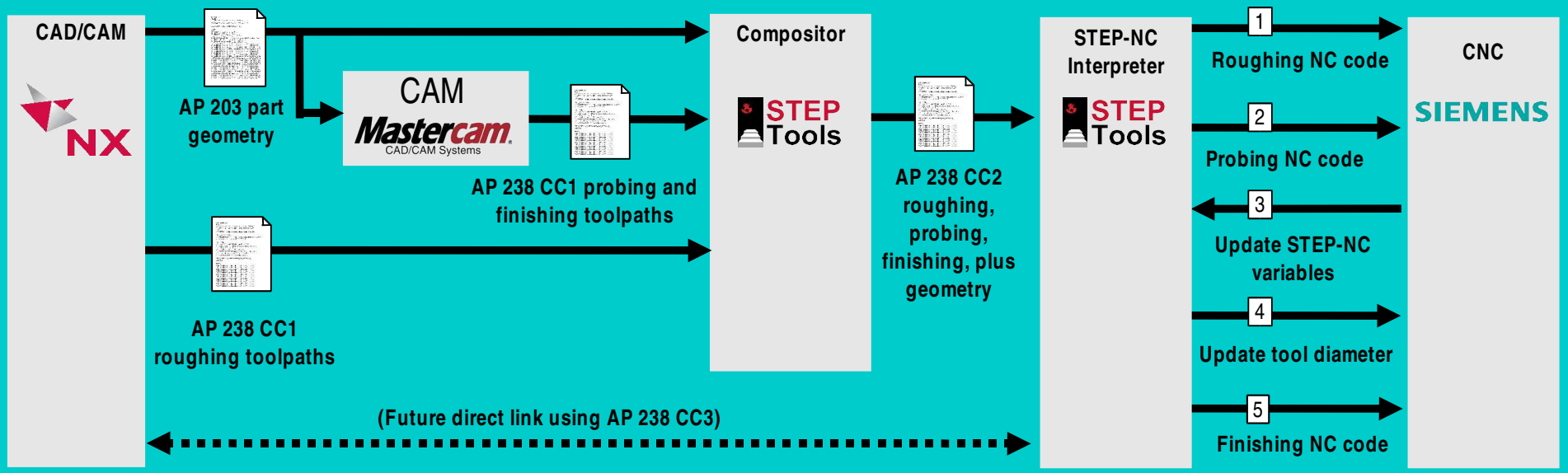
Closed-Loop Manufacturing with STEP NC



1 The Interpreter extracts roughing workingsteps from the STEP-NC data file and generates NC code automatically.

2 When roughing is done, the Interpreter extracts probing workingsteps and generates NC code, including the setting of NC variables with the results.

3 When probing is done, the Interpreter reads out the NC variables and stores them in the STEP-NC database, providing uniform access to data for either on-machine compensation or upstream analysis and planning.

4 The Interpreter computes the actual tool diameter, and updates the CNC's tool table accordingly.

5 Finishing workingsteps are extracted and converted to NC code. Unlike with wear registers, fewer or more tool paths may be calculated depending on the degree of wear.

STEP is the ISO 10303 standard for product data exchange, with Application Protocols (APs) for various technology areas:

- AP 203 for full boundary representation part geometry
- AP 238 Conformance Class 1 (CC1) for curves, splines and other geometry needed for high-accuracy machining
- AP 238 CC2, a superset of CC1 that includes AP 203 geometry needed for dynamic toolpath generation
- AP 238 CC3, a superset of CC2 that includes machining features (pockets, bosses) needed for on-machine measurement