Alarm Codes	Information	Content
1	TH error	A TH error was detected during the reading of the input device. The read code that causes the TH error and the first few characters from the block number can be confirmed by the diagnostic screen.
2	TV Parity error	An error was detected in the TV detection of a single block. By setting the parameter TVC (No.0000#0) to 0, the system does not perform TV detection.
3	Too many digits	More digits are allowed than words of the NC instruction. This allowed number of bits varies depending on the function and address.
4	Address not found	The address + value of the NC statement does not belong to the word format. This alarm is also issued when there are no reserved words in the user macro or if they do not match the syntax.
5	No data after address	Not the word format of the address + value of the NC statement. This alarm is issued when there is no reserved word in the user macro or if the syntax is not met.
6	Negative use illegal	A negative sign is specified in the word and system variable of the NC instruction.
7	Fractional use illegal	A decimal point is specified in an address that does not allow decimal points. Or specified 2 or more decimal points.
9	NC address is wrong	An address that cannot be specified in the NC statement is specified. Or the parameter has not been set (No.1020).
10	G code is incorrect	An unusable G code was specified.
11	Cutting speed is 0 (not commanded)	The command to cut the feedrate F code is set to 0. In the case of a rigid tapping command, when the F command is very small relative to the S command, this alarm is issued because the tool cannot be cut under the programmed lead.
15	At the same time control too many axes	A movement command is issued that is more than the number of axes that can be simultaneously controlled. Split the moving axis of the program command into two blocks.
20	Radius value is out of tolerance	An arc with a larger difference between the radius values of the start and end points than the set value of the parameter (No.3410) is specified. Please check the arc center command I, J, K of the program. The movement path when the value of the parameter (No. 3410) is increased becomes a spiral shape.
21	Illegal plane selection	Plane selection G17～G19 are incorrect. Review the program and check if there are no parallel axes that specify 3 basic axes at the same time. This alarm is issued when an arc command other than plane selection is included in the case of circular interpolation. In the case of <<T>>0i-TD, it is necessary to have a helical interpolation option to be able to execute three or more commands on the G02/G03 block.

22	R or I, J, K command not found	In circular interpolation, R (arc radius) or I, J and K (distance from the starting point to the arc center) are not set.
25	In the fast moving mode arc cutting	In the circular interpolation mode (G02, G03), F0 (F1 bit feed or reverse feed rapid movement) is commanded.
27	There is no axis command in G43/G44	The axis is not specified for the C-type tool length compensation in the G43/G44 block. The offset is not canceled, but another axis attempts to compensate for the C-type tool length. Multiple axis commands are specified for the C-type tool length compensation in the same block.
28	Illegal plane selection	Plane selection G17～G19 are incorrect. Review the program and check if there are no parallel axes that specify 3 basic axes at the same time. In the case of circular interpolation, this alarm is issued if an axis command other than plane selection is included. <> In the case of 0i -TD, it is necessary to have a three-axis or more command for the G02/G03 block, and a helical interpolation option is required.
29	Knife bias value is illegal	The offset number is incorrect.
30	Illegal knife number	An uncontrollable offset number was commanded.
31	Illegal P command in G10	The data input or the corresponding function of the L number of G10 is not in the valid state. There is no data to set the address P, R, etc. instructions. There are address instructions that are not related to data settings. According to the L number, the specified addresses are different. The symbol, decimal point, and range of the command address value are incorrect.
32	The knife offset value in G10 is illegal	The specified offset value is too large in the offset value program input (G10) or when the offset value is written with the system variable.
33	G41/G42 no intersection	The intersection point cannot be obtained for tool nose radius compensation or nose radius compensation. Please modify the program.
34	Do not allow cutting arcs in the starting/retracting section	In the tool radius compensation/tool nose radius compensation, an attempt was made to execute the start or cancel command in a mode other than G00/G01. Please modify the program.
35	Cannot command G31	1) It is in a state where G31 cannot be commanded. This alarm is issued when the G code of group 07 (tool radius compensation or tool nose radius compensation, etc.) cannot be canceled. 2) The torque limit has not been specified in the torque limit jump command (G31P98/P99). Please specify in the PMC window, etc.
37	Cannot change plane in G41/G42	Switch to the compensation plane G17/G18/G19 in tool radius compensation or nose radius compensation. Please modify the program.
38	Arc segment has interference	Because the start and end points of the arc coincide in the arc center, overcutting may occur under the tool radius or tool nose radius compensation. Please modify the program.