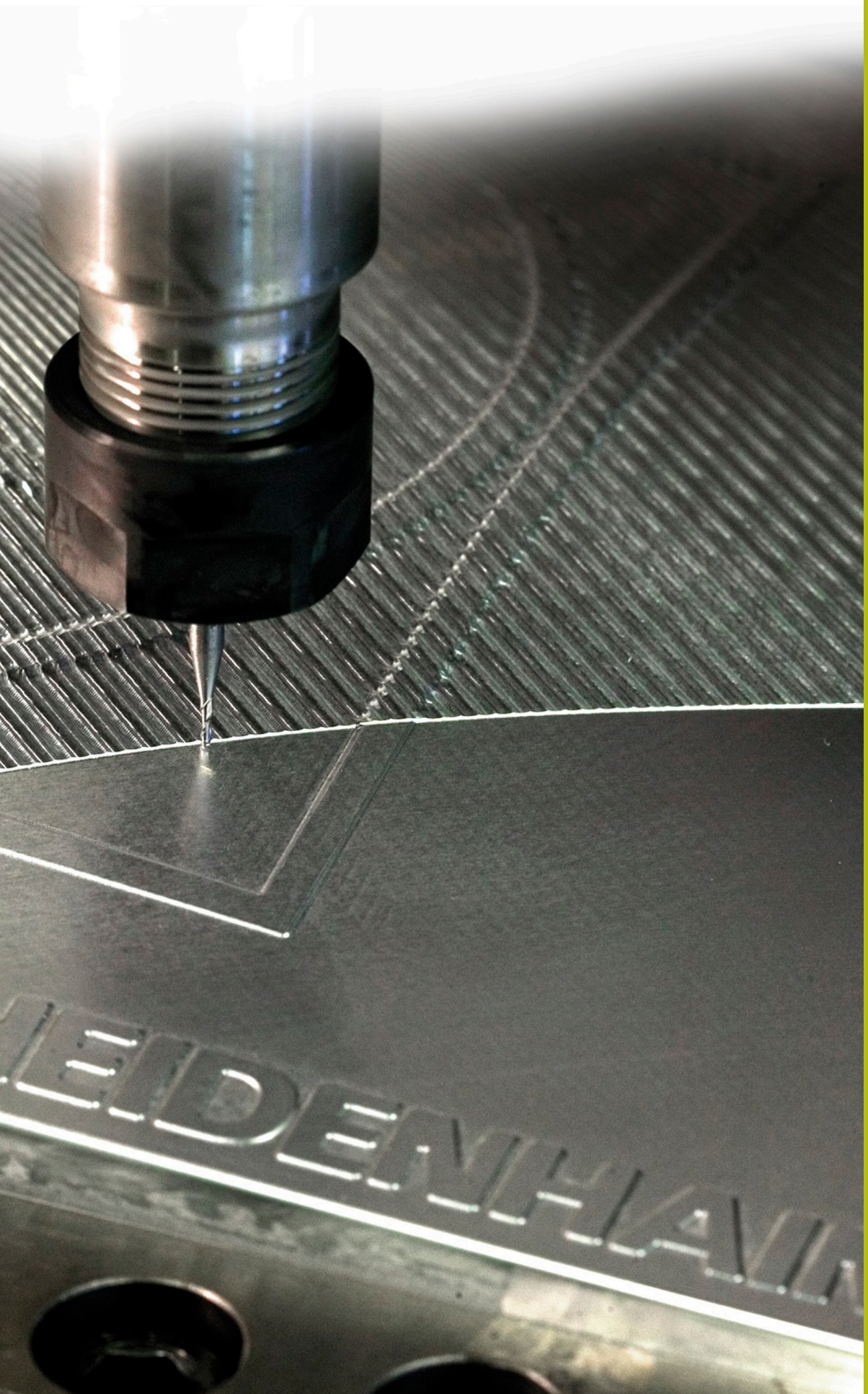
HEIDENHAIN



NC Solutions

English (en)
4/2017

1 Notes on using NC programs

Notes on downloading

The program name tells you what kind of NC program it is:

- Programs ending in **_en** include comments.
- Programs without the ending **_en** do not include comments, and for this reason alone can already run on TNCs older than the TNC 407.
- Some NC programs call additional files; e.g. **5010_en.h** calls the program **50101_en.h**. In this case download all files.

For some programs only simple changes are necessary in order to use them on older control models:

- Delete comments
- Resolve formulas
- Delete BLK FORM
- Replace cycles

Required modifications

NC programs published in the database are suggestions for solutions. The NC programs must be adapted before being used on a machine.

Modifications are always necessary for:

- Tools
- Cutting parameters
- Feed rates
- Clearance height
- Machine-specific positions (e.g. **M91**)
- Paths of program calls

Some NC programs depend on the machine kinematics. Adapt these NC programs to your machine kinematics before the first test run.

Tool data used

The tool data for milling programs come from the programming station with NC software version 340594-05.

For turning programs the tools used are shown in the turret list of the NC program.

You can download the tool list with the most important tool data as a PDF file. This information is also under the program description of every NC program.