Options and Accessories
For TNC Controls
Options and accessories for TNC controls

HEIDENHAIN controls are known for their complete range of features and extensive functionality. Thanks to a series of control options and accessories, they can also be optimally adapted to the given application. This brochure provides a broad overview of the available options and the most important hardware enhancements for the current control versions. Along with a detailed description of its functionality, each option or hardware enhancement contains a list of the software versions with which it is available.

Options are functions already integrated into the control, allowing you to tailor the TNC’s functionality to your specific needs at any time. Some options, however, must be adapted by the machine manufacturer. Options are conveniently enabled via a keyword.

HEIDENHAIN offers software as practical tools for applications external to the TNC, from supporting data transfer or creating a PLC program all the way to a complete programming station.

Hardware enhancements make work on the machine easier, faster, and more reliable. An electronic handwheel, for example, permits fine manual control, and a workpiece touch probe significantly reduces workpiece setup times.

The version schema was simplified, starting with NC software version 16:
• The publication period determines the version number.
• All control models of a publication period have the same version number.
• The version number of the programming stations corresponds to the version number of the NC software.

Table of contents

<table>
<thead>
<tr>
<th>Overview</th>
<th>Overview tables</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dynamic Precision</td>
<td>10</td>
</tr>
<tr>
<td>Dynamic Efficiency</td>
<td>11</td>
</tr>
<tr>
<td>Connected Machining</td>
<td>12</td>
</tr>
<tr>
<td>Digital Shop Floor</td>
<td>13</td>
</tr>
<tr>
<td>Options</td>
<td>Programming and operation</td>
</tr>
<tr>
<td>Machine accuracy</td>
<td>14</td>
</tr>
<tr>
<td>Machining functions</td>
<td>34</td>
</tr>
<tr>
<td>Communication</td>
<td>41</td>
</tr>
<tr>
<td>Adaptation to the machine</td>
<td>53</td>
</tr>
<tr>
<td>Software</td>
<td>56</td>
</tr>
<tr>
<td>Hardware enhancements</td>
<td>63</td>
</tr>
</tbody>
</table>

Please also note the page references in the overview tables.
### Overview

#### 20 Advanced Graphic Features
- Verification graphics, program run graphics
- Plan view
- Projection in three planes
- 3D view
- Detailed 3D view

#### 19 Advanced Programming Features
- Expanded programming functions
- Free contour programming
- Canned cycles
- Peck drilling, reaming, boring, counterboring, centering
- Milling internal and external threads
- Clearing level and oblique surfaces
- Multi-operation machining of straight and circular slots
- Complete machining of rectangular and circular pockets
- Circular and linear point patterns
- Contour train, contour pocket, including contour-parallel machining
- Integration of cycles created by machine manufacturer
- Engraving cycle: engrave text or numbers in a straight line or on an arc
- Contour slot with trochoidal milling

#### 17 Touch Probe Functions
- Probing cycles
- Workpiece misalignment compensation, preset setting
- Automatic tool and workpiece measurement
- Touch-probe input enabling for non-HEIDENHAIN systems

#### 9 Advanced Function Set 2
- Interpolation, linear in 5 axes
- 5-axis simultaneous machining
- 3D tool compensation via surface normal vectors
- Changing the swivel-head angle with the electronic handwheel (TCPM = Tool Center Point Management)
- Keeping the tool perpendicular to the contour
- Tool radius compensation perpendicular to the tool direction
- Manual traverse in the active tool-axis system

#### 8 Advanced Graphic Features
- Verification graphics, program run graphics
- Plan view
- Projection in three planes
- 3D view

### Machine Functions

#### Machining Functions
- Turning:
  - Interpolation turning
  - Interpolation milling
- Turning-specific contour elements
- Milling:
  - 3D radius compensation based on tool inclination
  - Tool radius compensation
  - Tool radius compensation perpendicular to the tool direction
  - Tool center point management
  - Tool compensation, pre-calculating

#### Motion Functions
- Motion interpolation
- Interpolation turning
- Interpolation milling
- Interpolation threading
- Interpolation drilling
- Interpolation tapping

### Option Table

<table>
<thead>
<tr>
<th>Option Number</th>
<th>Option Description</th>
<th>TNC 320</th>
<th>TNC 620</th>
<th>TNC 640</th>
<th>TNC 7</th>
<th>Must be adapted by the OEM</th>
<th>ID</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>Touch Probe Functions</td>
<td>☑</td>
<td>☑</td>
<td>☑</td>
<td>☑</td>
<td>Yes</td>
<td>634063-01</td>
<td>16</td>
</tr>
<tr>
<td>19</td>
<td>Advanced Programming Features</td>
<td>☑</td>
<td>☑</td>
<td>☑</td>
<td>☑</td>
<td>No</td>
<td>626252-01</td>
<td>18</td>
</tr>
<tr>
<td>20</td>
<td>Advanced Graphic Features: verification graphics, program run graphics</td>
<td>☑</td>
<td>☑</td>
<td>☑</td>
<td>☑</td>
<td>No</td>
<td>626253-01</td>
<td>19</td>
</tr>
<tr>
<td>21</td>
<td>Advanced Function Set 3: handwheel superimpositioning, handwheel positioning during program run</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>Yes</td>
<td>628254-01</td>
<td>42</td>
</tr>
<tr>
<td>22</td>
<td>Pallet Management</td>
<td>–</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>Yes</td>
<td>628255-01</td>
<td>21</td>
</tr>
<tr>
<td>40</td>
<td>Dynamic Collision Monitoring (DCM)</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>526452-01</td>
<td>44</td>
</tr>
<tr>
<td>42</td>
<td>CAD Import: DXF converter, importing contours and machining positions from DXF files</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>No</td>
<td>526450-01</td>
<td>23</td>
</tr>
<tr>
<td>44</td>
<td>Global PGM Settings: global program settings</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>576057-01</td>
<td>46</td>
</tr>
<tr>
<td>45</td>
<td>Adaptive Feed Control (AFC)</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>571948-01</td>
<td>47</td>
</tr>
<tr>
<td>50</td>
<td>Turning: turning functions</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>634060-01</td>
<td>26</td>
</tr>
<tr>
<td>52</td>
<td>3D-ToolComp: 3D radius compensation based on tool inclination (only in conjunction with option 9)</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>No</td>
<td>679678-01</td>
<td>31</td>
</tr>
<tr>
<td>53</td>
<td>Advanced Spindle Interpolation: interpolating spindle, interploting turning</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>751953-01</td>
<td>30</td>
</tr>
<tr>
<td>55</td>
<td>Spindle Synchronism: spindle synchronism of two or more spindles</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>808247-01</td>
<td>31</td>
</tr>
<tr>
<td>56</td>
<td>DCM v2: fixture measurement (automatically enables option 40)</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>1353266-01</td>
<td>48</td>
</tr>
<tr>
<td>57</td>
<td>Active Chatter Control (ACC)</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>800547-01</td>
<td>48</td>
</tr>
<tr>
<td>58</td>
<td>CAD Model Optimizer</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>135918-01</td>
<td>25</td>
</tr>
<tr>
<td>59</td>
<td>Batch Process Manager: convenient interface for pallet management</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>121562-01</td>
<td>22</td>
</tr>
<tr>
<td>60</td>
<td>Grinding: grinding and dressing functions</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>1237232-01</td>
<td>32</td>
</tr>
</tbody>
</table>

= Available as option
= Not available
= Standard function
<table>
<thead>
<tr>
<th>Option number</th>
<th>Option</th>
<th>TNC 320</th>
<th>TNC 620</th>
<th>TNC 640</th>
<th>TNC 7</th>
<th>Must be adapted by the OEM</th>
<th>ID</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>157</td>
<td>Gear Cutting: function for gear manufacturing</td>
<td>–</td>
<td>–</td>
<td>•</td>
<td>Yes</td>
<td>1237229-01</td>
<td>30</td>
<td></td>
</tr>
<tr>
<td>158</td>
<td>Turning v2: Extended turning cycles and functions (automatically enables option 50)</td>
<td>–</td>
<td>–</td>
<td>•</td>
<td>Yes</td>
<td>1359035-01</td>
<td>28</td>
<td></td>
</tr>
<tr>
<td>159</td>
<td>Model Aided Setup: graphically supported setup of workpieces</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>No</td>
<td>1364052-01</td>
<td>17</td>
<td></td>
</tr>
<tr>
<td>167</td>
<td>Optimized Contour Milling (OCM): optimize roughing processes</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>1288547-01</td>
<td>49</td>
<td></td>
</tr>
<tr>
<td>48</td>
<td>KinematicsOpt: touch-probe cycles for automatic measurement of rotary axes</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>630016-01</td>
<td>34</td>
<td></td>
</tr>
<tr>
<td>52</td>
<td>KinematicsComp: 3D spatial compensation</td>
<td>–</td>
<td>–</td>
<td>•</td>
<td>Yes</td>
<td>661679-01</td>
<td>35</td>
<td></td>
</tr>
<tr>
<td>141</td>
<td>Cross Talk Compensation (CTC): compensation of position errors through axis coupling</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>800542-01</td>
<td>36</td>
<td></td>
</tr>
<tr>
<td>142</td>
<td>Position Adaptive Control (PAC): position-dependent adaptation of control parameters</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>800544-01</td>
<td>27</td>
<td></td>
</tr>
<tr>
<td>143</td>
<td>Load Adaptive Control (LAC): load-dependent adaptation of control parameters</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>800545-01</td>
<td>38</td>
<td></td>
</tr>
<tr>
<td>144</td>
<td>Motion Adaptive Control (MAC): motion-dependent adaptation of control parameters</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>800546-01</td>
<td>29</td>
<td></td>
</tr>
<tr>
<td>146</td>
<td>Machine Vibration Control (MVC): active vibration damping</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>800548-01</td>
<td>40</td>
<td></td>
</tr>
<tr>
<td>155</td>
<td>Component Monitoring: monitoring for component overload and wear</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>1226833-01</td>
<td>50</td>
<td></td>
</tr>
<tr>
<td>168</td>
<td>Process Monitoring: reference-based monitoring of the machining process</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>1302488-01</td>
<td>51</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Communication</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>HEIDENHAIN DNC: communication with external Windows applications via COM components</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>No</td>
<td>526451-01</td>
<td>54</td>
<td></td>
</tr>
<tr>
<td>56</td>
<td>OPC UA NC Server 1 to 6: standardized interface for access to data and functions of the control</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>1291434-01</td>
<td>52</td>
<td></td>
</tr>
<tr>
<td>61</td>
<td>Remote Desktop Manager: display and remote operation of external computers (e.g., Windows PC)</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>894423-01</td>
<td>55</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Option number</th>
<th>Option</th>
<th>TNC 320</th>
<th>TNC 620</th>
<th>TNC 640</th>
<th>TNC 7</th>
<th>Must be adapted by the OEM</th>
<th>ID</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>157</td>
<td>Gear Cutting: function for gear manufacturing</td>
<td>–</td>
<td>–</td>
<td>•</td>
<td>Yes</td>
<td>1237229-01</td>
<td>30</td>
<td></td>
</tr>
<tr>
<td>158</td>
<td>Turning v2: Extended turning cycles and functions (automatically enables option 50)</td>
<td>–</td>
<td>–</td>
<td>•</td>
<td>Yes</td>
<td>1359035-01</td>
<td>28</td>
<td></td>
</tr>
<tr>
<td>159</td>
<td>Model Aided Setup: graphically supported setup of workpieces</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>No</td>
<td>1364052-01</td>
<td>17</td>
<td></td>
</tr>
<tr>
<td>167</td>
<td>Optimized Contour Milling (OCM): optimize roughing processes</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>1288547-01</td>
<td>49</td>
<td></td>
</tr>
<tr>
<td>48</td>
<td>KinematicsOpt: touch-probe cycles for automatic measurement of rotary axes</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>630016-01</td>
<td>34</td>
<td></td>
</tr>
<tr>
<td>52</td>
<td>KinematicsComp: 3D spatial compensation</td>
<td>–</td>
<td>–</td>
<td>•</td>
<td>Yes</td>
<td>661679-01</td>
<td>35</td>
<td></td>
</tr>
<tr>
<td>141</td>
<td>Cross Talk Compensation (CTC): compensation of position errors through axis coupling</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>800542-01</td>
<td>36</td>
<td></td>
</tr>
<tr>
<td>142</td>
<td>Position Adaptive Control (PAC): position-dependent adaptation of control parameters</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>800544-01</td>
<td>27</td>
<td></td>
</tr>
<tr>
<td>143</td>
<td>Load Adaptive Control (LAC): load-dependent adaptation of control parameters</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>800545-01</td>
<td>38</td>
<td></td>
</tr>
<tr>
<td>144</td>
<td>Motion Adaptive Control (MAC): motion-dependent adaptation of control parameters</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>800546-01</td>
<td>29</td>
<td></td>
</tr>
<tr>
<td>146</td>
<td>Machine Vibration Control (MVC): active vibration damping</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>800548-01</td>
<td>40</td>
<td></td>
</tr>
<tr>
<td>155</td>
<td>Component Monitoring: monitoring for component overload and wear</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>1226833-01</td>
<td>50</td>
<td></td>
</tr>
<tr>
<td>168</td>
<td>Process Monitoring: reference-based monitoring of the machining process</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>1302488-01</td>
<td>51</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Communication</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>HEIDENHAIN DNC: communication with external Windows applications via COM components</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>No</td>
<td>526451-01</td>
<td>54</td>
<td></td>
</tr>
<tr>
<td>56</td>
<td>OPC UA NC Server 1 to 6: standardized interface for access to data and functions of the control</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>1291434-01</td>
<td>52</td>
<td></td>
</tr>
<tr>
<td>61</td>
<td>Remote Desktop Manager: display and remote operation of external computers (e.g., Windows PC)</td>
<td>–</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>894423-01</td>
<td>55</td>
<td></td>
</tr>
</tbody>
</table>

Available as option: •
Not available: –
Standard function: ✓
### Software

<table>
<thead>
<tr>
<th>Software</th>
<th>TNC 320</th>
<th>TNC 620</th>
<th>TNC 640</th>
<th>TNC 7</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ConfigDesign: configuration of the machine parameters</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>–</td>
</tr>
<tr>
<td>PCtoolsUpdateManager: update manager for HEIDENHAIN software</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>–</td>
</tr>
<tr>
<td>PLCtext: processing and managing of language-sensitive texts</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>–</td>
</tr>
<tr>
<td>TNCAnalyzer: analysis and evaluation of service files</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>–</td>
</tr>
<tr>
<td>TNC OEMOption: key generator for enabling options for a limited time</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>–</td>
</tr>
<tr>
<td>TNC OEMNumber: day code generator for the machine manufacturer area</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>–</td>
</tr>
<tr>
<td>TNCtest: creation and execution of acceptance tests</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>–</td>
</tr>
<tr>
<td>RemoTools SDK: function library for the development of one's own communication applications (option 18 required)</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>54</td>
</tr>
<tr>
<td>TNCremo: data transfer software</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>63</td>
</tr>
<tr>
<td>TNCremoPlus: data transfer software with “live” screen</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>63</td>
</tr>
<tr>
<td>RemoteAccess: software for remote diagnostics</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>64</td>
</tr>
<tr>
<td>PLCDesign: PLC diagnostics for all HEIDENHAIN controls</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>65</td>
</tr>
<tr>
<td>StateMonitor</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>66</td>
</tr>
<tr>
<td>KinematicsDesign: creation of control kinematic models</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>69</td>
</tr>
<tr>
<td>M3D Converter: creation of high-resolution collision bodies</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>69</td>
</tr>
<tr>
<td>CycleDesign: incorporation of developed NC subprograms as cycles</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>69</td>
</tr>
<tr>
<td>TNCscope: recording and evaluation of, for example, axis curves of HEIDENHAIN controls</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>70</td>
</tr>
<tr>
<td>TNCdiag: evaluation of diagnostic information of digital motor systems</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>71</td>
</tr>
<tr>
<td>TNCopt: initial setup of digital control loops</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>72</td>
</tr>
<tr>
<td>Toconfig: configuration of, for example, handwheels</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>73</td>
</tr>
<tr>
<td>TNCkeygen: contains TNC OEMOption and TNC OEMNumber</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>74</td>
</tr>
<tr>
<td>BMXdesign: designing language-sensitive soft keys</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>75</td>
</tr>
<tr>
<td>Programming station</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>76</td>
</tr>
<tr>
<td>virtualTNC: for PC-based machine simulation</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>77</td>
</tr>
</tbody>
</table>

### Hardware enhancements

<table>
<thead>
<tr>
<th>Hardware enhancements</th>
<th>TNC 320</th>
<th>TNC 620</th>
<th>TNC 640</th>
<th>TNC 7</th>
<th>Must be adapted by the OEM</th>
<th>ID</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Handwheel</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HR 130 TTL: panel mounted</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>No</td>
<td>540940-01</td>
<td>78</td>
</tr>
<tr>
<td>HR 510 portable handwheel</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>No</td>
<td>1120313-xx</td>
<td>1119871-xx</td>
</tr>
<tr>
<td>HR 510FS portable handwheel</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>No</td>
<td>1119874-xx</td>
<td>1120311-xx</td>
</tr>
<tr>
<td>HR 520 portable handwheel with display</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>No</td>
<td>670200-xx</td>
<td>670202-xx</td>
</tr>
<tr>
<td>HR 520FS portable handwheel with display</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>No</td>
<td>670205-xx</td>
<td>670204-xx</td>
</tr>
<tr>
<td>HR 550FS portable handwheel</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>No</td>
<td>1183021-xx</td>
<td>1200459-xx</td>
</tr>
<tr>
<td>HRA 551FS handwheel holder for HR 550FS</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>No</td>
<td>1119052-xx</td>
<td></td>
</tr>
<tr>
<td>Workpiece touch probe</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TS 260 with cable</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>738283-xx</td>
<td>79</td>
</tr>
<tr>
<td>TS 460 with radio or infrared transmission</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>1178530-xx</td>
<td>80</td>
</tr>
<tr>
<td>TS 642 with infrared transmission</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>653217-xx</td>
<td>80</td>
</tr>
<tr>
<td>TS 760 with radio or infrared transmission</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>1283267-xx</td>
<td></td>
</tr>
<tr>
<td>Tool touch probe</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TT 160 with cable</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>729763-xx</td>
<td>80</td>
</tr>
<tr>
<td>TT 460 with radio or infrared transmission</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>•</td>
<td>Yes</td>
<td>1192582-xx</td>
<td>80</td>
</tr>
<tr>
<td>Additional operating station</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ITC 362 with 24-inch touchscreen (separate TNC keyboard required)</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>1349871-xx</td>
<td>81</td>
</tr>
<tr>
<td>ITC 895 with 15-inch touchscreen and ASCII keyboard</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>1370459-xx</td>
<td>81</td>
</tr>
<tr>
<td>ITC 960 19-inch touchscreen (separate TNC keyboard required)</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>1174935-xx</td>
<td>81</td>
</tr>
<tr>
<td>Industrial PC</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>IPC 306 with 8 GB RAM</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>1179066-01</td>
<td>82</td>
</tr>
<tr>
<td>Vision system</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>VT 121 for tool inspection</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>Yes</td>
<td>1243468-01</td>
<td>83</td>
</tr>
</tbody>
</table>

○ = Available as option
– = Not available
✓ = Standard function
The term Dynamic Precision refers to optional functions for HEIDENHAIN controls that reduce the dynamic error of machine tools. By improving the machine’s dynamic performance and increasing TCP accuracy, these functions take milling to the limits of the technologically possible—regardless of the machine’s age and load or machining position. No mechanical modifications are needed.

Attaining precise workpieces with high surface quality no longer requires working slowly. That’s because, with Dynamic Precision, machine tools work fast and accurately at the same time.

High precision and fast machining are the basis for increased productivity. Unit cost is therefore reduced without sacrificing accuracy or surface quality. Dynamic Precision even ensures that accuracy is maintained regardless of the operating time and load. As a result, feed rates no longer need to be reduced on account of machine age or load.

The functions of Dynamic Precision are available as options for HEIDENHAIN controls. The machine manufacturer can deploy individual functions alone or in combination:
- **CTC**: compensation of acceleration-dependent position error at the tool center point, thus increasing accuracy during acceleration phases
- **MVC**: damping of machine vibrations for improved surface finishes
- **PAC**: position-dependent adaptation of control parameters for high accuracy, regardless of machine age and load
- **MAC**: motion-dependent adaptation of control parameters

The functions of Dynamic Precision are adapted to the movements and loads of the machine tool through a high clock rate of the controller unit (a component of HEIDENHAIN controls).

Because Dynamic Precision is made up of software functions, it requires no intervention in the machine’s mechanics or power train. Nevertheless, the machine manufacturer must enable the individual functions, enter their parameters, and adapt them to the machine.

Adjustments in the controller unit (a component of HEIDENHAIN controls).

Overview
Dynamic Precision

- **Motion Adaptive Control (MAC)**: motion-dependent adaptation of control parameters
- **Load Adaptive Control (LAC)**: load-dependent adaptation of controller parameters
- **Position Adaptive Control (PAC)**: position-dependent adaptation of control parameters
- **Machine Vibration Control (MVC)**: damping of machine oscillations
- **Cross Talk Compensation (CTC)**: compensation of position errors arising from machine compliance

**Dynamic Efficiency**

At HEIDENHAIN, the term Dynamic Efficiency refers to innovative TNC functions that help make heavy machining and roughing processes more efficient and reliable. These software functions not only support the machine operator but also make the manufacturing process itself faster, more stable, and more predictable—in short, more efficient. Dynamic Efficiency helps boost the metal removal rate and reduce machining time.

Dynamic Efficiency encompasses four software functions:
- **ACC**: reduces chatter susceptibility, thus permitting higher feed rates and greater infeeds
- **AFC**: controls the feed rate based on the machining situation
- **Trochoidal milling**: a function for tool-friendly roughing of slots and pockets
- **OCM**: optimizes roughing processes for fast and low-wear machining

The AFC, ACC, and OCM functions are described in greater detail later in this brochure and are identifiable by their Dynamic Efficiency logo.

Individually, each of these solutions delivers key improvements to the machining process. But in combination, these TNC functions bring out the full potential of the machine and tool while reducing the mechanical load. They are also of great value under changing machining conditions, such as during interrupted cuts, various plunging processes, or simple roughing operations. In practice, the removal rate can be increased by 20 to 25 percent.

By permitting higher removal rates, Dynamic Efficiency boosts productivity without requiring operators to resort to special tools. The avoidance of tool overloading and premature tool wear, as well as the additional gain in process reliability, significantly contributes to more economical production.

**Dynamic Efficiency**

<table>
<thead>
<tr>
<th>Dynamic Efficiency</th>
<th>TNC7</th>
<th>TNC 640</th>
<th>TNC 620</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>36</td>
</tr>
<tr>
<td>Cross Talk Compensation (CTC)</td>
<td>Option</td>
<td>Option</td>
<td>Option</td>
<td>36</td>
</tr>
<tr>
<td>Machine Vibration Control (MVC)</td>
<td>Option</td>
<td>Option</td>
<td>Option</td>
<td>40</td>
</tr>
<tr>
<td>Position Adaptive Control (PAC)</td>
<td>Option</td>
<td>Option</td>
<td>Option</td>
<td>37</td>
</tr>
<tr>
<td>Load Adaptive Control (LAC)</td>
<td>Option</td>
<td>Option</td>
<td>Option</td>
<td>38</td>
</tr>
<tr>
<td>Motion Adaptive Control (MAC)</td>
<td>Option</td>
<td>Option</td>
<td>Option</td>
<td>39</td>
</tr>
</tbody>
</table>

**Dynamic Efficiency**

For more information, see the Dynamic Efficiency Technical Information document, and visit www.klartext-portal.com.

**Technical Information**

For more information, see the Dynamic Precision Technical Information document, and visit www.klartext-portal.com.

**Overview**

Dynamic Precision

**Overview**

Dynamic Efficiency

<table>
<thead>
<tr>
<th>Dynamic Precision</th>
<th>TNC7</th>
<th>TNC 640</th>
<th>TNC 620</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAC: motion-dependent adaptation of control parameters</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>LAC: load-dependent adaptation of control parameters for high accuracy, regardless of machine age and load</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>PAC: position-dependent adaptation of control parameters</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Dynamic Efficiency</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
</tbody>
</table>

**Overview**

Dynamic Efficiency

**Overview**

Dynamic Efficiency

**Overview**

Dynamic Efficiency
With the Connected Machining package of functions, the HEIDENHAIN control streamlines operations thanks to easy data usage, time-saving workflows, and transparent processes. These solutions put the operator at the center of digital job management through the control on his milling machine or lathe. The HEIDENHAIN control can be linked as desired to every production-related area of the company, featuring adaptability to already existing structures and extensibility for future innovation.

The benefits are time-saving workflows, workload reduction through easy data usage, and transparent processes in all areas of the company—not just on the shopfloor but also in design, production planning, management, logistics, servicing, and more. Purely digital job management with Connected Machining augments the strong manufacturing capabilities of modern equipment and machines.

**Efficient and lossless data transmission**
Use the TNC control to directly access data in your network or to transfer data from your PC to the TNC control. The program manager manages access to network drives through the TNC. The task-oriented method of operation reduces the time required and allows interaction without in-depth knowledge about the control.

- HEIDENHAIN DNC (option 18) for monitoring and remote operation of older machines.
- Remote Desktop Manager (option 133) for access from the control to Windows PCs and the software installed there.
- Firefox web browser for accessing web-based applications, such as ERP and MES-clients, e-mail-clients, and HTML5 applications.

**Remote operation**
The following functions are available for remote operation of HEIDENHAIN controls or third-party CNC controls:
- HEIDENHAIN OPC UA NC Server (options 56 to 61) for cross-platform remote monitoring and controlling of a TNC. The task-oriented method of operation reduces the time required and allows interaction without in-depth knowledge about the control.
- HEIDENHAIN DNC (option 18) for monitoring and remote operation of older machines.
- Remote Desktop Manager (option 133) for access from the control to Windows PCs and the software installed there.
- Firefox web browser for accessing web-based applications, such as ERP and MES-clients, e-mail-clients, and HTML5 applications.

The following solutions and options are offered as well:
- Remote Desktop Manager (option 133) allows operators to access PCs and their installed software from the control.
- The HEIDENHAIN DNC interface (option 18) links the control to enterprise-resource planning and production-activity control systems.
- Extended Workspace displays processes and depicts job data at the machine in a large and clear format.
- The OPC UA NC Server option enables a convenient cross-platform connection to your IT system landscape.

**Supported file formats**
Being able to display, edit and evaluate common file types opens up many possibilities for using digital job data directly on the control. The Extended Workspace display provides a clear representation of your job data without losing track of important information about the control.
- PDF viewer for displaying task lists and manufacturing notes, for example (*.pdf)
- CAD viewer for displaying 3D models and design drawings, for example (*.step, *.stl, *.dxf, *.iges)
- Graphics files for displaying clamping situations, for example (*.gif, *.bmp, *.jpg, *.png)
- Spreadsheets for displaying working steps and tool data, for example (*.xls, *.xlsx, *.odf, *.ods)
- HTML viewer for displaying documentation and help-files, for example (*.html, *.htm, *.chm)

**Remote operation**

**Efficient and lossless data transmission**
Use the TNC control to directly access data in your network or to transfer data from your PC to the TNC control. The program manager manages access to network drives through the TNC. The task-oriented method of operation reduces the time required and allows interaction without in-depth knowledge about the control.

- HEIDENHAIN OPC UA NC Server (options 56 to 61) for cross-platform remote monitoring and controlling of a TNC. The task-oriented method of operation reduces the time required and allows interaction without in-depth knowledge about the control.
- HEIDENHAIN DNC (option 18) for monitoring and remote operation of older machines.
- Remote Desktop Manager (option 133) for access from the control to Windows PCs and the software installed there.
- Firefox web browser for accessing web-based applications, such as ERP and MES-clients, e-mail-clients, and HTML5 applications.

**Remote operation**
The following functions are available for remote operation of HEIDENHAIN controls or third-party CNC controls:
- HEIDENHAIN OPC UA NC Server (options 56 to 61) for cross-platform remote monitoring and controlling of a TNC. The task-oriented method of operation reduces the time required and allows interaction without in-depth knowledge about the control.
- HEIDENHAIN DNC (option 18) for monitoring and remote operation of older machines.
- Remote Desktop Manager (option 133) for access from the control to Windows PCs and the software installed there.
- Firefox web browser for accessing web-based applications, such as ERP and MES-clients, e-mail-clients, and HTML5 applications.

The following solutions and options are offered as well:
- Remote Desktop Manager (option 133) allows operators to access PCs and their installed software from the control.
- The HEIDENHAIN DNC interface (option 18) links the control to enterprise-resource planning and production-activity control systems.
- Extended Workspace displays processes and depicts job data at the machine in a large and clear format.
- The OPC UA NC Server option enables a convenient cross-platform connection to your IT system landscape.

**Supported file formats**
Being able to display, edit and evaluate common file types opens up many possibilities for using digital job data directly on the control. The Extended Workspace display provides a clear representation of your job data without losing track of important information about the control.
- PDF viewer for displaying task lists and manufacturing notes, for example (*.pdf)
- CAD viewer for displaying 3D models and design drawings, for example (*.step, *.stl, *.dxf, *.iges)
- Graphics files for displaying clamping situations, for example (*.gif, *.bmp, *.jpg, *.png)
- Spreadsheets for displaying working steps and tool data, for example (*.xls, *.xlsx, *.odf, *.ods)
- HTML viewer for displaying documentation and help-files, for example (*.html, *.htm, *.chm)

### Installation

**Remote Desktop Manager**
- Option

**HEIDENHAIN DNC**
- Option

**OPC UA NC Server**
- Option

**Extended Workspace**
- 

For more information, see the TNC brochures.
Programming and operation
Machining with a rotary table

Many five-axis operations that initially seem very complex can actually be reduced to conventional 2D movements executed on a cylindrical surface. For creating and editing these programs quickly and easily without a CAM system, the TNC comes to your aid with helpful functions.

Cylinder surface machining
The TNC makes it easy for you to program straight-line and arc contours on cylindrical surfaces for rotary and tilting tables by conveniently allowing you to program the contour on the plane of an unrolled cylinder (axis-independently on the TNC-7, TNC 640, TNC 620 and TNC 320). The TNC then executes the programmed machining operation on the cylinder surface.

The TNC provides four cycles for cylinder surface machining:
- Slot milling (the slot width and tool diameter are equivalent)
- Guide-groove milling (the slot width is greater than the tool diameter)
- Ridge milling
- Outside contour milling

Feed rate for rotary axes and tables in mm/min
By default, rotary axis feed rates are programmed in degrees/min, but the TNC can also interpret these feed rates in mm/min. This makes the feed rate at the contour independent of the distance between the tool center point and the center of the rotary axis.

Programming and operation
Coordinate transformation: tilting the working plane, and the PLANE function

Programs for contours and holes on inclined surfaces are usually very complex, requiring significant computing resources and programming work. In cases like these, the TNC can save you a great deal of programming time. You simply program the machining operation as usual in the main plane (e.g., in XY). The machine then runs the program in a plane that has been tilted with respect to the main plane about one or more rotary axes.

The PLANE function makes it easy to define a tilted working plane and, based on the information in the workpiece drawing, offers seven different methods for doing so. In order to keep this complex function as easy to use as possible, each plane definition has its own animation that you can view prior to selecting the function. Intuitive help graphics assist you during programming.

You can also use the PLANE function to define the positioning behavior during tilting to avoid unpleasant surprises when the program is run. The settings for defining the positioning behavior are identical for all PLANE functions, making operation much easier.

Advanced Function Set 1
TNC-7/TNC 640/TNC 620
TNC 320

Option 8
ID 617920-01
ID 536146-01
ID 536164-01
ID 536146-01
ID 536146-01
ID 536146-01
ID 536146-01

TNC7
TNC 640
TNC 620
TNC 320
NC SW 81762x-16 or later
NC SW 34056x-01 or later
NC SW 34056x-01/73498x-01/81760x-01 or later
NC SW 34055x-01/771851-01 or later
NC SW 81762x-16 or later
NC SW 34056x-01 or later
NC SW 34056x-01/73498x-01/81760x-01 or later
NC SW 34055x-01/771851-01 or later
Installation by the machine manufacturer
For more information, see the TNC brochures.
Programming and operation

Touch probe cycles

Workpiece alignment
The HEIDENHAIN touch probes and probing functions of the TNC reduce the need for tedious manual alignment of the workpiece:
- Workpiece clamped in any position
- The touch probe determines the actual clamping situation by probing a surface
- The TNC compensates for this misalignment with a “basic rotation,” either by rotating the NC program to the ascertained angle or by correcting the actual misalignment through a turn of the rotary table.
- The TNC offers manual, automatic, and semiautomatic cycles for correcting misalignments in two or three dimensions
- Some automatic probing cycles feature semiautomatic mode, tolerance monitoring and actual-to-nominal value transfer
- Automatic probing cycle with which touch points in one direction can be repeated over a specified distance

Preset setting
Being able to define the preset quickly and reliably reduces nonproductive time and increases machining accuracy. The TNC features a myriad of probing cycles for automatic preset setting.

Workpiece inspection
The TNC features numerous measuring cycles for inspecting the geometry of machined workpieces. This allows you to do the following:
- Identify a workpiece and call an appropriate part program
- Check for correct execution of all machining operations
- Detect and compensate for tool wear, etc.

Tool measurement
In conjunction with the TT touch probes for tool measurement, the TNC makes it possible to automatically measure tools inside the machine. The measured tool length and radius are stored by the TNC in its central tool memory. In-process tool measurement allows you to immediately identify wear or breakage, thus avoiding scrap and rework.

On the TNC 620, the touch probe cycles are available only with option 17. This option is automatically enabled with the use of HEIDENHAIN touch probes featuring the EnDat interface (NC SW 81760x-06 or later). The TNC7, TNC 640 and TNC 320 already include the touch probe cycles as part of their standard functions.

For more information, see the TNC brochures and the Touch Probes brochure.

Graphical 6D workpiece setup

For single parts and small lot sizes without a special holder with a defined location, the position of the workpiece blank almost always needs to be ascertained before machining. With this intuitive probing function, the TNC7 offers the possibility of setting up workpieces quickly, easily and safely with graphic guidance. The exact position of the workpiece blank is measured in the machine’s working space and reported to the control.

The simulation view depicts a 3D model of the workpiece blank in the machine’s working space. After the operator has roughly aligned the model manually, a green arrow indicates that the model is ready to be probed. The axis keys or the electronic handwheel are used to position the touch probe to the blank in order to record the touch points. The control automatically selects the probing direction. All of the six degrees of freedom can thus be measured with only one single function.

During the entire setup process, the control informs the operator about the quality of the probing points for determining the position and orientation of the workpiece. The operator can thus quickly see when the measurement of the actual position and orientation of the workpiece is complete. The operator can also move the rotary axes during the setup procedure in order to probe undercuts, inclined surfaces, or rounded surfaces, for example. This allows the operator to align even complex workpiece blanks relative to pre-machined features, such as is needed for mold repair or 3D-printed workpiece blanks.

For more information, see the TNC7 brochure, and visit www.klartext-portal.com
Programming and operation

Advanced programming functions: FK free contour programming, canned cycles

**FK free contour programming**

Not all workpieces are suitably dimensioned for conventional NC programming. In this case, the FK free contour programming function lets you simply enter data from the drawing without performing additional conversions or calculations! In the process, you can leave individual contour elements undefined as long as the overall contour is defined. If the data allow for more than one mathematical solution, then the TNC programming graphics helpfully render the possible variants to choose from. On the TNC 7, graphical programming (as standard feature) replaces FK programming.

**Standard cycles**

In addition to the canned cycles for drilling and tapping (with or without a floating tap holder), option 19 also gives you cycles for thread milling, reaming, boring, and hole patterns, as well as milling cycles for plane-surface face milling and for roughing and finishing pockets, slots, and studs.

**Cycles for complex contours**

Special help for roughing pockets with any contour is provided by the Subcontour List (SL) cycles and the Optimized Contour Milling (OCM) software option. These groups of cycles each contain canned cycles for roughing and finishing. The contour or subcontours are defined in subprograms. As a result, a single contour description can be used for various operations with different tools.

**OEM cycles**

As original equipment manufacturers, machine manufacturers can contribute their special manufacturing know-how by designing additional fixed cycles and saving them in the TNC. The end user is likewise given the option of programming his own cycles. HEIDENHAIN makes this possible with its PC software CycleDesign, which allows you to create input parameters and arrange the menu key structure of the cycles as desired.

**Graphical contour programming** *

Couple the familiar Klartext functions with the graphical programming capability to streamline your daily work at the machine. With the graphical programming feature, the operator draws the desired contours directly on the touchscreen. The TNC 7 then converts the drawing into a Klartext program. This function makes it very easy to modify existing programs.

* Available only for the TNC 7

---

**Programmation and operation**

Verification graphics, program-run graphics

**Programming graphics**

HEIDENHAIN controls support you with detailed graphical graphics. These graphics are a feature on the TNC 320, TNC 620 and TNC 640, and are described in their respective brochures.

Other graphical views are available as options:

- **Simulation**
  - To be on the safe side before running a program, the TNC can graphically simulate the machining operation. The TNC can display this simulation in the following ways:
    - In plan view with different shades of depth
    - In different projections (as in the workpiece drawing)
    - In 3D view
  - Details can be shown under magnification. In addition, the TNC displays the calculated machining time in hours, minutes, and seconds. During the simulation, the TNC 640 and TNC 620 controls can also detect collisions between the workpiece and the tool holder, and can issue appropriate warning messages.

- **Model comparison**
  - The TNC 7 has a feature for comparing the workpiece blank and the finished part.
  - Based on the different colors, you easily see where residual material still remains or where too much material was removed. There is also a measuring function that lets you ascertain the position and depth at any location on the workpiece, together with the current NC block and the associated tool.

- **Program-run graphics**
  - Direct observation of the workpiece within the machine is usually impossible due to coolant and the safety enclosure. With the program-run graphics you can always see the current machining status of the workpiece as well as the positions of the collision objects in the machine on the control’s display. During machining, you can switch between various operating modes at any time, for example in order to create programs.

---

**Advanced Programming Features**

<table>
<thead>
<tr>
<th>Option</th>
<th>ID 628252-01</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>Standard function</td>
</tr>
<tr>
<td>TNC 640</td>
<td>Standard function</td>
</tr>
<tr>
<td>TNC 620</td>
<td>NC SW 34056x-01/73498x-01/61760x-01 or later</td>
</tr>
<tr>
<td>TNC 320</td>
<td>Standard function</td>
</tr>
</tbody>
</table>

**Installation by the operator**

For more information, see the TNC brochures, and visit www.klartext-portal.com
**Finely detailed 3D view**

The simulation and program-run graphics of the controls also include additional viewing functions with detailed 3D rendering. These functions help you evaluate the workpiece quality in the simulation prior to machining, as well as in real time during machining.

The high-resolution 3D view with strong zoom factors reveals even the tiniest program errors on the workpiece surface. For an even more precise view of the NC data analysis, you can also display the tool paths and their corresponding block numbers. Of particular help is the selectable workpiece transparency feature, which allows you to see hidden cavities and undercuts. The TNC7 offers a sectional view along a plane. This is useful, for example, when looking for residual material.

Enhanced graphics also make tool-specific machining operations easier to see: each machining operation performed with the same tool is shown in its own unique color. In 3D view, the TNC also includes a measuring function that displays the coordinates when you shift the focus within the graphics by touch.

If you just need a quick overview of the contour and the machining time, you can change the resolution and simulation mode to boost the calculation speed.

The TNC 640 (with NC SW 34059x-09 or later) and the TNC 7 simulate not only material removal but also the complete 3D machine model, allowing the operator to precisely evaluate processes in the working space before machining. The machine's collision objects are configured and activated by the machine manufacturer.

The TNC 640 (with NC SW 34059x-11 or later) and TNC 620 (with NC SW 81760x-08 or later) enable the convenient incorporation of complex workpiece blanks and finished parts as STL files (such as 3D models from CAM systems). In addition, the current machining status can be saved as an STL file during the simulation in order to integrate it into other programs as a workpiece blank.

<table>
<thead>
<tr>
<th>Advanced Graphic Features</th>
<th>Option 20</th>
<th>ID 628253-01</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>Standard function</td>
<td></td>
</tr>
<tr>
<td>TNC 640</td>
<td>Standard feature with NC SW 34059x-04 or later NC SW 81760x-01 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 620</td>
<td>Standard feature in NC SW 771851-01 or later</td>
<td></td>
</tr>
</tbody>
</table>

**Pallet Management**

For any sequence of different workpieces loaded on pallets, the TNC is able to identify the proper machining program and datum shift.

When a pallet is loaded for machining, the TNC automatically calls the correct program. This allows different parts to be machined in any order desired.

Pallet insertions can be controlled via PLC axes. The desired sequence and presets for the pallet and workpieces are defined in the pallet table by the operator. The pallet table is freely configurable by the machine manufacturer, which means that any information can be stored in the tables and called later by the PLC.

Execution of the pallet tables with the TNC7 can be either workpiece- or tool-oriented (with NC SW 34059x-08 or later on the TNC 640, with NC SW 81760x-05 or later on the TNC 620).

For more information, see the TNC brochures, and visit www.klartext-portal.com
Programming and operation

Batch Process Manager: organization and fine tuning of job sequences on the machine

Batch Process Manager is a powerful function for pallet machining and series production. With its intuitive user interface, you can plan your production process and receive important information about upcoming machining operations.

Batch Process Manager makes it possible to easily change the sequence of pending jobs. Batch Process Manager also performs a duration calculation for all planned jobs or NC programs. It checks whether, for example, all NC programs will be able to run without error and whether all necessary tools are available with sufficient remaining service life. Batch Process Manager displays the results of these checks in the status overview, thereby ensuring the smooth execution of the planned jobs.

Batch Process Manager shows the following information in advance:
- The machining sequence
- The time of the next manual intervention
- Program duration and end time of program run
- Status information about preset, tool, and program

On the TNC 620, Batch Process Manager requires the additional enabling of Pallet Management (option 22). On controls with NC software versions up to 34059x-08 or 81760x-05, Extended Tool Management (option 93) must be enabled as well.

Batch Process Manager Option 154
Option 22
ID 1219521-01
ID 628255-01
TNC7
TNC 640
TNC 620
TNC 320
NC SW 81762x-16 or later
NC SW 81760x-05 or later
NC SW 81760x-05 or later
NC SW 81760x-05 or later
Installation by the machine manufacturer

For more information, see the TNC brochures, and visit www.klartext-portal.com

Programming and operation

CAD Import: import contours and machining positions from DXF files

Why program contours when you already have the drawing as a DXF file? You can open DXF files directly on the TNC in order to extract contours or machining positions from them. This not only saves time otherwise spent on programming and testing but also provides assurance that the finished contour precisely complies with the design engineer’s specifications.

DXF files usually contain multiple layers used by the design engineer to organize the drawing. In order to minimize unneeded information on your screen during contour selection, you can hide all of the inessential layers in the DXF file with just a gesture. This feature requires an operating panel with a touchpad or an external pointing device. The TNC can select a contour train even if it has been stored across different layers.

The TNC also supports you in defining the workpiece preset. For this purpose, the TNC provides a function that allows you to move the drawing’s datum to a useful position by simply selecting an element. Selecting contours is also very convenient: you can select any element by touch. Once you select a second element, the TNC detects your desired machining direction and initiates automatic contour detection. In doing so, the TNC automatically selects all clearly identifiable contour elements until the contour closes or branches out. At that point, you can select the next contour element. In this way you can quickly and easily define even extensive contours by touch. You can even shorten, lengthen, or interrupt the contour elements as needed.

In addition, you can easily select machining positions and save them as point files, especially for transferring drilling positions or starting points for pocket machining. Of course, the TNC saves the machining positions in such a way that they are approached via the shortest path.

With the CAD Import option, you can also import contours and machining positions from 3D models.

CAD Import Option 42
ID 526450-01
TNC7
TNC 640
TNC 620
TNC 320
NC SW 81762x-16 or later
NC SW 81760x-05 or later
NC SW 73498x-02/81760x-01 or later
NC SW 771851-01 or later
Installation by the operator

For more information, see the TNC brochures, and visit www.klartext-portal.com
Programming and operation

CAD Import: import contours from 3D models

CAD Import (option 42)
CAD viewer enables the display of 2D and 3D models right on the TNC (e.g., DXF, Step, STL or IGES files). Using the CAD Import option, you can easily incorporate contours and machining positions from these CAD files directly into your Klartext program. You can thereby reduce the amount of programming work and avoid input errors resulting from transposed digits, incorrectly placed decimal points, and the like.

In particular, extracting machining information directly from CAD data provides additional possibilities for creating NC programs with a tilted working plane. You can also define the preset on the 3D model with a 3D basic rotation. In addition, you can place a datum with the appropriate 3D rotation on the desired working plane.

The working plane can be conveniently saved to the clipboard and transferred to the NC program with the proper transformation and relevant PLANE command. On the defined working plane, you can extract contours and machining positions, and transfer them to the NC program.

Selecting contours is very convenient: you can select any element by clicking it with the mouse. Once you select a second element, the TNC detects your desired machining direction and initiates automatic contour detection. In doing so, the TNC automatically selects all clearly identifiable contour elements until the contour closes or branches out. This method lets you define elaborate contours with just a few clicks of the mouse. You can then easily copy the selected contour via the clipboard to an existing Klartext program.

The CAD Import option is an expansion to the DXF Converter. All previous functions have been adopted and enhanced to include 3D import functions. Nearly all of the common DXF, Step and IGES file formats can be used.

<table>
<thead>
<tr>
<th>CAD Import</th>
<th>Option 42</th>
<th>ID 526450-01</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>NC SW 81762x-16 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 640</td>
<td>NC SW 34069x-08 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 620</td>
<td>NC SW 81790x-05 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 320</td>
<td>NC SW 771861-05 or later</td>
<td></td>
</tr>
</tbody>
</table>

Installation by the operator

For more information, see the TNC brochures, and visit www.klartext-portal.com

Programming and operation

Optimization of CAD models

The CAD Model Optimizer software option offers the operator the power to simplify 3D models and heal contours, thus creating valid STL files for collision monitoring or simulation.

The initial model can be imported through the CAD viewer. The 3D mesh function simplifies the model and autonomously corrects errors such as small holes in the solid model or self-intersecting lines on surfaces. The control covers the opened 3D model with a mesh of triangles. The control then generates an STL file that you can use for various functions of the control. For example, you can then quickly and easily repair faulty files of fixtures or tool holders.

CAD Model Optimizer (Option 152) ID 1363918-01

| Installation by the operator |

For more information, see the TNC brochures, and visit www.klartext-portal.com
Programming and operation

Turning functions: milling and turning on the same machine

The TNC7 and TNC 640 provide powerful functions for NC program-driven switching between turning mode and milling mode. You are completely free to decide how and when you wish to combine these two machining modes.

Machining simple contours
Various cycles are available for the longitudinal and transverse machining of simple contours. The area to be machined can also be inclined, thus requiring a plunging movement. Of course, the TNC7 and TNC 640 automatically take the angle of the turning tool into account.

Machining any contours
If the contours to be machined are complex and can no longer be defined with simple cycle parameters, you can still describe them using contour subprograms. This process is identical to the procedure used for SL cycles in milling, for which you use Cycle 14 to define the subprogram describing the finished contour. You can define the technology parameters in the respective turning cycle. You use exactly the same Klartext functions for the contour description as you would for defining a milling contour.

Turning-specific recess and undercut contour elements are also available, which you can insert between contour elements in the same manner as chamfers and rounding arcs. Besides radial and axis recesses, E, F, H, K, and U undercuts are available, as are thread undercuts.

Depending on the cycle, the TNC7 and TNC 640 cut parallel to the axis or the contour. You define the machining operations (roughing, finishing) or the oversize with dialog guidance in the corresponding parameters.

Other machining operations:
• Recessing
• Thread machining
• Blank form update
• Orientation of the turning tool

<table>
<thead>
<tr>
<th>Turning with a facing slide</th>
</tr>
</thead>
<tbody>
<tr>
<td>A facing slide allows you to perform turning operations on a stationary workpiece. This, in turn, makes it possible to implement off-center or tilted-plane turning operations. With a facing slide, the cutting edge rotates in the spindle, while an axis integrated in the facing slide controls the turning tool (plan stroke). With the TNC7 and TNC 640, you don’t need to worry about the complex motion involved. You simply select the facing-slide mode with a programming command and program as always with standard turning cycles. The TNC7 and TNC 640 handle all the conversions for you and automatically perform all motion sequences.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Programming and operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turning functions: eccentric turning</td>
</tr>
</tbody>
</table>

The eccentric turning function lets you perform turning operations even when the workpiece axis is misaligned with the axis of rotation due to the setup. During machining, the TNC7 and TNC 640 then compensate for any eccentricity through corrective movements of the rotary spindle coupled with the linear axis. This can significantly reduce your setup times.

* On controls with NC software version 34059x-04, RTC (option 135) must be enabled.

Installation by the machine manufacturer

For more information, see the TNC brochures, and visit www.klartext-portal.com

<table>
<thead>
<tr>
<th>Turning</th>
<th>Option 50</th>
<th>ID 634608-01</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>NC SW 81762x-16 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 640</td>
<td>NC SW 34059x-04 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 620</td>
<td>–</td>
<td></td>
</tr>
<tr>
<td>TNC 320</td>
<td>–</td>
<td></td>
</tr>
</tbody>
</table>

For more information, see the TNC brochures, and visit www.klartext-portal.com
**Programming and operation**

**Turning functions: simultaneous turning**

This option allows you to produce complex contours that would otherwise require machining at varying tool angles. This way you can, for example, produce undercut contours with just one tool. For simple contours, this option can prolong tool life by using a larger area of the indexable insert. This involves at least three-axis motion (two linear axes and one rotary axis).

The cycles for simultaneous turning monitor the workpiece contour relative to the tool and the tool carrier. To obtain optimal surface finishes, these cycles avoid unnecessary tilting movements. If you wish to activate specific tilting movements, you can define start and finish angles on the TNC7 and TNC 640. A cycle for simultaneous roughing is also available (with NC SW 34059x-11 or later).

The control now permits the use of FreeTurn tools for inclined and simultaneous turning operations. FreeTurn tools are lathe tools with a special tooth arrangement. Depending on the design, a single FreeTurn tool can perform both axis-parallel and contour-parallel roughing and finishing. The use of FreeTurn tools reduces machining times because fewer tool changes are needed (with NC software version 16 or later).

**Advanced Function Set Turning**

* Advanced Function Set Turning additionally requires option 50

<table>
<thead>
<tr>
<th>TNC 7</th>
<th>TNC 640</th>
<th>TNC 620 / TNC 320</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option 158</td>
<td>ID 1237237-01</td>
<td></td>
</tr>
</tbody>
</table>

**Extended Tool Management**

This software option extends tool management by the two tables **Tooling list** and **T usage order**. The **Tooling list** shows the tool requirements of the NC program or pallet to be run. The **Tooling list** contains information about all the tools called within an NC program. Before starting the program, you can check, for example, whether all tools are contained in the magazine. When you select an NC program in the **Program run** operating mode, the control automatically creates the **Tooling list** table. The control lists all the tools called within the active NC program in numerical order.

The **T usage order** table shows the tool sequence of the pending pallet or NC program. Before starting the program, you can see, for example, when a manual tool change will take place. When you select an NC program in the **Program run** operating mode, the control automatically creates the **T usage order** table. The control lists all the tools called within the active NC program in chronological order.

Extended tool management enables you to detect the tool requirements in time and thus prevent interruptions during program run.

---

For more information, see the TNC brochures, and visit www.klartext-portal.com

* Turning v2 includes option 50
Programming and operation

Interpolating spindle: interpolation turning

In interpolation turning, the tool’s cutting edge describes a circular motion, with the cutting edge always oriented either toward the center of the circle (outside machining) or away from the center (inside machining). By varying the circle radius and the axial position, any rotationally symmetrical object can be produced in any working plane.

With the interpolation turning cycle, the TNC can create a rotationally symmetrical shoulder defined by its start and end point in the active working plane. The center of rotation is the starting point in the working plane when the cycle is called. The rotational surfaces can be inclined or rounded away from each other.

This cycle can be used only for finishing work and does not permit roughing operations with multiple passes. The machining strategy can be selected as needed: both inside machining and outside machining are possible. The TNC7 and TNC 640 also let you machine any rotationally symmetrical contour (without undercuts).

For certain machining operations, the rotations of the tool spindle must be synchronized with the movement of other axes. This is the case, for example, during the production of external gears via hobbing.

In connection with option 50 (Turning) or option 158 (Turning v2) and option 131 (Spindle Synchronism), the control offers Cycle 880 GEAR HOBBING, which allows you to machine external cylindrical gears or helical gears at any angle. During hobbing, the rotation of the tool spindle and rotary table are synchronized while the gear hob moves axially along the workpiece.

Cycle 880 automatically controls these complex movements and lets you easily enter all of the relevant values. You can utilize the tooth parameters straight out of your drawing, which the cycle then uses to calculate the required five-axis motion.

The TNC 640 (with NC SW 34059x-11 or later) and TNC7 controls support tapping with one or more spindles. The spindles can be coupled for this purpose. Tapping is performed with the master spindle. The coupled slave spindles follow the machining operation.
Programming and operation
Grinding and dressing functions

With the TNC7 and TNC 640, you can even perform grinding operations on your machine. Cycles for jig grinding and dressing allow you to program the respective function with great convenience. The TNC7 and TNC 640 can also superimpose a tool-axis reciprocating stroke onto the programmed movements. Optimized tool management supports the operator during every process, including grinding and dressing. The TNC7 and TNC 640 are therefore the ideal basis for attaining exceptional surface quality and accuracy in your applications.

Jig grinding
Jig grinding is the grinding of a 2D contour. On a milling machine, jig grinding is primarily used for finishing holes or pre-machined contours using a grinding tool. The grinding contour can be programmed in the defined grinding cycles and then machined.

Grinding open and closed contours
You also have the option of superimposing a tool-axis reciprocating stroke onto the tool’s motion. Special cycles allow you to define, start, and stop the reciprocating stroke. The reciprocating stroke enables uniform wear on the grinding tool and precise geometries on ground surfaces.

Dressing
Dressing cycles allow you to “true up” grinding tools inside the machine. During dressing, the grinding tool is machined with a special dressing tool. Klartext cycles are available for dressing the grinding tool’s diameter or profile.

Gear Cutting
Option 157 ID 1237235-01
TNC7 NC SW 81762x-16 or later
TNC 640 NC SW 34059x-09 or later
TNC 620 –
TNC 320 –
Installation by the machine manufacturer
For more information, see the TNC brochures, and visit www.klartext-portal.com

Programming and operation
Gear machining in a single setup

Because gears are typically manufactured on specialized machine tools, time-consuming rechucking is often required.

With the Gear Cutting function, however, you can completely machine spur and helical gears in a single setup using skiving or hobbing operations. If you wish to use the Gear Cutting option in turning mode, then option 50 or option 158 (Turning v2) will need to be enabled. The Gear Cutting function helps you program complex sequences by letting you define just the gear geometry and tool data. The control will take care of the remaining calculations, particularly the complex synchronization of motions. As a result, the production of internal gears turns into an easy-to-master standard application.

Skiving
Skiving owes its current success to its significantly higher efficiency and throughput compared with traditional gear shaping.

With skiving, external and internal gears can be produced on machines with synchronized spindles.

Hobbing
Hobbing is particularly well suited to external gear machining. Its benefits include high throughput and a wide variety of tooth shapes that can be produced with relatively easy-to-make tools.

Lift-off
This option not only makes programming easier but also provides better protection for your equipment. To prevent damage during unforeseen program interruptions (e.g., a power outage), its cycles support optimized lift-off capability. These cycles automatically define the tool direction and path for retraction from the workpiece.
Machine accuracy

KinematicsOpt: easy calibration of rotary axes

Accuracy requirements are becoming increasingly stringent, particularly when it comes to five-axis machining. Complex parts are expected to be machined to exact specifications with reproducible accuracy over extended periods.

The KinematicsOpt function for TNC controls is a key ingredient in helping you meet these high demands. First, a probing cycle automatically measures your machine’s rotary axes with a HEIDENHAIN touch probe.

To measure the rotary axes, a calibration sphere is fixed on the machine table and probed with the HEIDENHAIN touch probe. Prior to probing, you specify the touch point grid and define a different measuring range for each rotary axis.

Based on the measured values, the TNC determines the spatial errors that arise from axis tilting. The cycle then calculates an optimized kinematic machine description that minimizes these errors and saves it as the machine’s kinematic model.

Of course, a detailed log file is also available containing the actual measured values, the measured and optimized dispersion (for the static tilting accuracy), and the actual compensation values.

Together with option 52 (KinematicsComp), the KinematicsOpt option permits compensation for spatial misalignment of the rotary axes and makes 3D compensation of rotary swivel heads on a rotary axis grid possible.

Optimal use of KinematicsOpt requires a highly rigid calibration sphere for reducing the deformations caused by probing forces. For this purpose, HEIDENHAIN offers calibration spheres with highly rigid holders of various lengths.

Calibration spheres are available as accessories:
- KKH 80 Height: 80 mm ID 655475-03
- KKH 250 Height: 250 mm ID 655475-01

KinematicsComp: 3D spatial compensation

Tight workpiece tolerances require high machine accuracy. However, machine tools inevitably exhibit installation- or production-related errors.

The more axes a machine has, the more sources of error there are. The ISO 230-1 standard, for example, lists eight relevant types of error for a linear axis (three component errors and two relevant position errors) and eleven for a rotary axis (six component errors and five relevant position errors). Countering such errors at the mechanical level requires extraordinary design effort. Moreover, these errors are particularly noticeable on five-axis machines and very large machines. What should also not to be neglected is thermal expansion, which can cause highly complex changes in the geometry of machine components.

The KinematicsComp function gives the machine manufacturer a way to significantly improve his machine’s accuracy. The TNC’s standard kinematic model describes the machine’s degrees of freedom and the rotational centers of its rotary axes. The enhanced kinematics description of KinematicsComp additionally includes the option of considering positioning, straightness and angular errors of all axes through compensation value tables. In addition to featuring multidimensional tables, KinematicsComp also uses formulas with which even errors beyond the scope of the rigid body error model can be compensated for. This is done so well that the tool center point (TCP) can exactly follow the ideal nominal contour. Even thermally induced errors can be compensated for after being detected by sensors and the PLC. The spatial errors of the tool center point, for example, can be measured with a laser tracer or laser interferometer and converted into compensation tables. Option 48 (KinematicsOpt) uses KinematicsComp to compensate for the misalignment of rotary axes and for the grid compensation of rotary swivel heads.

KinematicsComp

Option 52 ID 661879-01

TNC7
TNC 640
TNC 620
TNC 320
 NC SW 81762x-16 or later
 NC SW 34059x or later
 NC SW 34056x-03/73498x or later

Installation by the machine manufacturer

For more information, see the TNC brochures, and visit www.klartext-portal.com

Ascertaining geometric error with a laser coordinate measuring device (source: PTB Notification 117)
**Machine accuracy**

CTC: compensation of position errors due to axis couplings

During dynamic acceleration processes, a machine tool's structure undergoes forces capable of deforming machine components, resulting in errors at the tool center point (TCP). The dynamic acceleration of an axis causes not only axial deformations but also deformations that are lateral to the direction of acceleration (due to mechanical axis couplings). This is especially the case if the point of force applied to an axis does not coincide with its center of gravity, thus potentially resulting in pitching motion during braking and acceleration phases. The resulting position error at the TCP in the direction of the accelerated axis and lateral axes is proportional to the amount of acceleration.

If the dynamic position error as a function of the axis acceleration is known from measurements at the TCP then this acceleration-dependent error can be compensated for with the CTC (Cross Talk Compensation) servo-control option in order to prevent negative effects on the surface quality and accuracy of the workpiece.

A grid encoder (KGM) in the plane defined by two mechanically coupled axes can be used to measure the acceleration-dependent position error of these axes. The resulting error at the TCP often depends not only on the acceleration but also on the position of the axes in the working space. The CTC servo-control option can take this into account as well.

---

**Machine accuracy**

PAC: position-dependent adaptation of control parameters

Depending on the positions of the axes in the working space, a machine’s kinematics give rise to variable dynamic behavior that can adversely affect servo-control stability.

To fully exploit the machine’s dynamic performance, you can use the PAC option (Position Adaptive Control) to change machine parameters based on position.

This makes it possible to assign the respective optimal loop gain to defined interpolation points. Additional position-dependent filter parameters can be defined in order to further increase control loop stability.

---

**Cross Talk Compensation**

<table>
<thead>
<tr>
<th>Option 141</th>
<th>ID 800542-01</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>NC SW 81762x-16 or later</td>
</tr>
<tr>
<td>TNC 640</td>
<td>NC SW 34058x-02 or later</td>
</tr>
<tr>
<td>TNC 620</td>
<td>NC SW 34066x-04/73458x-02/81760x-01 or later</td>
</tr>
<tr>
<td>TNC 320</td>
<td>–</td>
</tr>
</tbody>
</table>

Installation by the machine manufacturer

For more information, see the TNC brochures, see the Dynamic Precision Technical Information document, and visit www.klartext-portal.com

---

**Position Adaptive Control**

<table>
<thead>
<tr>
<th>Option 142</th>
<th>ID 800544-01</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>NC SW 81762x-16 or later</td>
</tr>
<tr>
<td>TNC 640</td>
<td>NC SW 34058x-02 or later</td>
</tr>
<tr>
<td>TNC 620</td>
<td>NC SW 34056x-04/73498x-02/81760x-01 or later</td>
</tr>
<tr>
<td>TNC 320</td>
<td>–</td>
</tr>
</tbody>
</table>

Installation by the machine manufacturer

For more information, see the TNC brochures, see the Dynamic Precision Technical Information document, and visit www.klartext-portal.com
The dynamic behavior of machines with moving tables can vary depending on the mass or mass moment of inertia of the clamped workpiece.

The **LAC** option (Load Adaptive Control) enables the control to automatically ascertain the current workpiece mass or mass moment of inertia and the acting frictional forces. In order to ensure an optimum response to changes in machine behavior at different loads, adaptive feedforward controls can be activated, and the control loop gain can be changed based on the load.

In order to allow rapid adjustment in response to sudden changes in the load (e.g., from loading and unloading the workpiece), Cycle 239 is provided for ascertaining the current load status.

In addition to the position-dependent adaptation of control parameters through the PAC option, the **MAC** option (Motion Adaptive Control) provides a way of changing machine parameters based on other input quantities such as speed or motor acceleration. This motion-dependent adaptation of the control parameters makes it possible, for example, to implement speed-dependent adaptation of the \( kv \) factor for drive systems exhibiting stability changes at different traversing speeds.

A further application is the acceleration-dependent change of the tensioning torque between the master and slave axes for master-slave torque control.

With the MAC option, this configuration makes it possible to attain a significantly higher maximum acceleration at rapid traverse, for example, through parameter-based reduction of the tensioning torque at increasing acceleration.

---

**Machine accuracy**

LAC: load-dependent adaptation of control parameters

**Machine accuracy**

MAC: motion-dependent adaptation of control parameters

---

Optimal feedforward control for rotary tables without additional load and with following error within the tolerance band (±0.001°)

With change in load

- Without LAC: with unchanged feedforward control, the following error (±0.008°) is outside of the tolerance band
- With LAC: with feed-forward control and active LAC, the following error is within the tolerance band (±0.001°)

---

<table>
<thead>
<tr>
<th>Load Adaptive Control</th>
<th>Option 143</th>
<th>ID: 800545-01</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>NC SW 81762x-16 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 640</td>
<td>NC SW 34059x-02 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 620</td>
<td>NC SW 34056x-04/73498x-02/81760x-01 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 320</td>
<td>--</td>
<td></td>
</tr>
</tbody>
</table>

Installation by the machine manufacturer

For more information, see the TNC brochures, see the Dynamic Precision Technical Information document, and visit www.klartext-portal.com

---

<table>
<thead>
<tr>
<th>Motion Adaptive Control</th>
<th>Option 144</th>
<th>ID: 800546-01</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>NC SW 81762x-16 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 640</td>
<td>NC SW 34059x-02 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 620</td>
<td>NC SW 34056x-04/73498x-02/81760x-01 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 320</td>
<td>--</td>
<td></td>
</tr>
</tbody>
</table>

Installation by the machine manufacturer

For more information, see the TNC brochures, see the Dynamic Precision Technical Information document, and visit www.klartext-portal.com
Machine accuracy
MVC: active vibration damping

In the presence of low-frequency vibrations on a machine tool, inclined or curved surfaces can often exhibit surface-finish problems in the form of visible shadows or inconsistent contrast. Peaks as small as 1 µm or smaller may be visually noticeable on a workpiece. These disturbances often necessitate surface rework, which comes with additional cost.

Low-frequency disturbances are often caused by elasticity in the drive train, such as vibrations between the motor and the slides, or machine setup vibrations, in which high accelerations of the machine axes cause disturbances in the machine tool through its fastening elements or connected base.

Excitations due to high accelerations can be lowered by reducing the jerk, but this results in longer machining times.

The Machine Vibration Control (MVC) option suppresses low-frequency vibrations via the control loop of the control. To this end, the Active Vibration Damping (AVD) and Frequency Shaping Control (FSC) functions are available with this option. MVC works in two ways: it causes a clean workpiece surface because any vibrations that would be visible on the workpiece surface are suppressed, and it also enables fast and low-vibration milling.

MVC thus increases the productivity of a machine tool and/or improves the surface quality of workpieces.

Machining functions
5-axis simultaneous machining

The TNC provides many powerful functions specifically developed for five-axis simultaneous machining.

The NC programs for five-axis simultaneous machining are created with CAM systems in conjunction with postprocessors. In principle, such programs contain either all of the coordinates of the machine’s existing NC axes or NC blocks with surface normal vectors. During five-axis machining with three linear axes and two additional tilting axes, the tool is always perpendicular to the workpiece surface or is tilted at a specific angle relative to it (inclined tool machining).

Regardless of the type of five-axis programs you wish to run, the TNC performs all of the necessary compensating movements in the linear axes arising from movements in the tilting axes. The TNC’s tool center point management feature (TCPM)—an improvement on the proven TNC function M128—provides optimal tool guidance and prevents contour gouging.

---

**Machine Vibration Control**

<table>
<thead>
<tr>
<th>Option</th>
<th>ID</th>
</tr>
</thead>
<tbody>
<tr>
<td>146</td>
<td>800548-01</td>
</tr>
</tbody>
</table>

**TNC7**
NC SW 81762x-16 or later

**TNC 640**
NC SW 34058x-04 or later

**TNC 620**
NC SW 34066x-04/73498x-02/81760x-01 or later

**TNC 320**

*Installation by the machine manufacturer*

*For more information, see the TNC brochures, see the Dynamic Precision Technical Information document, and visit www.klartext-portal.com*

**Advanced Function Set 2**

<table>
<thead>
<tr>
<th>Option</th>
<th>ID</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>617921-01</td>
</tr>
</tbody>
</table>

**TNC7**
NC SW 81762x-16 or later

**TNC 640**
NC SW 34056x-01 or later

**TNC 620**
NC SW 34056x-01/73498x-01/81760x-01 or later

**TNC 320**

*Installation by the machine manufacturer*

*For more information, see the TNC brochures*
**Machining functions**

**Handwheel superimpositioning: superimpose handwheel positioning during program run**

The **Handwheel Superimpositioning** function (M118) enables you to make manual corrections with the handwheel during program run. This is particularly helpful when you wish to change the inclination angles of rotary axes: in externally written NC programs such changes often cause collisions between the tilting head and the workpiece. You can also use handwheel superimpositioning to adjust the offset compensation in linear axes without having to change the NC program.

**Machining functions**

**Tool compensation: radius-compensated contour precalculation (LOOK AHEAD)**

The **LOOK AHEAD** function in the TNC’s geometry processing system monitors radius-compensated contours for undercuts and intersections, and calculates the tool path in advance starting from the current block. Sections of the contour that would be damaged by the tool are simply not machined (dark areas in the figure) and can be reworked with a smaller tool at a later time. You can also use this function to add tool radius compensation to NC programs created with an external programming system and output as an uncompensated contour. Consequently, inaccuracies in the NC programs arising from calculations in the CAM system can be compensated for.

<table>
<thead>
<tr>
<th>Advanced Function Set 3</th>
<th>Option 21</th>
<th>ID 629254-01</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>Standard function</td>
<td></td>
</tr>
<tr>
<td>TNC 640</td>
<td>Standard function</td>
<td></td>
</tr>
<tr>
<td>TNC 620</td>
<td>NC SW 34056x-01/73498x-01/81760x-01 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 320</td>
<td>Standard function</td>
<td></td>
</tr>
<tr>
<td><strong>Installation</strong></td>
<td>by the machine manufacturer</td>
<td></td>
</tr>
<tr>
<td><strong>For more information</strong></td>
<td>see the TNC brochures</td>
<td></td>
</tr>
</tbody>
</table>
The complex machine movements and normally high traversing speeds of five-axis machining make axis movements difficult to predict. Collision monitoring is thus a valuable function that makes things easier for the machine operator and protects the machine from damage.

In these cases, the machine operator is supported by the TNC’s Dynamic Collision Monitoring (DCM) option. The control interrupts the machining process whenever a collision is imminent, thereby improving safety for both the machine and its operator. This, in turn, helps prevent machine damage and any resulting costly downtime. Unattended shifts also become more reliable.

The DCM option works not only in automatic mode but is also active in manual mode. If, for example, the machine operator is on a collision course during setup, the TNC detects this, stops axis movement, and issues an error message. Before actually machining a part, you can also check for collisions in the simulation, with a real preset and real tools.

Of course, the TNC also shows the machine operator which machine components are at risk, both by means of an error message and graphically. If a collision warning occurs, the TNC permits tool retraction only in directions that will increase the clearance between the colliding objects.

The TNC 640 (with NC SW 34059v.05 or later) and TNC7 controls also offer a convenient way to import collision objects from standard CAD models (e.g., STL) into the control as M3D data. This permits highly detailed modeling of the machine components and better utilization of the machine’s working space.

On the TNC 640 (with NC SW 34059v.11 or later) and the TNC7, clamping setups can be incorporated in *.cfg, *.m3d, or *.stl formats, including via KlarText commands.

**Machining functions**

**Dynamic Collision Monitoring (DCM)**

DCMv2 includes all features of option 40. The expanded collision monitoring of the TNC7 not only prevents tools from colliding with machine components, but also with fixtures. The TNC7 features a probing function that graphically and interactively assists you in determining the exact position of the fixture. That way you no longer have to worry about either the sequence or the actual probing functions. Colored arrows in a virtualized work envelope show you whether the position is suitable and the probing operation can be performed. This graphically supported measurement can be used for any workholding equipment. An accurate 3D model is all that is needed.

---

**DCMv2 includes all features of option 40.**

The expanded collision monitoring of the TNC7 not only prevents tools from colliding with machine components, but also with fixtures. The TNC7 features a probing function that graphically and interactively assists you in determining the exact position of the fixture. That way you no longer have to worry about either the sequence or the actual probing functions. Colored arrows in a virtualized work envelope show you whether the position is suitable and the probing operation can be performed. This graphically supported measurement can be used for any workholding equipment. An accurate 3D model is all that is needed.

---

**Dynamic Collision Monitoring**

**Option 40**

| TNC 640 | NC SW 81762x.06 or later |
| TNC 620 | NC SW 34059v.06 or later |
| TNC 320 | – |

Installation by the machine manufacturer

For more information, see the TNC brochures, and visit www.klarText-portal.com

---

**DCMv2**

**Option 140**

| TNC 640 | NC SW 81762x.16 or later |
| TNC 620 | – |
| TNC 320 | – |

Installation by the machine manufacturer

For more information, see the TNC7 brochure, and visit www.klarText-portal.com
Machining functions
Global Program Settings

The global program settings, which come into play particularly in large-scale mold making, are available in the Program Run and MDI modes. This option lets you define global and overriding coordinate transformations and settings for the NC program without actually changing the NC program.

You can change the Global Program Settings during a program stop, even mid-program. A clearly structured form is provided for this. Upon starting, the TNC then moves as needed to a new position in accordance with a positioning logic that you can define. The following functions are available:

- Additional rotation based on basic rotation or 3D basic rotation
- Shifting the workpiece preset in a single axis
- Mirroring individual axes
- Additional shift of an already shifted workpiece datum
- Rotation about the active tool axis
- Handwheel superimposition
- Feed rate factor

Handwheel superimposition is possible in various coordinate systems:

- Machine coordinate system
- Workpiece coordinate system (the active basic rotation is taken into account)
- Tilted coordinate system

You can select the desired coordinate system in a clearly structured form.

Adaptive Feed Control (AFC)
automatically regulates the feed rate of the TNC based on the present spindle power and other process data. During a teach-in cut, the TNC records the maximum occurring spindle power. Prior to machining, you define the limits (in a table within which the TNC is allowed to influence the feed rate in “control” mode. Naturally, various overload reactions can be specified as well, including by your machine manufacturer.

The TNC’s Adaptive Feed Control option offers a series of benefits:

- Optimization of the machining time
  Particularly in the case of cast parts, oversizes and material irregularities (cavities) can occur to varying degrees. By controlling the feed rate, the control strives to stay within the bounds of the previously “learned” maximum spindle power throughout the entire machining time. Increasing the feed rate in areas requiring less material removal shortens the overall machining time.

- Tool monitoring
  The Adaptive Feed Control option continuously compares the spindle power with the feed rate. As a tool becomes blunt, the spindle power increases, causing the TNC to reduce the feed rate. As soon as the feed rate falls below a specified minimum, the TNC reacts with an NC stop or warning, or automatically replaces the tool. This prevents secondary damage after tool breakage or wear.

- Protecting machine components
  When the feed rate is reduced in response to the exceeded learned maximum spindle power, machine components undergo less strain. This protects the main spindle from overloading.

Adaptive Feed Control (AFC) automatically regulates the feed rate of the TNC based on the present spindle power and other process data. During a teach-in cut, the TNC records the maximum occurring spindle power. Prior to machining, you define the limits (in a table within which the TNC is allowed to influence the feed rate in “control” mode. Naturally, various overload reactions can be specified as well, including by your machine manufacturer.

The TNC’s Adaptive Feed Control option offers a series of benefits:

- Optimization of the machining time
  Particularly in the case of cast parts, oversizes and material irregularities (cavities) can occur to varying degrees. By controlling the feed rate, the control strives to stay within the bounds of the previously “learned” maximum spindle power throughout the entire machining time. Increasing the feed rate in areas requiring less material removal shortens the overall machining time.

- Tool monitoring
  The Adaptive Feed Control option continuously compares the spindle power with the feed rate. As a tool becomes blunt, the spindle power increases, causing the TNC to reduce the feed rate. As soon as the feed rate falls below a specified minimum, the TNC reacts with an NC stop or warning, or automatically replaces the tool. This prevents secondary damage after tool breakage or wear.

- Protecting machine components
  When the feed rate is reduced in response to the exceeded learned maximum spindle power, machine components undergo less strain. This protects the main spindle from overloading.
Machining functions
Active Chatter Control (ACC)

Strong forces come into play during roughing (power milling). Depending on the rotational tool speed, the resonances in the machine tool, and the chip volume (metal removal rate during milling), a phenomenon known as “chatter” may occur. Chatter induces heavy strain on the machine and causes ugly marks on the workpiece surface. It also increases the progress and irregularity of tool wear, even leading to tool breakage under extreme circumstances.

With Active Chatter Control (ACC), HEIDENHAIN now provides an effective control function for reducing a machine’s tendency to chatter. This control function is particularly advantageous when used during heavy machining. Thanks to ACC, substantially higher removal rates are possible. Depending on the type of machine, the removal rate can be increased by up to 25 percent or more. At the same time, you are reducing strain on the machine and increasing tool life.

The TNC 640 (with NC SW 34059x-11 or later), the TNC 620 (with NC SW 81760x-07 or later) and the TNC 7 provide OCM cycles for chamfering or deburring and for standard shapes.

Efficient machining strategies are a key basis for economical NC manufacturing. Roughing processes, in particular, offer great potential for optimization because they usually make up a sizeable portion of the total machining time.

Process reliability and maximum removal rates require optimal adaptation of the cutting data to the characteristics of the tool and workpiece material. For this purpose, Optimized Contour Milling (OCM) provides a cutting data calculator that draws on an extensive integrated material database. The operator can adapt the automatically calculated cutting values with regard to the mechanical and thermal load on the tool. Tool service life can be managed with process reliability even at the highest possible removal rates.

With OCM, you can rough out any pocket and stand with high process reliability and lower tool wear thanks to highly consistent process conditions. You simply program the contour as usual directly in Klartext or make use of the convenient CAD Import function. The control then automatically calculates the complex movements required for maintaining consistent process conditions. OCM takes void areas into account, which significantly reduces machining times (with NC software version 16 or later).

Advantages of OCM over conventional machining:
• Reduced thermal load on the tool
• Superior chip removal
• Consistent tool contact conditions (higher cutting parameters and higher material removal rates)

OCM is an effective, reliable, and convenient way to improve your throughput:
• Create programs on the shop floor for any pocket or stand
• Significantly higher machining speed
• Considerable reduction in tool wear
• More chips in less time

The OCM option provides practical cycles for roughing, side-wall finishing, and floor finishing.

Machining functions
Optimized Contour Milling (OCM)

In the following machining example, both the machining time and tool wear were reduced by a factor of three.

Tool: VHM end mill Ø 10 mm
Workpiece material: 1.4104

Optimized Contour Milling Option 167 ID 1289547-01
TNC7
TNC 640
TNC 620
TNC 320
NC SW 81760x-07 or later
NC SW 81762x-16 or later
NC SW 34059x-10 or later
NC SW 34056x-04/73498x-02/81760x-01 or later
–
Installation by the machine manufacturer
For more information, see the TNC brochures, and visit www.klartext-portal.com
Machining functions

Component Monitoring

Overload often leads to damaged machine components, and therefore to machine downtime. For example, the spindle bearing is often subject to a very high load in machining strategies that have been optimized for maximum efficiency, and can therefore sustain undetected damage. Component Monitoring can detect these dangers, and issue warnings or even stop the machine tool if necessary. Thanks to continuous monitoring of the bearing load and visualization of these values, the machining processes can be optimized correspondingly.

However, the production quality of a machine tool is influenced not only by overload. Components that are constantly under load, such as guideways or recirculating ball screws, are subject to wear and thus have an influence on the finished product. The TNC7 uses the Component Monitoring option to monitor and document the momentary machine status. The machine manufacturer can read and evaluate these data, and react using predictive maintenance, thereby avoiding unplanned machine downtimes. The MONITORING HEATMAP function allows you to individually color, from within the NC program, the workpiece view of the concurrent simulation. That way you quickly see in the workpiece view where components were subject to a strong load.

Cycle 238, MEASURE MACHINE STATUS*, is part of software option 155 (Component Monitoring). This cycle allows you to ascertain and record the current machine status. Through data comparison, deviations in machine parameters can be detected and documented over extended periods of time, allowing you to track the machine aging processes.

* Available on the following controls: TNC7, TNC 640 (with NC SW 34059x-10 or later), TNC 620 (with NC SW 81760x-07 or later)

Process Monitoring

The fully-integrated process monitoring capability of the TNC7 is a unique feature for making production processes safer and more reliable. Reliable process monitoring is essential for avoiding damage resulting from unavoidable disturbances during automated production. For example, if breakage inspection is performed for tools during the machining operations, then valuable productive time is lost. Here the operator can save time and costs with the fully-integrated process monitoring feature from HEIDENHAIN. Process monitoring entails the recording of one or more reference machining operations. During subsequent operations, the control monitors the process and can react to deviations in a timely manner. For example, if there is a disturbance in the process, the machine can be stopped and the tool disabled. In conjunction with pallet machining, the next workpiece can automatically be started with a replacement tool. That way the machine tool remains productive despite the disturbance, and no manual intervention is necessary. Process Monitoring uses signals from within the control and does not require any additional sensors.

Process Monitoring increases safety and efficiency:

- Detect deviations from a reference machining run
- Enjoy reliable monitoring thanks to robust program synchronization down to the block level
- Ensure productivity through an extensive range of possible reactions, such as inserting a replacement tool
- Easy checking of the process result via 3D visualization of the workpiece and a 2D graph
- Easy programming
- Benefit from zero installation effort

Component Monitoring

Option 155
ID 1229683-01

<table>
<thead>
<tr>
<th>TNC7</th>
<th>TNC 640</th>
<th>TNC 620</th>
<th>TNC 320</th>
</tr>
</thead>
<tbody>
<tr>
<td>NC SW 81762x-16 or later</td>
<td>NC SW 81760x-09 or later</td>
<td>NC SW 81760x-06 or later</td>
<td>–</td>
</tr>
</tbody>
</table>

Installation by the machine manufacturer

For more information, see the Information for the Machine Tool Builder brochures

Process Monitoring

Option 168
ID 1302488-01

<table>
<thead>
<tr>
<th>TNC7</th>
<th>TNC 640</th>
<th>TNC 620</th>
<th>TNC 320</th>
</tr>
</thead>
<tbody>
<tr>
<td>NC SW 81762x-16 or later</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
</tbody>
</table>

Installation by the machine manufacturer

For more information, see the TNC7 brochure, and visit www.klartext-portal.com
Machining functions

3D-ToolComp: 3D radius compensation based on tool inclination

3D-ToolComp is a powerful option for three-dimensional tool radius compensation. In a compensation table, angle-dependent delta values are defined describing the tool’s deviation from an ideal circular form (see figure).

The TNC then connects the radius value defined for the tool’s current point of contact with the workpiece. In order for the exact point of contact to be determined, the NC program must be created by the CAM system with surface-normal blocks (LN blocks). The surface-normal blocks specify the theoretical center point of the radius cutter and the tool orientation relative to the workpiece surface as needed.

The ideal method is to create the compensation table fully automatically. This is done by measuring the shape of the tool with a laser system and a special cycle such that the TNC can make immediate use of the table. If the form errors of the tool are already available to you in a calibration chart provided by the tool manufacturer, then you can also create a compensation table manually.

Measuring 3D geometries

The control features a cycle that allows you to measure points on 3D geometries. To do so, you simply enter the point to be measured, along with its coordinates and associated normal vectors, into Cycle 444, 3D PROBING. After probing, the TNC automatically calculates whether the measured point is within a specified tolerance. You can then call the result via a compensation table manually.

To obtain even more accurate results, you can perform a 3D calibration of the touch probe prior to running Cycle 444. The cycle then compensates for the switching behavior of the individual touch probe in all directions.

Software option 92 (3D-ToolComp) is required for 3D calibration.

Communication

HEIDENHAIN OPC UA NC Server: standardized communication

OPC UA NC Server (options 56 to 61)

Anyone wishing to digitally network their manufacturing environment needs effective technology with an assured future. Discover the HEIDENHAIN OPC UA NC Server, which provides HEIDENHAIN controls with an interface based on the OPC UA standard. This internationally standardized and widespread communication protocol makes it fast and easy to connect machines to your production IT setup. OPC UA is a cross-operating-system interface that can connect the HEIDENHAIN control not only to widespread Windows systems but also to Linux-based systems and Apple computers with macOS.

Numerous developer toolkits are available for OPC UA, RemoTools SDK is not needed. This option’s standardized protocol, freely selectable toolkit, and application-oriented HEIDENHAIN information model allow standard software solutions and highly individualized applications to be developed with significantly less time to market.

The HEIDENHAIN OPC UA NC Server supports the following functions:

- Reading and writing variables
- Subscribing to value changes
- Executing methods
- Subscribing to events
- Transferring of files

Six SIK options are available for the HEIDENHAIN OPC UA NC Server. Each of the six SIK options enables an incoming OPC UA connection. Multiple connections can be configured and activated as needed on the control.

Explore the full HEIDENHAIN OPC UA NC Server yourself and test new OPC UA applications on a virtual machine such as the HEIDENHAIN programming station. Current versions of the programming station feature a free, full-version demo of the OPC UA NC Server.

OPC UA NC Server (options 56 to 61)

<table>
<thead>
<tr>
<th>Classification</th>
<th>Control</th>
<th>Software</th>
</tr>
</thead>
<tbody>
<tr>
<td>• Machine identification</td>
<td>TNC7</td>
<td>81.762x-06 or later</td>
</tr>
<tr>
<td>• Production data acquisition</td>
<td>TNC 640</td>
<td>34056x-16 or later</td>
</tr>
<tr>
<td>• Machine messages</td>
<td>TNC 620</td>
<td>81.762x-08 or later</td>
</tr>
<tr>
<td>• Program sequence check</td>
<td>TNC 640</td>
<td>81.762x-10 or later</td>
</tr>
<tr>
<td>• File transfer</td>
<td>TNC 620</td>
<td>34056x-11 or later</td>
</tr>
<tr>
<td>• Remote management of file systems</td>
<td>TNC7</td>
<td>81.762x-12 or later</td>
</tr>
<tr>
<td>• Machine StateMonitor</td>
<td>TNC 640</td>
<td>34056x-13 or later</td>
</tr>
<tr>
<td>• Automation</td>
<td>TNC 620</td>
<td>81.762x-14 or later</td>
</tr>
<tr>
<td>• OEM enhancements (e.g., PLC)</td>
<td>TNC7</td>
<td>81.762x-16 or later</td>
</tr>
<tr>
<td>• Tool data management</td>
<td>TNC 640</td>
<td>34056x-17 or later</td>
</tr>
<tr>
<td>• Connectivity</td>
<td>TNC 620</td>
<td>81.762x-18 or later</td>
</tr>
</tbody>
</table>

For more information, visit www.heidenhain.de/opcua-nc-server

For more information, see the TNC brochures, and visit www.klartext-portal.com
Paperless job management requires the seamless exchange of data about all process steps in the production process. The HEIDENHAIN DNC option allows a Windows application to access data on the control and change them as needed. Possible fields of application include the following:

- Software solutions for controlling the manufacturing process
  - Machine data acquisition systems and production data acquisition systems (MDA/PDA)
  - Connection to higher-level ERP/MES systems
  - Preventive maintenance planning based on the machine’s actual condition
- Standardized or customized PC software
  - Greater process reliability and equipment availability
  - Error reporting systems (e.g., notifying the customer’s smartphone when the current machining process encounters problems)
  - Current status overviews of all production machines
  - Database creation for extensive data mining

Since HEIDENHAIN DNC has been around for many years already, the interface is also particularly suitable for connecting older machine tools.

RemoTools SDK development package
In order for you to make use of the HEIDENHAIN DNC software interface, HEIDENHAIN offers the RemoTools SDK software development package. RemoTools SDK provides a Microsoft COM component for development environments on Windows operating systems, thus enabling communication with the HEIDENHAIN control. The COM component is registered in the Windows operating system during installation of RemoTools SDK.

Remote Desktop Manager: view and remotely operate external computers
A typical work day may often require data entry into planning and control systems or diagnostic testing with Windows-based software. The Remote Desktop Manager option provides the operator with the opportunity to operate one or more Windows PCs directly from the TNC. This option fully integrates the Windows PC operating interface into the control interface on the TNC’s screen.

This functionality is available regardless of whether the Windows PC is an industrial PC (e.g., the IPC 306) in the machine’s control cabinet, in the local network as a server, or at the operator’s office workstation.

Possible applications range from centralized job, tool, and NC program management to the remote operation of CAD/CAM systems from the machine. With this option, the machine tool operating panel becomes a versatile and efficient workspace for tasks ranging from specialized manufacturing processes to decentralized order processing.

Extended Workspace Comfort
With Extended Workspace Comfort, you can add a second 19-inch HERO screen next to or above the control screen of the TNC 640. While the current NC program is displayed on the control screen, Extended Workspace Comfort offers, for example, a view of an office PC that is connected via Remote Desktop Manager.

Extended Workspace Compact
ExtendedWorkspaceCompact lets any additional applications run in an extra window on a 24-inch touchscreen of the TNC 640, for example, the screen of a Windows PC can be shown. Its position alongside the familiar control screen gives you an optimum overview.

Extended Workspace
With Extended Workspace, you can add a second 19-inch HERO screen next to or above the control screen of the TNC 640. While the current NC program is displayed on the control screen, Extended Workspace offers, for example, a view of an office PC that is connected via Remote Desktop Manager.

Extended Workspace Compact
Extended Workspace Compact lets any additional applications run in an extra window on a 24-inch touchscreen of the TNC 640; for example, the screen of a Windows PC can be shown. Its position alongside the familiar control screen gives you an optimum overview.

For more information, see the TNC brochures, see the Connected Machining brochure.
Adaptation to the machine
Additional control loops

The number of enabled control loops depends on the SIK being used and on any additionally enabled control loops (which can be ordered anytime as needed).

Additional control loops can be enabled either singly or in groups. Through combinations of control-loop groups and single control loops, any number of control loops can be enabled.

The maximum possible number of control loops depends on the control:
- TNC7: 24 control loops
- TNC 640: 24 control loops
- TNC 620: 8 control loops
- TNC 320: 6 control loops

Controls featuring integrated functional safety (FS) from HEIDENHAIN can attain Safety Integrity Level 2 (SL 2) as per EN 61508, and Performance Level “d”, Category 3, as per EN ISO 13849-1. In these standards, safety-related systems are assessed based on criteria such as the failure probabilities of integrated components and subsystems. This modular approach aids the manufacturers of safety-related machines in implementing their systems, since they can then build upon prequalified subsystems.

The controls with functional safety (FS) are based on two redundant, mutually independent safety channels. All safety-relevant signals are collected, processed, and output over two channels. Errors are detected through a reciprocal data comparison of the two channels’ states. Consequently, the occurrence of a single error in the control does not cause a loss in safety functionality.

For Gen 3 drives, integrated functional safety (FS) is enabled through software options and is scalable. When purchasing these options, you can enable the exact number of safe control loops you actually need. For this purpose, new software options (160 to 166) are being introduced exclusively for the Gen 3 drives. Up to now, integrated functional safety (FS) has been enabled in conjunction with a PLB 62xx FS.

For Gen 3 drives, integrated functional safety (FS) is enabled through software options and is scalable. When purchasing these options, you can enable the exact number of safe control loops you actually need. For this purpose, new software options (160 to 166) are being introduced exclusively for the Gen 3 drives. Up to now, integrated functional safety (FS) has been enabled in conjunction with a PLB 62xx FS.

Control-loop groups

| 4 Additional Axes | Option 0 364540-01 |
| Additional Axes | Option 1 364540-01 |
| Additional Axes | Option 2 364540-01 |
| Additional Axes | Option 3 364540-01 |
| Additional Axes | Option 4 364540-01 |
| Additional Axes | Option 5 364540-01 |
| Additional Axes | Option 6 364540-01 |
| Additional Axes | Option 7 364540-01 |

Option number | Option | ID | Comment
--- | --- | --- | ---
160 | Integrated FS: Basic | 1249928-01 | Only for Gen 3: enabling of functional safety (FS) and enabling of four safe control loops
161 | Integrated FS: Full | 1249929-01 | Only for Gen 3: enabling of functional safety (FS) and enabling of the maximum number of safe control loops (≥ 10)
162 | Add. FS Ctrl. Loop 1 | 1249930-01 | Only for Gen 3: additional safe control loop 1
163 | Add. FS Ctrl. Loop 2 | 1249931-01 | Only for Gen 3: additional safe control loop 2
164 | Add. FS Ctrl. Loop 3 | 1249932-01 | Only for Gen 3: additional safe control loop 3
165 | Add. FS Ctrl. Loop 4 | 1249933-01 | Only for Gen 3: additional safe control loop 4
166 | Add. FS Ctrl. Loop 5 | 1249934-01 | Only for Gen 3: additional safe control loop 5
169 | Add. FS Full | 1316991-01 | Enabling all remaining FS control loops

Software option 169 enables all remaining FS axes if the following requirements are fulfilled:
- Software option 160 is enabled
- Software options 162 to 168 are enabled

For more information, see the Technical Information documents
Adaptation to the machine
Synchronized axes: gantry axes, tandem tables

Synchronized axes are synchronously moving axes that are programmed using the same axis designation.

With HEIDENHAIN controls, parallel axis systems (gantry axes) such as those on portal-type machines or tilting tables can be moved synchronously thanks to high-accuracy, dynamic position control. The fast and particularly precise positioning movements are perfectly harmonized, thus enabling five-axis simultaneous motion for highly demanding tasks. Multiple slave axes can be assigned to a single master gantry axis.

Master-slave torque systems are typically used when large parts need to be moved or when rack and pinion drive systems require pre-stressing for backlash-free motion. Up to six drive systems can be combined into a single master-slave network and mutually pre-stressed as desired. As a result, fast and precise axis positioning becomes attainable even on large machine tools.

Adaptation to the machine
Python OEM Process: realize special functions

The Python OEM Process option gives machine manufacturers a powerful tool for using an object-oriented programming language within the control. Python is an easy-to-learn script language supporting all necessary high-level language elements. Python OEM Process can be employed universally for machine functions, complex calculations, and the display of special user interfaces. User-specific or machine-specific solutions can be efficiently implemented. Regardless of whether you intend to create special algorithms for specialized functions or separate solutions (e.g., interfaces for machine maintenance software), a myriad of existing Python- and GTK-based functions are at your disposal.

You can integrate your newly created applications through the PLC in familiar PLC windows, or you can display them in your own windows incorporated into the TNC interface, even making them as large as the control’s full screen.

Simple Python scripts (e.g., for display screens) can also be executed without enabling Python OEM Process (software option 46). 10 MB of dedicated memory are reserved for this function. For more information, refer to the “Python in HEIDENHAIN Controls” Technical Manual.
Adaptation to the machine

Double Speed: short control-loop cycle times for direct drive motors

Single-speed control loops are usually sufficient for linear motors, torque motors, and conventional axes. **Double-speed control loops**, however, are the go-to choice for HSC spindles and difficult-to-control axes. In the control’s default settings, all axes are set to single speed. Each axis that is switched from single speed to double speed can reduce the number of available control loops by one. PWM frequencies greater than 5 kHz require double-speed control loops. Option 49 must be enabled for this.

Double-speed control loops permit higher PWM frequencies and shorter speed-controller cycle times, thereby enabling improved current control for spindles and increased controller performance for linear and torque motors.

**Control loop cycle times**

<table>
<thead>
<tr>
<th></th>
<th>Single speed</th>
<th>Double speed (with option 49)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Fine interpolation</strong></td>
<td>0.2 ms</td>
<td>0.1 ms</td>
</tr>
<tr>
<td><strong>Position controller</strong></td>
<td>0.2 ms</td>
<td>0.1 ms</td>
</tr>
<tr>
<td><strong>Speed controller</strong></td>
<td>0.2 ms</td>
<td>0.1 ms</td>
</tr>
<tr>
<td><strong>Current controller</strong></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

\[\frac{f_{PWM}}{T_{INT}}\]

- 3333 Hz: 120 μs
- 4000 Hz: 120 μs
- 5000 Hz: 100 μs
- 6666 Hz: 100 μs
- 8000 Hz: 75 μs with option 49
- 10 000 Hz: 75 μs with option 49

**Double-Speed Axes**

<table>
<thead>
<tr>
<th>Machine</th>
<th>Option 49</th>
<th>ID 032223-01</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>NC SW 81760x=16 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 640</td>
<td>NC SW 34059x=01 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 620</td>
<td>NC SW 34058x=01/73498x=01/81760x=01 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 320</td>
<td>–</td>
<td></td>
</tr>
</tbody>
</table>

Installation by the machine manufacturer

For more information, see the Information for the Machine Tool Builder brochures

---

Adaptation to the machine

OEM Option

Machine manufacturers often equip their machine tools with added useful functions stored in the control’s configuration parameters (e.g., PLC). These functions are then offered to the operator as options. To give the operator the greatest possible flexibility in enabling these options, HEIDENHAIN can reserve a range within the options menu (SIK menu) for use by the machine manufacturer as desired.

Options 101 to 130 provide thirty options that can be enabled by the machine manufacturer and enabled through his own PLC program. A particular benefit is the operator’s ability to enable options via the SIK menu without the need for on-site support by the machine manufacturer.

**OEM Option**

<table>
<thead>
<tr>
<th>Options 101 to 130</th>
<th>ID 579661-01 to ID 579661-30</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>–</td>
</tr>
<tr>
<td>TNC 640</td>
<td>NC SW 34059x=02 or later</td>
</tr>
<tr>
<td>TNC 620</td>
<td>–</td>
</tr>
<tr>
<td>TNC 320</td>
<td>–</td>
</tr>
</tbody>
</table>

Installation by the machine manufacturer

Further information –
Adaptation to the machine

RTC: real-time coupling function for synchronizing axes and spindles

The Real-Time Coupling function (RTC) enables the cyclical calculation of a position offset for an axis based on the actual and nominal values of any other axes within the system. This function allows complex, simultaneous movements of multiple NC or PLC axes to be implemented. The interdependencies of the axes are defined in mathematical formulas. Possible applications, for example, are in PLC axes that need to be synchronized with an NC axis during a tool change in order to avoid collisions with the tool holders. The machine manufacturer defines these movements with RTC. The real-time coupling function makes complex traversing movements possible through the coupling of principal and secondary axes. Many new solutions are thus made possible, ranging from process-specific movements to tool changes with special requirements.

Synchronizing Functions

<table>
<thead>
<tr>
<th>Synchronizing Functions</th>
<th>Option 135</th>
<th>ID 1085731-01</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>NC SW 81762x-16 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 640</td>
<td>NC SW 34056x-04 or later</td>
<td></td>
</tr>
<tr>
<td>TNC 620</td>
<td>–</td>
<td></td>
</tr>
<tr>
<td>TNC 320</td>
<td>–</td>
<td></td>
</tr>
</tbody>
</table>

Installation by the machine manufacturer

For more information, see the Information for the Machine Tool Builder brochures

Software

TNCremo: programs for data transfer

The free PC software package TNCremo supports the operator in transferring data from a PC to HEIDENHAIN controls or programming stations. TNCremo lets you bidirectionally transfer externally stored machining programs, tool tables, pallet tables, and backups of the hard drive, as well as query the operating status of the machine.

Functions:
- Data transfer and file management
- Creation of screenshots of the control
- Reading out of the control log
- Data backup for the control
- Creation of service files

In addition to the features already familiar to you from TNCremo, TNCremoPlus can transfer the current content of the control’s screen to the PC (live screen). This allows you to implement convenient machine monitoring.

TNCremo

Free download

TNCremoPlus

Enabling module requires a fee ID 340447-xx

TNC7
NC SW 81762x-16 or later
NC SW 34056x-01 or later
NC SW 34056x-01/73498x-01/81760x-01 or later
NC SW 34055x-01/771851-01 or later

Installation by the operator

For more information, visit www.klartext-portal.com
RemoteAccess: remote diagnostics for HEIDENHAIN controls

PC software for remote diagnostics, monitoring and operation
RemoteAccess grants quick and easy access to HEIDENHAIN controls that are installed within the same local network (intranet).
RemoteAccess offers the following functions:
• Display of the control’s user interface on the PC
• Operating the control directly through the live view as well as with the integrated keyboard
• HEIDENHAIN PC tools are integrated automatically
• Can be enhanced with OEM-specific applications

Single station license ID 1339577-01
Network license (14 stations) 1339577-02
Network license (20 stations) 1339577-03

Extension: Secure Remote Access (SRA)
The Secure Remote Access enhancement permits setting up an encrypted remote connection to a HEIDENHAIN control via the Internet.
Once the SRA connection has been set up, RemoteAccess behaves like a local network connection.
Possible applications when using SRA:
• User support
• Online seminars
• Diagnostics and service
• Other OEM services
Secure Remote Access enhancement ID 1356741-01

RemoteAccess in the corporate network
Remote connection over PC remote control software

PLCdesign: PLC program development software
The PLCdesign software is a convenient way to create PLC programs. It also comes with an extensive selection of sample PLC programs.

Functions
• Management of all project files with integrated project management in an intuitive tree structure
• Creation of PLC programs in IL
• Integrated editor with syntax highlighting for PLC programs
• Context-sensitive output windows
• Extensive help functions
• PLC beautifier
• Soft-key creation and generation
• Postprocessor for resolving symbolic operands in generic files
• Support for machine-specific configuration
• Uploading of all files to the control
• Debug functions:
  – Source text monitoring
  – Storage table
  – Monitoring list for symbolic operands
• Source-code browser for input wizard and navigation
• Supports source-code control systems

RemoteAccess
CD-ROM with dongle
ID 340449-xx
ID 340454-xx
ID 340455-xx

TNC7
NC SW 81762x-16 or later
TNC 640
NC SW 34058x-01 or later
TNC 620
NC SW 34056x-01/73498x-01/81760x-01 or later
TNC 320
NC SW 34055x-01/771851-01 or later

PLCdesign
NC SW 81762x-16 or later
NC SW 34058x-01 or later
NC SW 34056x-01/73498x-01/81760x-01 or later
NC SW 34055x-01/771851-01 or later

Installation by the machine manufacturer
Further information

For more information, see the Information for the Machine Tool Builder brochures
Software
StateMonitor: collect and evaluate machine data

The StateMonitor software records and visualizes the statuses of machines in a manufacturing environment. By evaluating important data, such as the current machine status, machine messages, override positions, and utilization history, StateMonitor provides in-depth information about the machine’s utilization level. Based on these collected data, it is possible to uncover optimization potential.

StateMonitor can record and visualize the following information:
- Operating modes
- Override positions (spindle, rapid traverse, feed rate)
- Program status and program name, as well as subprograms if applicable
- Program run time
- SIK number and software number
- Machine messages
- Maintenance entries and the maintenance history
- Job times
- Other machine-specific signal values

Machines with different controls can be connected to StateMonitor. The software supports the protocol types HEIDENHAIN DNC, OPC UA, umati, MTConnect, Modbus TCP and FOCAS.

For more information, please contact the team of the Digital Shop Floor from HEIDENHAIN.

StateMonitor can record and evaluate the following information:
- Operating modes
- Override positions (spindle, rapid traverse, feed rate)
- Program status and program name, as well as subprograms if applicable
- Program run time
- SIK number and software number
- Machine messages
- Maintenance entries and the maintenance history
- Job times
- Other machine-specific signal values

Installation by the machine manufacturer or end user

For more information, visit www.digital-shop-floor.heidenhain.com and www.klartext-portal.com, and see the Digital Shop Floor flyer.
Software

KinematicsDesign: create machine kinematic models

KinematicsDesign is a PC software application for the graphically supported creation of kinematic descriptions for HEIDENHAIN controls. It also offers a convenient way to configure and start using the DCM collision monitoring function.

KinematicsDesign can create all objects of the kinematic chain, transfer them to and from the control, and edit them. Extensive visualization possibilities allow the operator to display the information that he currently needs in a clear format. Transformations, collision objects, and designations, for example, can be shown or hidden independently of each other. Collision objects can be shown as wire models, transparently, in full, etc. Colored highlighting makes it easy in KinematicsDesign to discern which collision objects have been checked in relation to each other or are excluded from the collision inspection. These functions are also available in the KinematicsDesign tool directly in the control.

KinematicsDesign visualizes the kinematic model with the current axis positions of a connected control (or programming station) or with manually entered axis positions and displays collisions. In addition, the current tool or a specified tool from a tool table can be shown along with a kinematic model of its tool carrier. KinematicsDesign can import kinematic models from the iTNC 530 and convert them into the format of the NCK controls, as well as generate clamping equipment and tool carrier kinematic models.

The TNC7 and TNC 640 let you import collision objects from a CAD file and incorporate them as M3D data into the machine kinematics. For the creation of the M3D data, the installation package of KinematicsDesign contains the free-standing PC program M3D Converter.

M3D Converter

This tool can upload, check, repair, simplify, and optimize Step and STL files, thereby preparing them for use as collision objects. In the process, the operator can always keep the difference from the original object in view. M3D Converter requires a software release module (ID 1124969-xx).

<table>
<thead>
<tr>
<th>M3D Converter</th>
<th>Enabling module</th>
</tr>
</thead>
<tbody>
<tr>
<td>ID 1124969-xx</td>
<td>requires a fee</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>TNC7</th>
<th>TNC 640</th>
<th>TNC 620</th>
<th>TNC 320</th>
</tr>
</thead>
<tbody>
<tr>
<td>NC SW 81762x-16 or later</td>
<td>NC SW 34056x-01 or later</td>
<td>NC SW 34056x-01/73498x-01/81760x-01 or later</td>
<td>NC SW 34055x-01/771851-01 or later</td>
</tr>
</tbody>
</table>

Installation by machine manufacturers or servicing providers

For more information, see the Information for the Machine Tool Builder brochures

Software

CycleDesign: save NC subprograms as cycles

For frequently recurring operations, HEIDENHAIN controls provide you with NC subprograms containing configurable parameters. These subprograms are referred to as cycles. The TNC guides you through the process of entering these parameters with prompts, questions, and help graphics.

You access the cycles on the control by pressing the CYCL DEF key.

Using CycleDesign, you can even incorporate your own NC subprograms as cycles into the soft-key structure of the control (or the menu structure of the TNC7). You can choose whether to add your cycles to the HEIDENHAIN cycle bar on the side or completely replace the HEIDENHAIN cycle bar.

CycleDesign lets you save this cycle data in the control’s memory.

<table>
<thead>
<tr>
<th>CycleDesign</th>
<th>Registration required</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>NC SW 81762x-16 or later</td>
</tr>
<tr>
<td>TNC 640</td>
<td>NC SW 34056x-01 or later</td>
</tr>
<tr>
<td>TNC 620</td>
<td>NC SW 34056x-01/73498x-01/81760x-01 or later</td>
</tr>
<tr>
<td>TNC 320</td>
<td>NC SW 34055x-01/771851-01 or later</td>
</tr>
</tbody>
</table>

Installation by operators or machine manufacturers

For more information, see the Information for the Machine Tool Builder brochures

<table>
<thead>
<tr>
<th>KinematicsDesign</th>
<th>Registration required</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>NC SW 81762x-16 or later</td>
</tr>
<tr>
<td>TNC 640</td>
<td>NC SW 34056x-01 or later</td>
</tr>
<tr>
<td>TNC 620</td>
<td>NC SW 34056x-01/73498x-01/81760x-01 or later</td>
</tr>
<tr>
<td>TNC 320</td>
<td>NC SW 34055x-01/771851-01 or later</td>
</tr>
</tbody>
</table>

Installation by machine manufacturers or servicing providers

For more information, see the Information for the Machine Tool Builder brochures
Software

TNCscope: export oscilloscope files to a PC

TNCscope is a PC oscilloscope software program for the machine manufacturer for recording and evaluating control signals, drive signals, and PLC operands from HEIDENHAIN controls. TNCscope enables convenient diagnosis from a Windows PC. With a simple Ethernet connection, you can also use TNCscope to perform remote maintenance on the control. The TNCscope software also lets you display oscilloscope files that have been recorded on the control (offline mode). An extensive selection of mathematical functions is available for analysis.

• Multi-channel recording
• Various triggering options
• Convenient analysis of measured values
• Display of circular interpolation test and XY files of the control
• Display of logic-trace files
• Cursor and auxiliary cursors
• Setting of marks
• Measurement and comparison of curves
• Gating of curves with a formula
• Calculation of integrals and derivatives
• XY view
• Frequency analysis with Fast Fourier Transformation (FFT)
• Copying of curves to another file
• Printing of a graph with additional information
• Import function for any ASCII tables
• Long-term measurements
• Recording in batch mode
• Support during the initial setup of various control functions (PAC, LAC, CTC, MAC, ACC, Component Monitoring)

TNCscope Registration required

TNC7
TNC 640
TNC 620
TNC 320
NC SW 81762x-16 or later
NC SW 34059x-01 or later
NC SW 34056x-01/771851-01 or later
NC SW 34055x-01/771851-01 or later
Installation by machine manufacturers or servicing providers

For more information, see the Information for the Machine Tool Builder brochures

Software

TNCdiag: for evaluating status information and diagnostic information

Based on the HFL interface (HEIDENHAIN Fibre Link) between the components of the Gen 3 drives, detailed diagnostic possibilities are offered during initial setup and later servicing. To this end, the visualization and presentation of the available diagnostic data have been improved in the TNCdiag software. TNCdiag provides all of the relevant information and statuses of the components in the control system all the way down to the encoders. This depiction can be component- or axis-specific and is optimized for touch operation. TNCdiag is available as an application on the control for displaying live data and as a PC tool for evaluating diagnostic data, including offline from service files.

• Status and diagnostic information about the HEIDENHAIN components (drive electronics, encoders, input/output devices, etc.) connected to the control
• History of the recorded data
• Replacement of DriveDiag for Gen 3

TNCdiag Registration required

TNC 640
TNC 620
TNC 320
NC SW 34059x-10 or later
NC SW 81760x-07 or later
Installation by machine manufacturers or servicing providers

For more information, see the Information for the Machine Tool Builder brochures

Graphically supported, dynamic display of status signals

Based on the HFL interface (HEIDENHAIN Fibre Link) between the components of the Gen 3 drives, detailed diagnostic possibilities are offered during initial setup and later servicing. To this end, the visualization and presentation of the available diagnostic data have been improved in the TNCdiag software. TNCdiag provides all of the relevant information and statuses of the components in the control system all the way down to the encoders. This depiction can be component- or axis-specific and is optimized for touch operation. TNCdiag is available as an application on the control for displaying live data and as a PC tool for evaluating diagnostic data, including offline from service files.

• Status and diagnostic information about the HEIDENHAIN components (drive electronics, encoders, input/output devices, etc.) connected to the control
• History of the recorded data
• Replacement of DriveDiag for Gen 3

TNCdiag Registration required

TNC 640
TNC 620
TNC 320
NC SW 34059x-10 or later
NC SW 81760x-07 or later
Installation by machine manufacturers or servicing providers

For more information, see the Information for the Machine Tool Builder brochures

Graphically supported, dynamic display of status signals
**Software**

**TNCopt:** for the initial setup of digital control loops

Machine tools with HEIDENHAIN controls must always meet the highest performance standards (surfaces, accuracy, fast and efficient machining, stable machining processes, etc.). As part of this, exceptional performance is required of the drive system with servo motors and spindles. That’s why HEIDENHAIN makes efficient and intelligent closed-loop technology its highest priority. Of particular importance is having the correct control-loop optimization and function settings in the control.

**TNCopt** conveniently helps you keep an overview and follow the proper sequence while setting up the axes.

**Functions:**
- Initial setup/optimization of the cascade control
- Compensation of peripheral machine influences (friction, hysteresis, etc.)
- System identification
- Dynamic Efficiency/Dynamic Precision
- Measuring functions (circular interpolation test, etc.)
- Automated Controller Tuning (ACT)

---

**Software**

**IOconfig:** configuration software for inputs and outputs

Modern machine tools are becoming ever more complex and extensively equipped. For this reason, IOconfig allows you to configure all bus systems for HEIDENHAIN controls with all symbolic PLC operands. All required information is stored in a project file (iopc).

**Functions:**
- All information is included in one project file
- All device descriptions are contained in the project file
- Generation of symbol definition files for PLC and SPLC
- Comprehensive option handling for including optional elements in the project
- Simulation mode for direct overview of active and inactive elements
- Convenient operation thanks to modern user interface
- Easy exchange of modules
- Symbol table for straightforward management of PLC symbols
- Definition of PLC start addresses for all elements
- Catalog for device descriptions
- Library for frequently used elements and branches
- Reference project for transferring elements from other projects
- Embedded in PLCdesign as of PLCdesign 3.1

With IOconfig 3, new bus systems are supported by simply adding further bus plug-ins. IOconfig currently includes plug-ins for SPI, PROFINET, PROFINET, and AS interface, as well as MOPLS1x and handwheels. The current device description files and the HEIDENHAIN PROFIBUS PL are also included in delivery.

---

### **TNCopt**

<table>
<thead>
<tr>
<th>TNC</th>
<th>Registration required</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>NC SW 81762x-16 or later</td>
</tr>
<tr>
<td>TNC 640</td>
<td>NC SW 34059x-04 or later</td>
</tr>
<tr>
<td>TNC 620</td>
<td>NC SW 81790x-01 or later</td>
</tr>
<tr>
<td>TNC 320</td>
<td></td>
</tr>
</tbody>
</table>

Installation by machine manufacturers or servicing providers

For more information, see the Information for the Machine Tool Builder brochures

### **IOconfig**

<table>
<thead>
<tr>
<th>TNC</th>
<th>Registration required</th>
</tr>
</thead>
<tbody>
<tr>
<td>TNC7</td>
<td>NC SW 81762x-16 or later</td>
</tr>
<tr>
<td>TNC 640</td>
<td>NC SW 34059x-01 or later</td>
</tr>
<tr>
<td>TNC 620</td>
<td>NC-SW 34056x-01/73495x-01/81760x-01 or later</td>
</tr>
<tr>
<td>TNC 320</td>
<td>NC-SW 77185x-01 or later</td>
</tr>
</tbody>
</table>

Installation by machine manufacturers or servicing providers

For more information, see the Information for the Machine Tool Builder brochures
Software

TN Cindygen: enabling keys for software options

TN Cindygen is a collection of PC software tools for generating enabling keys for HEIDENHAIN controls for a limited period of time.

OEM key generator

The OEM key generator PC tool makes it possible to generate an enabling key for software options on HEIDENHAIN controls. The selected option is enabled for a limited time (10 to 90 days). It can only be enabled once. You can generate the desired enabling key by entering the SIK number, the option to be enabled, the duration, and a manufacturer-specific password. This allows you to test the available options at your own pace without being forced to buy them.

OEM daily key generator

This application generates an enabling key for the protected machine manufacturer area on HEIDENHAIN controls. This key gives you access on the same day the key is generated.

BMXdesign

BMXdesign allows you to interactively create BMX project files and finished BMX files.

BMX files contain graphical and text elements that can be combined by the control into variants during program run. In this manner, help graphics or soft keys with language-sensitive texts or status-sensitive variants can be combined into just a single file.

BMX files are described in a BMX project file (*.BPJ). BMXdesign generates the final BMX file from the BPJ file.

Available functions:
- Interactive creation of BMX project files (WYSIWYG)
- Display of BMX files as on the control
- Adding and editing text and graphic elements
- Positioning of text fields with the mouse
- Automatic alignment of fields
- Adding of variants
- Replacement of database IDs with plain-language texts from multi-lingual files
- Printout with page preview
- Generation of BMX files
- Export function as bitmap file (*.BMP)
- Extracting graphic elements from BMX files
- Integration of PLCtext for managing BMX files
- Integration in PLCdesign
What’s the purpose of a programming station?
As easy as it is to write a part program on a TNC at the machine while a different part is being machined, short reloading times or high machine utilization may hinder attentive programming work on the shop floor. The programming station lets you program the same way you would at the machine but away from all the noise and distractions.

Creating programs
Creating, testing, and optimizing of HEIDENHAIN Klartext or ISO programs for the TNC on the programming station reduces machine idle times. And it’s easy to get used to because every keystroke feels the same: the programming station’s keyboard is just like the one on the machine.

Testing programs created offline
Naturally, you can also test programs written on a CAM system. The high-resolution program verification graphics help you easily spot contour damage and hidden details, even in complex 3D programs.

Training with the TNC programming station
Since the programming station is based on the same software as the TNC, it is ideal for apprentice and advanced-level training. Programming is performed on the original keyboard unit. Even the Test Run mode runs exactly as it would on the machine. This gives the trainee the confidence needed to operate a real machine. The programming station is also optimally suited for teaching TNC programming skills in schools, enabling programming in HEIDENHAIN Klartext and even ISO code.

The workstation
The programming station software runs on a PC and differs only marginally from a real TNC on a machine. You can still work with a TNC operating panel, except that it now includes the soft keys normally built into the monitor. The TNC operating panel is connected to the USB port of your PC allowing your PC to display the familiar TNC interface. Alternatively, you can also operate the programming station without a keyboard. In this case, operation is performed with a virtual keyboard displayed along with the TNC control panel. It gives you the TNC’s most important dialog activation keys.

Which programming stations are available?
Programming stations are available for all current TNC controls. A further version for operation with the VBox virtualization software is available as an option. The programming station DVDs include the programming station software, the necessary drivers, and the current User’s Manuals in all available languages.

The simulation of NC programs on the control has long been a part of HEIDENHAIN contouring controls. The virtualTNC solution makes it possible to use the TNC as a control component for machine simulation applications (virtual machines) on external computer systems.

How a virtual machine with virtualTNC works
Machine simulation applications (virtual machines) can fully simulate production units, allowing real-world production processes to be optimized in advance. The virtualTNC software can control the axes of a virtual machine as if it were a real system. Operators program and operate the control just as they would an actual HEIDENHAIN TNC.

The virtualTNC solution is a programming station software application of the TNC that, thanks to a special interface, is able to ascertain the current axis positions of the running “virtual” control.

Connecting virtualTNC over HEIDENHAIN DNC
Software manufacturers who wish to simulate a production system can connect their virtual machine to virtualTNC through HEIDENHAIN DNC. The COM component (AxisStreaming object) required for the programming and configuration of the interface to virtualTNC and its interface description are included in the RemTools SDK development package and the relevant help system.

Software
Programming station

<table>
<thead>
<tr>
<th>Programming station</th>
<th>TNC 640</th>
<th>TNC 620 / TNC 320</th>
</tr>
</thead>
<tbody>
<tr>
<td>Demo version</td>
<td>ID 1114029-xx</td>
<td>ID 1114030-xx</td>
</tr>
<tr>
<td>With TNC operating panel</td>
<td>ID 1113967-02</td>
<td></td>
</tr>
<tr>
<td>With virtual keyboard</td>
<td>ID 1113967-02</td>
<td></td>
</tr>
<tr>
<td>Single station license</td>
<td>ID 1113964-03</td>
<td></td>
</tr>
<tr>
<td>Network license, 1 station</td>
<td>ID 1125665-03</td>
<td></td>
</tr>
<tr>
<td>Network license, 14 stations</td>
<td>ID 1113908-03</td>
<td></td>
</tr>
<tr>
<td>Network license, 20 stations</td>
<td>ID 1113928-03</td>
<td></td>
</tr>
<tr>
<td>Operating panel without software release module</td>
<td>ID 1113967-62</td>
<td></td>
</tr>
</tbody>
</table>

Installation by the operator

For more information, see the Programming Station for TNC Controls brochure

Software
virtualTNC: control of virtual machines

<table>
<thead>
<tr>
<th>TNC 640</th>
</tr>
</thead>
<tbody>
<tr>
<td>Single station license</td>
</tr>
<tr>
<td>Network license, 1 station</td>
</tr>
<tr>
<td>Network license, 14 stations</td>
</tr>
<tr>
<td>Network license, 20 stations</td>
</tr>
</tbody>
</table>

Installation by manufacturers of machine simulation applications

For more information, see the HEIDENHAIN DNC brochure
You can move TNC-controlled machine axes by simply pressing the axis direction keys. A simpler and more sensitive method, however, is to use an electronic handwheel from HEIDENHAIN. In this case, the feed motor moves the axis slide in direct relation to the handwheel’s rotation. For delicate operations, you can simply place it in the large machines. When you aren’t using the handwheel, the HR 550 is ideal for use on operating statuses. As a wireless handwheel, you can simply place it in another location on the machine.

**HR 130 panel-mounted handwheels**

The panel-mounted handwheels from HEIDENHAIN can be integrated into the machine operating panel or mounted at another location on the machine.

**HR 510, HR 520, and HR 550 portable handwheels**

The HR 510, HR 520, and HR 550 portable handwheels are particularly helpful when you work near the machine’s working space. The axis keys and certain function keys are built into the housing. This lets you work near the machine’s working surface. In this case, the touch probes for workpiece measurement are inserted into the tool holder either manually or by a tool changer. Based on the machine, the touch probes can be equipped with various tool shanks. The stylus of a TS touch trigger probe is deflected upon contact with a workpiece surface. In that instant, the TS generates a triggering signal that, depending on the model, is transmitted to the control either by cable, radio transmission, or an infrared beam.

**Workpiece touch probes from HEIDENHAIN**

Workpiece touch probes help you reduce cost during shopfloor and serial production. Together with the TNC, touch probes can be used for manual or automatic setup, measuring, and inspection functions:

- Workpiece alignment
- Preset setting
- Workpiece measurement

The touch probes for workpiece measurement are inserted into the tool holder either manually or by a tool changer. Based on the machine, the touch probes can be equipped with various tool shanks. The stylus of a TS touch trigger probe is deflected upon contact with a workpiece surface. In that instant, the TS generates a triggering signal that, depending on the model, is transmitted to the control either by cable, radio transmission, or an infrared beam.

**Cable-bound touch probes**

For machines requiring manual tool changes, as well as for grinding machines and lathes:

- **TS 260**: new generation; axial or radial cable connection
- **TS 460**: new-generation standard touch probe for radio or infrared transmission; for workpiece inspection functions: automatic setup, measuring, and setup during machining

**Wireless touch probes**

With radio or infrared signal transmission for machines with automatic tool changes:

- **TS 460**: new-generation standard touch probe for radio and infrared transmission; for both the TS 460 and TT 460; EnDat functionality for transmission of the switching state, diagnostic information, and additional data.

*The SE 661 is only for the TNC7, TNC 640 and TNC 620.*

**Workpiece touch probes TNC7**

<table>
<thead>
<tr>
<th>SE 660 for radio or infrared transmission (hybrid technology)</th>
<th>For the TS 460 and TT 460; EnDat functionality for transmission of the switching state, diagnostic information, and additional data.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum NC SW</td>
<td>Minimum NC SW</td>
</tr>
<tr>
<td>TNC 320</td>
<td>771851-02</td>
</tr>
<tr>
<td>TNC 640</td>
<td>81762x-16</td>
</tr>
<tr>
<td>TNC 620</td>
<td>81760x-01</td>
</tr>
<tr>
<td>TNC 660</td>
<td>81760x-05</td>
</tr>
<tr>
<td>TNC 760</td>
<td>81760x-08</td>
</tr>
<tr>
<td>TNC 864</td>
<td>81760x-09</td>
</tr>
<tr>
<td>Installation by the machine manufacturer</td>
<td>For more information, see the Touch Probe for Machine Tool brochure</td>
</tr>
</tbody>
</table>
Hardware enhancements

**TT: tool touch probes**

A key factor in obtaining consistently high production quality is, of course, the tool. This requires the exact measurement of tool dimensions and a cyclical inspection of the tool for wear, breakage, and tooth shape. For tool measurement, HEIDENHAIN offers the TT triggering tool touch probes.

These touch probes are installed directly within the machine's workspace, where they enable tool measurement either before machining or during interruptions.

The TT tool touch probes measure the tool length and radius. During probing of a rotating or stationary tool (e.g., for individual tooth measurement), the contact plate is deflected and a triggering signal is transmitted directly to the TNC. The TT 160 uses cable-bound signal transmission, whereas the TT 460 uses radio or infrared transmission. This makes the latter particularly well suited for use on rotary and tilting tables.

![TT tool touch probe](Image)

**Tool length and radius measurement with stationary or rotating spindle**

![Tool wear measurement and tool breakage monitoring](Image)

**ITC: additional operating station**

The additional ITC (Industrial Thin Client) operating stations from HEIDENHAIN are convenient additional remote station solutions for operating the machine or machine units such as tool-changing stations. With a remote operating design tailored to the TNC, the ITC can be readily connected via a standard Ethernet connection with a 100 m (max.) cable.

The ITC 855 is a compact additional operating station for control systems with a 15-inch or 19-inch main screen. In addition to having an ASCII keyboard and a touchscreen, it is also equipped with the most important function keys of the TNC. The ITC 855 adjusts its resolution automatically to fit the size of the main screen. The soft keys are operated on the touchscreen.

Together with the TE 73k or TE 74k keyboard unit, the ITC 362 (24-inch screen with multitouch operation) or ITC 860 (19-inch touchscreen) each make up a full-fledged second operating station. Operation is identical to that of the TNC. The ITC 362 or ITC 860 can also be used as a second-screen solution for an extended workspace.

**Convenient plug-and-play technology**

As soon as the TNC detects an ITC, the control provides it with an up-to-date operating system. After the ITC has booted, the main screen is mirrored one-to-one. Thanks to this plug-and-play capability, configuration by the machine manufacturer is not needed. With a standard configuration of the X116 Ethernet interface, the TNC automatically integrates the ITC into its system.

You can switch between the TNC and the ITC either by direct takeover or in accordance with a configurable handover principle. Both the booting and shutdown processes are fully controlled by the TNC for maximum operational reliability.

---

### Tool touch probes

<table>
<thead>
<tr>
<th>Tool touch probes</th>
<th>TNC7 Minimum NC SW</th>
<th>TNC 640 Minimum NC SW</th>
<th>TNC 620 Minimum NC SW</th>
<th>TNC 320 Minimum NC SW</th>
</tr>
</thead>
<tbody>
<tr>
<td>TT 160</td>
<td>ID 729763-xx</td>
<td>81762x-16</td>
<td>34059x-05</td>
<td>771851-02</td>
</tr>
<tr>
<td>TT 460</td>
<td>ID 1192582-xx</td>
<td>81762x-16</td>
<td>34059x-05</td>
<td>771851-02</td>
</tr>
</tbody>
</table>

*Installation by the machine manufacturer*

*For more information, see the Touch Probes for Machine Tools brochure*

### Additional operating station

<table>
<thead>
<tr>
<th>Additional operating station</th>
<th>TNC7 Minimum NC SW</th>
<th>TNC 640 Minimum NC SW</th>
<th>TNC 620 Minimum NC SW</th>
<th>TNC 320 Minimum NC SW</th>
</tr>
</thead>
<tbody>
<tr>
<td>ITC 362</td>
<td>ID 1340971-xx</td>
<td>81762x-16</td>
<td>34059x-07 SP2</td>
<td>–</td>
</tr>
<tr>
<td>ITC 855</td>
<td>ID 1370459-01</td>
<td>–</td>
<td>34059x-08</td>
<td>81760x-05</td>
</tr>
<tr>
<td>ITC 860</td>
<td>ID 1174035-01</td>
<td>–</td>
<td>34059x-07 SP2</td>
<td>–</td>
</tr>
</tbody>
</table>

*Installation by the machine manufacturer*

*For more information, see the Information for the Machine Tool Builder brochures*
Hardware enhancements
IPC: industrial PC

With the IPC 306 industrial PC, you can start and remotely operate Windows-based applications via the TNC's user interface, for example. The user interface is displayed on the control screen. Option 133 is required for this.

Since Windows runs on the industrial PC, Windows has no effect on the NC machining process. The IPC is connected to the NC main computer via Ethernet. A second screen is not needed, since the Windows applications are displayed on the TNC's screen via remote access.

In addition to the IPC 306, a separately ordered hard disk is required for operation. The operating systems Windows 8, 10, or 11 can be installed on this empty data medium.

For more information, see the Information for the Machine Tool Builder brochures.

Hardware enhancements
VT 121: vision system for tool inspection

The vision system for tool inspection consists of two components:
- Camera with two objectives (VT 121)
- Touch-operated PC software (VTC)

The vision system enables tool imaging during machining. Besides documenting tool status and wear, the following applications are possible as well:
- Tool inspection before critical machining steps
- Optimization of cutting parameters
- Optimization of NC programs
- Breakage inspection
- Tool inspection after service life

The camera takes close-up images of each tooth as well as detailed panoramic images of the entire tool circumference. During inspection with the VTC, the lighting angle can be varied for these panoramic images, enabling optimal illumination of individual teeth. Tools can also be imaged from below. The VTC can run automatically during unattended shifts (with cycles for the TNC7 or TNC 640). Via an interface to the TNC's tool table, the PC software can even lock tools as needed.

The vision system for tool inspection not only helps you avoid expensive damage to the tool, workpiece, and machine, but even the tool costs themselves can be reduced in the long term through targeted minimization of tool wear. The vision system also provides the following benefits:
- Automated imaging during machining
- Time savings through keeping the tool in the machine
- PC software can be used on its own for later inspection
- Compact system usable for even large tools
- Rugged design
- Controlled compressed-air cleaning

For more information, see the VT 121 Vision System for Tool Inspection Product Information document.

Controls

<table>
<thead>
<tr>
<th>Controls</th>
<th>TNC 640</th>
<th>TNC 620</th>
</tr>
</thead>
<tbody>
<tr>
<td>Industrial PC</td>
<td>Prerequisite: Remote Desktop Manager option 133</td>
<td></td>
</tr>
<tr>
<td>IPC 306 With 8 GB of RAM</td>
<td>ID 1179966-01</td>
<td></td>
</tr>
<tr>
<td>SSDR IPC 306</td>
<td>ID 1282884-51</td>
<td>Data medium for operating system</td>
</tr>
<tr>
<td>Installation by the machine manufacturer</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Vision system Tool Inspection

<table>
<thead>
<tr>
<th>Vision system Tool Inspection</th>
<th>TNC7 Minimum NC SW</th>
<th>TNC 640 Minimum NC SW</th>
<th>TNC 620 Minimum NC SW</th>
<th>TNC 320 Minimum NC SW</th>
</tr>
</thead>
<tbody>
<tr>
<td>VT 121</td>
<td>1249466-01</td>
<td>81762x16</td>
<td>34039x10</td>
<td>–</td>
</tr>
<tr>
<td>Installation by the machine manufacturer</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

For more information, see the Information for the Machine Tool Builder brochures.