HEIDENHAIN

DataPilot 4110

CNC Programming and
Training on a PC

DataPilot 4110 –
Programming and Storing on a PC

DataPilot 4110 is the PC programming station for the MANUALplus 4110 and the organization system for the workshop and design office. By shifting the programming, program testing, and program optimization to the PC, machine idle times are greatly reduced.

Having the same conditions and same functions as the MANUALplus contouring control gives you the assurance that a program created with DataPilot can be run on the machine immediately.
That is why DataPilot 4110 is the ideal supplement to the control for program creation, archiving, and apprentice and advanced training.

Creating programs
Programming, testing and optimizing cycle programs and DIN programs with DataPilot on your PC substantially reduces idle machine times. You do not need to adjust your way of thinking, since you program and test with DataPilot in exactly the same way as on the lathe. DataPilot has the same software as the control. This ensures that a program created with DataPilot can be run on the machine immediately.

Storing programs
Even though the MANUALplus has a nearly inexhaustible memory capacity, you should also back up your programs on an external system. The MANUALplus has both an Ethernet and a serial RS-232-C interface. This enables you to integrate the MANUALplus into your existing network or to connect the DataPilot PC directly to the control.
Convenient program-transfer functions support both external programming and archiving on the DataPilot PC.

Training with DataPilot 4110
Because DataPilot 4110 is based on the same software as the MANUALplus, it is ideally suited for apprentice and advanced training. Programming and program testing on the DataPilot PC function exactly the same as they do on the machine. Even setup functions such as workpiece datum definition, tool measurement or machining of individual cycles, cycle programs or DIN programs are simulated with DataPilot. This gives the trainee the experience needed for safe operation of the machine.
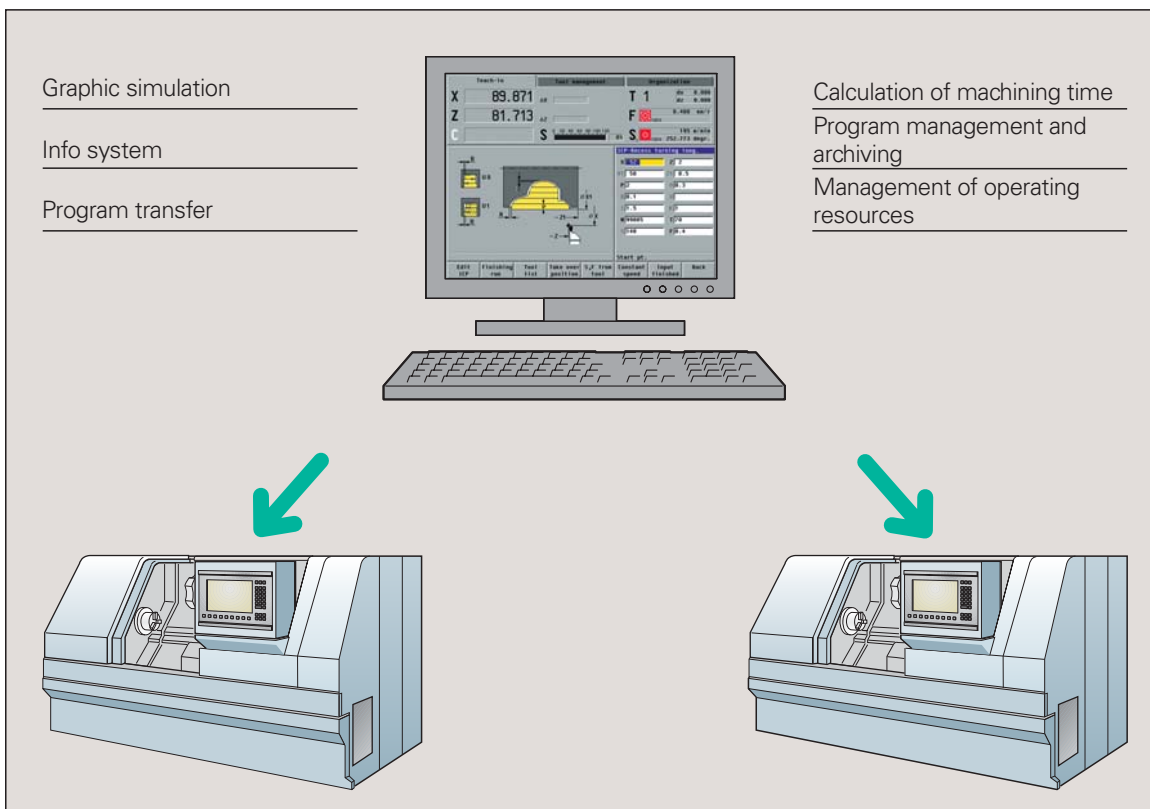
Working as you do on a lathe

The operation and programming of the DataPilot 4110 is identical with that of an NC machine. DataPilot 4110 even simulates axis and spindle movements and the execution of NC programs. Thus you can train students how to operate a lathe and familiarize them with NC programming for workpiece machining on a readily available PC.

Conversational languages

You can install DataPilot 4110 with several national languages and switch among them during operation of DataPilot. The following languages are available:

- Danish
- German
- English
- Finnish
- French
- Italian
- Dutch

- Polish
- Russian
- Spanish
- Swedish
- Czech
- Hungarian

Graphic simulation

Info system

Program transfer

Calculation of machining time

Program management and archiving

Management of operating resources

System requirements

DataPilot runs on PCs with one of the following operating systems:

- Windows 98 SE
- Windows ME
- Windows 2000
- Windows XP

A USB interface is required for connecting the software-enabling dongle. An Ethernet interface enables fast data exchange with CNC machines.

Depending on the installed languages, DataPilot requires approx. 30 to 50 MB of hard disk space.

Fast Creation of NC Programs Using DataPilot

DataPilot 4110 offers the same functions as the MANUALplus 4110 contouring control. This way you can program as quickly and reliably as on the machine—without having to relearn.

Cycle programming

Recurring operations such as area clearance, slot recessing, recess turning, undercutting, parting, thread cutting, milling and drilling are stored as fixed cycles. You only need to enter the desired positions, dimensions and values—the rest is automatic.

Available cycles include:
- Standard and ICP roughing
- Standard and ICP recessing
- Standard and ICP recess turning
- Thread cutting (single and multiple threads, longitudinal, transverse, and taper threads, undercuttings)
- Drilling cycles (drilling, deep-hole drilling, tapping, axial and radial hole patterns)
- Milling cycles for axial and radial contours and pocket milling (figures, ICP contours)
- Slot-milling cycles (axial and radial, helical), surface milling, centric polygon milling, thread milling

Interactive Contour Programming (ICP)

The ICP Interactive Contour Programming feature assists you in performing machining tasks for which standard cycles cannot be used due to missing data or highly compex contours. In most cases you can desribe the contour elements with the dimensions given in the production drawing.

The contours are defined using linear and circular contour elements as well as form elements like chamfers, roundings, and undercuts. DataPilot 4110 calculates missing data. If the entered data can result in several mathematically possible solutions, you are shown the various solutions for selection.

DXF import of contours

Why should you still painstakingly enter contour elements when the data already exists in the CAD system? ICP makes it possible to import contours in DXF format directly into DataPilot 4110. This doesn't just eliminate time consumed by programming and testing, but you can also be sure that the finished contour is exactly according to the designer's specifications.

DXF contours can describe workpiece blanks, finished parts, contour trains and milling contours. They must exist as two-dimensional elements in a separate layer, i.e. without dimension lines, wraparound edges, etc. Since the DXF format is fundamentally different from the ICP format, the contour is converted from DXF to ICP format during the import. It can then be treated as a normal ICP contour, and is available for roughing, recessing and milling cycles.
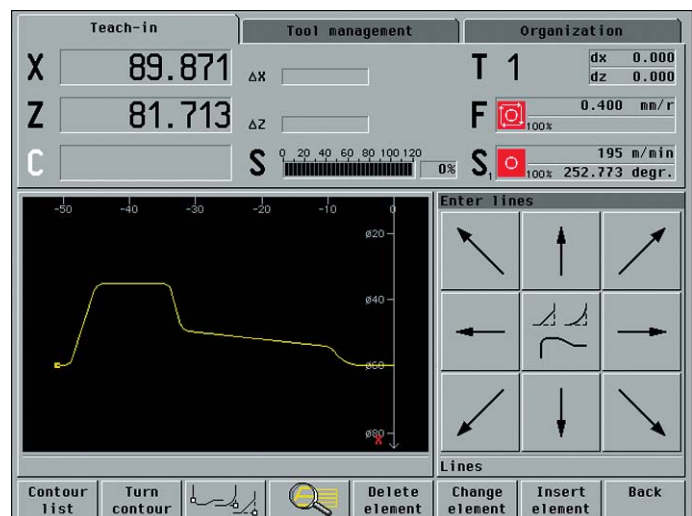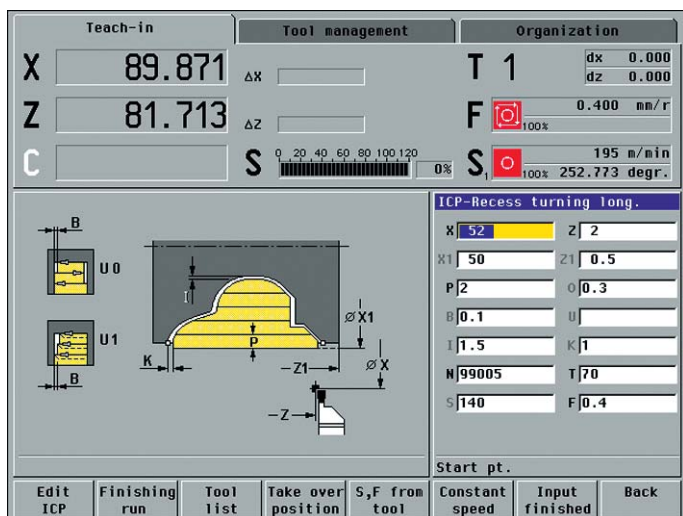
DIN programming

With the help of DIN programming, you can create independent DIN programs or DIN macros. Independent DIN programs contain all the necessary traversing and switching commands. DIN macros can be used for individual machining tasks within cycle programs.

The range of DIN commands includes:
• Traversing commands
• Cycles for roughing, recessing, recess turning, thread machining, milling and drilling
• Switching commands
• Mathematic operations for calculation of address parameters or variables
• Commands for program organization (branches, program-section repeats, subprograms)

DIN conversion

The transformation of a cycle program to a DIN program is referred to as DIN conversion. DIN conversion is often the most effective way to make a DIN program. You describe the contours with ICP, program the roughing, recessing and milling operations with ICP cycles, and define the drilling and thread machining with cycles as well.

DIN conversion provides you with an easy-to-read NC part program that you, the experienced NC programmer, can optimize. The graphic simulation and the simulated program execution of DataPilot 4110 enable you to analyze each individual cycle or every traverse movement.
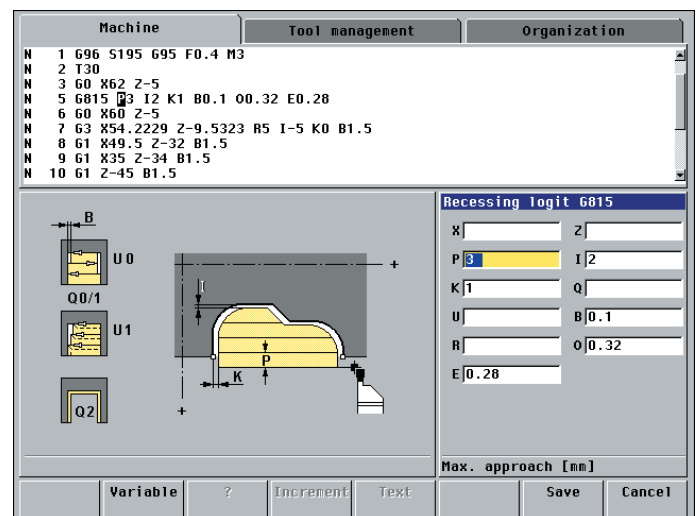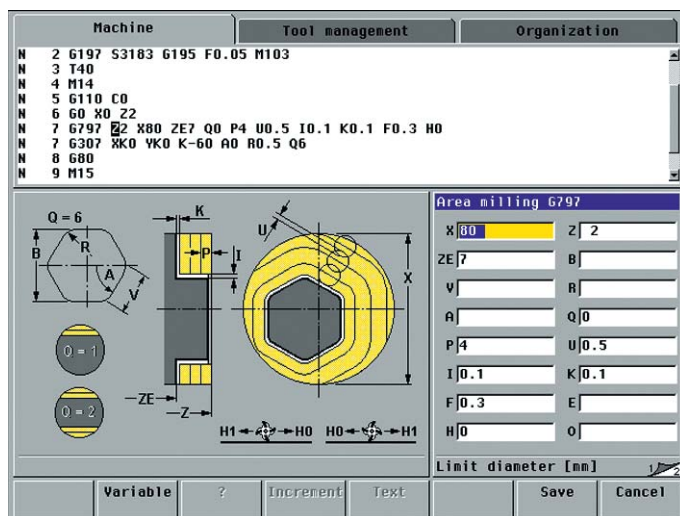
Program Simulation and Management with DataPilot

Graphic simulation for realistic testing before the machining process is especially useful at the programming station. DataPilot simultaneously determines the machining time, manages both programs and tools, and organizes the exchange of data, guaranteeing efficient production planning.

Graphic simulation and time calculation

The graphic simulation feature provides suitable functions for testing and optimizing cycle and DIN programs. In the case of cycle programs, the programmed contours and every traverse path are shown on your screen graphically. At the same time, the system displays infomation on the respective cycle or DIN command.

In addition, DataPilot 4110 shows the range machined by the tool. Thus it is possible to monitor the machining process exactly, especially for slopes, curves, recesses or bore holes.

You can inspect the milling and drilling operations with the C axis in the face or lateral view.

During simulation, DataPilot 4110 calculates the machining times and idle times. The times are displayed per cycle or tool use (in DIN programs) graphically or in tabular form.

Program and tool management, data exchange

DataPilot 4110 manages cycle programs, ICP contour definition programs, DIN programs and the tools individually for each lathe or every machine group.

Features such as listing programs by number or date, and copying, renaming or deleting programs, provide all the prerequisites for efficient program management.

You only need a few parameters to describe the tools used. Each tool is assigned a tool description (designation, identification number, etc.) to help you find it more easily. The tool parameters are managed in conjunction with the cutting data and tool life data.

DataPilot uses the commonly known Ethernet interface networks for data exchange. This allows you to integrate DataPilot into your company network and conveniently exchange data between the DataPilot PC and the MANUALplus 4110 control.

Graphic simulation
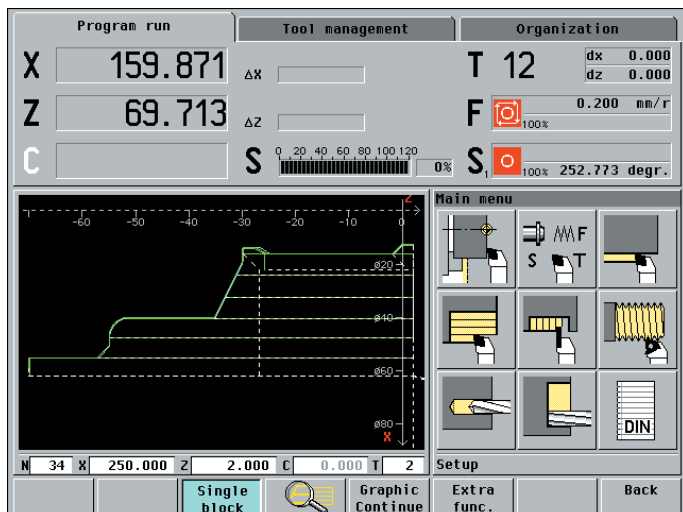
Tool management
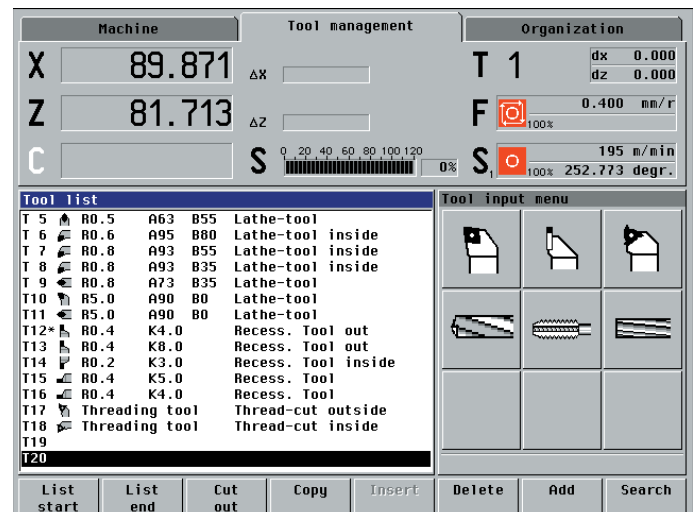
DataPilot for Practice-Oriented NC Training

The use of CNC-controlled machine tools, personal computers and IT systems in all areas of production requires a practice-oriented system for apprentice and advanced training. The modular setup of DataPilot makes it ideal for NC training.

With the realistic simulation feature, it is possible to graphically represent and monitor each programming step. While NC programs are running, the machine display shows the actual values, tool data, etc. The block display feature shows the commands of the cycle or NC part program. You can set the program execution so that program run stops after each cycle or DIN command or displays every positioning and switching command in DIN format (basic block display).

Thus you can convey to the trainee the whole programming sequence with ICP contour definition, cycle programming and DIN programming as well as the operation of a lathe and the execution of NC programs.

NC instructor

Printer

NC machine

Trainees