New functions and customer benefits

1. CYCLE982 has been expanded for tool measuring with orientable tool carrier and for calibrating turning tools

1.1. Cycle CYCLE982: Measuring turning and milling tools, calibrating tool probes

Tool measuring cycle CYCLE982 is used for measuring tools on turning machines. This cycle provides the following measuring functions:
- Tool probe calibration
- Measuring tool lengths L1 and L2 for turning tools with cutting edge positions 1 to 8,
- Measuring the tool lengths of millers and drills on turning machines
- and, for millers, also radius measurement.

New: Measuring cycle CYCLE982 has been expanded by a B and C axis (suitable for applications with orientable tool carrier) for tool measuring (turning, milling and drilling tools) on turning machines.

This software version allows the measurement of turning, milling and drilling tools for 90° multiple tool positions. To activate the new functions, set _CBIT[7]=1 (in the GUD6 block). The tolerance value for permissible angular positions is defined by a new parameter in the GUD6 block _CM[8] (preassigned with 0.5°).

In this parameter, the user can enter the "maximum admissible tolerance" of the position of the adjustable tool carrier (tool lengths L1, 2, 3) in respect of the configuration of the geometry axis, corresponding to the individual system! The tool carrier positions 0, 90, 180, 270° are supported!

New: Turning tools can now be calibrated using cutting edge positions 1 to 4 (previously only cutting edge position 3). From now on, calibration is therefore also possible on a counterspindle.
2. **Workpiece measurement expansion**

**New: CYCLE114 allows to offset tools used with adapter transformation.**
With the specific definition of the tool length to be offset in _KNUM, the offset is now performed in the transformed length for tools used with adapter transformation.

**New: Calibration status monitoring (for workpiece probes)**
Within the framework of workpiece measurement, the calibration status is monitored regarding the probe length with reference to the center point / end point (_CHBIT[14]) and probe type monoprobe. Only the workpieces calibrated can be measured. Otherwise, the following alarms are output:
61419 "Check probe ball calibration with regard to the center point/end point"
61420 "Check probe calibration with regard to the multi/monoprobe".

3. **Measuring in JOG expanded**

**New: Selection of offset targets to be displayed on the user interface**
In the commissioning screen, the operator can set the offset targets to be selected for zero offset during JOG measurement. He can also set the order in which they are displayed.

The following settings are possible:
- Basic reference
- G54-G57 and G505-G599
- Channel-specific basic zero offset
- Global basic zero offset
- “Measuring only”

When exiting the screenform via the vertical softkey “Accept data”, the data are integrated. The “Abort” softkey allows to exit the screenform without saving changes.
Compatibility

**CCU/NCU software:**
- SINUMERIK 810D/DE CCU: SW 06.05.36 or later
- SINUMERIK 840D/DE NCU: SW 06.05.36 or later
- NCK: 51.17.00 or later
- SINUMERIK 840Di/DiE: SW 03.02.00 or later

**HMI software:**
- HMI-Advanced: SW 7.3 or later

**840D sl software:**
- NCU 7x0.1: SW 1.4 or later
- with HMI-Embedded: SW 7.2 or later
- with ShopMill-HMI: SW 7.2 or later
- with ShopTurn-HMI: SW 7.2 or later

Supply

The measuring cycles described above cannot be ordered individually. They are either part of HMI-Advanced SW 7.3 or included in 840D sl Software Version 1.4. We would like to wish you a lot of success when using these new cycles!

We implement new software versions and patches according to customer requirements! No matter which cycle you select for your particular machining task, the measuring cycles from Siemens allow you to machine with the optimum efficiency!