGE PMC CONTROL AXIS	An axis is selected in commanding by PMC axis control. Modify the program.
141	CAN NOT COMMAND G51 IN CRC (M series)	G51 (Scaling ON) is commanded in the tool offset mode. Modify the program.
142	ILLEGAL SCALE RATE (M series)	Scaling magnification is commanded in other than 1 – 999999. Correct the scaling magnification setting (G51 P_p ..... or parameter 5411 or 5421).
143	SCALED MOTION DATA OVERFLOW (M series)	The scaling results, move distance, coordinate value and circular radius exceed the maximum command value. Correct the program or scaling mangification.
144	ILLEGAL PLANE SELECTED (M series)	The coordinate rotation plane and arc or cutter compensation C plane must be the same. Modify the program.
145	ILLEGAL CONDITIONS IN POLAR COORDINATE INTERPOLATION	The conditions are incorrect when the polar coordinate interpolation starts or it is canceled. 1) In modes other than G40, G12.1/G13.1 was specified. 2) An error is found in the plane selection. Parameters No. 5460 and No. 5461 are incorrectly specified. Modify the value of program or parameter.
146	IMPROPER G CODE	G codes which cannot be specified in the polar coordinate interpolation mode was specified. See section II–4.4 and modify the program.
148	ILLEGAL SETTING DATA (M series)	Automatic corner override deceleration rate is out of the settable range of judgement angle. Modify the parameters (No.1710 to No.1714)

Number	Message	Contents
149	FORMAT ERROR IN G10L3	A code other than Q1,Q2,P1 or P2 was specified as the life count type in the extended tool life management.
150	ILLEGAL TOOL GROUP NUMBER	Tool Group No. exceeds the maximum allowable value. Modify the program.
151	TOOL GROUP NUMBER NOT FOUND	The tool group commanded in the machining program is not set. Modify the value of program or parameter.
152	NO SPACE FOR TOOL ENTRY	The number of tools within one group exceeds the maximum value registerable. Modify the number of tools.
153	T–CODE NOT FOUND	In tool life data registration, a T code was not specified where one should be. Correct the program.
154	NOT USING TOOL IN LIFE GROUP (M series)	When the group is not commanded, H99 or D99 was commanded. Correct the program.
155	ILLEGAL T–CODE IN M06 (M series)	In the machining program, M06 and T code in the same block do not correspond to the group in use. Correct the program.
	ILLEGAL T–CODE IN M06 (T series)	Group No.ΔΔ which is specified with TΔΔ 88 of the machining program do not included in the tool group in use. Correct the program.
156	P/L COMMAND NOT FOUND	P and L commands are missing at the head of program in which the tool group is set. Correct the program.
157	TOO MANY TOOL GROUPS	The number of tool groups to be set exceeds the maximum allowable value. (See parameter No. 6800 bit 0 and 1) Modify the program.
158	ILLEGAL TOOL LIFE DATA	The tool life to be set is too excessive. Modify the setting value.
159	TOOL DATA SETTING INCOMPLETE	During executing a life data setting program, power was turned off. Set again.
160	MISMATCH WAITING M–CODE (T series (At two–path))	Diffrent M code is commanded in heads 1 and 2 as waiting M code. Modify the program.
	MISMATCH WAITING M–CODE (T series (At three–path))	1) Although the same P command is specified, the waiting M codes do not match. 2) Although the waiting M codes match, the P commands do not match. 3) Two–path wait and three–path wait are specified simultaneously. Modify the program.
	G72.1 NESTING ERROR (M series)	A subprogram which performs rotational copy with G72.1 contains another G72.1 command.
161	ILLEGAL P OF WAITING M–CODE (T series (three–path control)	1) The value of address P is a negative value, 1, 2, 4, or a value not smaller than 8. 2) The value specified in P is not consistent with the system configuration. Modify the program.
	G72.1 NESTING ERROR (M series)	A subprogram which performs parallel copy with G72.2 contains another G72.2 command.
163	COMMAND G68/G69 INDEPENDENTLY (T series (At two–path))	G68 and G69 are not independently commanded in balance cut. Modify the program.
169	ILLEGAL TOOL GEOMETRY DATA (At two–path)	Incorrect tool figure data in interference check. Set correct data, or select correct tool figure data.
175	ILLEGAL G107 COMMAND	Conditions when performing circular interpolation start or cancel not correct. To change the mode to the cylindrical interpolation mode, specify the command in a format of "G07.1 rotation–axis name radius of cylinder."

Number	Message	Contents
176	IMPROPER G–CODE IN G107 (M series)	Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified. 1) G codes for positioning: G28,, G73, G74, G76, G81 – G89, including the codes specifying the rapid traverse cycle 2) G codes for setting a coordinate system: G52,G92, 3) G code for selecting coordinate system: G53 G54–G59 Modify the program.
	IMPROPER G–CODE IN G107 (T series)	Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified. 1) G codes for positioning: G28, G76, G81 – G89, including the codes specifying the rapid traverse cycle 2) G codes for setting a coordinate system: G50, G52 3) G code for selecting coordinate system: G53 G54–G59 Modify the program.
177	CHECK SUM ERROR (G05 MODE)	Check sum error Modify the program.
178	G05 COMMANDED IN G41/G42 MODE	G05 was commanded in the G41/G42 mode. Correct the program.
179	PARAM. (NO. 7510) SETTING ERROR	The number of controlled axes set by the parameter 7510 exceeds the maximum number. Modify the parameter setting value.
180	COMMUNICATION ERROR (REMOTE BUF)	Remote buffer connection alarm has generated. Confirm the number of cables, parameters and I/O device.
181	FORMAT ERROR IN G81 BLOCK (Hobbing machine, EGB) (M series)	G81 block format error (hobbing machine) 1) T (number of teeth) has not been instructed. 2) Data outside the command range was instructed by either T, L, Q or P. 3) An overflow occurred in synchronization coefficient calculation. Modify the program.
182	G81 NOT COMMANDED (Hobbing machine) (M series)	G83 (C axis servo lag quantity offset) was instructed though synchronization by G81 has not been instructed. Correct the program. (hobbing machine)
183	DUPLICATE G83 (COMMANDS) (Hobbing machine) (M series)	G83 was instructed before canceled by G82 after compensating for the C axis servo lag quantity by G83. (hobbing machine)
184	ILLEGAL COMMAND IN G81 (Hobbing machine, EGB) (M series)	A command not to be instructed during synchronization by G81 was instructed. (hobbing machine) 1) A C axis command by G00, G27, G28, G29, G30, etc. was instructed. 2) Inch/Metric switching by G20, G21 was instructed.
185	RETURN TO REFERENCE POINT (Hobbing machine) (M series)	G81 was instructed without performing reference position return after power on or emergency stop. (hobbing machine) Perform reference position return.
186	PARAMETER SETTING ERROR (Hobbing machine, EGB) (M series)	Parameter error regarding G81 (hobbing machine) 1) The C axis has not been set to be a rotary axis. 2) A hob axis and position coder gear ratio setting error Modify the parameter.
187	HOB COMMAND IS NOT ALLOWED	Error in the modal state when G81.4 or G81 is specified 1. The canned cycle mode (G81 to G89) is set. 2. The thread cutting mode is set. 3. The C–axis is under synchronous, composite, or superimposed control.

Number	Message	Contents
190	ILLEGAL AXIS SELECT	In the constant surface speed control, the axis specification is wrong. (See parameter No. 3770.) The specified axis command (P) contains an illegal value. Correct the program.
194	SPINDLE COMMAND IN SYNCHRO–MODE	A contour control mode, spindle positioning (Cs–axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.
197	C–AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs–axis when the signal CON(DGN=G027#7) was off. Correct the program, or consult the PMC ladder diagram to find the reason the signal is not turned on.
199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.
200	ILLEGAL S CODE COMMAND	In the rigid tap, an S value is out of the range or is not specified. Modify the program.
201	FEEDRATE NOT FOUND IN RIGID TAP	In the rigid tap, no F value is specified. Correct the program.
202	POSITION LSI OVERFLOW	In the rigid tap, spindle distribution value is too large. (System error)
203	PROGRAM MISS AT RIGID TAPPING	In the rigid tap, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
204	ILLEGAL AXIS OPERATION	In the rigid tap, an axis movement is specified between the rigid M code (M29) block and G84 or G74 for M series (G84 or G88 for T series) block. Modify the program.
205	RIGID MODE DI SIGNAL OFF	1. Although a rigid M code (M29) is specified in rigid tapping, the rigid mode DI signal (DGN G061.0) is not ON during execution of the G84 (G88) block. 2. In a system with the multi–spindle option, the spindle used for rigid tapping is not selected (by DI signal G27#0 and #1, or G61#4 and #5). Check the PMC ladder diagram to find the reason why the DI signal is not turned on.
206	CAN NOT CHANGE PLANE (M series)	Plane changeover was instructed in the rigid mode. Correct the program.
207	RIGID DATA MISMATCH	The specified distance was too short or too long in rigid tapping.
210	CAN NOT COMAND M198/M199	M98 and M99 are executed in the schedule operation. M198 is executed in the DNC operation. Modify the program. 1) The execution of an M198 or M99 command was attempted during scheduled operation. Alternatively, the execution of an M198 command was attempted during DNC operation. Correct the program. The execution of an M99 command was attempted by an interrupt macro during pocket machining in a multiple repetitive canned cycle.
211	G31 (HIGH) NOT ALLOWED IN G99 (T series)	G31 is commanded in the per revolution command when the high–speed skip option is provided. Modify the program.
212	ILLEGAL PLANE SELECT (M series)	The arbitrary angle chamfering or a corner R is commanded or the plane including an additional axis. Correct the program.
	ILLEGAL PLANE SELECT (T series)	The direct drawing dimensions programming is commanded for the plane other than the Z–X plane. Correct the program.

Number	Message	Contents
213	ILLEGAL COMMAND IN SYNCHRO–MODE (M series)	Movement is commanded for the axis to be synchronously controlled. Any of the following alarms occurred in the operation with the simple synchronization control. 1) The program issued the move command to the slave axis. 2) The program issued the manual continuous feed/manual handle feed/incremental feed command to the slave axis. 3) The program issued the automatic reference position return command without specifying the manual reference position return after the power was turned on. 4) The difference between the position error amount of the master and slave axes exceeded the value specified in parameter NO.8313.
	ILLEGAL COMMAND IN SYNCHRO–MODE (T series)	A move command has been specified for an axis subject to synchronous control.
214	ILLEGAL COMMAND IN SYNCHRO–MODE	Coordinate system is set or tool compensation of the shift type is executed in the synchronous control. Correct the program.
217	DUPLICATE G51.2 (COMMANDS) (T series)	G51.2/G251 is further commanded in the G51.2/G251 mode. Modify the program.
218	NOT FOUND P/Q COMMAND IN G251 (T series)	P or Q is not commanded in the G251 block, or the command value is out of the range. Modify the program.
219	COMMAND G250/G251 INDEPENDENTLY (T series)	G251 and G250 are not independent blocks.
220	ILLEGAL COMMAND IN SYNCHR–MODE (T series)	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis.
221	ILLEGAL COMMAND IN SYNCHR–MODE (T series)	Polygon machining synchronous operation and axis control or balance cutting are executed at a time. Modify the program.
222	DNC OP. NOT ALLOWED IN BG.–EDIT (M series)	Input and output are executed at a time in the background edition. Execute a correct operation.
224	RETURN TO REFERENCE POINT (M series)	Reference position return has not been performed before the automatic operation starts. Perform reference position return only when bit 0 of parameter 1005 is 0.
	TURN TO REFERENCE POINT (T series)	Reference position return is necessary before cycle start.
225	SYNCHRONOUS/MIXED CONTROL ERROR (T series (At two–path))	This alarm is generated in the following circumstances. (Searched for during synchronous and mixed control command. 1 When there is a mistake in axis number parameter (No. 1023) setting. 2 When there is a mistake in control commanded. During hobbing synchronization, a command to bring the C–axis under synchronous, composite, or superimposed control is made. Modify the program or the parameter.
226	ILLEGAL COMMAND IN SYNCHRO–MODE (T series (At two–path))	A travel command has been sent to the axis being synchronized in synchronous mode. Modify the program or the parameter.
229	CAN NOT KEEP SYNCHRO–STATE (T series)	This alarm is generated in the following circumstances. 1 When the synchro/mixed state could not be kept due to system overload. 2 The above condition occurred in CMC devices (hardware) and synchro–state could not be kept. (This alarm is not generated in normal use conditions.)
230	R CODE NOT FOUND (Grinding machine) (M series)	The infeed quantity R has not been instructed for the G161 block. Or the R command value is negative. Correct the program.

Number	Message	Contents
231	ILLEGAL FORMAT IN G10 OR L50	Any of the following errors occurred in the specified format at the programmable–parameter input. 1 Address N or R was not entered. 2 A number not specified for a parameter was entered. 3 The axis number was too large. 4 An axis number was not specified in the axis–type parameter. 5 An axis number was specified in the parameter which is not an axis type. Correct the program. 6 In the locked state set by the password function, an attempt was made to set bit 4 (NE9) of parameter No. 3204 to 0 or change the contents of parameter No. 3210. 7 An attempt was made to change a program encryption parameter (parameter No. 3220 to 3223).
232	TOO MANY HELICAL AXIS COMMANDS	Three or more axes (in the normal direction control mode (M series) two or more axes) were specified as helical axes in the helical interpolation mode.
233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS–232–C interface, other users were using it.
239	BP/S ALARM	While punching was being performed with the function for controlling external I/O units ,background editing was performed.
240	BP/S ALARM	Background editing was performed during MDI operation.
241	ILLEGAL FORMAT IN G02.2/G03.2 (M series)	The end point, I, J, K, or R is missing from a command for involute interpolation.
242	ILLEGAL COMMAND IN G02.2/G03.2 (M series)	An invalid value has been specified for involute interpolation. • The start or end point is within the basic circle. • I, J, K, or R is set to 0. • The number of rotations between the start of the involute curve and the start or end point exceeds 100.
243	OVER TOLERANCE OF END POINT (M series)	The end point is not on the involute curve which includes the start point and thus falls outside the range specified with parameter No. 5610.
244	P/S ALARM (T series)	In the skip function activated by the torque limit signal, the number of accumulated erroneous pulses exceed 32767 before the signal was input. Therefore, the pulses cannot be corrected with one distribution. Change the conditions, such as feed rates along axes and torque limit, and try again.
245	T–CODE NOT ALOWEE IN THIS BLOCK (T series)	One of the G codes, G50, G10, and G04, which cannot be specified in the same block as a T code, was specified with a T code.
246	ENCODE PROGRAM NUMBER ERROR	During read of an encrypted program, an attempt was made to store the program with a number exceeding the protection range. (See parameter Nos. 3222 and 223.)
247	ILLEGAL CODE USED FOR OUTPUT	When an encrypted program is output, EIA is set for the punch code. Specify ISO.
250	Z AXIS WRONG COMMAND (ATC) (M series)	Movement along the Z–axis is specified in a block specifying a tool change command (M06T_). (Only for ROBODRILL)

Number	Message	Contents
251	ATC ERROR (M series)	This alarm is issued in the following cases: • An M06T_ command contains an unusable T code. • An M06 command has been specified when the Z machine coordi nate is positive. • The parameter for the current tool number (No. 7810) is set to 0. • An M06 command has been specified in canned cycle mode. • A reference position return command (G27 to G44) and M06 com-mand have been specified in the same block. • An M06 command has been specified in tool compensation mode (G41 to G44). • An M06 command has been specified without performing reference position return after power–on or the release of emergency stop. • The machine lock signal or Z–axis ignore signal has been turned on during tool exchange. • A pry alarm has been detected during tool exchange. Refer to diagnosis No. 530 to determine the cause. (Only for ROBO-DRILL)
252	ATC SPINDLE ALARM (M series)	An excessive error arose during spindle positioning for ATC. For details, refer to diagnosis No. 531. (Only for ROBODRILL)
253	G05 IS NOT AVAILABLE (M series)	Alarm details Binary input operation using high–speed remote buffer (G05) or high–speed cycle machining (G05) has been specified in advance control mode (G08P1). Execute G08P0; to cancel advance control mode, be-fore executing these G05 commands.
4500	REPOSITIONING INHIBITED	A repositioning command was specified in the circular interpolation (G02, G03) mode.
4502	ILLEGAL COMMAND IN BOLT HOLE	In a bolt hole circle (G26) command, the radius (I) was set to zero or a negative value, or the number of holes (K) was set to zero. Alternatively, I, J, or K was not specified.
4503	ILLEGAL COMMAND IN LINE AT ANGLE	In a line-at-angle (G76) command, the number of holes (K) was set to zero or a negative value. Alternatively, I, J, or K was not specified.
4504	ILLEGAL COMMAND IN ARC	In an arc (G77) command, the radius (I) or the number of holes (K) was set to zero or a negative value. Alternatively, I, J, K, or P was not speci-fied.
4505	ILLEGAL COMMAND IN GRID	In a grid (G78, G79) command, the number of holes (P, K) was set to zero or a negative value. Alternatively, I, J, K, or P was not specified.
4506	ILLEGAL COMMAND IN SHARE PROOFS	In a shear proof (G86) command, the tool size (P) was set to zero, or the blanking length (I) was 1.5 times larger than the tool size (P) or less. Al-ternatively, I, J, or P was not specified.
4507	ILLEGAL COMMAND IN SQUARE	In a square (G87) command, the tool size (P,Q) was set to zero or a neg-ative value, or the blanking length (I, J) was three times larger than the tool size (P, Q) or less. Alternatively, I, J, P, or Q was not specified.
4508	ILLEGAL COMMAND IN RADIUS	In a radius (G88) command, the traveling pitch (Q) or radius (I) was set to zero or a negative value, or the traveling pitch (Q) was greater than or equal to the arc length. Alternatively, I, J, K, P, or Q was not specified.
4509	ILLEGAL COMMAND IN CUT AT ANGLE	In a cut-at-angle (G89) command, the traveling pitch (Q) was set to zero, negative value, or another value larger than or equal to the length (I). Alternatively, I, J, P, or Q was not specified.
4510	ILLEGAL COMMAND IN LINE-PUNCH	In a linear punching (G45) command, the traveling distance was set to zero or a value 1.5 times larger than the tool size (P) or less. Alternative-ly, P was not specified.

Number	Message	Contents
4511	ILLEGAL COMMAND IN CIRCLE-PUNCH	In a circular punching (G46, G47) command, the same position was specified for both start and end points of the arc, radius (R) of the arc was set to zero, or the pitch (Q) was set to a value exceeding the arc length. Alternatively, R or Q was not specified.
4520	T, M INHIBITED IN NIBBLING-MODE	T code, M code, G04, G70 or G75 was specified in the nibbling mode.
4521	EXCESS NIBBLING MOVEMENT (X, Y)	In the nibbling mode, the X-axis or Y-axis traveling distance was larger than or equal to the limit (No. 16188 to 16193).
4522	EXCESS NIBBLING MOVEMENT (C)	In the circular nibbling (G68) or usual nibbling mode, the C-axis traveling distance was larger than or equal to the limit (No. 16194).
4523	ILLEGAL COMMAND IN CIRCLE-NIBBL	In a circular nibbling (G68) command, the traveling pitch (Q) was set to zero, a negative value, or a value larger than or equal to the limit (No. 16186, 16187), or the radius (I) was set to zero or a negative value. Alternatively, I, J, K, P, or Q was not specified.
4524	ILLEGAL COMMAND IN LINE-NIBBL	In a linear nibbling (G69) command, the traveling pitch (Q) was set to zero, negative value, or a value larger than or equal to the limit (No. 16186, 16187). Alternatively, I, J, P, or Q was not specified.
4530	A/B MACRO NUMBER ERROR	The number for storing and calling by an A or B macro was set to a value beyond the range from 1 to 5.
4531	U/V MACRO FORMAT ERROR	An attempt was made to store a macro while storing another macro using a U or V macro. A V macro was specified although the processing to store a macro was not in progress. A U macro number and V macro number do not correspond with each other.
4532	IMPROPER U/V MACRO NUMBER	The number of an inhibited macro (number beyond the range from 01 to 99) was specified in a U or V macro command.
4533	U/V MACRO MEMORY OVERFLOW	An attempt was made to store too many macros with a U or V macro command.
4534	W MACRO NUMBER NOT FOUND	Macro number W specified in a U or V macro command is not stored.
4535	U/V MACRO NESTING ERROR	An attempt was made to call a macro which is defined three times or more using a U or V macro command. An attempt was made to store 15 or more macros in the storage area for macros of number 90 to 99.
4536	NO W, Q COMMAND IN MULTI-PIECE	W or Q was not specified in the command for taking multiple workpieces (G73, G74).
4537	ILLEGAL Q VALUE IN MULTI-PIECE	In the command for taking multiple workpieces (G73, G74), Q is set to a value beyond the range from 1 to 4.
4538	W NO. NOT FOUND IN MULTI-PIECE	Macro number W specified in the command for taking multiple workpieces (G73, G74) is not stored.
4539	MULTI-PIECE SETTING IS ZERO	The command for taking multiple workpieces (G73, G74) was specified although zero is specified for the function to take multiple workpieces (No. 16206 or signals MLP1 and MLP2 (PMC address G231, #0 and #1)).
4540	MULTI-PIECE COMMAND WITHIN MACRO	The command for taking multiple workpieces (G73, G74) was specified when a U or V macro was being stored.
4542	MULTI-PIECE COMMAND ERROR	Although G98P0 was specified, the G73 command was issued. Although G98K0 was specified, the G74 command was issued.

Number	Message	Contents
4543	MULTI-PIECE Q COMMAND ERROR	Although G98P0 was specified, the Q value for the G74 command was not 1 or 3. Although G98K0 was specified, the Q value for the G73 command was not 1 or 2.
4544	MULTI-PIECE RESTART ERROR	In the command for resuming taking multiple workpieces, the resume position (P) is set to a value beyond the range from 1 to total number of workpieces to be machined.
4549	ILLEGAL TOOL DATA FORMAT	The quantity of tool data patterns to be saved is too large to fit the usable area (16 KB).
4600	T, C COMMAND IN INTERPOLATION	In the linear interpolation (G01) mode or circular interpolation (G02, G03) mode, a T command or C-axis command was specified.
4601	INHIBITED T, M COMMAND	In the block of G52, G72, G73, or G74, a T or M command was specified.
4602	ILLEGAL T-CODE	The specified T command is not cataloged on the tool register screen.
4603	C AXIS SYNCHRONOUS ERROR	The difference between the position deviation value of C1 axis and C2 axis exceeds the parameter value (No. 16364, 16365) with the C–axis synchronous control function.
4604	ILLEGAL AXIS OPERATION	A C-axis command was specified in the block containing a T command for multiple tools.
4605	NEED ZRN	C–axis synchronization failed.
4630	ILLEGAL COMMAND IN LASER MODE	In the laser mode, a nibbling command or pattern command was specified. In the tracing mode, an attempt was made to make a switch to the punching mode.
4650	IMPROPER G-CODE IN OFFSET MODE	In the cutter compensation mode, an inhibited G code (pattern command, G73, G74, G75, etc.) was specified.
4700	PROGRAM ERROR (OT +)	The value specified in the X-axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4701	PROGRAM ERROR (OT –)	The value specified in the X-axis move command exceeded the negative value of stored stroke limit 1. (Advance check)
4702	PROGRAM ERROR (OT +)	The value specified in the Y-axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4703	PROGRAM ERROR (OT –)	The value specified in the Y-axis move command exceeded the negative value of stored stroke limit 1. (Advance check)
4704	PROGRAM ERROR (OT +)	The value specified in the Z-axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4705	PROGRAM ERROR (OT –)	The value specified in the Z-axis move command exceeded the negative value of stored stroke limit 1. (Advance check)
5000	ILLEGAL COMMAND CODE (M series)	The specified code was incorrect in the high–precision contour control (HPCC) mode.
5003	ILLEGAL PARAMETER (HPCC) (M series)	There is an invalid parameter.
5004	HPCC NOT READY (M series)	High–precision contour control is not ready.
5006	TOO MANY WORD IN ONE BLOCK (M series)	The number of words specified in a block exceeded 26 in the HPCC mode.
5007	TOO LARGE DISTANCE (M series)	In the HPCC mode, the machine moved beyond the limit.
5009	PARAMETER ZERO (DRY RUN) (M series)	The maximum feedrate (parameter No. 1422) or the feedrate in dry run (parameter No. 1410) is 0 in the HPCC model.
5010	END OF RECORD	The end of record (%) was specified. I/O is incorrect. modify the program.

Number	Message	Contents
5011	PARAMETER ZERO(CUT MAX) (M series)	The maximum cutting feedrate (parameter No. 1422, No. 1430, No. 1431, No. 1432) is 0 in the HPCC mode.
5012	G05 P10000 ILLEGAL START UP (HPCC) (M series)	Function category: High–precision contour control Alarm details: G05 P10000 has been specified in a mode from which the system cannot enter HPCC mode.
5013	HPCC: CRC OFS REMAIN AT CANCEL (M series)	G05P0 has been specified in G41/G42 mode or with offset remaining.
5014	TRACE DATA NOT FOUND	Transfer cannot be performed because no trace data exists.
5015	NO ROTATION AXIS (M series)	The specified rotation axis does not exist for tool axis direction handle feed.
5016	ILLEGAL COMBINATION OF M CODE	M codes which belonged to the same group were specified in a block. Alternatively,an M code which must be specified without other M codes in the block was specified in a block with other M codes.
5018	POLYGON SPINDLE SPEED ERROR (T series)	Function category: Polygon turning Alarm details: In G51.2 mode, the speed of the spindle or polygon synchronous axis either exceeds the clamp value or is too small. The specified rotation speed ratio thus cannot be maintained.
5020	PARAMETER OF RESTART ERROR	An erroneous parameter was specified for restarting a program. A parameter for program restart is invalid.
5030	ILLEGAL COMMAND (G100) (T series)	The end command (G110) was specified before the registratioin start command (G101, G102, or G103) was specified for the B–axis.
5031	ILLEGAL COMMAND (G100, G102, G103) (T series)	While a registration start command (G101, G102, or G103) was being executed, another registration start command was specified for the B–axis.
5032	NEW PRG REGISTERED IN B–AXS MOVE (T series)	While the machine was moving about the B–axis, at attempt was made to register another move command.
5033	NO PROG SPACE IN MEMORY B–AXS (T series)	Commands for movement about the B–axis were not registered because of insufficient program memory.
5034	PLURAL COMMAND IN G110 (T series)	Multiple movements were specified with the G110 code for the B–axis.
5035	NO FEEDRATE COMMANDED B–AXS (T series)	A feedrate was not specified for cutting feed about the B–axis.
5036	ADDRESS R NOT DEFINED IN G81–G86 (T series)	Point R was not specified for the canned cycle for the B–axis.
5037	ADDRESS Q NOT DEFINED IN G83 (T series)	Depth of cut Q was not specified for the G83 code (peck drilling cycle). Alternatively, 0 was specified in Q for the B–axis.
5038	TOO MANY START M–CODE COMMAND (T series)	More than six M codes for starting movement about the B–axis were specified.
5039	START UNREGISTERED B–AXS PROG (T series)	An attempt was made to execute a program for the B–axis which had not been registered.
5040	CAN NOT COMMANDED B–AXS MOVE (T series)	The machine could not move about the B–axis because parameter No.8250 was incorrectly specified, or because the PMC axis system could not be used.
5041	CAN NOT COMMANDED G110 BLOCK (T series)	Blocks containing the G110 codes were successively specified in tool–tip radius compensation for the B–axis.

Number	Message	Contents
5043	TOO MANY G68 NESTING (M series)	Three–dimensional coordinate conversion G68 has been specified three or more times.
	TOO MANY G68 NESTING (T series)	Three–dimensional coordinate conversion G68.1 has been specified three or more times.
5044	G68 FORMAT ERROR (M series)	A G68 command block contains a format error. This alarm is issued in the following cases: 1. I, J, or K is missing from a G68 command block (missing coordinate rotation option). 2. I, J, and K are 0 in a G68 command block. 3. R is missing from a G68 command block.
	G68 FORMAT ERROR (T series)	A G68.1 command block contains a format error. This alarm is issued in the following cases: 1. I, J, or K is missing from a G68.1 command block (missing coordinate rotation option). 2. I, J, and K are 0 in a G68.1 command block. 3. R is missing from a G68.1 command block.
5046	ILLEGAL PARAMETER (ST.COMP)	The parameter settings for straightness compensation contain an error. Possible causes are as follows: 1. A parameter for a movement axis or compensation axis contains an axis number which is not used. 2. More than 128 pitch error compensation points exist between the negative and positive end points. 3. Compensation point numbers for straightness compensation are not assigned in the correct order. 4. No straightness compensation point exists between the pitch error compensation points at the negative and positive ends. 5. The compensation value for each compensation point is too large or too small. 6 The settings of parameters Nos. 13881 to 13886 are illegal (in the interpolation type straightness compensation).
5050	ILL–COMMAND IN CHOPPING MODE (M series)	A command for switching the major axis has been specified for circular threading. Alternatively, a command for setting the length of the major axis to 0 has been specified for circular threading.
5051	M–NET CODE ERROR	Abnormal character received (other than code used for transmission)
5052	M–NET ETX ERROR	Abnormal ETX code
5053	M–NET CONNECT ERROR	Connection time monitoring error (parameter No. 175)
5054	M–NET RECEIVE ERROR	Polling time monitoring error (parameter No. 176)
5055	M–NET PRT/FRT ERROR	Vertical parity or framing error
5057	M–NET BOARD SYSTEM DOWN	Transmission timeout error (parameter No. 177) ROM parity error CPU interrupt other than the above
5058	G35/G36 FORMAT ERROR (T series)	A command for switching the major axis has been specified for circular threading. Alternatively, a command for setting the length of the major axis to 0 has been specified for circular threading.
5059	RADIUS IS OUT OF RANGE	A radius exceeding nine digits has been specified for circular interpolation with the center of the arc specified with I, J, and K.

Number	Message	Contents
5060	ILLEGAL PARAMETER IN G02.3/G03.3 (M series)	There is a parameter setting error. Parameter No. 5641 (setting of the linear axis) is not set. The axis set in parameter No. 5641 is not a linear axis. Parameter No. 5642 (setting of a rotation axis) is not set. The axis set in parameter No. 5642 is not a rotation axis. The linear and rotation axes cannot be controlled by the CNC. (The value set in parameter No. 1010 is exceeded.)
5061	ILLEGAL FORMAT IN G02.3/G03.3 (M series)	The exponential interpolation command (G02.3/G03.3) has a format error. Address I, J, or K is not specified. The value of address I, J, or K is 0.
5062	ILLEGAL COMMAND IN G02.3/G03.3	The value specified in an exponential interpolation command (G02.3/03.3) is illegal. A value that does not allow exponential interpolation is specified. (For example, a negative value is specified in In.)
5063	IS NOT PRESET AFTER REF. (M series)	Function category: Workpiece thickness measurement Alarm details The position counter was not preset before the start of workpiece thickness measurement. This alarm is issued in the following cases: (1) An attempt has been made to start measurement without first establishing the origin. (2) An attempt has been made to start measurement without first presetting the position counter after manual return to the origin.
5064	DIFFERRENT AXIS UNIT (IS–B, IS–C) (M series)	Circular interpolation has been specified on a plane consisting of axes having different increment systems.
5065	DIFFERENT AXIS UNIT (PMC AXIS) (M series)	Axes having different increment systems have been specified in the same DI/DO group for PMC axis control. Modify the setting of parameter No. 8010.
5067	G05 PO COMMANDED IN G68/G51 MODE (HPCC) (M series)	HPCC mode cannot be canceled during G51 (scaling) or G68 (coordinate system rotation). Correct the program.
5068	G31 FORMAT ERROR (M series)	The continuous high–speed skip command (G31 P90) has one of the following errors: 1. The axis along which the tool is moved is not specified. 2. More than one axis is specified as the axis along which the tool is moved. Alternatively, the EGB skip command (G31.8) or continuous high–speed skip command (G31.9) has one of the following errors: 1. A move command is specified for the EGB axis (workpiece axis). 2. More than one axis is specified. 3. P is not specified. 4. The specified Q value exceeds the allowable range. Correct the program.
5069	WHL–C:ILLEGA P–DATA (M series)	The P data in selection of the grinding–wheel wear compensation center is illegal.
5073	NO DECIMAL POINT	No decimal point has been specified for an address requiring a decimal point.
5074	ADDRESS DUPLICATION ERROR	The same address has been specified two or more times in a single block. Alternatively, two or more G codes in the same group have been specified in a single block.
5082	DATA SERVER ERROR	This alarm is detailed on the data server message screen.

Number	Message	Contents
5085	SMOOTH IPL ERROR 1	A block for specifying smooth interpolation contains a syntax error.
5096	MISMATCH WAITING M–CODE (M series)	Different wait codes (M codes) were specified in HEAD1 and HEAD2. Correct the program.
5110	NOT STOP POSITION (G05.1 G1) (M series)	An illegal G code was specified in AI contour control mode. A command was specified for the index table indexing axis in AI control mode.
	NOT STOP POSITION (G05.1 G1) (21*i*–M)	An illegal G code was specified in AI look–ahead control mode. A command was specified for the index table indexing axis in AI look–ahead control mode.
5111	IMPROPER MODEL G–CODE (G05.1 G1) (M series)	An illegal G code is left modal when AI contour control mode was specified.
	IMPROPER MODEL G–CODE (G05.1 G1) (21*i*–M)	An illegal G code is left modal when AI look–ahead control mode was specified.
5112	G08 CAN NOT BE COMMANDED (G05.1 G1) (M series)	Look–ahead control (G08) was specified in AI contour control mode.
	G08 CAN NOT BE COMMANDED (G05.1 G1) (21*i*–M)	Look–ahead control (G08) was specified in AI look–ahead control mode.
5114	NOT STOP POSITION (G05.1 Q1) (M series)	At the time of restart after manual intervention, the coordinates at which the manual intervention occurred have not been restored.
	CAN NOT ERROR IN MDI MODE (G05.1) (21*i*–M)	AI contour control (G05.1) was specified in MDI mode.
5115	SPL : ERROR (M series)	There is an error in the specification of the rank.
		No knot is specified.
		The knot specification has an error.
		The number of axes exceeds the limits.
		Other program errors
5116	SPL : ERROR (M series)	There is a program error in a block under look–ahead control.
		Monotone increasing of knots is not observed.
		In NURBS interpolation mode, a mode that cannot be used together is specified.
5117	SPL : ERROR (M series)	The first control point of NURBS is incorrect.
5118	SPL : ERROR (M series)	After manual intervention with manual absolute mode set to on, NURBS interpolation was restarted.

Number	Message	Contents
5122	ILLEGAL COMMAND IN SPIRAL (M series)	A spiral interpolation or conical interpolation command has an error. Specifically, this error is caused by one of the following: 1) L = 0 is specified. 2) Q = 0 is specified. 3) R/, R/, C is specified. 4) Zero is specified as height increment. 5) Three or more axes are specified as the height axes. 6) A height increment is specified when there are two height axes. 7) Conical interpolation is specified when the helical interpolation function is not selected. 8) Q < 0 is specified when radius difference > 0. 9) Q > 0 is specified when radius difference < 0. 10) A height increment is specified when no height axis is specified.
5123	OVER TOLERANCE OF END POINT (M series)	The difference between a specified end point and the calculated end point exceeds the allowable range (parameter 3471).
5124	CAN NOT COMMAND SPIRAL (M series)	A spiral interpolation or conical interpolation was specified in any of the following modes: 1) Scaling 2) Programmable mirror image 3) Polar coordinate interpolation In cutter compensation C mode, the center is set as the start point or end point.
5134	FSSB : OPEN READY TIME OUT	Initialization did not place FSSB in the open ready state.
5135	FSSB : ERROR MODE	FSSB has entered error mode.
5136	FSSB : NUMBER OF AMPS IS SMALL	In comparison with the number of controlled axes, the number of amplifiers recognized by FSSB is not enough.
5137	FSSB : CONFIGURATION ERROR	FSSB detected a configuration error.
5138	FSSB : AXIS SETTING NOT COMPLETE	In automatic setting mode, axis setting has not been made yet. Perform axis setting on the FSSB setting screen.
5139	FSSB : ERROR	Servo initialization did not terminate normally. The optical cable may be defective, or there may be an error in connection to the amplifier or another module. Check the optical cable and the connection status.
5155	NOT RESTART PROGRAM BY G05	During servo leaning control by G05, an attempt was made to perform restart operation after feed hold or interlock. This restart operation cannot be performed. (G05 leaning control terminates at the same time.)
5156	ILLEGAL AXIS OPERATION (AICC) (M series)	In AI contour control mode, the controlled axis selection signal (PMC axis control) changes. In AI contour control mode, the simple synchonous axis selection signal changes.
	ILLEGAL AXIS OPERATION (AICC) (21*i*–M)	In AI look–ahead control mode, the controlled axis selection signal (PMC axis control) changes. In AI look–ahead control mode, the simple synchonous axis selection signal changes.
5157	PARAMETER ZERO (AICC) (M series)	Zero is set in the parameter for the maximum cutting feedrate (parameter No. 1422 or 1432). Zero is set in the parameter for the acceleration/deceleration before interpolation (parameter No. 1770 or 1771). Set the parameter correctly.

Number	Message	Contents
5195	DIRECTION CAN NOT BE JUDGED (T series)	When the touch sensor with a single contact signal input is used in the direct input B function for tool offset measurement values, the stored pulse direction is not constant. One of the following conditions exists: · The stop state exists in offset write mode. · Servo off state · The direction varies. · Movement takes place simultaneously along two axes.
5196	ILLEGAL OPERATION (HPCC) (M series)	Detach operation was performed in HPCC mode. (If detach operation is performed in HPCC mode, this alarm is issued after the currently executed block terminates.)
5197	FSSB : OPEN TIME OUT	The CNC permitted FSSB to open, but FSSB was not opened.
5198	FSSB : ID DATA NOT READ	Temporary assignment failed, so amplifier initial ID information could not be read.
5199	FINE TORQUE SENSING PARAMETER	A parameter related to the fine torque sensing function is illegal. · The storage interval is invalid. · An invalid axis number is set as the target axis. Correct the parameter.
5212	SCREEN COPY : PARAMETER ERROR	There is a parameter setting error. Check that 4 is set as the I/O channel.
5213	SCREEN COPY : COMMUNICATION ERROR	The memory card cannot be used. Check the memory card. (Check whether the memory card is write–protected or defective.)
5214	SCREEN COPY : DATA TRANSFER ERROR	Data transfer to the memory card failed. Check whether the memory card space is insufficient and whether the memory card was removed during data transfer.
5218	ILLEGAL PARAMETER (INCL. COMP)	There is an inclination compensation parameter setting error. Cause: 1. The number of pitch error compensation points between the negative (–) end and positive (+) end exceeds 128. 2. The relationship in magnitude among the inclination compensation point numbers is incorrect. 3. An inclination compensation point is not located between the negative (–) end and positive (+) end of the pitch error compensation points. 4. The amount of compensation per compensation point is too large or too small. Correct the parameter.
5219	CAN NOT RETURN	Manual intervention or return is not allowed during three–dimensional coordinate conversion.
5220	REFERENCE POINT ADJUSTMENT MODE	A parameter for automatically set a reference position is set. (Bit 2 of parameter No. 1819 = 1) Perform automatic setting. (Position the machine at the reference position manually, then perform manual reference position return.) Supplementary: Automatic setting sets bit 2 of parameter No. 1819 to 0.
5222	SRAM CORRECTABLE ERROR	The SRAM correctable error cannot be corrected. Cause: A memory problem occurred during memory initialization. Action: Replace the master printed circuit board (SRAM module).

Number	Message	Contents
5227	FILE NOT FOUND	A specified file is not found during communication with the built–in Handy File.
5228	SAME NAME USED	There are duplicate file names in the built–in Handy File.
5229	WRITE PROTECTED	A floppy disk in the built–in Handy File is write protected.
5231	TOO MANY FILES	The number of files exceeds the limit during communication with the built–in Handy File.
5232	DATA OVER–FLOW	There is not enough floppy disk space in the built–in Handy File.
5235	COMMUNICATION ERROR	A communication error occurred during communication with the built–in Handy File.
5237	READ ERROR	A floppy disk in the built–in Handy File cannot be read from. The floppy disk may be defective, or the head may be dirty. Alternatively, the Handy File is defective.
5238	WRITE ERROR	A floppy disk in the built–in Handy File cannot be written to. The floppy disk may be defective, or the head may be dirty. Alternatively, the Handy File is defective.
5242	ILLEGAL AXIS NUMBER (M series)	The axis number of the synchronous master axis or slave axis is incorrect. (This alarm is issued when flexible synchronization is turned on.) Alternatively, the axis number of the slave axis is smaller than that of the master axis.
5243	DATA OUT OF RANGE (M series)	The gear ratio is not set correctly. (This alarm is issued when flexible synchronization is turned on.)
5244	TOO MANY DI ON (M series)	Even when an M code was encountered in automatic operation mode, the flexible synchronization mode signal was not driven on or off. Check the ladder and M codes.
5245	OTHER AXIS ARE COMMANDED (M series)	One of the following command conditions was present during flexible synchronization or when flexible synchronization was turned on: 1. The synchronous master axis or slave axis is the EGB axis. 2. The synchronous master axis or slave axis is the chopping axis. 3. In reference position return mode
5251	ILLEGAL PARAMETER IN G54.2 (M series)	A fixture offset parameter (No. 7580 to 7588) is illegal. Correct the parameter.
5252	ILLEGAL P COMMAND IN G54.2 (M series)	The P value specifying the offset number of a fixture offset is too large. Correct the program.
5257	G41/G42 NOT ALLOWED IN MDI MODE (M series)	G41/G42 (cutter compensation C: M series) was specified in MDI mode. (Depending on the setting of bit 4 of parameter No. 5008)
	G41/G42 NOT ALLOWED IN MDI MODE (T series)	G41/G42 (tool–nose radius compensation: T series) was specified in MDI mode. (Depending on the setting of bit 4 of parameter No. 5008)
5300	SET ALL OFFSET DATAS AGAIN	After the inch/metric automatic conversion function (OIM: Bit 0 of parameter No. 5006) for tool offset data is enabled or disabled, all the tool offset data must be reset. This message reminds the operator to reset the data. If this alarm is issued, reset all the tool offset data. Operating the machine without resetting the data will result in a malfunction.
5302	ILLEGAL COMMAND IN G68 MODE	A command to set the coordinate system is specified in the coordinate system rotation mode.

Number	Message	Contents
5303	TOUCH PANEL ERROR	A touch panel error occurred. Cause: 1. The touch panel is kept pressed. 2. The touch panel was pressed when power was turned on. Remove the above causes, and turn on the power again.
5306	MODE CHANGE ERROR	In a one–touch macro call, mode switching at the time of activation is not performed correctly.
5307	INTERNAL DATA OVER FLOW (M series)	In the following function, internal data exceeds the allowable range. 1) Improvement of the rotation axis feedrate
5311	FSSB:ILLEGAL CONNECTION	A connection related to FSSB is illegal. This alarm is issued when either of the following is found: 1. Two axes having adjacent servo axis numbers (parameter No. 1023), odd number and even number, are assigned to amplifiers to which different FSSB systems are connected. 2. The system does not satisfy the requirements for performing HRV control, and use of two pulse modules connected to different FSSB systems having different FSSB current control cycles is specified.
5321	S–COMP. VALUE OVERFLOW	The straightness compensation value has exceeded the maximum value of 32767. After this alarm is issued, make a manual reference position return.
5400	SPL:ILLEGAL AXIS COMMAND (M series)	An axis specified for spline interpolation or smooth interpolation is incorrect. If an axis that is not the spline axis is specified in spline interpolation mode, this alarm is issued. The spline axis is the axis specified in a block containing G06.1 or the next block. For smooth interpolation, the axis specified in G5.1Q2 is incorrect.
5401	SPL:ILLEGAL COMMAND (M series)	In a G code mode in which specification of G06.1 is not permitted, G06.1 is specified.
5402	SPL:ILLEGAL AXIS MOVING (M series)	A movement is made along an axis that is not the spline interpolation axis. For example, in three–dimensional tool compensation mode using an offset vector of which components are the X–, Y–, and Z–axes, when two–axis spline interpolation is performed with the two spline axes set to the X– and Y–axes, a movement along the Z–axis occurs, resulting in this alarm.
5403	SPL:CAN NOT MAKE VECTOR (M series)	Three–dimensional tool compensation vectors cannot be generated. · When a three–dimensional tool compensation vector is created for the second or subsequent point, that point, previous point, and next point are on the same straight line, and that straight line and the three–dimensional tool compensation vector for the previous point are in parallel. · When a three–dimensional tool compensation vector is created at the end point of smooth interpolation or spline interpolation, the end point and the point two points before are the same.
5405	ILLEGAL PARAMETER IN G41.2/ G42.2 (M series)	The parameter setting that determines the relationship between the rotation axis and rotation plane is incorrect.
5406	G41.3/G40 FORMAT ERROR (M series)	1) A G41.3 or G40 block contains a move command. 2) A G1.3 block contains a G code or M code for which buffering is suppressed.

Number	Message	Contents
5407	ILLEGAL COMMAND IN G41.3 (M series)	1) A G code that belongs to group 01 except G00 and G01 is specified in G41.3 mode. 2) An offset command (a G code belonging to group 07) is specified in G41.3 mode. 3) The block next to G41.3 (startup) contains no movement.
5408	G41.3 ILLEGAL START_UP (M series)	1) In a mode of group 01 except G00 and G01, G41.3 (startup) is specified. 2) At startup, the included angle of the tool direction vector and move direction vector is 0 or 180 degrees.
5409	ILLEGAL PARAMETER IN G41.3 (M series)	The parameter setting (No. xxxx to xxxx) that determines the relationship between the rotation axis and rotation plane is incorrect.
5411	NURBS:ILLEGAL ORDER (M series)	The number of steps is specified incorrectly.
5412	NURBS:NO KNOT COMMAND (M series)	No knot is specified. Alternatively, in NURBS interpolation mode, a block not relating to NURBS interpolation is specified.
5413	NURBS:ILLEGAL AXIS COMMAND (M series)	An axis not specified with controlled points is specified in the first block.
5414	NURBS:ILLEGAL KNOT (M series)	The number of blocks containing knots only is insufficient.
5415	NURBS:ILLEGAL CANCEL (M series)	Although NURBS interpolation is not completed yet, the NURBS interpolation mode is turned off.
5416	NURBS:ILLEGAL MODE (M series)	A mode that cannot be used with NURBS interpolation mode is specified in NURBS interpolation mode.
5417	NURBS:ILLEGAL MULTI–KNOT (M series)	As many knots as the number of steps are not specified at the start and end points.
5418	NURBS:ILLEGAL KNOT VALUE (M series)	Knots do not increase in monotone.
5420	ILLEGAL PARAMETER IN G43.4/ G43.5 (M series)	A parameter related to pivot tool length compensation is incorrect.
5421	ILLEGAL COMMAND IN G43.4/ G43.5 (M series)	In pivot tool length compensation (type 2) mode, a rotation axis is specified.
5422	EXCESS VELOCITY IN G43.4/G43.5 (M series)	As a result of pivot tool length compensation, an attempt was made to move the tool along an axis at a feedrate exceeding the maximum cutting feedrate.
5425	ILLEGAL OFFSET VALUE (M series)	The offset number is incorrect.
5430	ILLEGAL COMMAND IN 3–D CIR (M series)	In a modal state in which three–dimensional circular interpolation cannot be specified, a three–dimensional circular interpolation (G02.4/G03.4) is specified. Alternatively, in three–dimensional circular interpolation mode, a code that cannot be specified is specified.
5432	G02.4/G03.4 FORMAT ERROR (M series)	A three–dimensional circular interpolation command (G02.4/G03.4) is incorrect.
5433	MANUAL INTERVENTION IN 3–D CIR (M series)	In three–dimensional circular interpolation mode (G02.4/G03.4), manual intervention was made when the manual absolute switch was on.
5435	PARAMETER OUT OF RANGE (TLAC) (M series)	Incorrect parameter setting (set value range)
5436	PARAMETER SETTING ERROR 1 (TLAC) (M series)	Incorrect parameter setting (setting of the rotation axis)
5437	PARAMETER SETTING ERROR 2 (TLAC) (M series)	Incorrect parameter setting (setting of the tool axis)
5440	ILLEGAL DRILLING AXIS SELECTED (M series)	The drilling axis specified for the drilling canned cycle is incorrect. The G code command block of the canned cycle does not specify the Z point of the drilling axis. When there is a parallel axis with the drilling axis, the parallel axis is also specified at the same time.

Number	Message	Contents
5445	CRC:MOTION IN G39 (M series)	Corner circular interpolation (G39) of cutter compensation is not specified alone but is specified with a move command.
5446	CRC:NO AVOIDANCE (M series)	Because there is no interference evade vector, the interference check evade function of cutter compensation cannot evade interference.
5447	CRC:DANGEROUS AVOIDANCE (M series)	The interference check evade function of cutter compensation determines that an evade operation will lead to danger.
5448	CRC:INTERFERENCE TO AVD. (M series)	In the interference check evade function of cutter compensation, a further interference occurs for an already created interference evade vector.
5452	IMPROPER G–CODE (5AXIS MODE) (M series)	A G code that cannot be specified is found. (5–axis mode) This alarm is issued when: 1) Three–dimensional cutter compensation (side–face offset and leading–edge offset) is applied during cutter compensation, or cutter compensation is applied during three–dimensional cutter compensation (side–face offset and leading–edge offset). 2) A leading–edge offset of three–dimensional cutter compensation is applied during side–face offsetting of three–dimensional cutter compensation, or a side–face offset of three–dimensional cutter compensation is applied during leading–edge offsetting of three–dimensional cutter compensation. 3) Tool axis direction tool length compensation is applied during tool length compensation, or tool length compensation is applied during tool axis direction tool length compensation. 4) Tool center point control is provided during tool length compensation, or tool length compensation is applied during tool center point control. 5) Tool center point control is provided during tool axis direction tool length compensation, or tool axis direction tool length compensation is applied during tool center point control. If this alarm is issued, cancel the relevant mode, then specify a different mode.
5453	NOTE: G68 IS CANCELED (HPCC) (M series)	When bit 2 of parameter No. 5400 is set to 1, and a reset does not cancel G68, this alarm is issued at the time of program restart. To release this alarm, press <RESET> and <CAN>. Once this operation is performed, the alarm will not be issued at the next restart.
5455	ILLEGAL ACC. PARAMETER (M series)	A permissible acceleration parameter for optimum torque acceleration/deceleration is incorrect. The cause is one of the following: 1) The ratio of the deceleration rate to the acceleration rate is below the limit. 2) The time required for deceleration to a speed of 0 exceeds the maximum value.

NOTE
HPCC designates High Precision Contour Control.
AICC designates AI Contour Control.

(2) Background edit alarm

Number	Message	Contents
???	BP/S alarm	BP/S alarm occurs in the same number as the P/S alarm that occurs in ordinary program edit. (070, 071, 072, 073, 074 085,086,087 etc.)
140	BP/S alarm	It was attempted to select or delete in the background a program being selected in the foreground. **(Note)** Use background editing correctly.

NOTE
Alarm in background edit is displayed in the key input line of the background edit screen instead of the ordinary alarm screen and is resettable by any of the MDI key operation.

(3) Absolute pulse coder (APC) alarm

Number	Message	Contents
300	APC alarm: nth–axis origin return	Manual reference position return is required for the nth–axis (n=1 – 8).
301	APC alarm: nth–axis communication	nth–axis (n=1 – 8) APC communication error. Failure in data transmission Possible causes include a faulty APC, cable, or servo interface module.
302	APC alarm: nth–axis over time	nth–axis (n=1 – 8) APC overtime error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
303	APC alarm: nth–axis framing	nth–axis (n=1 – 8) APC framing error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
304	APC alarm: nth–axis parity	nth–axis (n=1 – 8) APC parity error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
305	APC alarm: nth–axis pulse error	nth–axis (n=1 – 8) APC pulse error alarm. APC alarm.APC or cable may be faulty.
306	APC alarm: nth–axis battery voltage 0	nth–axis (n=1 – 8) APC battery voltage has decreased to a low level so that the data cannot be held. APC alarm. Battery or cable may be faulty.
307	APC alarm: nth–axis battery low 1	nth–axis (n=1 – 8) axis APC battery voltage reaches a level where the battery must be renewed. APC alarm. Replace the battery.
308	APC alarm: nth–axis battery low 2	nth–axis (n=1 – 8) APC battery voltage has reached a level where the battery must be renewed (including when power is OFF). APC alarm .Replace battery.
309	APC ALARM: n AXIS ZRN IMPOSSIBL	Return to the origin has been attempted without first rotating the motor one or more times. Before returning to the origin, rotate the motor one or more times then turn off the power.

(4) Inductsyn alarms

Number	Message	Description
330	INDUCTOSYN:DATA ALARM	The absolute–position data (offset data) from Inductosyn cannot be detected.
331	INDUCTOSYN:ILLEGAL PRM	Parameter No. 1874, 1875, or 1876 is set to 0.

(5) Serial pulse coder (SPC) alarms

No.	Message	Description
360	n AXIS : ABNORMAL CHECKSUM (INT)	A checksum error occurred in the built–in pulse coder.
361	n AXIS : ABNORMAL PHASE DATA (INT)	A phase data error occurred in the built–in pulse coder.
362	n AXIS : ABNORMAL REV.DATA (INT)	A rotation speed count error occurred in the built–in pulse coder.
363	n AXIS : ABNORMAL CLOCK (INT)	A clock error occurred in the built–in pulse coder.
364	n AXIS : SOFT PHASE ALARM (INT)	The digital servo software detected invalid data in the built–in pulse coder.
365	n AXIS : BROKEN LED (INT)	An LED error occurred in the built–in pulse coder.
366	n AXIS : PULSE MISS (INT)	A pulse error occurred in the built–in pulse coder.
367	n AXIS : COUNT MISS (INT)	A count error occurred in the built–in pulse coder.
368	n AXIS : SERIAL DATA ERROR (INT)	Communication data from the built–in pulse coder cannot be received.
369	n AXIS : DATA TRANS. ERROR (INT)	A CRC or stop bit error occurred in the communication data being received from the built–in pulse coder.
380	n AXIS : BROKEN LED (EXT)	The LED of separate detector is erroneous.
381	n AXIS : ABNORMAL PHASE (EXT LIN)	A phase data error occurred in the separate linear scale.
382	n AXIS : COUNT MISS (EXT)	A pulse error occurred in the separate detector.
383	n AXIS : PULSE MISS (EXT)	A count error occurred in the separate detector.
384	n AXIS : SOFT PHASE ALARM (EXT)	The digital servo software detected invalid data in the separate detector.
385	n AXIS : SERIAL DATA ERROR (EXT)	Communication data from the separate detector cannot be received.
386	n AXIS : DATA TRANS. ERROR (EXT)	A CRC or stop bit error occurred in the communication data being received from the separate detector.
387	n AXIS : ABNORMAL ENCODER (EXT)	An error occurs in the separate detector. For details, contact the manufacturer of the scale.

- **The details of serial pulse coder alarm**

The details of serial pulse coder alarm are displayed in the diagnosis display (No. 202 and No.203) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
202		CSA	BLA	PHA	PCA	BZA	CKA	SPH

#6 (CSA) : The serial pulse coder is defective. Replace it.

#5 (BLA) : The battery voltage is low. Replace the batteries.

#4 (PHA) : The serial pulse coder or feedback cable is defective. Replace the serial pulse coder or cable.

#3 (PCA) : The serial pulse coder is defective. Replace it.

#2 (BZA) : The pulse coder was supplied with power for the first time.
Make sure that the batteries are connected.
Turn the power off, then turn it on again and perform a reference position return.

#1 (CKA) : The serial pulse coder is defective. Replace it.

#0 (SPH) : The serial pulse coder or feedback cable is defective. Replace the serial pulse coder or cable.

	#7	#6	#5	#4	#3	#2	#1	#0
203	DTE	CRC	STB	PRM				

#7 **(DTE)** : The serial pulse coder encountered a communication error.
The pulse coder, feedbak cable, or feedback receiver circuit is defective.
Replace the pulse coder, feedback cable, or NC–axis board

#6 **(CRC)** : The serial pulse coder encountered a communication error.
The pulse coder, feedback cable, or feedback receiver circuit is defective.
Replace the pulse coder, feedback cable, or NC–axis board.

#5 **(STB)** : The serial pulse coder encountered a communication error.
The pulse coder, feedback cable, or feedback receiver circuit is defective.
Replace the pulse coder, feedback cable, or NC–axis board.

#4 **(PRM)** : An invalid parameter was found. Alarm No. 417 (invalid servo parameter) is also issued.

(6) Servo alarms(1/2)

Number	Message	Contents
401	SERVO ALARM: n–TH AXIS VRDY OFF	The n–th axis (axis 1–8) servo amplifier READY signal (DRDY) went off. Refer to procedure of trouble shooting.
402	SERVO ALARM: SV CARD NOT EXIST	The axis control card is not provided.
403	SERVO ALARM: CARD/SOFT MISMATCH	The combination of the axis control card and servo software is illegal. The possible causes are as follows: · A correct axis control card is not provided. · Correct servo software is not installed on flash memory.
404	SERVO ALARM: n–TH AXIS VRDY ON	Even though the n–th axis (axis 1–8) READY signal (MCON) went off, the servo amplifier READY signal (DRDY) is still on. Or, when the power was turned on, DRDY went on even though MCON was off. Check that the servo interface module and servo amp are connected.
405	SERVO ALARM: (ZERO POINT RETURN FAULT)	Position control system fault. Due to an NC or servo system fault in the reference position return, there is the possibility that reference position return could not be executed correctly. Try again from the manual reference position return.
407	SERVO ALARM: EXCESS ERROR	The following error occurred during simple synchronous control: The difference in machine coordinates between the synchronized axes exceeds the value set in parameter No. 8314.
409	SERVO ALARM: n AXIS TORQUE ALM	Abnormal servo motor load has been detected. Alternatively, abnormal spindle motor load has been detected in Cs mode.
410	SERVO ALARM: n–TH AXIS – EXCESS ERROR	One of the following errors occurred: 1) The positional deviation value when the n–th axis stops exceeds the value set in parameter No. 1829. 2) In simple synchronous control, the compensation amount for synchronization exceeds the value set in parameter No. 8325. This alarm is issued only for the slave axis.
411	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1–8) moves is larger than the set value. Refer to procedure of trouble shooting.
413	SERVO ALARM: n–th AXIS – LSI OVERFLOW	The contents of the error register for the n–th axis (axis 1–8) exceeded $\pm 2^{31}$ power. This error usually occurs as the result of an improperly set parameters.

Number	Message	Contents
415	SERVO ALARM: n–TH AXIS – EXCESS SHIFT	A speed higher than 524288000 units/s was attempted to be set in the n–th axis (axis 1–8). This error occurs as the result of improperly set CMR.
417	SERVO ALARM: n–TH AXIS – PARAMETER INCORRECT	This alarm occurs when the n–th axis (axis 1–8) is in one of the conditions listed below. (Digital servo system alarm) 1) The value set in Parameter No. 2020 (motor form) is out of the specified limit. 2) A proper value (111 or –111) is not set in parameter No.2022 (motor revolution direction). 3) Illegal data (a value below 0, etc.) was set in parameter No. 2023 (number of speed feedback pulses per motor revolution). 4) Illegal data (a value below 0, etc.) was set in parameter No. 2024 (number of position feedback pulses per motor revolution). 5) Parameters No. 2084 and No. 2085 (flexible field gear rate) have not been set. 6) A value outside the limit of {1 to the number of control axes} or a non–continuous value (Parameter 1023 (servo axis number) contains a value out of the range from 1 to the number of axes, or an isolated value (for example, 4 not prceded by 3).was set in parameter No. 1023 (servo axisnumber). 7) A torque control parameter is set incorrectly in PMC axis control. (The torque constant parameter is set to 0.)
420	SERVO ALARM: n AXIS SYNC TORQUE (M series)	During simple synchronous control, the difference between the torque commands for the master and slave axes exceeded the value set in parameter No. 2031.
421	SERVO ALARM: n AXIS EXCESS ER (D)	The difference between the errors in the semi–closed loop and closed loop has become excessive during dual position feedback. Check the values of the dual position conversion coefficients in parameters No. 2078 and 2079.
422	SERVO ALARM: n AXIS	In torque control of PMC axis control, a specified allowable speed has been exceeded.
423	SERVO ALARM: n AXIS	In torque control of PMC axis control, the parameter–set allowable cumulative travel distance has been exceeded.
430	n AXIS : SV. MOTOR OVERHEAT	A servo motor overheat occurred.
431	n AXIS : CNV. OVERLOAD	1) PSM: Overheat occurred. 2) β series SVU: Overheat occurred.
432	n AXIS : CNV. LOW VOLT CONTROL	1) PSM: Control power voltage has dropped. 2) PSMR: The control power supply voltage has dropped. 3) β series SVU: The control power supply voltage has dropped.
433	n AXIS : CNV. LOW VOLT DC LINK	1) PSM: The DC link voltage has dropped. 2) PSMR: The DC link voltage has dropped. 3) α series SVU: The DC link voltage has dropped. 4) β series SVU: The DC link voltage has dropped.
434	n AXIS : INV. LOW VOLT CONTROL	SVM: The control power supply voltage has dropped.
435	n AXIS : INV. LOW VOLT DC LINK	SVM: The DC link voltage has dropped.
436	n AXIS : SOFTTHERMAL (OVC)	The digital servo software detected the soft thermal state (OVC).
437	n AXIS : CNV. OVERCURRENT POWER	PSM: Overcurrent flowed into the input circuit.

Number	Message	Contents
438	n AXIS : INV. ABNORMAL CURRENT	1) SVM: The motor current is too high. 2) α series SVU: The motor current is too high. 3) β series SVU: The motor current is too high.
439	n AXIS : CNV. OVERVOLT POWER	1) PSM: The DC link voltage is too high. 2) PSMR: The DC link voltage is too high. 3) α series SVU: The C link voltage is too high. 4) β series SVU: The link voltage is too high.
440	n AXIS : CNV. EX DECELERATION POW.	1) PSMR: The regenerative discharge amount is too large. 2) α series SVU: The regenerative discharge amount is too large. Alternatively, the regenerative discharge circuit is abnormal.
441	n AXIS : ABNORMAL CURRENT OFFSET	The digital servo software detected an abnormality in the motor current detection circuit.
442	n AXIS : CNV. CHARGE FAILURE	1) PSM: The spare discharge circuit of the DC link is abnormal. 2) PSMR: The spare discharge circuit of the DC link is abnormal.
443	n AXIS : CNV. COOLING FAN FAILURE	1) PSM: The internal stirring fan failed. 2) PSMR: The internal stirring fan failed. 3) β series SVU: The internal stirring fan failed.
444	n AXIS : INV. COOLING FAN FAILURE	SVM: The internal stirring fan failed.
445	n AXIS : SOFT DISCONNECT ALARM	The digital servo software detected a broken wire in the pulse coder.
446	n AXIS : HARD DISCONNECT ALARM	A broken wire in the built–in pulse coder was detected by hardware.
447	n AXIS : HARD DISCONNECT (EXT)	A broken wire in the separate detector was detected by hardware.
448	n AXIS : UNMATCHED FEEDBACK ALARM	The sign of feedback data from the built–in pulse coder differs from that of feedback data from the separate detector.
449	n AXIS : INV. IPM ALARM	1) SVM: IPM (intelligent power module) detected an alarm. 2) α series SVU: IPM (intelligent power module) detected an alarm.
453	n AXIS : SPC SOFT DISCONNECT ALARM	Software disconnection alarm of the α pulse coder. Turn off the power to the CNC, then remove and insert the pulse coder cable. If this alarm is issued again, replace the pulse coder.
456	ILLEGAL CURRENT LOOP	The current control cycle settings (parameter No. 2004, bit 0 of parameter No. 2003, and bit 0 of parameter No. 2013) are incorrect. Possible problems are as follows. – For the two axes whose servo axis numbers (settings of parameter No. 1023) are an odd number followed by an even number (a pair of axes 1 and 2 or axes 5 and 6, for example), a different current control cycle is set for each of the axes. – The requirements for slaves needed for the set current control cycle, including the number, type, and connection method of them, are not satisfied.
457	ILLEGAL HI HRV (250US)	Use of high–speed HRV is specified although the current control cycle is 200 µs.
458	CURRENT LOOP ERROR	The current control cycle setting does not match the actual current control cycle.
459	HI HRV SETTING ERROR	Of two axes having adjacent servo axis numbers (parameter No. 1023), odd number and even number, high–speed HRV control can be performed for one axis and not for the other.

Number	Message	Contents
460	n AXIS : FSSB DISCONNECT	FSSB communication was disconnected suddenly. The possible causes are as follows: 1) The FSSB communication cable was disconnected or broken. 2) The power to the amplifier was turned off suddenly. 3) A low–voltage alarm was issued by the amplifier.
461	n AXIS : ILLEGAL AMP INTERFACE	The axes of the 2–axis amplifier were assigned to the fast type interface.
462	n AXIS : SEND CNC DATA FAILED	Because of an FSSB communication error, a slave could not receive correct data.
463	n AXIS : SEND SLAVE DATA FAILED	Because of an FSSB communication error, the servo system could not receive correct data.
464	n AXIS : WRITE ID DATA FAILED	An attempt was made to write maintenance information on the amplifier maintenance screen, but it failed.
465	n AXIS : READ ID DATA FAILED	At power–up, amplifier initial ID information could not be read.
466	n AXIS : MOTOR/AMP COMBINATION	The maximum current rating for the amplifier does not match that for the motor.
467	n AXIS : ILLEGAL SETTING OF AXIS	The servo function for the following has not been enabled when an axis occupying a single DSP (corresponding to two ordinary axes) is specified on the axis setting screen. 1. Learning control (bit 5 of parameter No. 2008 = 1) 2. High–speed current loop (bit 0 of parameter No. 2004 = 1) 3. High–speed interface axis (bit 4 of parameter No. 2005 = 1)
468	HI HRV SETTING ERROR(AMP)	Use of high–speed HRV is specified for a controlled axis of an amplifier which does not support high–speed HRV.

- **Details of servo alarm**

The details of servo alarm are displayed in the diagnosis display (No. 200 and No.204) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
200	OVL	LV	OVC	HCA	HVA	DCA	FBA	OFA

#7 (OVL) : An overload alarm is being generated.
(The details are indicated in diagnostic data No.201).

#6 (LV) : A low voltage alarm is being generated in servo amp.
Check LED.

#5 (OVC) : A overcurrent alarm is being generated inside of digital servo.

#4 (HCA) : An abnormal current alarm is being generated in servo amp.
Check LED.

#3 (HVA) : An overvoltage alarm is being generated in servo amp.
Check LED.

#2 (DCA) : A regenerative discharge circuit alarm is being generated in servo amp.
Check LED.

#1 (FBA) : A disconnection alarm is being generated.
(The details are indicated in diagnostic data No. 201)

#0 (OFA) : An overflow alarm is being generated inside of digital servo.

	#7	#6	#5	#4	#3	#2	#1	#0
201	ALD			EXP				

When OVL equal 1 in diagnostic data No.200 (servo alarm No. 400 is being generated):

#7 **(ALD)** 0 : Motor overheating

1 : Amplifier overheating

When FBAL equal 1 in diagnostic data No.200 (servo alarm No. 416 is being generated):

ALD	EXP	Alarm details
1	0	Built–in pulse coder disconnection (hardware)
1	1	Separately installed pulse coder disconnection (hardware)
0	0	Pulse coder is not connected due to software.

	#7	#6	#5	#4	#3	#2	#1	#0
204		OFS	MCC	LDA	PMS			

#6 (OFS) : A current conversion error has occured in the digital servo.

#5 (MCC) : A magnetic contactor contact in the servo amplifier has welded.

#4 (LDA) : The LED indicates that serial pulse coder C is defective

#3 (PMS) : A feedback pulse error has occured because the feedback cable is defective.

(7) Over travel alarms

Number	Message	Contents
500	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke check I. (Parameter No.1320 or 1326 **NOTE)**
501	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke check I. (Parameter No.1321 or 1327 **NOTE)**
502	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke check II. (Parameter No.1322)
503	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke check II. (Parameter No.1323)
504	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke check III. (Parameter No.1324)
505	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke check III. (Parameter No.1325)
506	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side hardware OT.
507	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side hardware OT.
508	INTERFERENCE: +n (T series (two–path control))	A tool moving in the positive direction along the n axis has fouled another tool post.
509	INTERFERENCE: –n (T series (two–path control))	A tool moving in the negative direction along the n axis has fouled another tool post.
510	OVER TRAVEL: +n	Alarm for stroke check prior to movement. The end point specified in a block falls within the forbidden area defined with the stroke check in the positive direction along the N axis. Correct the program.
511	OVER TRAVEL: –n	Alarm for stroke check prior to movement. The end point specified in a block falls within the forbidden area defined with the stroke check in the negative direction along the N axis. Correct the program.

Number	Message	Contents
514	INTERFERENCE : +n	The rotation area interference check function found interference on the plus side of the n axis.
515	INTERFERENCE : –n	The rotation area interference check function found interference on the minus side of the n axis.

NOTE
Parameters 1326 and 1327 are effective when EXLM(stroke check switch signal) is on.

(8) Servo alarms

Number	Message	Contents
600	n AXIS: INV. DC LINK OVER CURRENT	DC link current is too large.
601	n AXIS: INV. RADIATOR FAN FAILURE	The external dissipator stirring fan failed.
602	n AXIS: INV. OVERHEAT	The servo amplifier was overheated.
603	n AXIS: INV. IPM ALARM(OH)	The IPM (intelligent power module) detected an overheat alarm.
604	n AXIS: AMP. COMMUNICATION ERROR	Communication between the SVM and the PSM failed.
605	n AXIS: CNV. EX. DISCHARGE POW.	PSMR: Regenerative power is too large.
606	n AXIS: CNV. RADIATOR FAN FAILURE	PSM: The external dissipator stirring fan failed. PSMR: The external dissipator stirring fan failed.
607	n AXIS: CNV. SINGLE PHASE FAILURE	PSM: Input voltage is in the open–phase condition. PSMR: Input voltage is in the open–phase condition.

(9) Overheat alarms

Number	Message	Contents
700	OVERHEAT: CONTROL UNIT	Control unit overheat Check that the fan motor operates normally, and clean the air filter.
701	OVERHEAT: FAN MOTOR	The fan motor on the top of the cabinet for the contorl unit is overheated. Check the operation of the fan motor and replace the motor if necessary.
704	OVERHEAT: SPINDLE	Spindle overheat in the spindle fluctuation detection (1) If the cutting load is heavy, relieve the cutting condition. (2) Check whether the cutting tool is share. (3) Another possible cause is a faulty spindle amp.

(10)Rigid tapping alarms

Number	Message	Contents
740	RIGID TAP ALARM: EXCESS ERROR	The positional deviation of the stopped spindle has exceeded the set value during rigid tapping.
741	RIGID TAP ALARM: EXCESS ERROR	The positional deviation of the moving spindle has exceeded the set value during rigid tapping.
742	RIGID TAP ALARM: LSI OVERFLOW	An LSI overflow has occurred for the spindle during rigid tapping.

(11) Serial spindle alarms

Number	Message	Contents
749	S–SPINDLE LSI ERROR	It is serial communication error while system is executing after power supply on. Following reasons can be considered. 1) Optical cable connection is fault or cable is not connected or cable is cut. 2) MAIN CPU board or option 2 board is fault. 3) Spindle amp. printed board is fault. 4) The spindle amplifier is under an abnormal condition. (The SPM indication is A, A1, A2, or the like, depending on the type of the abnormality.) If this alarm occurs when CNC power supply is turned on or when this alarm can not be cleared even if CNC is reset, turn off the power supply also turn off the power supply in spindle side. If the spindle amplifier is under an abnormal condition, check the SPM indication (A, A1, A2, or the like). Then, refer to the FANUC SERVO MOTOR αi series MAINTENANCE MANUAL (B–65285EN) or FANUC SERVO MOTOR α series MAINTENANCE MANUAL (B–65165E) to solve the problem.
750	SPINDLE SERIAL LINK START FAULT	This alarm is generated when the spindle control unit is not ready for starting correctly when the power is turned on in the system with the serial spindle. The four reasons can be considered as follows: 1) An improperly connected optic cable, or the spindle control unit's power is OFF. 2) When the NC power was turned on under alarm conditions other than SU–01 or AL–24 which are shown on the LED display of the spindle control unit. In this case, turn the spindle amplifier power off once and perform startup again. 3) Other reasons (improper combination of hardware) This alarm does not occur after the system including the spindle control unit is activated. 4) The second spindle (when SP2, bit 4 of parameter No. 3701, is 1) is in one of the above conditions 1) to 3). See diagnostic display No. 409 for details.
752	FIRST SPINDLE MODE CHANGE FAULT	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contouring, spindle positioning, rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.
754	SPINDLE–1 ABNORMAL TORQUE ALM	Abnormal first spindle motor load has been detected.
762	SECOND SPINDLE MODE CHANGE FAULT	Refer to alarm No. 752.(For 2nd axis)
764	SPINDLE–2 ABNORMAL TORQUE ALM	Same as alarm No. 754 (for the second spindle)
772	SPINDLE–3 MODE CHANGE ERROR	Same as alarm No. 752 (for the third spindle)
774	SPINDLE–3 ABNORMAL TORQUE ALM	Same as alarm No. 754 (for the third spindle)
782	SPINDLE–4 MODE CHANGE ERROR	Same as alarm number 752 (for the fourth spindle)
784	SPINDLE–4 ABNORMAL TORQUE ALM	Same as alarm number 754 (for the fourth spindle)

- **The details of spindle alarm No.750**

- **1st and 2nd spindles**

	#7	#6	#5	#4	#3	#2	#1	#0
409					SPE	S2E	S1E	SHE

#3 (SPE) 0 : In the spindle serial control, the serial spindle parameters fulfill the spindle unit startup conditions.

1 : In the spindle serial control, the serial spindle parameters do not fulfill the spindle unit startup conditions.

#2 (S2E) 0 : The second spindle is normal during the spindle serial control startup.

1 : The second spindle was detected to have a fault during the spindle serial control startup.

#1 (S1E) 0 : The first spindle is normal during the spindle serial control startup.

1 : The first spindle was detected to have a fault during the spindle axis serial control startup.

#0 (SHE) 0 : The serial communications module in the CNC is normal.

1 : The serial communications module in the CNC was detected to have a fault.

- **3rd and 4th spindles**

The details of spindle alarm No. 750 are displayed in the diagnosis display (No. 409) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
409					SPE	S4E	S3E	SHE

#3 (SPE) 0 : In the spindle serial control, the serial spindle parameters fulfill the spindle unit startup conditions.

1 : In the spindle serial control, the serial spindle parameters do not fulfill the spindle unit startup conditions.

#2 (S2E) 0 : The fourth spindle is normal during the spindle serial control startup.

1 : The fourth spindle was detected to have a fault during the spindle serial control startup.

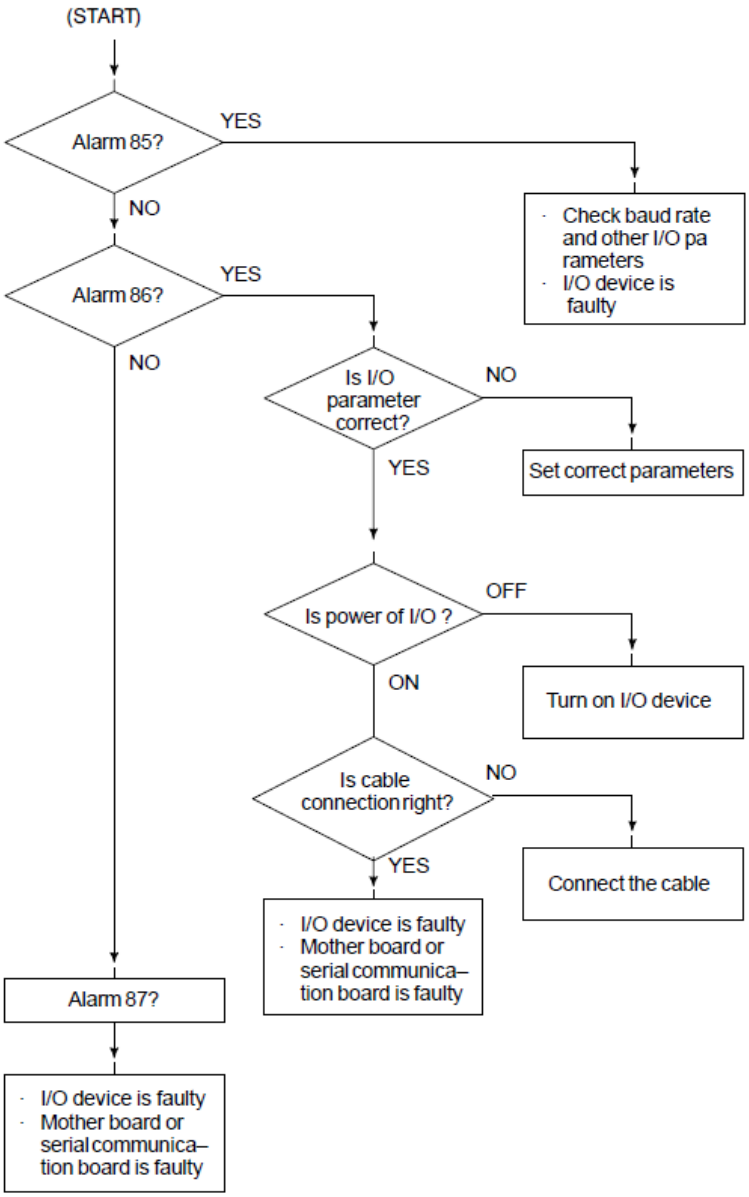
#1 (S1E) 0 : The third spindle is normal during the spindle serial control startup.

1 : The third spindle was detected to have a fault during the spindle axis serial control startup.

#0 (SHE) 0 : The serial communications module in the CNC is normal.

1 : The serial communications module in the CNC was detected to have a fault.

ALARM 85 TO 87 (READER/PUNCHER INTERFACE ALARM)



Causes

(a) Parameters on reader/puncher interface are not correct.
Check the following setting data and parameters.
(b) External I/O device or host computer is faulty.
(c) Mother board or serial communication board is faulty.
(d) Cable between NC and I/O device is faulty.

Countermeasures

(a) Parameters on reader/puncher interface are not correct.
Check the following setting data and parameters:
<Setting>
PUNCH CODE=0 OR 1 (0: EIA,1:ISO)
Select ISO or EIA according to the type of I/O device.
If punch code does not match, alarm 86 will generate.

<Parameter>

Function \ Value of parameter 0020		0	1	2	3	
Feed		0101#7	0111#7	0121#7	0131#7	
Data input code		0101#3	0111#3	0121#3	0131#3	
Stop bit		0101#0	0111#0	0121#0	0131#0	
Type of I/O device		102	112	122	132	
Baud rate		103	113	123	133	
Communication method	0135#3	–	–	–	0	1
		RS–232C				RS–422
Connector		MOTHER BOARD			SERIAL COMMUNICATION BOARD	
		JD36A		JD36B	JD28A	JD6A

NOTE
1 Numbers in the table indicate parameters and bit numbers.
Example) 101#7: bit7 of parameter 101.
2 For data communications by RS–422, refer to parameters 134 and 135.

	#7	#6	#5	#4	#3	#2	#1	#0
0101	NFD				ASI			SB2
0111								
0121								
0131								

#7(NFD) 0 : Feed is output before and after data in data output (FANUC PPR)
1 : Feed is not output (standard).

#3(ASI) 0 : Data input code is EIA or ISO (automatic recognition)
1 : Data input code is ASCII.

#0(SB2) 0 : No. of stop bits is 1.
1 : No. of stop bits is 2.

0102 0112 0122 0132	Type of I/O device

Value	TYPE OF I/O DEVICE
0	RS–232–C (if the following units are not used)
1	FANUC CASSETTE B1/B2 (bubble cassette)
2	FANUC CASSETTE F1 (Old type FLOPPY CASSETTE ADAP-TOR)
3	FANUC PROGRAM FILE Mate, FANUC FA CARD ADAPTOR FANUC FLOPPY CASSETTE ADAPTOR, FANUC SYSTEM P–MODEL H, FANUC Handy File
4	Not used
5	Portable tape reader
6	FANUC PPR, FANUC SYSTEM P–MODEL G, FANUC SYSTEM P–MODEL H

0103 0113 0123 0133	Baud rete

Value	Baud rate
7	600
8	1200
9	2400
10	4800
11	9600
12	19200

When bit#3 of parameter no. 0135=1 (RS–422 interface), the following setting is also available.

Value	Baud rate
13	38400
14	76800
15	86400

Check the following parameters also, when parameter no.0020 is 3.

	#7	#6	#5	#4	#3	#2	#1	#0
0134			CLK	NCD		SYN	PRY	

#5(CLK) 0 : Internal clock is used for baud rate clock of RS–422 interface.
1 : External clock is used for baud rate clock of RS–422 interface.

#4(NCD) 0 : CD (signal quality detection) of RS–232C interface is checked.
1 : CD (signal quality detection) of RS–232C interface is not checked.

#2(SYN) 0 : In protocol B, NC reset/alarm is not informed to the host.
1 : In protocol B, NC reset/alarm is informed to the host by SYN and NAK code.

#1(PRY) 0 : No parity bit
1 : With parity bit

	#7	#6	#5	#4	#3	#2	#1	#0
0135	RMS				R42	PRA	ETX	ASC

#7(RMS) In protocol A, status of remote / tape operation of SAT command is

0 : Always transmitted by 0.

1 : Transmitted by the contents of remote / tape switching request issued by SET command from the CNC.

#3(R42) 0 : Interface is of RS–232C.

1 : Interface is of RS–422.

#2(PRA) 0 : Communication protocol is protocol B

1 : Communication protocol is protocol A

#1(ETX) 0 : End code of protocol A or extended protocol A is CR of ASCII/ISO.

1 : End code of protocol A or extended protocol A is ETX of ASCII/ISO.

#0(ASC) 0 : All the communication codes except for NC data is ISO code.

1 : All the communication codes except for NC data is ASCII code.

(b) External I/O device or Host computer is in trouble

(i) Check whether the setting on communication of external I/O device or host computer is the same as that of the CNC. (baud rate, stop bits,etc.) If they are not the same, change the setting.

(ii) When spare I/O device presents, check whether it is possible to realize communication using the spare I/O device.

(c) Spindle module or communication control module is faulty

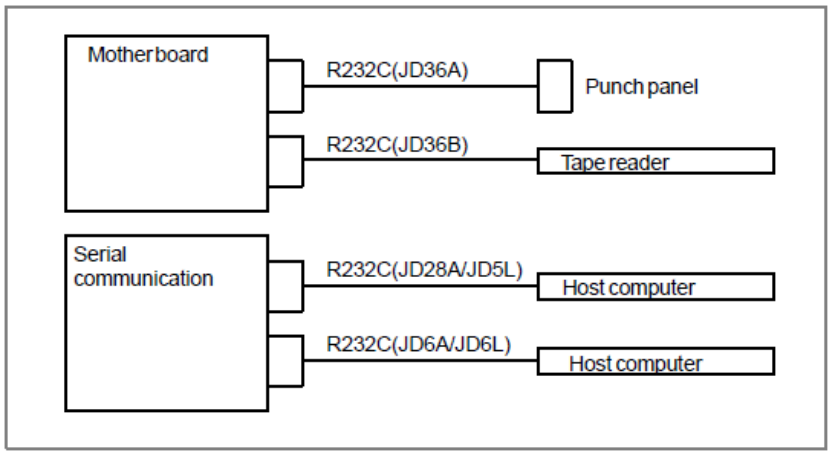
(i) When parameter no.0020 is 0 or 1 or 2 (JD36A,JD36B of Main CPU board) Replace the module since spindle module may be faulty.

(ii) When parameter no. 0020 is 3 (JD28A,JD6A of option 1 board) Because communication control module (5) may be faulty, replace the module.

(d) Cable between NC and I/O device is faulty.

Check the cable for disconnection or wrong connection.

<Connection>



ALARM 90 (REFERENCE POSITION RETURN IS ABNORMAL)

Contents

Reference position return was executed when the following condition is not satisfied:
The CNC received one rotation signal at least one time when the axis is moving to the reference position at a speed higher than a speed equivalent to 128 pulses of position error amount(DGN300).

Countermeasures

(START)

Check whether position gain is greater than 128 pulses (DGN 300) before or during reference position return.

Position error amount : DGN 300

128 or more — YES → (1) Next page

NO

Raise the speed

Check feed rate command:
PRM 1420 F : Rapid traverse rate (mm/min)
PRM 1424 Manual rapid traverse rate (mm/min)
PRM 1825 G : Servo loop gain ($0.01sec^{-1}$)

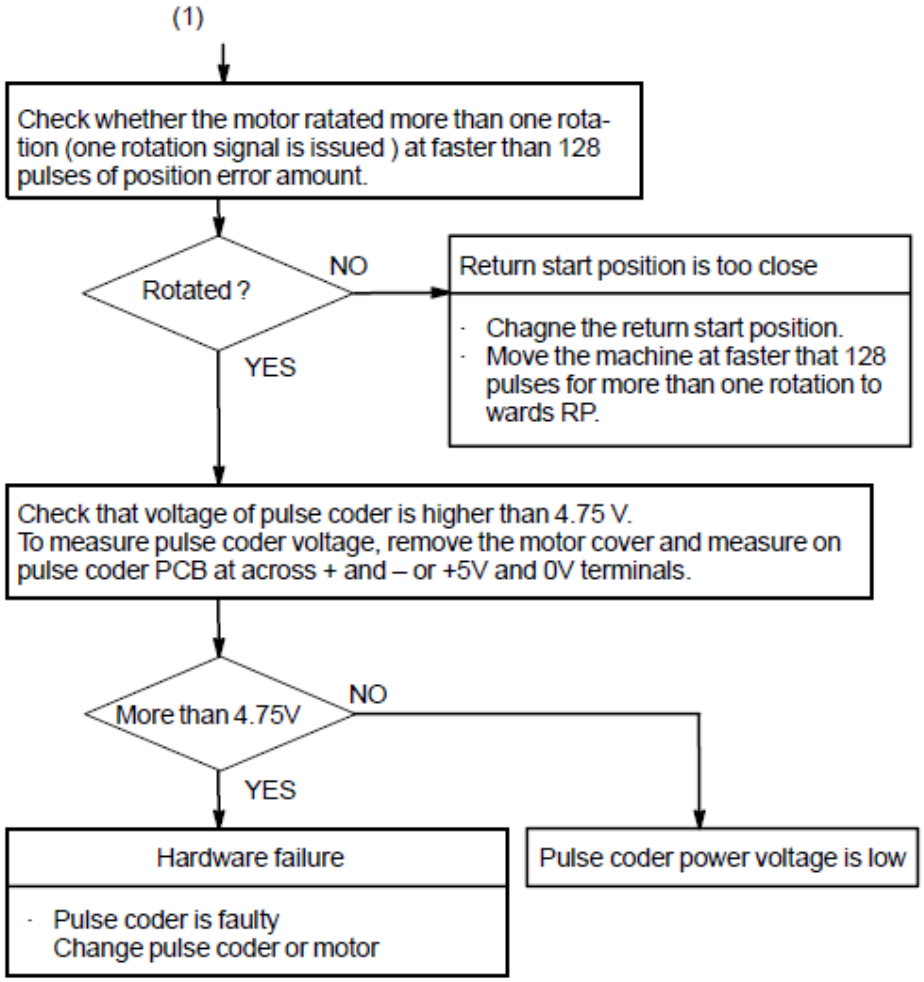
$$\text{Position error} = \frac{F \times 5000/3}{G \times \text{detection unit}[\mu m/\text{PLUSE}]}$$

Detection unit : Move amount to a command pulse (usually 1μm)
In metric machine, if the no. of digits below decimal point is 4 on the position display screen, detection unit is 0.1 μm.

Check rapid traverse override signals :
ROV1 DGN 014.0
1014.0 (For two–path control)
ROV2 DGN 014.1
1014.1 (For two–path control)
PRM 1421 Fo rate

ROV1	ROV2	Override
0	0	100%
0	1	50%
1	0	25%
1	1	Fo rate

Check reference position return deceleration signal
DEC1 to DEC8 DGN 009.0 to 009.7
When reference position return is started from deceleration signal 0, feed rate becomes FL rate.
PRM 1425 FL rate



CAUTION

After the pulse coder or motor is exchanged, reference position or machine's standard point may be different from former one. Please set it correctly.

A speed more than 128 pulses is required because if speed is lower that this, one–rotation signal does not function stably, causing improper position detection.
If bit 0 of parameter No. 2000 is set to 1, a speed corresponding to a positional deviation of 1280 pulses or more is required.
Parameter No. 1836 can be set to 128 or less, as the minimum positional deviation with which reference position return is possible. (If the parameter is set to 0, 128 is assumed as the minimum positional deviation. If bit 0 of parameter No. 2000 is set to 1, a value equal to ten times the set value is used for checking.)

11.14 ALARM 300 (REQUEST FOR REFERENCE POSITION RETURN)

Absolute position data in the serial pulse coder was lost.
(This alarm will be generated when serial pulse coder is exchanged or position feedback signal cable of the serial pulse coder is disconnected).

Remedies

Machine position must be memorized using the following method:

- **When reference position return function is present**

  (1) Execute manual reference position return only for an axis for which this alarm was generated.When manual reference position return cannot be executed because of an another alarm, set parameter 1815#5 to 0 and release the alarm and perform manual operation.

  (2) Press RESET key at the end of reference position return to release the alarm.

- **When reference position return function is not present**

  Execute dogless reference position setting to memorize the reference position.

- **When serial pulse coder is changed**

  Since the reference position is different from the former one, change the grid shift value (PRM 1850) to correct the position.

Related parameters

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APC x	APZx				

#5(APCx) 0 : Position detector is incremental pulse coder.
1 : Position detector is absolute pulse coder.

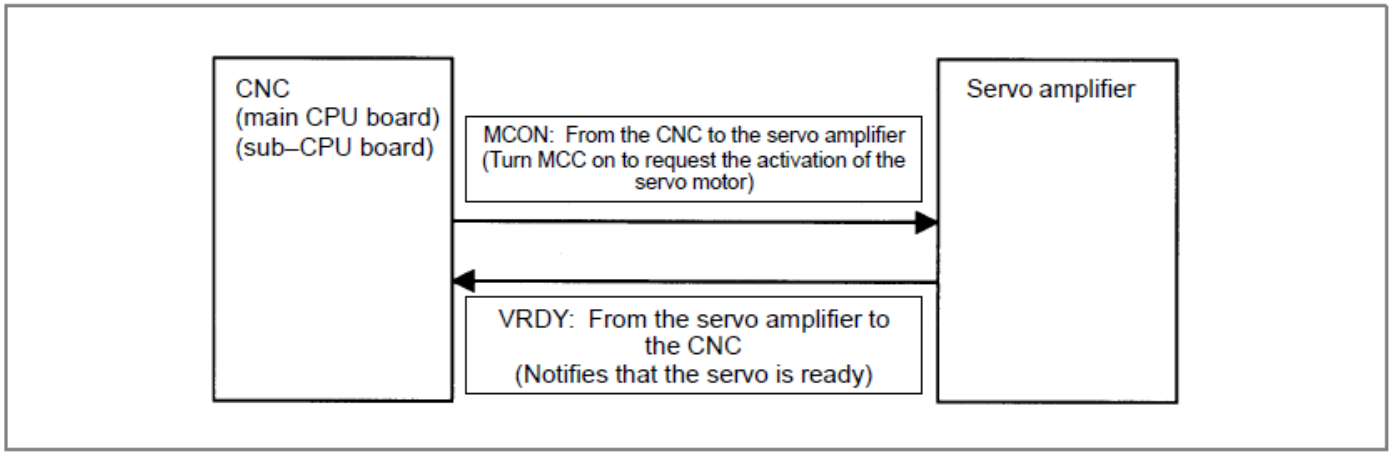
#4(APZx) Reference position of absolute pulse coder is :
0 : not established
1 : established

ALARM 401
(V READY OFF)

Causes and actions

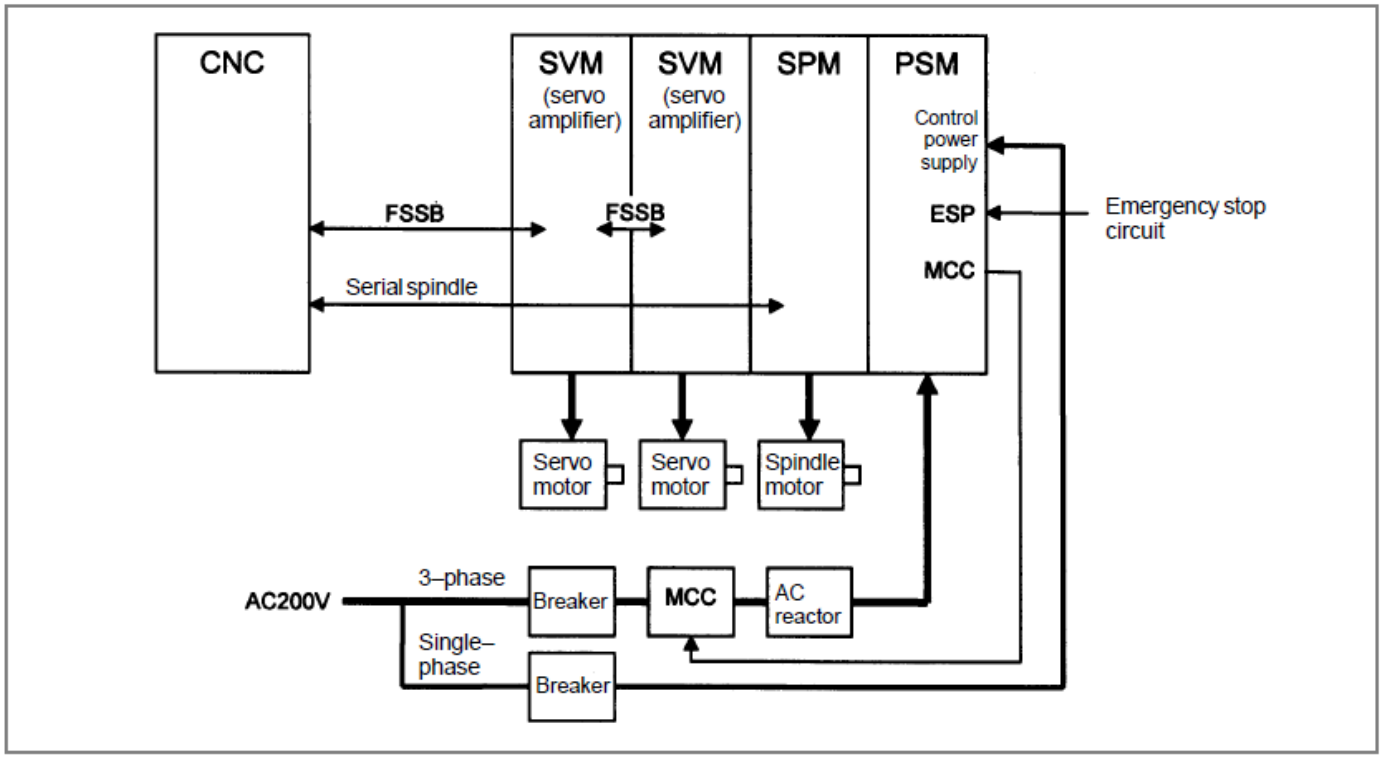
This alarm is issued if the servo ready signal (VRDY) of a servo amplifier does not turn on or if the signal turns off during operation.
There are cases in which this alarm is issued because another servo alarm is issued. If this occurs, first take the action for the first alarm.
Check the power magnetic circuit around the amplifier. The servo amplifier or the axis control cards on the CNC may be defective.

- **VRDY**



The exchange of this information is performed via the FSSB (optical cable).

- **Example of connection around the amplifier (Typical example)**



Check items

- Is the PSM control power supply on?
- Has an emergency stop been canceled?
- Is a terminating connector connected to the JX1B connector of the terminating amplifier?
- Is MCC on? If there is an external MCC sequence in addition to the MCC contact of the PSM, check that sequence also.
- Is the power for driving MCC supplied?
- Is the breaker on?
- Has some alarm been issued in the PSM or SPM?

- **Replacing the servo amplifier**

If no problem is found in the power magnetic circuit around the amplifier, replace the servo amplifier.

- **Replacing the axis control cards**

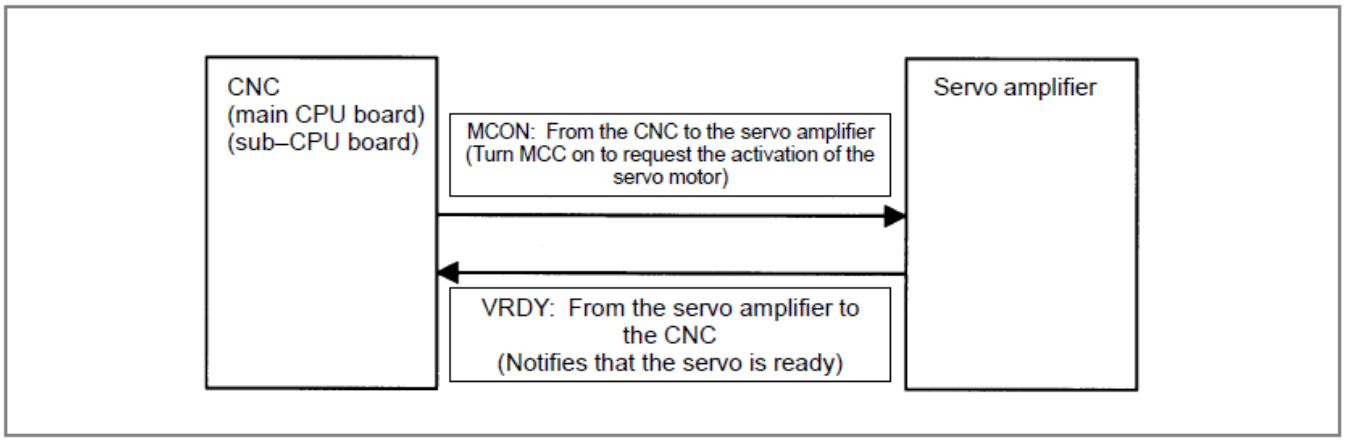
If the above action does not solve the problem, replace the axis control cards.

11.16 ALARM 404 (V READY ON)

Causes and actions

This alarm is issued if the servo ready signal (VRDY) of a servo amplifier remains on.
The servo amplifier or the axis control cards on the CNC may be defective.

- **VRDY**



The exchange of this information is performed via the FSSB (optical cable).
This alarm is issued if VRDY remains on when the CNC turns MCON off or if VRDY turns on before the CNC turns MCON on.

- **Replacing the servo amplifier**

The servo amplifier may be defective. Replace the servo amplifier.

- **Replacing the axis control cards**

If replacing the servo amplifier does not solve the problem, replace the axis control cards.

ALARM 462 (SEND CNC DATA FAILED) ALARM 463 (SEND SLAVE DATA FAILED)

Causes and actions

Alarm 462 is issued if a slave (servo amplifier) cannot receive correct data due to an FSSB communication error.
Alarm 463 is issued if the CNC cannot receive correct data due to an FSSB communication error.
If these alarms are issued, the alarm message indicates the number of the defective axis (axis name).

- **Servo amplifier or optical cable**

  Any of the optical cables between the CNC control unit and the amplifier corresponding to the axis number indicated in the alarm message may be defective.
  Or, any of the first amplifier to the amplifier corresponding to that axis number may be defective.

- **Axis control cards**

  The axis control cards installed on the CNC may be defective.

11.18 ALARM 417 (DIGITAL SERVO SYSTEM IS ABNORMAL)

Digital servo parameters are abnormal.
(Digital servo parameters are set incorrectly.)

- **Causes**

1 Confirm the setting value of the following parameters:
PRM 2020 : Motor format number
PRM 2022 : Motor rotation direction
PRM 2023 : Number of pulses of velocity feedbacks
PRM 2024 : Number of pulses of position feedback
PRM 1023 : Servo axis number
PRM 2084 : Flexible feed gear ratio
PRM 2085 : Flexible feed gear ratio
Confirm the details with diagnosis function of CNC side.
2 Change the setting of this parameter to 0.
PRM 2047 : Observer parameter
3 Perform initial setting of digital servo parameters.
Refer to setcion 6.1 "Initial Setting of Servo Parameters" .

This data indicates the cause of servo alarm No. 417, detected by the NC. If the alarm is detected by the servo, the PRM bit (bit 4 of DGN No. 0203) is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
0280		AXS		DIR	PLS	PLC		MOT

#0(MOT) : The motor type specified in parameter No. 2020 falls outside the predetermined range.

#2(PLC) : The number of velocity feedback pulses per motor revolution, specified in parameter No. 2023, is zero or less. The value is invalid.

#3(PLS) : The number of position feedback pulses per motor revolution, specified in parameter No. 2024, is zero or less. The value is invalid.

#4(DIR) : The wrong direction of rotation for the motor is specified in parameter No. 2022 (the value is other than 111 or –111).

#6(AXS) : In parameter No. 1023 (servo axis number), a value that falls outside the range of 1 to the number of controlled axes is specified. (For example, 4 is specified instead of 3.) Alternatively, the values specified in the parameter are not consecutive.

ALARM 700 (OVERHEAT: CONTROL UNIT)

Causes and actions

This alarm is issued if the ambient temperature of the CNC control unit is abnormally high. As an installation condition, the ambient temperature of the CNC must not exceed 58°C (for LCD–mounted type CNC) or 55°C (for stand–alone type CNC).

- **Ambient temperature**

A temperature monitoring circuit is installed on the motherboard (main CPU board), and causes this alarm to be issued if the ambient temperature is abnormally high.
Take appropriate action to the cabinet that houses the CNC control unit so that the temperature falls within the proper temperature range (0 to 58°C (for LCD–mounted type CNC) or 0 to 55°C (for stand–alone type CNC).
If it is obvious that the ambient temperature is not abnormal, the motherboard (main CPU board) may be defective.

ALARM 701 (OVERHEAT: FAN MOTOR)

Causes and actions

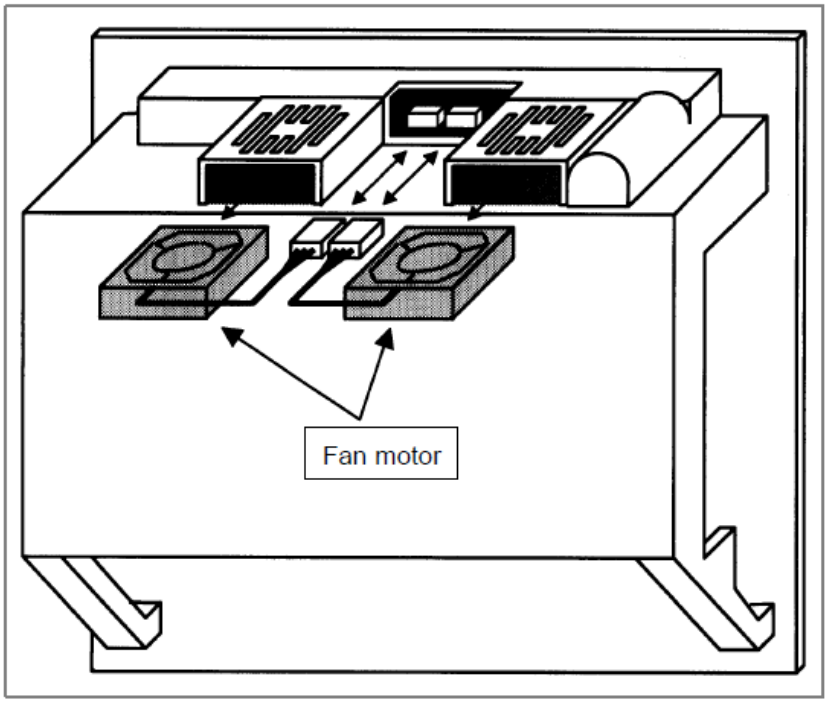
This alarm is issued if a fault occurs in any of the fan motors, such as the stoppage of a fan motor during the operation of the CNC.

- **Fan motors**

Fan motors are installed in the uppermost portion of the CNC control unit. Each fan motor is attached with an alarm detector circuit, which notifies the CNC of a fault such as the stoppage of the fan motor, thereby issuing this alarm.
If this alarm is issued, replace the fan motor.

[For the LCD–mounted type]

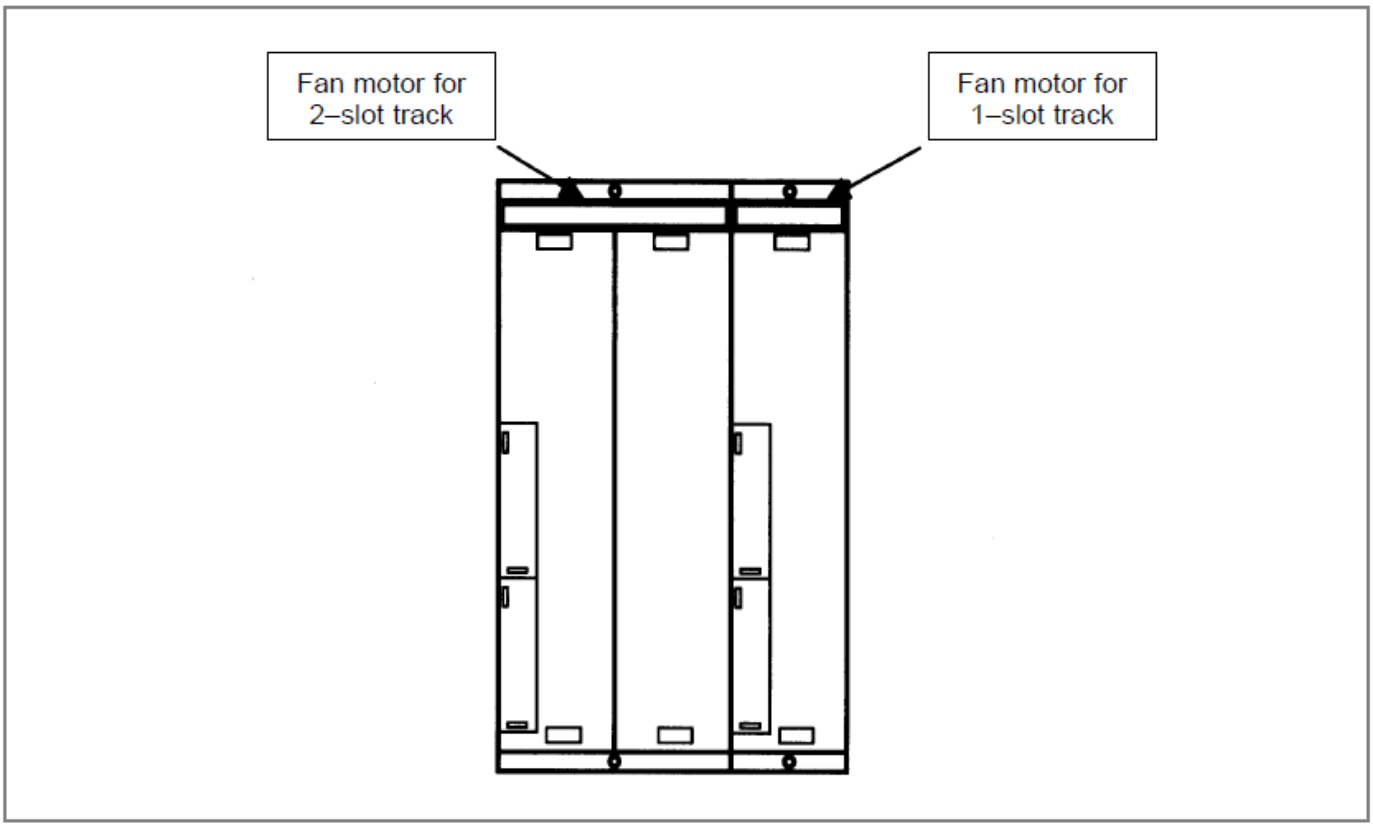
For units without option slots



Specifications of fan motors

	Ordering information	Quantity required
Unit without option slots	A02B–0236–K120	Two
Unit with two option slots	A02B–0281–K121	Two
Unit with three option slots	A02B–0281–K121	Two
	A02B–0236–K122	Two
Unit with four option slots	A02B–0281–K121	Four

[For the stand–alone type] For the stand–alone type, a fan can be replaced together with its case.



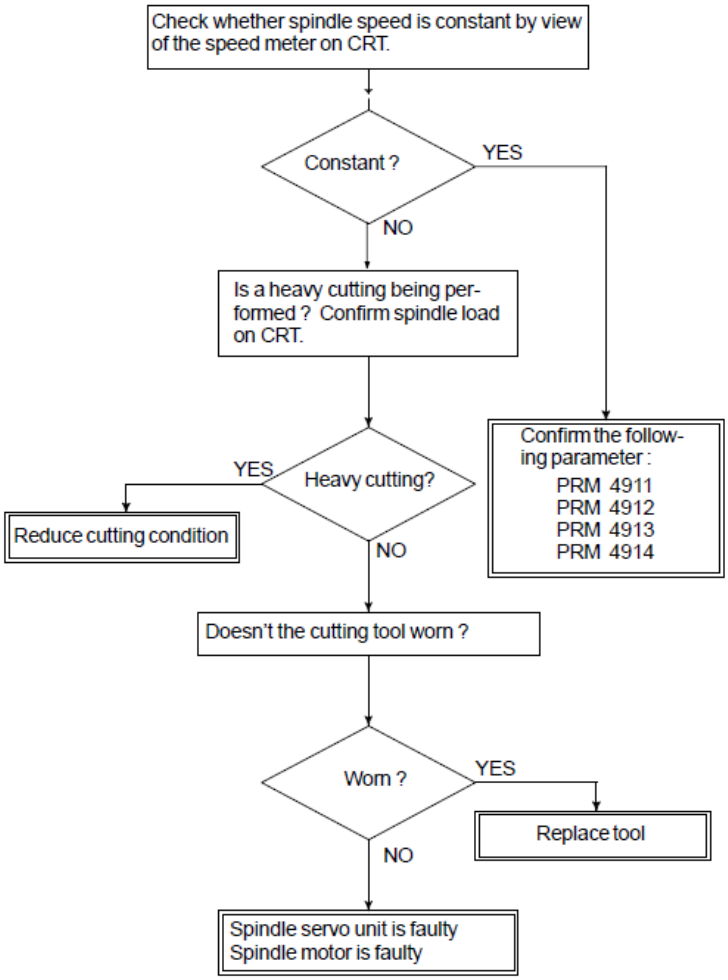
Specifications of fan motors

	Ordering information
For 1–slot track	A02B–0265–C101
For 2–slot track	A02B–0260–C021

11.21 ALARM 704 (SPINDLE SPEED FLUCTUATION DETECTION ALARM)

Spindle speed changes abnormally due to load.

Remedies



Remedies

PRM 4911 : A ratio of spindle speed at which actual spindle speed is regarded as arrived at a command spindle speed.

PRM 4912 : Spindle speed fluctuation ratio up to which the spindle speed fluctuation detection alarm is not issued.

PRM 4913 : Spindle speed fluctuation that is not regarded as the spindle speed fluctuation alarm.

PRM 4914 : Time when a spindle speed changed to when spindle speed fluctuation detection is started.

ALARM 749 (SERIAL SPINDLE COMMUNICATION ERROR)

Causes and actions

An error occurred in the communication between the serial spindle amplifier (SPM) and the CNC. The probable causes include:

- Contact failure of the connection cable
- Defective printed circuit board on the CNC
- Defective spindle amplifier
- Noise

- **Connection cable**

Check that the cable connecting the serial spindle amplifier (SPM) to the CNC is in contact.
Check that the cable is inserted firmly into the connectors and that it does not have any conductors likely to be cut off.
Check that the cable used is a twisted–pair cable and that it is connected as described in the connection manual.

- **Printed circuit boards on the CNC**

A spindle control circuit for the CNC is installed on the motherboard and the sub–CPU board. If this alarm is issued from the main CPU, replace the motherboard. If it is issued from the sub–CPU, replace the sub–CPU board.

- **Spindle amplifier module (SPM)**

When an error occurred on the spindle amplifier module (SPM) side, a code of A, A1, or A2 is indicated on the SPM depending on the nature of the error.
In this case, take appropriate actions in FANUC SERVO MOTOR αi series Maintenance Manual (B–65285EN) or FANUC SERVO MOTOR α series Maintenance Manual (B–65165E).

- **Noise environment**

If any of the above actions does not solve the problem, examine the noise environment of the connection cable.
See the section on the measures against noise, take appropriate actions such as the reinforcement of the cable shield and the separation of the cable from the power line.

ALARM 750 (SPINDLE SERIAL LINK STARTUP FAILURE)

Causes and actions

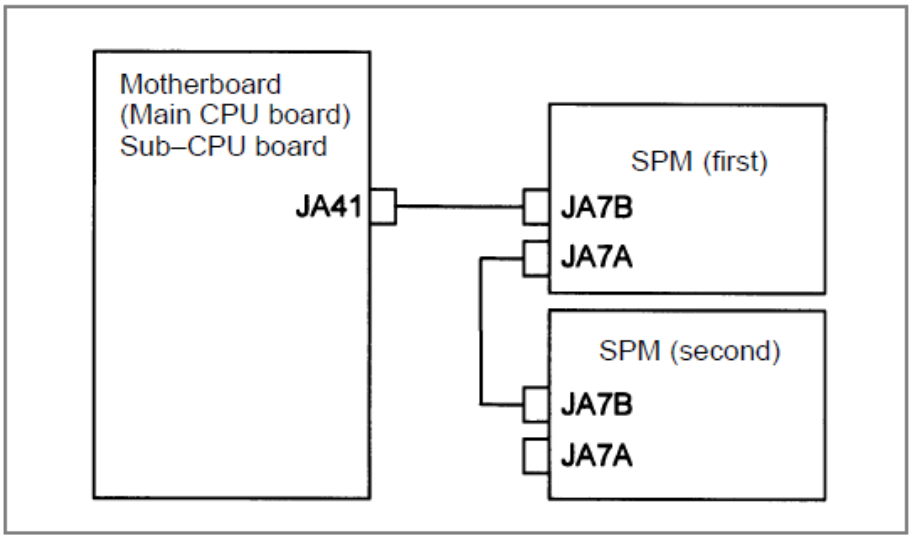
This alarm is issued if a serial spindle amplifier (SPM) does not enter the normal startup state when the CNC is turned on.
This alarm is not issued once the CNC system including the spindle amplifiers has started up normally. It is issued if a fault occurs in the power–on process.
The probable causes include the following:

- Contact failure, wiring error, or connection error of the connection cable
- The CNC is turned on when a spindle amplifier is in the alarm state.
- Parameter setting error
- Defective printed circuit board on the CNC
- Detective spindle amplifier

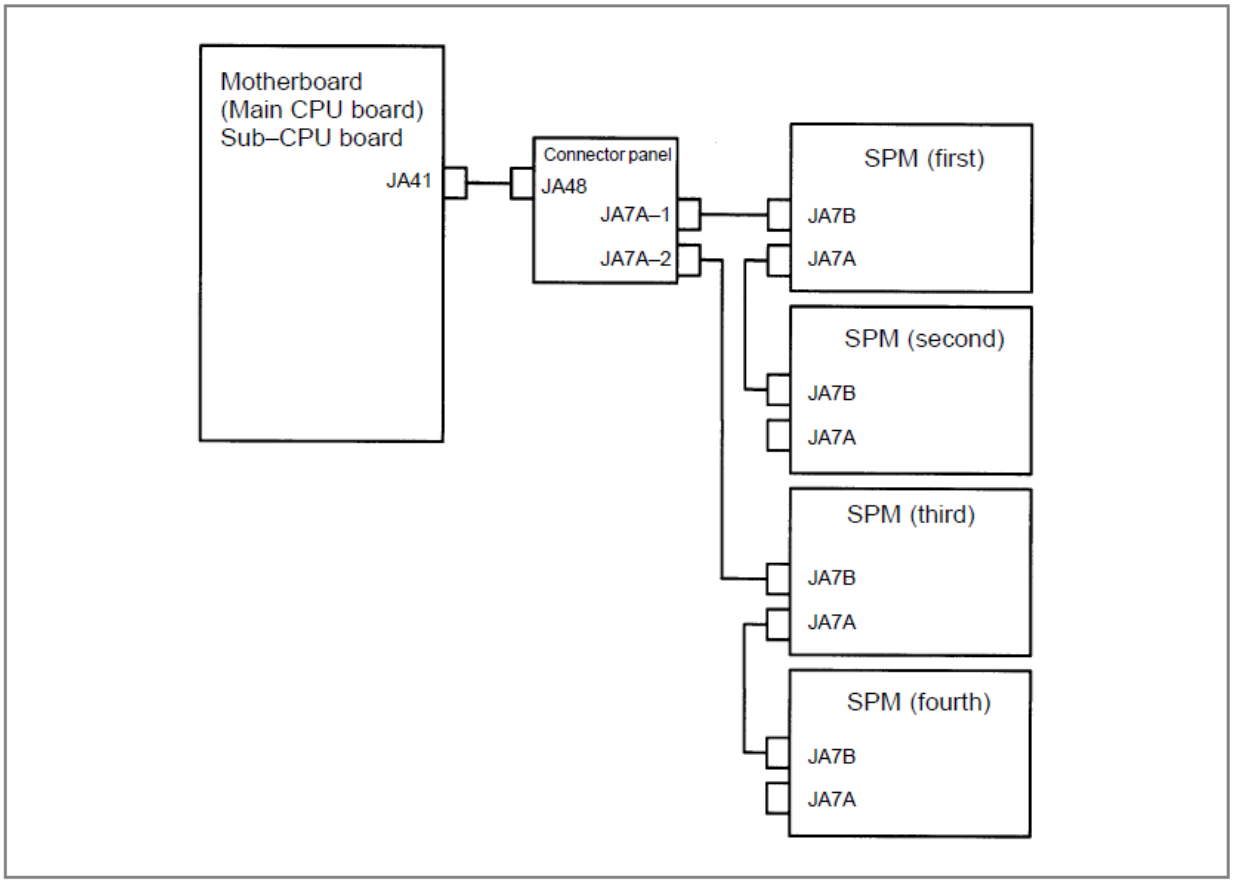
• Connection

Up to four serial spindle amplifiers (SPMs) can be connected per path. Note, however, the number of amplifiers that can be connected differs depending on the model, number of paths, and configuration. Refer to the Connection Manual (Hardware).

[Diagram of connection of up to two amplifiers per path]



[Diagram of connection of three or four amplifiers per path]



Check that the cables are connected as shown in the figure above. Check that JA7Bs and JA7As are connected correctly.
Check that the cables are latched firmly and are not loose.
Refer to the Connection Manual (Hardware) to check that the cables are connected correctly.

- **States of the spindle amplifiers**

This alarm is issued if the CNC is turned on when the LED of a spindle amplifier indicates a number other than "24".
On the spindle amplifier, remove the cause of the alarm. Turn off the spindle amplifier and the CNC, then turn on the system again.

- **Details of the alarm**

If this alarm is issued, its details can be checked with diagnosis numbers 409 and 439.

- **1st and 2nd spindles**

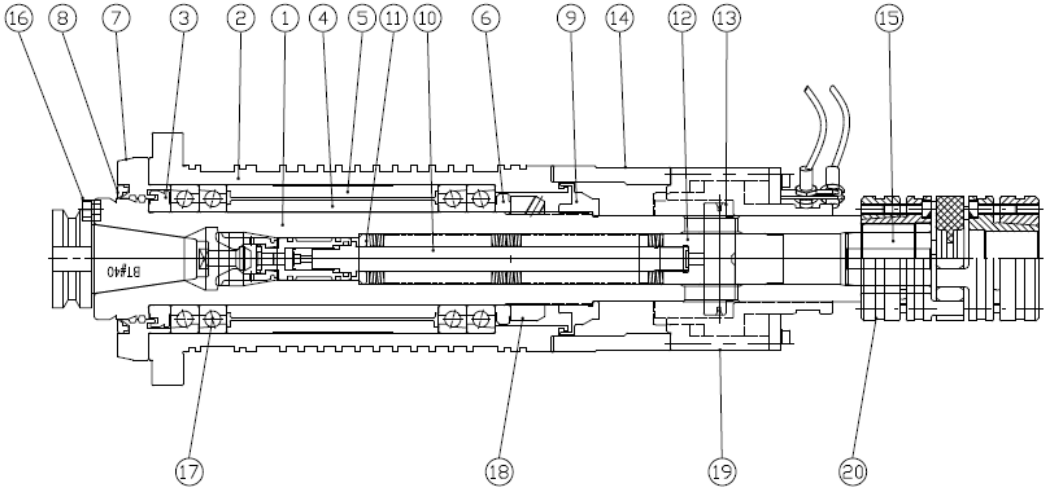
	#7	#6	#5	#4	#3	#2	#1	#0
0409					SPE	S2E	S1E	SHE

SPE: 0 : In the spindle serial control, the serial spindle parameters fulfill the spindle unit startup conditions.
1 : In the spindle serial control, the serial spindle parameters do not fulfill the spindle unit startup conditions.

7-5 List of the Machine Components

7-5-1 Spindle

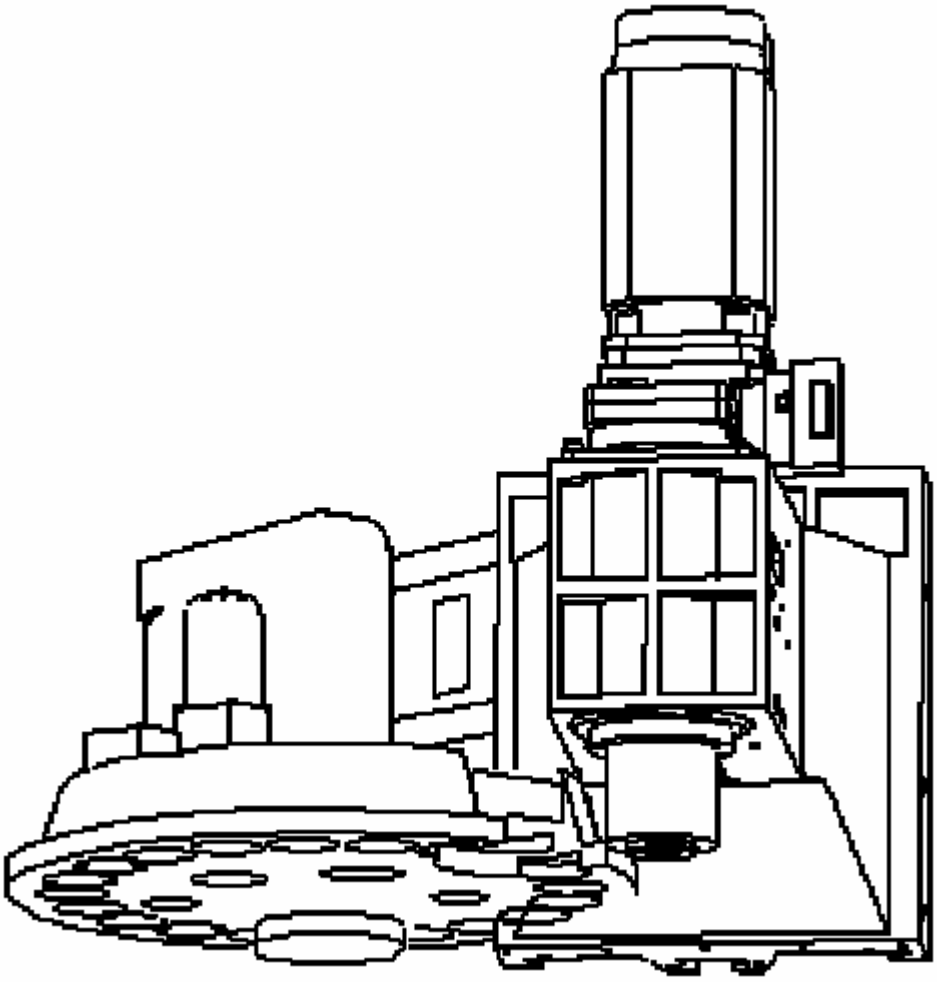
1. SVL2416SX

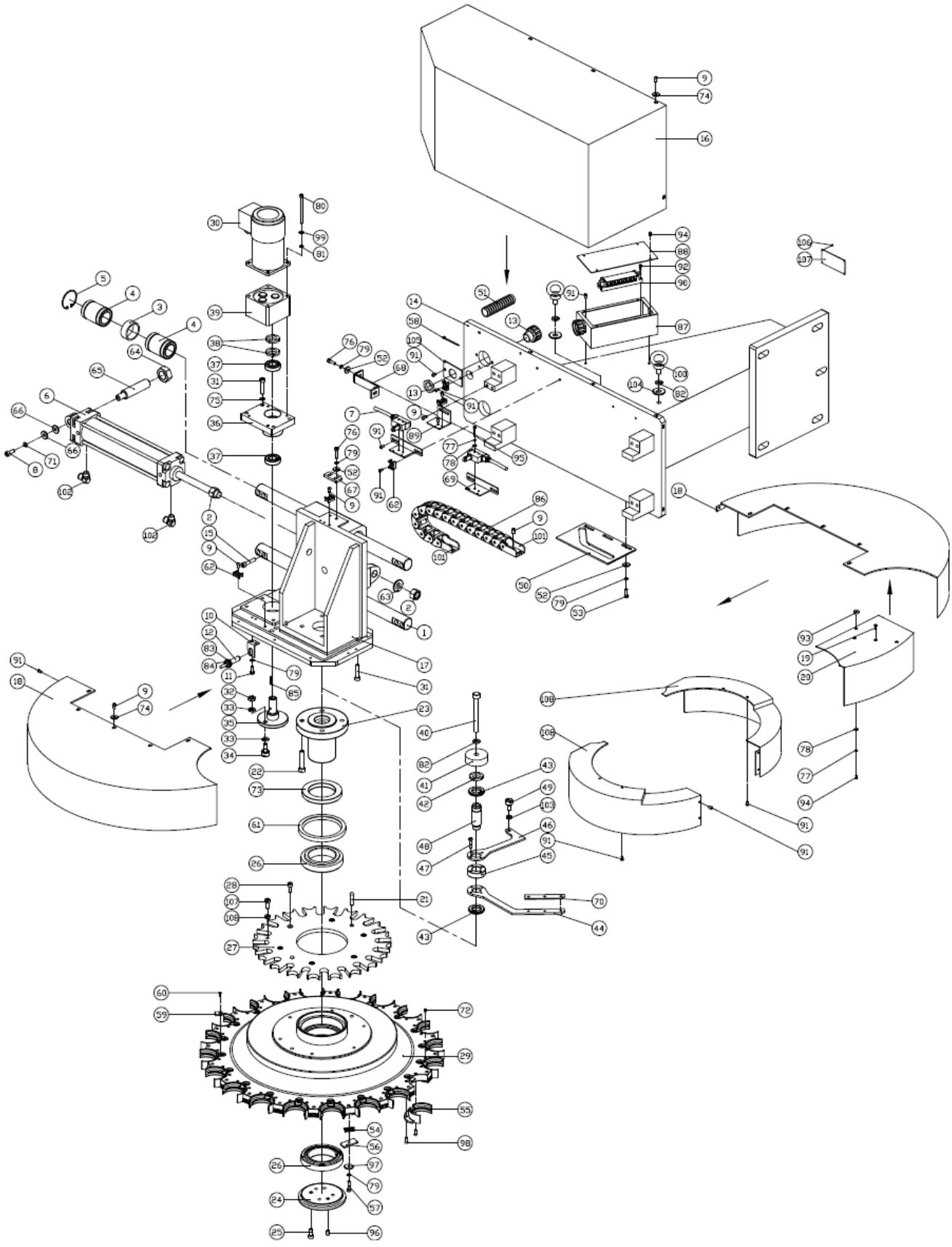


Item	NAME	Qty
1	Shaft	1
2	Spindle Body	1
3	Collar	1
4	Collar	1
5	Collar	1
6	Collar	1
7	Front Cover	1
8	Jacketing	1
9	Nut	1
10	Pull Stud	1
11	Coupling	1
12	Spring collar	1
13	Shaft bar	1
14	Cylinder seat	1
15	Shaft stopper	1
16	Round Key	2
17	Tac bearing	4
18	Nut	1
19	Cylinder	1
20	Coupling	1

7-5-2 Tool Magazine

1. Umbrella type



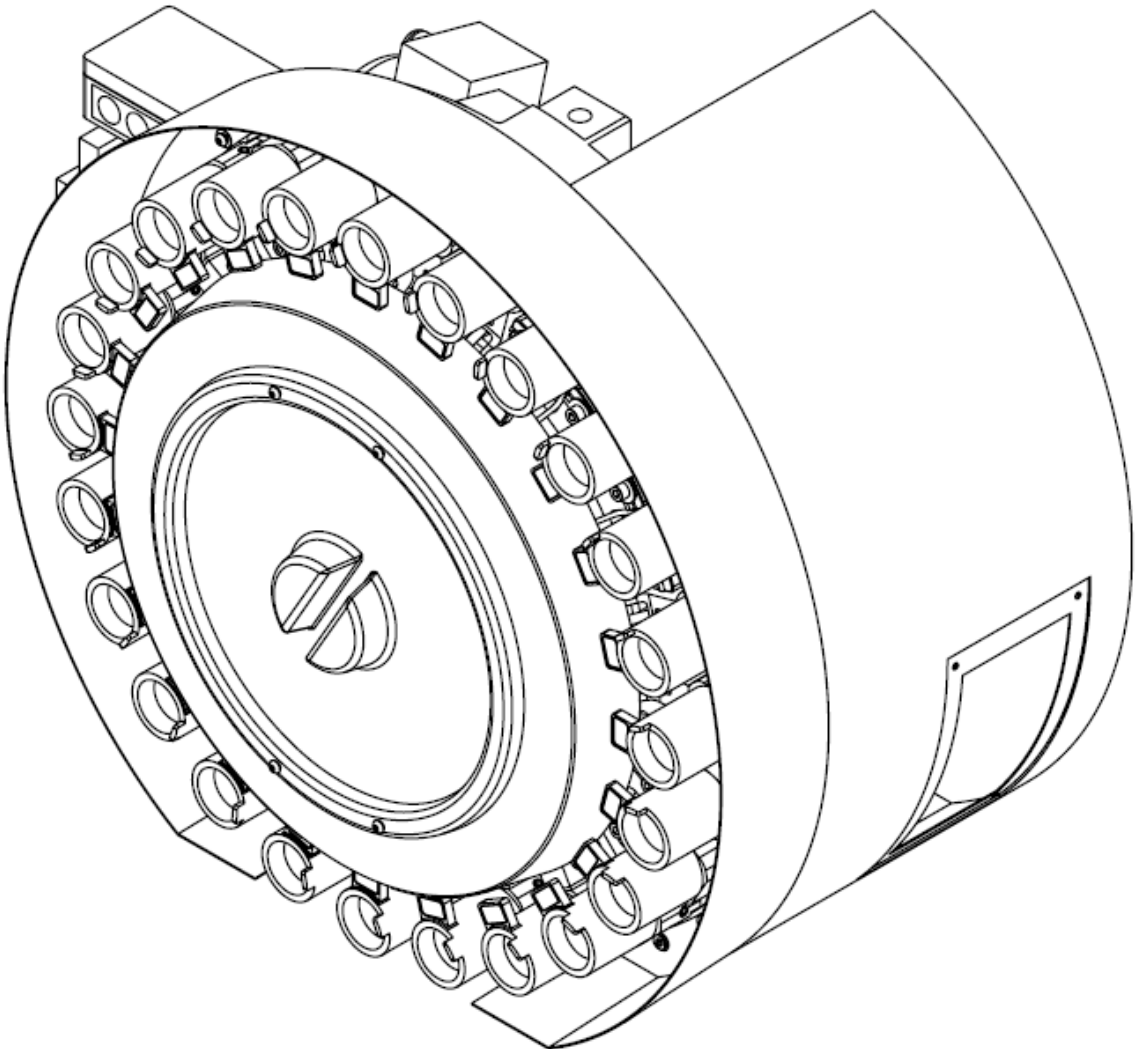


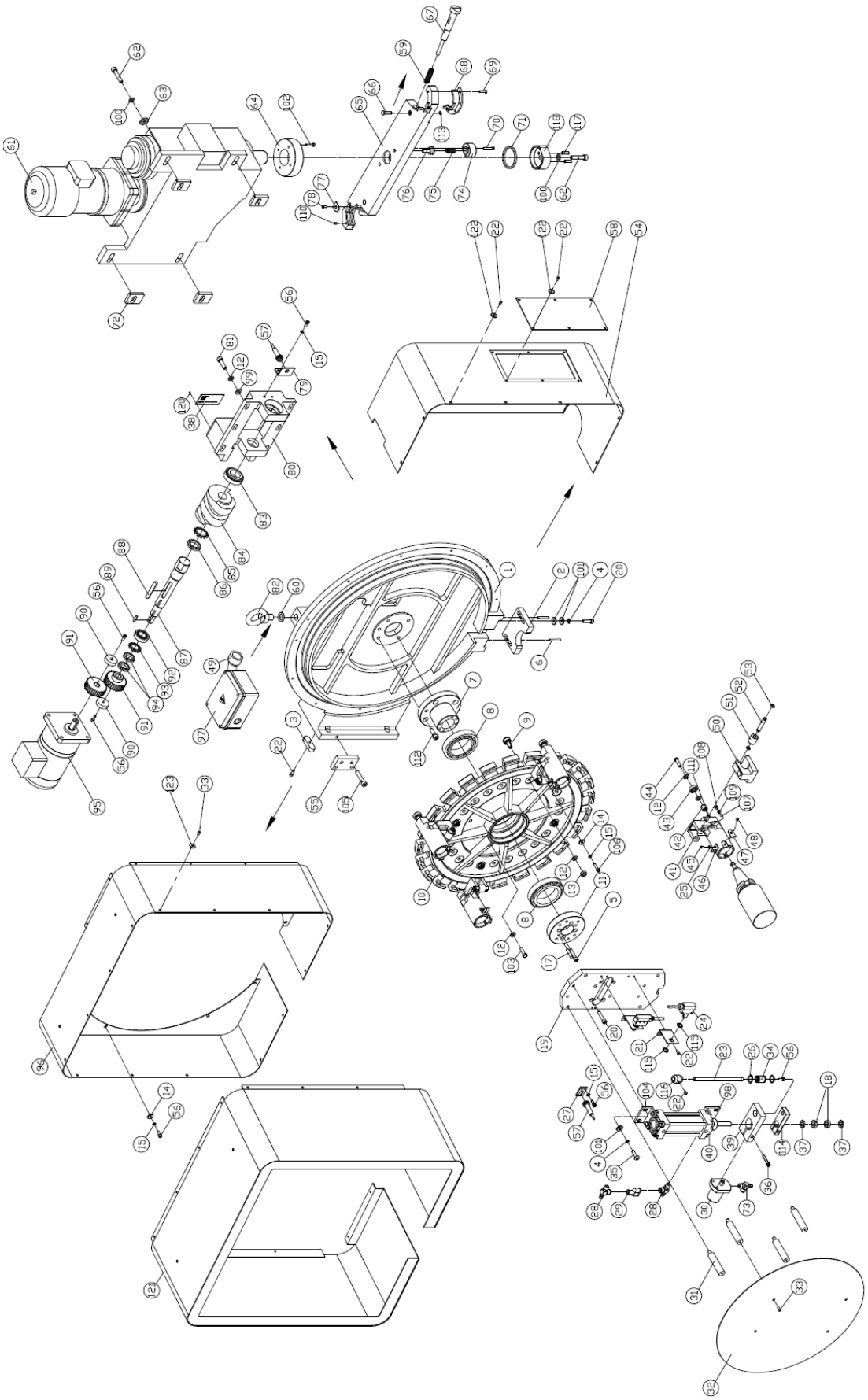
Item	NAME	SIZE	Qty
1	Guide rail	35*576L	2
2	Hex. Nut	M18	2
3	Spacer ring		2
4	Steel ball guide sleeve	35UU	4
5	C retainer	R52	2
6	Air cylinder	63*20*230	1
7	Limit switch	SHL-Q2255	2
8	Hex. socket screw	M8*1.25P*20L	1
9	Round head inside hex. screw	M6*1.0P*12L	20
10	Sensor seat		1
11	Hex. socket screw	M6*1.0P*12L	2
12	Sensor	3RG4012-0AB00	2
13	PVC Wave tube connection	3/4”	2
14	Support		1
15	Hex. socket screw	M8*1.25P*40L	8
16	Slide seat guard cover	724L	1
17	Slide seat		1
18	Tool magazine cover	420H	1
19	Flat head inside hex socket screw	M5*0.8P*14L	3
20	Safety door		1
21	Bevel pin	Φ10*38L	2
22	Hex. socket screw	M10*1.5P*55L	4
23	Fixing shaft		1
24	Lower press plate		1
25	Hex. socket screw	M8*1.25P*20L	4
26	Conical roller bearing	32015	2
27	Dividing plate		1
28	Hex. socket screw	M8*1.25P*20L	6
29	Tool magazine	424A	1
30	Induction motor	51K60GN-SMTS	1

Item	NAME	SIZE	Qty
31	Hex. socket screw	M8*1.25P*40L	8
32	Hex. Nut	M10*1.5P	1
33	Spring washer	M10	2
34	The bearing of handles	CF10	1
35	Driving shaft		1
36	Bearing seat		1
37	Groove needle roller bearing	6004 ZZ	2
38	Bearing locking nut	AN04	2
39	Reduction gear	5GN25K	1
40	Hex. socket screw	M12*1.75P*110L	1
41	Cover		1
42	Bearing locking nut	AN05	1
43	Ball thrust bearing	BMW51105	2
44	Rocking arm		1
45	Spacer ring		1
46	Rocking arm		1
47	Hex. socket screw	M5*0.8P*25L	4
48	Connecting rod		1
49	The bearing of handles	CF8	1
50	Cam plate		1
51	PVC Wave tube	3/4”	1
52	Flat washer	M6	7
53	Hex. socket screw	M6*1.0P*20L	3
54	Number plate		20
55	Hold tool carrier	Option	20
56	Flat washer	MT420H	20
57	Hex. socket screw	M6*1.0P*30L	20
58	Bind ribbon	ALT-150mm	8
59	Tool fixing block	For HSK-63 tool holder	20
60	Flat head inside hex. screw	M4*0.7P*10L	20

Item	NAME	SIZE	Qty
61	Oil seal	TC110*130*12	1
62	Bind ribbon fixing block	ATM-2	5
63	Air cylinder washer	416-520	1
64	Hex. Nut	M25*1.5P	2
65	Air cylinder fixing plug	416-424	1
66	Flat washer	M8	2
67	Bump block		1
68	Air cylinder stop seat		1
69	Limit switch seat		2
70	Spacer	MT420-MT424	1
71	Spring washer	M8	1
72	Stoppage screw	M5*0.8P*8L	40
83	Sleeve		1
74	Flat washer	M6	14
75	Spring washer	M8	4
76	Hex. socket screw	M6*1.0P*20L	4
77	Spring washer	M4	5
78	Flat washer	M4	5
79	Spring washer	M6	29
80	Hex. socket screw	M6*85L	4
81	Flat washer	M6*13*1	4
82	Spring washer	M12	3
83	Suns nut	M12	2
84	Hex. Nut	M12	2
85	Semicircle key	4*4*25L	1
86	Chain	320.42KR37	1M
87	Terminal box		1
88	Terminal box cover		1
89	Protest fixing seat		1
90	Terminal seat	10A12P	1

Item	NAME	SIZE	Qty
91	Round head inside hex. screw	M5*0.8P*10L	97
92	Hex. socket screw	M4*0.7P*10L	2
93	Shockproof ring		1
94	Round head inside hex. screw	M4*0.7P*8L	5
95	Hex. socket screw	M4*0.7P*25L	4
96	Stoppage screw	M8*1.25P*14L	2
97	Flat washer	M6	20
98	Flat head inside hex socket screw	M5*0.8P14L	40
99	Spring washer	M6	4
100	Ring	M12	2
101	Plastic connection		2
102	Tube connection	CTL-1003A	2
103	Hex. Nut	M8	1
104	Flat washer	M12	0
105	Protect seat		1
106	Motor screw	M2*5	4
107	Make No. plate		1
108	Dividing plate cover		1





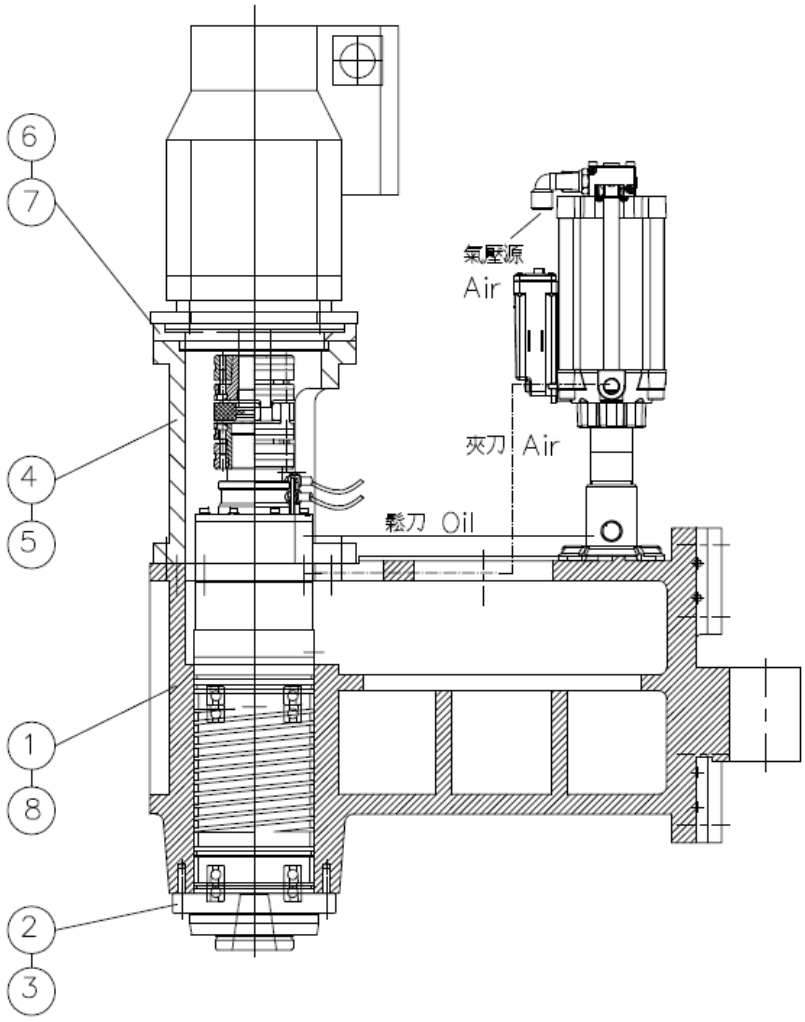
Item	NAME	SIZE	Qty
1	Main Body		1
2	Pocket Positioning Set		1
3	Position Key	ϕ20x12x60L	2
4	Spring Washer	M8	8
5	Hexagon Socket Button Head Screw	M8x1.25Px30L	4
6	Taper Pin	40L	2
7	Tool Disk Driver		1
8	Taper roll Bearing	32015	2
9	Bearing	CF10	24
10	Tool disk Module		1
11	Bearing Bracket		1
12	Spring Washer	M10	79
13	Nut	M10x1.5P	24
14	Plain Washer	M6	32
15	Spring Washer	M6	36
16	Hexagon Socket Button Head Screw	M6x1.0Px20L	8
17	Straight Pin	ϕ10x38L	5
18	Nut	M14x1.5P	2
19	Cylinder Mounting Plate Module		1
20	Hexagon Socket Button Head Screw	M8x1.25Px40L	8
21	Limit Switch Bracket		2
22	Hexagon Socket Button Head Screw	M5x0.8Px10L	13
23	Rod		1
24	Limit Switch	SHL-Q2255	2
25	Plain Washer	M4	24
26	Retaining Ring - C	S21	2
27	Sensor Bracket		1
28	Regulator		2
29	One-Touch fittings-Y		1
30	Cylinder Module		1
31	Support Rod		4
32	Cover		1
33	Hexagon Socket Button Head Screw	M6x1.0Px12L	20

Item	NAME	SIZE	Qty
34	Linear bearing	LM12UU	1
35	Hexagon Socket Button Head Screw	M8x1.25Px30L	4
36	Hexagon Socket Button Head Screw	M6x1.0Px40L	1
37	Spring Washer	M14	2
38	Name Labels		1
39	Linear bearing Bracket		1
40	Cylinder Module	ϕ50x18X105L	1
41	Pocket	65°	24
42	Bearing washer		24
43	Ball Bearing	6000ZZ	24
44	Hexagon Socket Button Head Screw	M10x150Px50L	24
45	Hexagon Socket Button Head Screw	M4x0.7Px8L	24
46	Name plate		24
47	Tool Mounting block		24
48	Socket Countersunk Head Screw	M4x0.7Px8L	24
49	PVC Joints -HE	1”(ϕ28)	1
50	Tool Tilt Block		1
51	Tool Roller		24
52	Tool Roller Pin		24
53	Retaining Ring - C	S - 10	48
54	ATC Cover		1
55	Magazine Adjustiment Block		1
56	Hexagon Socket Button Head Screw	M6x1.0Px16L	14
57	Sensor	3RG4012-OAF01 (PNP)	2
58	Cam box Window		1
59	Finger Spring	RS14x55(ϕ2.0T)	2
60	Spring Washer	M20	1
61	Cam Box	402 TYPE	1
62	Hexagon Socket Button Head Screw	M12x1.75Px50L	5
63	Washer	M12	4
64	Shaft ring		1
65	Arm	265L	1
66	Safety Pin		2

Item	NAME	SIZE	Qty
67	Finger	265L	2
68	Gripper		2
69	Socket Countersunk Head Screw	M6x1.0Px12L	8
70	Hexagon Socket Button Head Screw	M4x0.7Px30L	6
71	Taper Snap Ring	ϕ40xϕ45	1
72	Adapter Block		4
73	Regulator		1
74	Spring Cover		2
75	Pin Spring	RS14x55(ϕ2.0T)	2
76	Cap Scar		2
77	Key		2
78	Socket Countersunk Head Screw	M5x0.8Px14L	2
79	Sensor Bracket		1
80	Cam mounted plate		
81	Hexagon Socket Button Head Screw	M10x1.5Px35L	1
82	Hook Ring		6
83	Taper Roller Bearing		1
84	Cam		1
85	Washer	AW06	1
86	Locking Nut	AN06	1
87	Roller Cam		1
88	Key	8x10x70L	1
89	Key	5x5x25L	1
90	Gear Mounted plate		2
91	Gear		2
92	Taper Roller Bearing	32005	2
93	Washer	AW05	1
94	Locking Nut	AN05	2
95	Reducer Motor	200W	1
96	Cover		1
97	CE Box	10031-23P	1
98	Cylinder Mounting Plate		1
99	Washer	M10	12

Item	NAME	SIZE	Qty
100	Spring Washer	M12	9
101	Washer	M8	8
102	Hexagon Socket Button Head Screw	M6x1.0Px30L	4
103	Hexagon Socket Button Head Screw	M10x1.5Px20L	1
104	Cylinder Mounting Plate		1
105	Hexagon Socket Button Head Screw	M10x1.5Px50L (Full Pitch)	2
106	Hexagon Socket Button Head Screw	M6x1.0Px25L	24
107	Steel Ball	8mm	96
108	Hexagon Socket Button Head Screw	M10x1.5Px8L	96
109	Spring	ϕ 1.2xcd8x14xn6	96
110	Hexagon Socket Button Head Screw	M6x1.0Px8L	8
111	Position pin		24
112	Hexagon Socket Button Head Screw	M10x1.5Px30L	4
113	Retaining Ring - C	S8	2
114	Cylinder Mounting Adapter plate		1
115	Nut	M12	4
116	Sensor Dog		1
117	Hexagon Socket Button Head Screw	M8x1.25Px25L	2
118	Cover		1
119	Washer	M6	12
120	Taper Position Pin	M2x5	4
121	Cover		1
122	Washer	M5	11
123	Washer	M6	13

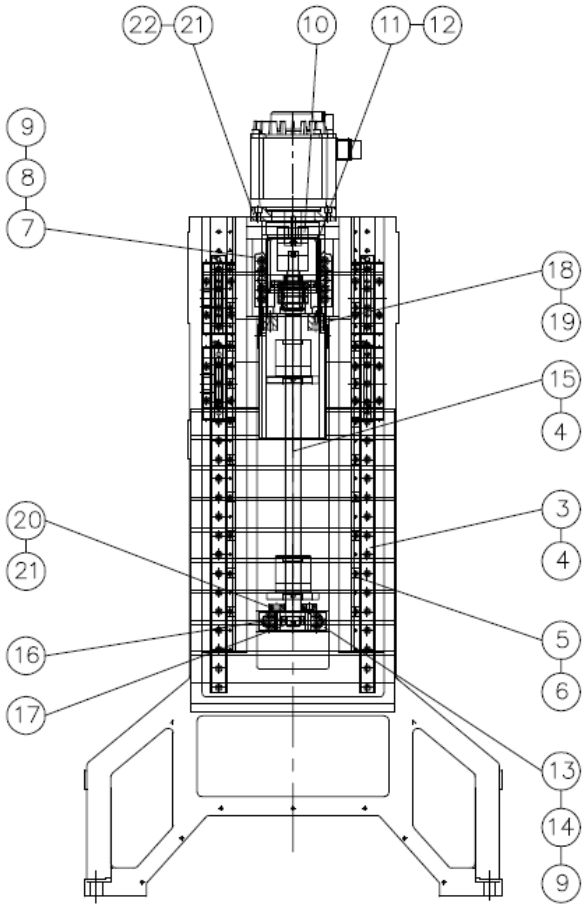
7-5-3 Spindle head

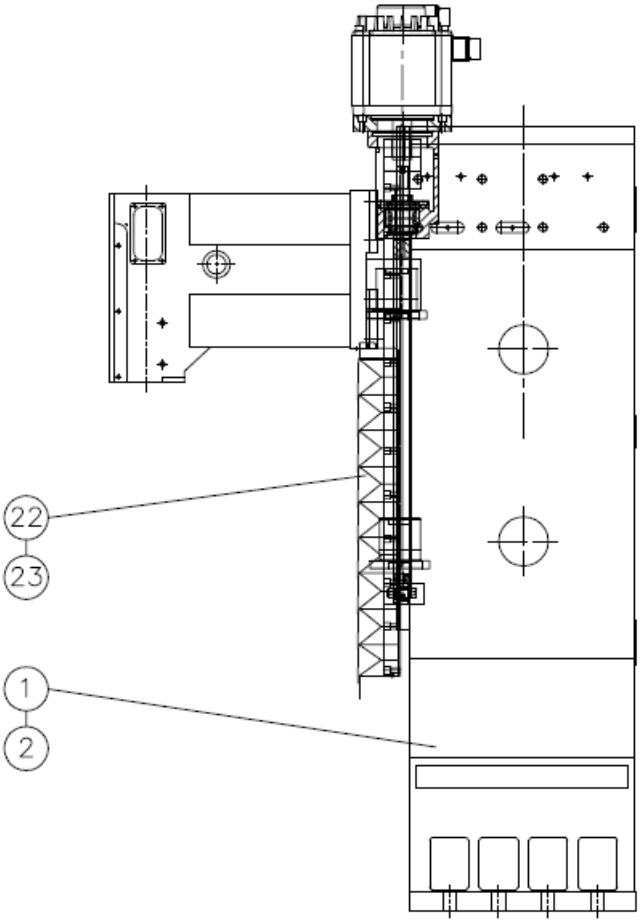


Item	NAME	Size	Qty
1	Main bracket		1
2	Spindle	ϕ120	1
3	Hexagonal Socket Head Screw	M8x1.25Px40L	6
4	Spindle Adapter Plate		1
5	Hexagonal Socket Head Screw	M12x1.75Px35L	4
6	Motor adjustment plate		1
7	Hexagonal Socket Head Screw	M10x1.5Px50L	4
8	Hexagonal Socket Head Screw	M8x1.25Px30L	16

7-5-4 Column

1. SVL2416

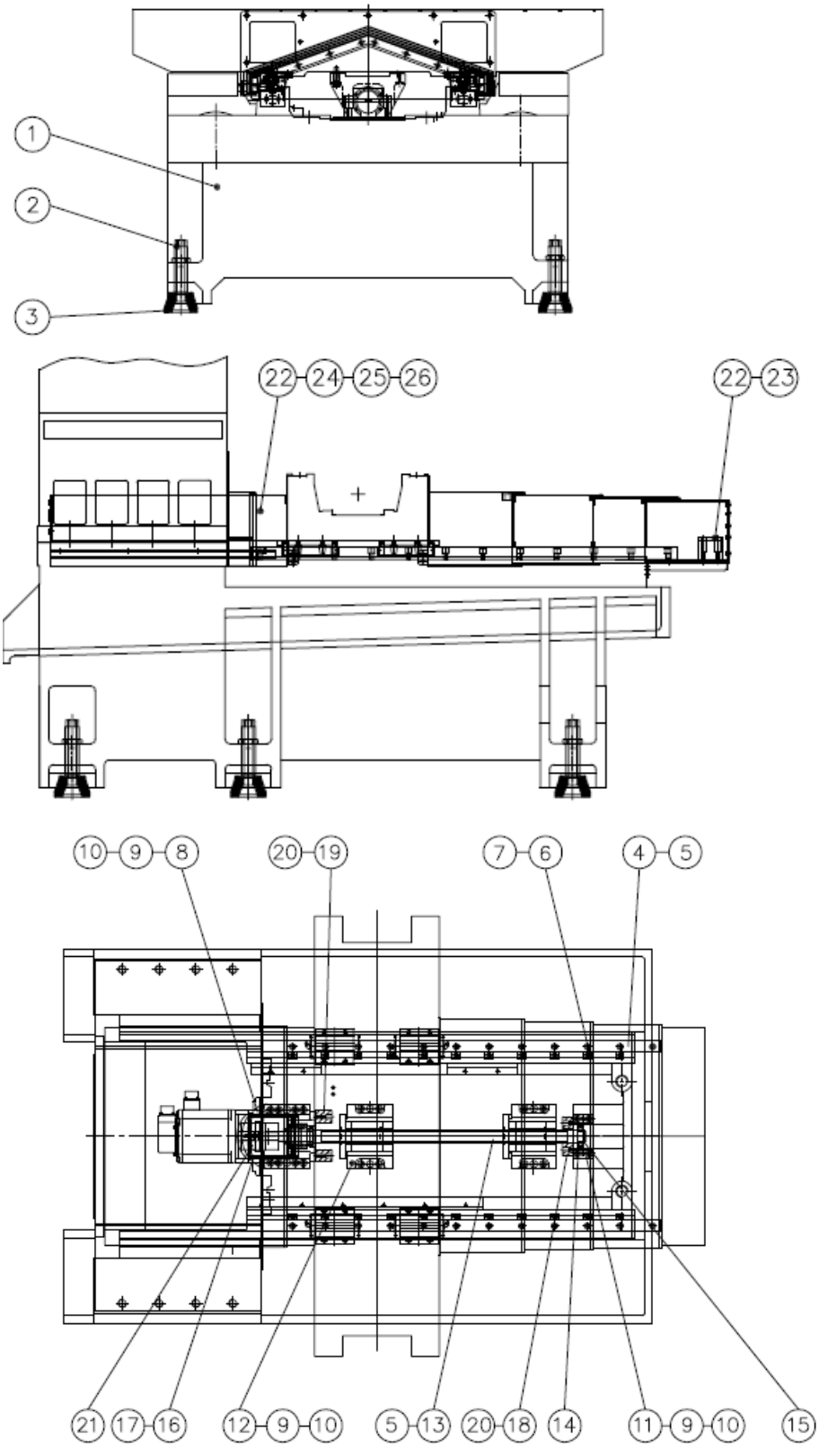




Item	NAME	Size	Qty
1	Column		1
2	Hexagonal Socket Head Screw	M20x2Px75L	8
3	Linear Guide Way		2
4	Hexagonal Socket Head Screw	M8x1.25Px30L	12
5	Wedge	T1	20
6	Hexagonal Socket Head Screw	M5x0.75Px12L	20
7	Z Motor plate	ϕ 110	1
8	Hexagonal Socket Head Screw	M10x1.5Px45L	8
9	Taper Pin	ϕ 8x40L	4
10	Coupling	ϕ 65(outside dimension) , ϕ 20x ϕ 35 (inside)	1
11	Z Motor plate Cover		1
12	Stainless Hexagonal Socket Head Screw	M4x0.7Px6L	4
13	Z Bearing Case		1
14	Hexagonal Socket Head Screw	M10x1.15Px35L	4
15	Ball Screw	R32-16K4-FSC-626-790-0.008	1
16	Ball Bearing	NTN 6205ZZC2/2AS	1
17	Retaining Ring - C	S25	1
18	Stopper	T=25	2
19	Hexagonal Socket Head Screw	M6x1.0Px35L	2
20	Stopper	T=15	2
21	Hexagonal Socket Head Screw	M6x1.0Px16L	2
22	Z Motor mounting Adapter	ϕ 114.3mm (T=30mm)	1
23	Hexagonal Socket Head Screw	M10x1.5Px35L	4
24	Z Telescopic cover	W=35	1
25	Hexagonal Socket Head Screw	M5x0.75Px12L	6

7.5.5 Base Plate

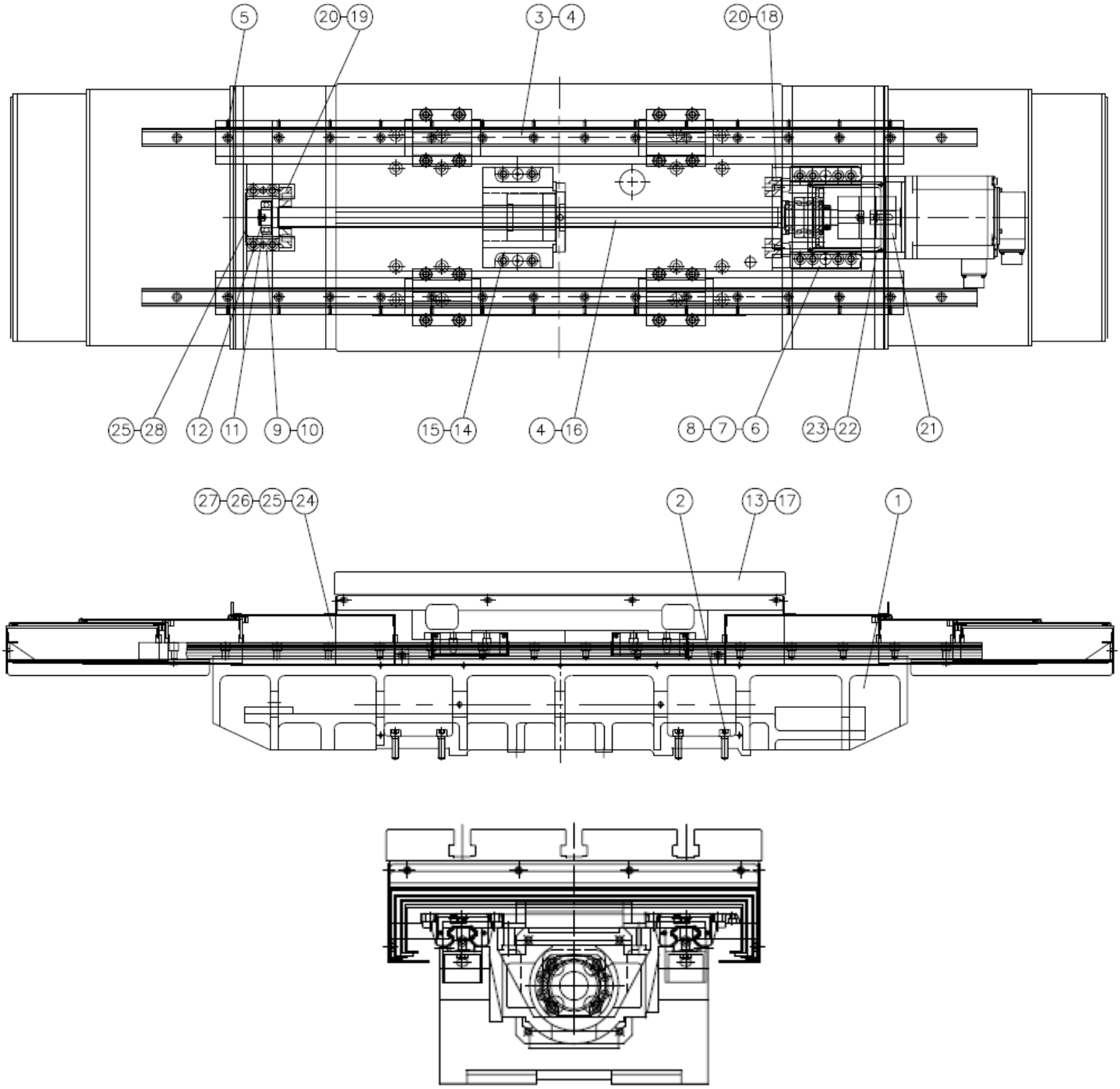
1. SVL2416



Item	NAME	Size	Qty
1	Base		1
2	Adjustable level bolt	M30x2Px115L	6
3	Adjustable level Base	T5	6
4	Linear Guide Way		2
5	Hexagonal Socket Head Screw	M8x1.25Px30L	26
6	Wedge	T1	22
7	Hexagonal Socket Head Screw	M5x0.8Px12L	22
8	Y Motor mounting Plate	ϕ 110mm	1
9	Hexagonal Socket Head Screw	M10x1.5Px35L	10
10	Taper Pin	ϕ 8x40L	6
11	Y Bearing Case	T=50mm	1
12	Ball Screw Nut Plate	115L	1
13	Ball Screw	R32-16K4-FSC-626-790-0.008	1
14	Ball Bearing	NTN 6205ZZC2/2AS	1
15	Retaining Ring - C	S25	1
16	Y Bearing Case Cover		1
17	Hexagonal Socket Head Screw	M4x0.7Px6L	4
18	Stopper	T=25	2
19	Stopper	T=40	2
20	Hexagonal Socket Head Screw	M6x1.0Px35L	2
21	Hexagonal Socket Head Screw	M6x1.0Px25L	8
22	Coupling	ϕ 65(outside dimention) , ϕ 20x ϕ 20 (inside)	1
23	Y Telescopic Cover unit	W = 35	1
24	Hexagonal Socket Head Screw	M6x1.0Px12L	33
25	Spring Washer	M6	33
26	Washer	M6	33

7-5-6 Saddle base / Workbench

1.SVL2416



Item	NAME	Size	Qty
1	Saddle base		1
2	Hexagonal Socket Head Screw	M10x1.5Px30L	28
3	Linear Guide Way		2
4	Hexagonal Socket Head Screw	M8x1.25Px30L	34
5	Hexagonal Socket Head Screw	M4x0.7Px12L	28
6	X Motor mounting Plate	ϕ 110mm	1
7	Hexagonal Socket Head Screw	M10x1.5Px45L	6
8	Taper Pin	ϕ 8x40L	6
9	X Bearing Case	T=50mm	1
10	Hexagonal Socket Head Screw	M12x1.75Px45L	4
11	Ball Bearing	NTN 6205ZZC2/2AS	1
12	Retaining Ring - C	S25	1
13	Table	W = 30	1
14	Ball Screw Nut Plate	X Axis	1
15	Hexagonal Socket Head Screw	M10x1.5Px35L	4
16	Ball Screw	R32-16K4-FSC-786-950-0.008	1
17	Hexagonal Socket Head Screw	M10x1.5Px25L	16
18	Stopper	T=25	2
19	Stopper	T=20	2
20	Hexagonal Socket Head Screw	M6x1.0Px35L	4
21	Coupling	ϕ 65(outside dimention) , ϕ 20x ϕ 35 (inside)	1
22	X Motor mounting Plate Cover		1
23	Hexagonal Socket Head Screw	M4x0.7Px6L	4
24	X Axis Telescopic Cover		1
25	Hexagonal Socket Head Screw	M6x1.0Px12L	24
26	Spring Washer	M6	20
27	Washer	M6	20
28	X Axis Bearing case Cover		1