NC FAULT DISPLAYS: TX 8 D		TRAUB

Display	Cause	Rectification
004 W.D.T 1	System monitoring time exceeded.	If LED1 lights up on print FX 01, exchange print. If the LED stays on after switching on again, change print FX 84-7.
005 MEMORY PARITY	Print FX 84-7 or FX 27 faulty.	If LEDs 2, 3 on print FX 84-7 or LED2 on print FX 27 are on, exchange prints in question.
009 MAIN CPU ERROR	Print FX 01 faulty.	If LED1 is on, exchange print.
010 PC ERROR	Print FX 15 faulty. Fault in PC monitoring program.	LEDs 1, 2, 3, 4 on print FX 15 light up (exchange print). Ring after-sales service (fault in machine data or PC program).
011 CPU CHECK SUM ERROR	Programming error in E Prom or axis control.	Exchange print FX 31 or E Prom on print FX 31.
012 PC CHECK SUM ERROR	E Prom memory PC is faulty.	Exchange print FX 25 or E Proms.
015 EMERGENCY	EMERGENCY STOP status operative.	Eliminate EMERGENCY STOP situation.
016 WRONG PARA DATA	Parameter data are not acceptable.	Check parameter data.
018 BATTERY ALARM	Buffer battery of memory below acceptable voltage limit.	LED1 on print FX 84-7 lights up. Replace battery. Check parameters, part programs, tool data and reload, if necessary.
019 PROGRAM MEMORY ALARM	Machine program destroyed.	After RESET, input or store part program again.
020 READ ERROR BUBBLE MEMORY	Bubble memory is defective or contains incorrect data.	Check bubble memory.
021 WRITE ERROR BUBBLE MEMORY	Bubble memory is defective.	Check bubble memory.

Display	Cause	Rectification
022 ERROR CORRECT. BUBBLE MEMORY		The fault can still be corrected.
101 OVER TRAVEL +X $1 501 " " +X $2	The axes are outside the acceptable traverse range of the machine in +X and +Z direction, respectively. Traverse in automatic or manual mode.	Clear axes in opposite direction. Check program in automatic mode and correct. Press RESET.
102 OVER TRAVEL +Z $1 502 " " +Z $2		

12

Display	Cause	Rectification
104 OVER TRAVEL +4 504	The axes are outside the acceptable traverse range of the machine in + direction.	Clear axes in opposite direction. In case of automatic mode, check and correct program. Press RESET.
105 OVER TRAVEL +5 505		
106 OVER TRAVEL $-X \not\$1$ 506 " " $-X \$2$	The axes are outside the acceptable traverse range of the machine in $-X$ and $-Z$ direction, respectively.	Clear axes in opposite direction. In case of automatic mode, check and correct program. Press RESET.
107 OVER TRAVEL $-Z \$1$ 507 " " $-Z \$2$		
109 OVER TRAVEL -4 509	The axes are outside the acceptable traverse range of the machine in "$-$" direction.	Clear axes in opposite "+" direction. In case of automatic mode, check and correct program. Press RESET.
110 OVER TRAVEL -5 510		
111 SOFT LIMIT $+X \$1$ 511 " " $+X \$2$	The end position value set by parameter has been reached in "+" direction.	Check parameter in CHECK, image 2. If parameter is correct, check part program. If parameter value is incorrect, correct parameter and switch OFF/ON.
112 SOFT LIMIT $+Z \$1$ 512 " " $+Z \$2$		

Display	Cause	Rectification
116 SOFT LIMIT −X $1 516 −X $2	The end position value set by parameter has been reached in '−' direction.	Check parameter in CHECK, image 2. If parameter is correct, check part program. If parameter value is incorrect, correct parameter and switch OFF/ON.
117 SOFT LIMIT −Z $1 517 −Z $2		
122 TAILSTOCK $1 522 BARRIER $2	The value of the protected tailstock zone set by parameters has been reached or exceeded.	Check appropriate parameter in SET UP, image 4. If values are correct, modify part program.
123 CHUCK $1 523 BARRIER $2	The value of the protected chuck zone set by parameters has been reached or exceeded.	Press RESET. If parameter values are incorrect, modify them and switch control system off.
124 RESET 524	RESET status.	Clear RESET status.
130 POSITION ERROR $1 530 OVER X $2	Trailing error (traverse rate too high and/or servo time delay too long).	Check cable and screw clamps and replace, if necessary. CHECK, image 2 – check servo constants.
131 POSITION ERROR $1 531 OVER Z $2	Cable breakage between NC and servo unit.	Exchange servo amplifier. Check feed-back unit or exchange it.
132 POSITION ERROR OVER C $1	Cable breakage between control circuit print (FX 31/61) and feed-back unit.	Check direction of motor, tachogenerator and resolver.
133 POSITION ERROR OVER 4	Wrong servo constants as parameter values.	
134 POSITION ERROR OVER 5	Wrong direction of rotation of motor with regard to traverse command. Loose connection between motor and recirculating ball screw.	
140 NO FEED BACK $1 540 SIGNAL X $2	No output signal from resolver.	Check resolver output signal.
	Cable breakage between feed-back unit and control prints (FX 61/26).	Check cable. Check connection for secure seating.
141 NO FEED BACK $1 541 SIGNAL Z $2		Exchange feed-back unit.

14.

Display	Cause	Rectification
142 NO FEED BACK 542 SIGNAL C		
143 NO FEED BACK 543 SIGNAL 4		
144 NO FEED BACK 544 SIGNAL 5		

NC FAULT DISPLAYS: TX 8 D		TRAUB

Display	Cause	Rectification
150 DRIVE UNIT $ 1 550 ALARM X $ 2	Motor circuit breaker or fuse of current supply faulty.	Check fuse. Check motor circuit braker.
151 DRIVE UNIT $ 1 551 ALARM Z $ 2	Excess transformer temperature.	Check alarm displays (LED) on servo amplifiers (for description of alarm status conditions refer to descrip-tion of servo amplifier faults).
152 DRIVE UNIT 552 ALARM C		
153 DRIVE UNIT 553 ALARM 4		
154 DRIVE UNIT 554 ALARM 5		
201 PROGRAM FILE 601 ERROR (M2/M30)	Program start cannot be found with SEARCH or END OF PROGRAM M2/M30.	Check part program. Has M30 been programmed for both slides? Check SET UP (activated part program).
202 PROGRAM 602 DESTROYED	Reading fault during auto-start and skip back to program start. M2/M30 illegible.	Check part program. Reload part program. Exchange print FX 84-7 if the same fault occurs with another program.
203 PROGRAM NOT 603 FOUND	Program cannot be found with external program call-up.	Check whether the correspond-ing program exists. Reload part program. Compare external call-up with corresponding part program number.
204 PROGRAM 604 DESTROYED (SEARCH)	Reading fault in case of external program call-up.	Check part program. Compare call-up with cor-responding part program number. Change print FX 84-7.
205 NO EXT. 605 SEARCH FUNCTION	External program search function is not set.	Set external search function in CHECK, image 6. Switch NC OFF/ON. Repeat search function process.

JULY 1985	Ref. 636 702	K.1-6

16

TRAUB

Display	Cause	Rectification
206 SPINDLE 606 LOWER CLAMP	The spindle speed is limited by the programmed lower min. value.	Only an information; the program sequence is not interrupted.
207 SPINDLE 607 UPPER CLAMP	The spindle speed is limited to the programmed upper value.	The upper limiting speed is displayed in SET UP, image 5. Lower limiting speed: see SET UP (5).
208 SPINDLE 608 UPPER OVER	Actual spindle speed exceeds limiting speed determined by parameter.	Actual spindle speed can be read in MONITOR, image 2. The admissible spindle speed tolerance can be set from 0 - 20 % in CHECK, image 4.
209 SPINDLE 609 LOWER OVER	Actual spindle speed is below limiting speed determined by parameter.	This alarm always appears with spindle speed START/STOP.
210 WAITING FOR 610 SYNCHRO	Slide 1 or slide 2 waits for synchronization mark of appropriate counter-slide.	Only an information; the program sequence is not interrupted. Check synchronization mark ! in part program.
211 OVERRIDE 611 ZERO	In automatic or manual mode the feed switch has been set to zero.	Set override (feed) switch to desired value.
212 NOT AUTO START 612 STATUS	A program that has the same program number as a program to be started is processed in the editor.	Allocate a different number to the program to be edited.
213 RST OR AUTOM. 613 NECESSARY	During data transfer from NC memory to GPS, the NC is not in RESET status or in automatic mode.	Check mode of operation. Press RESET and start transfer once more.

17

Display	Cause	Rectification
214 NEED R.P. 614 RETURN (GPS)	Prior to data transfer to the GPS, the reference points must be approached.	Approach reference points.
215 RST OR SETUP 615 AFTER EDIT	The program indicator is not at the beginning of the program. No actual program present.	Select required program by RESET or in SET UP status.
216 SERIAL OUT 616 ERROR (G.P.S.)	A wrong start/stop position has occured during data transfer. The interface unit has not been switched on.	Switch unit on. Exchange print FX 27.
217 NEED RESET 617 MEMORY (EXT)	Not in memory (automatic) mode. Not in RESET status.	Establish RESET status. Select automatic mode.
218 ILLEGAL PROG. 618 NO. (EXT)	Program number not acceptable with external program call-up.	Program number called up is not in work memory. Unacceptable program number from PC program present.
219 SEARCH $1 / $2 619 NOT FINISHED	Slide 1 or 2 have not been started.	Selection of both slides (1 + 2) in automatic mode.
220 FEED LIMIT	Programmed feed value exceeds rapid traverse or parameter value of max. feed.	Check programmed feed value.
223 SERVO OFF START ERROR	Axis is started while it is still in SERVO OFF status.	Check part program.
226 FEED LIMIT MILL.	Programmed feed value exceeds rapid traverse or parameter value of max. feed for milling.	Check programmed feed value.
301 NEED R.P. 701 RETURN	Reference point has not been approached.	Approach reference point.

18

Display	Cause	Rectification
302 ILLEGAL 702 G COMMAND	G command is not in guidance pattern.	Modify part program accordingly.
303 SAME ADDRESS 703 IN A BLOCK	The same address appears several times in a single block (more than 3 times with M-FCT).	Modify part program accordingly.

17

Display	Cause	Rectification
304 ILLEGAL ADDRESS 704 CODE	D or R has been programmed without geometry parameter.	Check OPTIONS parameters: e.g. geometry option 1 geometry option 2
	C traverse in a block with D or R command.	
	Angle program (A) without geometry parameter.	Block format
	The block following a geometry block with A has a format that differs from G1, G2 or G3.	Block sequence
	The following block sequence is unacceptable: G1 ...X ...D or R G1 Z ...A G2/G3	
	A block with an angle statement is followed by a format that differs from the angle statement or G1.	
	A G1 radius or chamfer is followed by another format than G1.	
	An arithmetical function actually used is not available.	
	Unacceptable address code with macro programming.	
305 LARGE DATA 705 VALUE	The data input have too many digits.	Reduce the number of digits.
306 OVER DIGIT 706 BELOW DECIMAL	Too many digits have been programmed after the decimal point.	Modify part program accordingly.
307 PROGRAM END 707 WITHOUT M2/M30	No M2 or M30 at end of program.	Modify part program accordingly.
308 ILLEGAL VARIABLE 708 DATA	The variables used for parameter programming are not acceptable.	Use acceptable variable format in part program.
309 TOOL LIFE 709 OVER	Tool life time elapsed.	Only indication. The part program will be processed and the last part will be activated by the PC program. Reactivate tool life by SET UP, image 2.

Display	Cause	Rectification
310 END POINT NOT 710 FOUND G01	An incremental value has been programmed after a block with angle programming (A). No point of intersection between angle and straight line. Intersection point shift during tool radius compensation and tangential intersection point calculation between angle and radius.	Modify part program accordingly.
311 G01 NEED 711 (X, Z, A)	After a block including an angle indication (A), there is no X or Z value. After a block with an angle indication only a G01 block without A format is programmed. After a block with an angle indication and unknown end point, no G01 block with angle (A) is programmed. The following (second) geometry block has neither X nor Z direction. The following (second) geometry block has no angle indication (A).	Modify part program accordingly.
312 TWO LINES HAVE 712 SAME ANGLE	Two straight lines are parallel and have no intersection.	Modify part program accordingly.
313 CORNER LENGTH > 713　LINE	The resulting chamfer is larger than the straight line.	Modify part program accordingly; see section transition chamfer/radius.
314 ILLEGAL TWO 714 ARC COMMANDS	The intersection between two blocks with angle indication cannot be calculated. The second geometry block has been programmed with incremental dimensions.	Modify part program acccordingly.

21

Display	Cause	Rectification
315 COEXIST 715 R AND I/K COMMAND	In a tangential intersec- tion point calculation, R and P or Q have been pro- grammed. With tangential transition straight line/angle, neither R nor P and Q have been programmed. I, K and R have been pro- grammed in an angle indi- cation. I/K and P/Q have been used in angle programming. R and P/Q have been used in angle programming.	Modify part program accordingly.
316 END POINT ERROR 716 G02/G03	End point error with G02/G03.	SET UP, image 6 – check parameter value (end point error). Check and correct part program.
317 T COMMAND 717 IN G02/G03	A "T" command was pro- grammed in circular arc block format G02/G03.	Input "T" call-up at another point in the part program.
318 OVER 8 TIMES 718 NESTING	Sub-routine G22 contains more than 8 loops.	Modify part program accordingly.
319 NO A AND P 719 IN G22/G71-73	"A" or "P" address is missing in G22, 71, 72 or 73 call-up.	Modify part program accordingly.
320 START BLOCK 720 NOT FOUND	The start address for finished contour is missing or cannot be found with G22, G71, G72 or G73.	Compare "A" or "P" address with start address of finished contour. Compare sub-routine call-up with "A" or "P" format. Modify part program accordingly.
321 RETURN BLOCK 721 Q NOT FOUND	End address "Q" not pro- grammed after "P" in G22, G71, G72 or G73. End/skip-back address in finished contour missing.	Compare "A" or "Q" address with appropriate end of part program (finished contour).

Display	Cause	Rectification
322 NO MDI DATA 722	No MDI data input	Input MDI data.
323 ILLEGAL COMMAND 723 IN MDI	Unacceptable program commands in MDI mode, for ex. G22, macro commands such as GOTO, DO, WHILE, etc.	Carry out these commands in automatic mode.
324 MACRO DATA 724 OVERFLOW	In instruction format of user macro, errors have occured with arithmetical operations.	Check format of user macro instructions and modify accordingly.
325 MACRO FORMAT 725 ERROR	User macro includes a format error.	Modify format in part program accordingly.
326 U < I, 726 W < K, (UWIK) = 0 (G74/75)	Cutting depth greater than acceptable or ZERO.	Modify depth of cut in G74/75 cycle.
327 I < D OR 727 K < D (G74/75)	Chip depth greater than programmed stock removal.	Compare chip depth with value for stock removal and modify accordingly.
328 A > 90, 728 K < D, (U/W/H) = 0	During threading, number of cuts is ZERO. Cutting angle exceeds 90 degrees. Thread length is ZERO. Depth of cut greater than thread depth.	Modify part program accordingly (programming instructions).
329 ILLEGAL AXIS 729 IN DRILL	No X or Z value programmed. X and Z value programmed for both axes.	Check drilling cycle format and modify part program accordingly.
330 D > X/Z 730 IN DRILL	Drilling depth greater than total value programmed for hole.	Modify part program in accordance with programming instructions
331 TOOL CHANGE 731 P. X=Z=0	Tool changing point in X or/and Z direction = 0.	Modify parameter value. Check part program.

23

NC FAULT DISPLAYS: TX & D   TRAUB

Display	Cause	Rectification
332 F COMMAND = 0 732	No feed rate programmed.	With DRY RUN (test run) only warning. State feed rate in program.
333 X = 0 733 AT V-COMMAND	The X value is ZERO with V programming in G97.	With V programming in G97, an X value is required (G97 X... V...).
334 MULTI NEED 734 CANNED CYCLE	G71/72/73 has to be programmed before G81 call-up (multiple cycle).	See programming instructions, G81 cycle.
335 S<Q IN SPINDLE 735 SPEED SET	The minimum spindle speed limit through G92 is less than the minimum speed set in parameter.	Modify "S" or "Q" value in part program.
336 PROGRAM IS BEING 736 OPERATED	The program called up is already being processed.	Warning only. In EDIT mode, call up a different program number.
337 737	free	
338 TAP CYCLE 738 FORMAT ERROR	Format error in deep-hole drilling cycle.	See programming instructions for deep-hole drilling cycle.
340 R-COMP. NO 740 CROSS P.(L-L)	Two straight lines have no intersection when applying tool radius compensation.	See section "Tool radius compensation" in programming instructions.
341 R-COMP. NO 741 CROSS P.(C-L)	No intersection between straight line and angle indication.	Modify part program accordingly.
342 R-COMP. NO 742 CROSS P.(C-C)	No intersection present with 2 angle indications.	
343 R-COMP. BOUNDING 743 DIRECT.	Direction of compensation with tool radius compensation is limiting direction.	
344 R-COMP. SIDE 744 REVERSED	Direction of tool radius compensation inverted.	
345 R-COMP. ILLEGAL 745 TIP POINT	A wrong quadrant has been allocated to the tool (1-8).	

Display	Cause	Rectification
346 R-COMP. START 746 IN GO2/03	G46 directly precedes GO2/GO3 block.	
347 R-COMP.2 747 MOTIONLESS BLOCK	Three or more blocks with- out traverse contained in program section with tool radius compensation (G46).	
348 R-COMP. TOO 748 SMALL GO1	Traverse programmed in GO1 following a GOO block is too small.	
349 R-COMP. TRI- 749 ANGLE PATH	A GO1 loop results in a quadrant change.	
350 NEED GOO 750 BEFORE G40	Before deleting tool radius compensation by G40, a GOO block must be programmed. No temporary deletion of tool radius compensation.	
351 R-COMP. T-COMMAND 751 COMES	"T" command while executing tool radius compensation.	
352 R-COMP. 1 BLOCK 752 SKIPPED	First block is deleted while tool radius compen- sation is processed.	Warning only. Processing of part program continues.
353 R-COMP. ILLEGAL 753 G-COMMAND	Unacceptable "G" function programmed within tool radius compensation pro- gram.	Use only acceptable "G" commands. Modify part program accord- ingly.
354 D=0 IN 754 G71/72/73	Depth of cut in stock removal cycles is ZERO.	Programming instructions, section stock removal cycles.
355 ILLEGAL (G) 755 IN FINISHING	Wrong "G" command pro- grammed in finished contour when using G71/72/73.	Modify part program accord- ingly.
356 OVER BLOCKS IN 756 FINISHING	When using stock removal cycles, finished contour has more than 50 blocks.	
357 FORMAT ERROR IN 757 FINISHING	When using stock removal (roughing) cycles G71-73, finished contour contains a format error.	

25

Display	Cause	Rectification
358 (U-I), (W-K) 758 < D IN G71-73	Depth of cut per pass greater than total depth of cut.	See programming instructions stock removal (roughing) cycles.
359 ERROR FINISHED 759 CONTOUR IN G91		
360 FINISH PROGR. 760 ERR IN G71	Wrong finished contour programmed for stock removal (roughing) cycle G71-73.	See programming instructions stock removal (roughing) cycles.
361 E> F IN G71/ 761 G72/G73	"E" command greater than "F" command with G71 to G73 command.	In rough turning cycle, set smaller "E" command than "F" command.
367 FORMAT ERROR 767 IN G79	G79 hole depth = ZERO. First cut = ZERO. X and Z are either both present or both absent.	See programming instructions G79 cycle.
368 W< I/K/I+K, I > A IN G79	Value absent or incorrect.	See programming instructions.
369 NEED M3/M4 769 COMMAND	No M3/M4 before G82 cycle command.	See programming instructions G82 cycle.
370 NEED AXIS 770 DATA	Corresponding axis data (traverse movements) missing in second geometry function.	See programming instructions second geometry function.
371 ANGLE IS NOT 771 CORRECT	Angle indication with regard to line has incorrect sign.	See programming instructions angle definition with geometry application.
372 MIDDLE POINT 772 IS NOT FOUND	No centre is found in geometry function.	See programming instructions geometry.
373 G02/G03 BECOME 773 CIRCLE	In case of G02 or G03 programming, a full circle is obtained.	Modify part program accordingly.
374 NOT SAME 774 DIRECTION	Monotonous increase or decrease not selected as parameter value.	Check parameter.

Display	Cause	Rectification
377 3 AXES LOCK 777 ERROR (COCOM)	3-D interpolation not permitted (COCOM).	Modify part program accordingly.
378 SERVO OFF 778 COMMAND ERROR	Control loop not closed C/R axis.	Modify part program accordingly.
379 NEED SEARCH 779 OR RESET	Automatic start key has been pressed without having first activated a program.	Activate program.
380 ERR RETURN 780 (1)	Display or error 1 in user macro program.	Check user macro part program.
381 ERR RETURN 781 (2)	Display or error 2 in user macro program.	These faults can be defined in user macro program.
382 ERR RETURN 782 (3)	Display error 3.	
383 ERR RETURN 783 (4)	Display error 4.	
384 ERR RETURN 784 (5)	Display error 5.	
385 ERR RETURN 785 (6)	Display error 6.	
386 ERR RETURN 786 (7)	Display error 7.	
387 ERR RETURN 787 (8)	Display error 8.	Fault displays can be freely defined.
388 ERR RETURN 788 (9)	Display error 9.	
389 ERR RETURN 789 (10)	Display or error 10 in a macro program defined by user.	
390 ILLEGAL "G" 790 CODE (MILL)	Incorrect "G" code during milling mode.	Check part program.
391 AXIS UNFIT 791 PLANE (MILL)	Plane selected in milling mode does not correspond to axis addresses.	Check part program.

27

Display	Cause	Rectification
392 CANCEL SHIFT X 792 (MILL)	When switching to milling mode there was still a co-ordinate shift in X axis.	Select coordinate shift be-fore switching to milling mode.
393 CANCEL R/C 793 COMP. (MILL)	No deletion of radius com-pensation before switching to milling mode.	Delete radius compensation by G40.
394 ILLEGAL AXIS 794 (MILL)	Programming of C, 4th and 5th axis not allowed in milling operation.	Check part program.
399 NEED R.P. 799 RETURN (MILL)	No reference point after switching to milling mode.	In milling mode, first ap-proach reference point.
801 INPUT 901 ERROR	Incorrect input.	See programming/operating instructions.
802 MAX/MIN 902 DATA ERROR	Too many digits behind de-cimal point. Incorrect use of decimal point. Data values too small/too great.	See programming/operating instructions.
803 KEY IS NOT 903 ACCEPTABLE	Key is not allowed in this operation.	See programming/operating instructions.
804 OPERATION 904 LOCKED	Key is not allowed in this operation.	See programming/operating instructions.
805 09000-09999 905 LOCKED	TRAUB sub-routines are locked.	If modification is desired, contact TRAUB.
806 NEED R.P. 906 RETURN	Reference point is re-quired in TEACH-IN.	See operating instructions.
807 NEED TEACH-IN 907 POSITION	Machine position has not been stored with TEACH-IN.	Memorize machine position with TEACH-IN before input of measuring values.
808 MAX. DATA 908 OVER (TEACH-IN)	Data value determined by TEACH-IN exceeds admissible value.	See operating instructions, try again.
810 SEARCH NOT 910 FOUND	Program to be called up is not found.	- Store this part program. - Correct wrong program number.
811 SEARCH NEED 911 RESET	RESET has not been pressed prior to search run.	RESET.

28

NC FAULT DISPLAYS: TX 8 D		TRAUB

Display	Cause	Rectification
815 SEARCH NOT 915 FOUND (EDIT)	Character sequence not found: - during search - during replacement.	Check character sequence.
816 NUMBER 916 ERROR	Number to be deleted or replaced is not acceptable.	Check the number.
818 OVER 8 BLOCKS 918	"Nested" more than 8 times in G22.	Correct part program.
820 ILLEGAL 920 PROGRAM NO.	Program number is faulty (parallel input).	Check and correct program number accordingly.
821 NO PROGRAM 921 NO.	Program number not stated during input via punched tape (parallel interface).	State program number during input in IN/OUT image.
822 OVER PROGRAM 922 AREA	Existing memory capacity has been exceeded.	Clear memory or punch different programs and then delete.
823 OVER NUMBER 923 OF PROGRAMS	Quantity of program numbers has been exceeded.	Output and deletion of programs that are not used.
824 SAME PROGRAM 924 NO. STORED	A program with the same number has already been stored in the work memory.	Check/change program number and store again.
825 PROGRAM NOT 925 FOUND	Program to be punched is not in work memory.	Check work memory. Check program number in OUT.
826 OVER PROGRAM 926 AREA (COPY)	Capacity of program memory exceeded during copying of program.	Check and reduce part program. Restart copying.
827 OVER 64 CH 927 IN A BLOCK	Block contains more than 64 characters.	Check part program.
828 ADDRESS PATTERN 928 ERROR	Address character used does not exist in pattern.	Check address data.

Display	Cause	Rectification
829 ILLEGAL 929 G-CODE	"G" command not included in specifications.	Correct part program and look up in programming instructions.
830 WAIT FOR I/O 939 OR EDIT END	Program cannot been edited yet.	Wait until program is completely read in or press RESET.
840 ILLEGAL CODE 940	Horizontal parity is incorrect.	Check whether format is allowed. Check punched tape reader.
841 PARITY V 941 ERROR	Vertical parity has been set.	Check whether format is allowed. Check number of characters per block. Check punched tape reader.
842 PARA/TOOL 942 FORMAT ERROR	Format error in parameter data or tool data.	Check parameter data or tool data and correct.
843 MAX./MIN. 943 DATA ERROR	(serial input) Data values input too great/too small. Observe admissible format.	Check values programmed in punched tape.
844 DATA IN IS 944 NOT ACCEPTED	Data input (program blocks, tool data and parameters) blocked in automatic mode. Work memory defective or data in actual part program have been modified during machining.	Select admissible operating mode.
845 TAPE COMPARE 945 ERROR	Data entered do not correspond to memory contents.	Incorrect contents shown on CRT.
846 $ NO. IS NOT 946 PROGRAMMED	(serial input) Tool data have not been allocated to slide.	Only warning - program is read in as for one slide.
847 ILLEGAL $ NO. 947	(serial input) Allocated tool address is not acceptable.	Check part program and correct, if necessary.
848 PARA/TOOL 948 ILLEGAL PAGE	(serial input) Allocation of parameter data to corresponding signification L ... = barrier	Check punched tape format.

30

NC FAULT DISPLAYS: TX 8 D		CRAUB

Display	Cause	Rectification
	points is not correct.	Correct parameter format in punched tape accordingly.
849 WRONG BLOCK 949 DATA	Addresses have been used that are not included in the menu.	Check part program.
850 PTR I/O 950 ERROR	Reader is not switched on. Reader is not ready to operate. Monitoring time of reader exceeded.	Switch on reader. Check reader. EQR required as last character - load punched tape end of block.
851 PTP I/O 951 ERROR	Punch is not switched on. Punch is not ready to operate.	Switch on punch. Check punch.
852 PTR TIME 952 EXCEEDED	Serial input/output unit has not given a signal within a certain time.	Check listing or parameter or input/output unit.
860 I/O ERROR 960 (S-IN)	Serial input error. No supply voltage on unit.	Check unit and switch it on.
861 I/O ERROR 961 (S-OUT)	Serial output error. No supply voltage on unit.	Check and/or switch on unit.
870 CMT I/O ERROR 970 (IN)	CMT input I/O error.	Check cassette.
871 CMT I/O ERROR 971 (OUT)	CMT output I/O error.	Check cassette.
872 CMT CHECK 972 SUM ERROR	CMT check sum error.	Try again.
873 CMT CRC ERROR 973 PROTECT	CMT CRC error (I/O).	Replace cassette tape.
874 CMT WRITE 974 PROTECT	CMT write protect error (CMT tabs have been broken out).	Replace by cassette whose tabs are intact.
875 END OF CMT 975 PROTECT	End of cassette.	Exchange cassette.