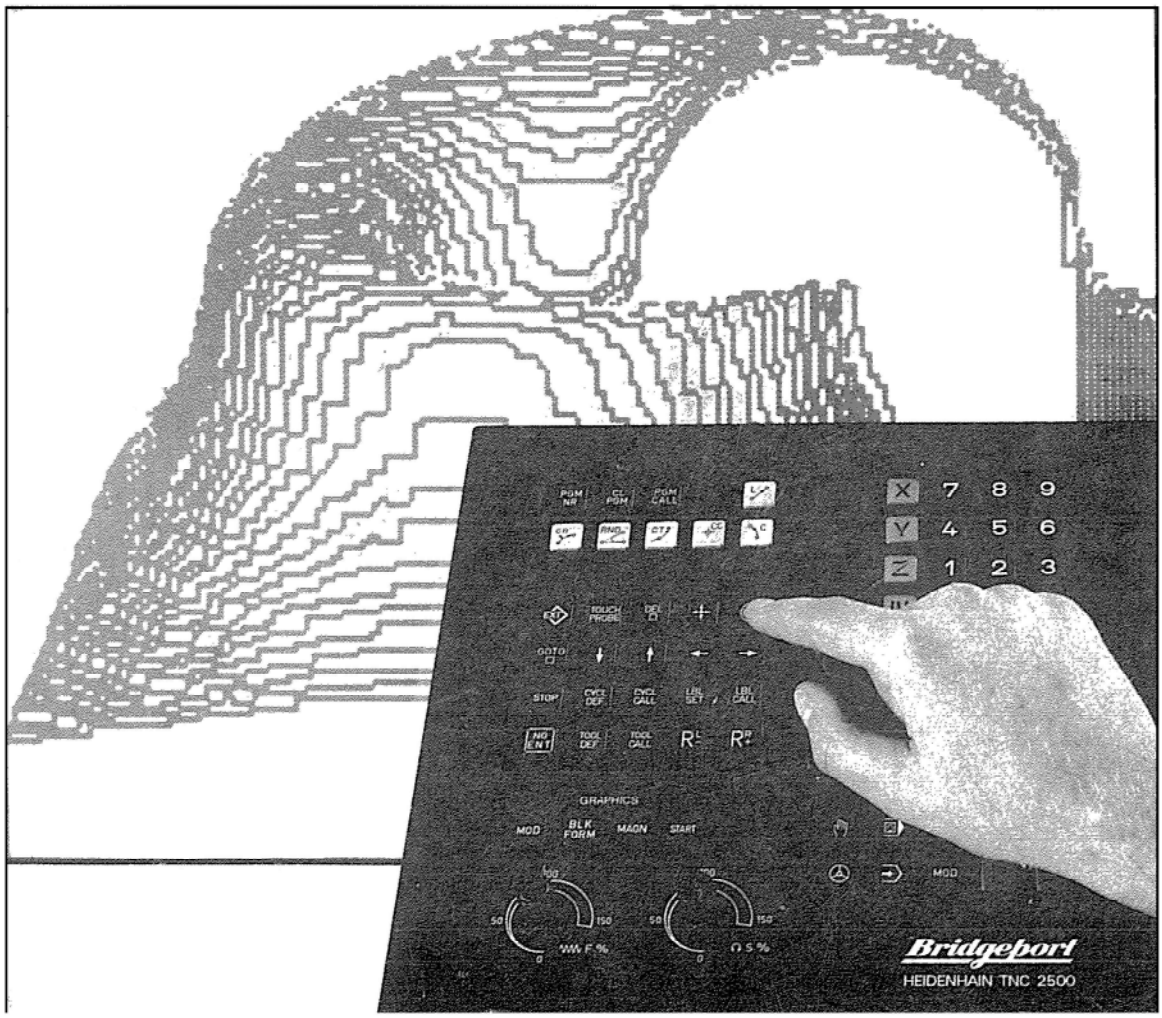
HEIDENHAIN

Operating Manual
Programming in Plain Language

HEIDENHAIN TNC 2500
Contouring Control



December '89

Contents

General	Introduction	A1
	Operating modes and displays	A4
	Workpiece coordinates	A6
	Machine axes	A8
	Cutting data	A11
Setting-up	Switch-on	E1
	Manual mode	E2
	Electronic handwheel, jog	E6
	Positioning with manual data input	E8
	3D Touch probe	E11
Programming in plain language	Program/The plain language concept	K1
	Dialogue/Editing	K5
	Tools	K10
	Tool path compensation	K14
	Auxiliary and supplementary functions	K20
	Movements in Cartesian coordinates	K22
	Movements in Polar coordinates	K43
	Approach to/Departure from a contour	K50
	Special behaviour of compensation *	K59
	Teach-in *	K66
	Program jumps *	K68
	Jumps within a program	K69
	Program call	K75
	Standard cycles, Introduction, Overview	K77
	– Machining cycles	K78
	– Variable-contour pockets	K88
	– Coordinate transformations *	K113
	– other cycles *	K122
	Parametric programming *	K125
Program run	Program tests	L1
	Graphic simulation	L2
	Automatic, Single block	L10
	– Faults, Interruptions *	L11
	– Background programming *	L15
	– Transfer blockwise *	L16
Supplementary information *	General displays, Settings	Z1
	Traversing ranges	Z3
	External data transmission	Z4
	User-parameters	Z14
	Error messages	Z16
	Technical data	Z17
	Accessories	Z18

* These topics can be disregarded when working through this manual for the first time.