

# NX CAD/CAM Turning Foundation

## Supporting functions for NC programming from Siemens Digital Industries Software

### Benefits

- Automated feature handling speeds common processes
- Boundary-based cutting provides flexibility to cut on minimal geometry
- Solids-based cutting cuts complex shapes intelligently
- Master model capability ensures that NC programming stays associative to the source model
- Synchronous modeling capabilities make it easy to adjust model for optimal NC programming
- Full design capability is provided with integrated CAD/CAM seat

### Benefits

- Synchronous technology
- 2D drafting
- Solid modeling
- Basic surface modeling
- Postprocessing and Post Configurator
- Shop documentation

### Summary

NX™ CAD/CAM Turning Foundation software facilitates numerical control (NC) programming of turning toolpaths in an integrated computer-aided design (CAD) environment. All of the supporting functions for NC programming are provided, from translators to toolpath visualization to postprocessing.

### Turning

NX provides comprehensive turning functionality driven by the in-process 3D solid part model.

### Associative turning profile

The software tracks allowable turning volumes precisely, even for millturn parts. You can maintain correct turning profiles through part updates. Spun solid profiles and merged multiple cross-sections provide the correct turning profiles for any part and automatically update when geometry changes.

### Comprehensive set of automatic operations

You can easily program a full range of lathe operations, including:

- On-center hole making: spot, drill, peck, chip, ream and tap

- Roughing: face, turn, back turn, bore, back bore and undercut – all with multiple patterns and depth of cut control and angle control
- Rough/finish grooving – with auto left/right tracking point control
- Outside diameter/inside diameter (OD/ID) threading
- OD/ID face
- Cut-off operation and bar feed operation types
- Part-off operation with preplunging and chamfering options

### Feature automation

NX Turning automates grooving with feature-based machining processes. You can produce threads according to standards with parameter-driven thread cutting processes. You can also break corners with arcs or chamfers that account for the finish status of the adjacent geometry.

### User control

You can customize and fine-tune turning boundaries to account for various finish requirements or in-process characteristics, such as grind stock. NX suppresses features as needed. Transitions to/from boundary edits ensure the desired result is easily achieved. Additionally, you can compensate for cutter diameter variance or use zero diameter tool programming (part dimension programming).

### Advanced options

NX provides special cutting patterns, such as finish all, down-only cutting and

# NX CAD/CAM Turning Foundation

toward/away from corner patterns. You can protect valuable work pieces by interrupting long cuts for insert changes. Safe exit and re-entry to the cut are handled automatically.

## Teach mode

NX Turning provides step-by-step motion instructions through a few simple mouse clicks. It supports all of your turning center's capabilities (including bar feeds or tail stocks) and visualizes their motion.

## Tools

You can specify the adapter, holder and insert as part of a tool assembly as well as multitools, which allows the combination of multiple inserts in a single tool. NX enables you to configure left- or right-handed tools for inside or outside diameters. You can cut either side of the centerline using either upper or lower turret configurations. Multiple drive points per tool facilitate flexible usage of tools versus various geometry. Probing tools are available for on-machine probing cycles.

## Standard supporting functions

### Translators

You can read data from practically any CAD system with a set of translators for the Parasolid® software, Initial Graphics Exchange Specification (IGES), Drawing Exchange Format (DXF), Standard for the Exchange of Product Data (STEP) formats and the JT™ data format. In addition you can write data in all of these formats, plus STL.

### Operation navigator

From the CAM application's information center, you can create, view, modify, verify, inspect and organize your NC programs. You can manage tools, methods, geometry and output from an information-rich program display.

## Toolpath visualization

You can stay confident with toolpath visualization and material removal verification tools. Integrated collision checking identifies any possible problems.

## Toolpath editing

You can make fast changes to toolpath results with graphical editing tools and protect your edits by locking toolpaths from future recalculations.

You can utilize the powerful template mechanism to create semi-automated processes customized to your environment. Highly familiar operation dialogs help you to streamline common tasks with the minimum user input.

NX outputs production-ready g-code via its integrated postprocessing. You can edit and fine-tune existing postprocessors with the included interactive Post Configurator application, or even build a postprocessor from scratch using Post Configurator templates. You can also use included sample posts or download a post from our support library.

## Shop documentation

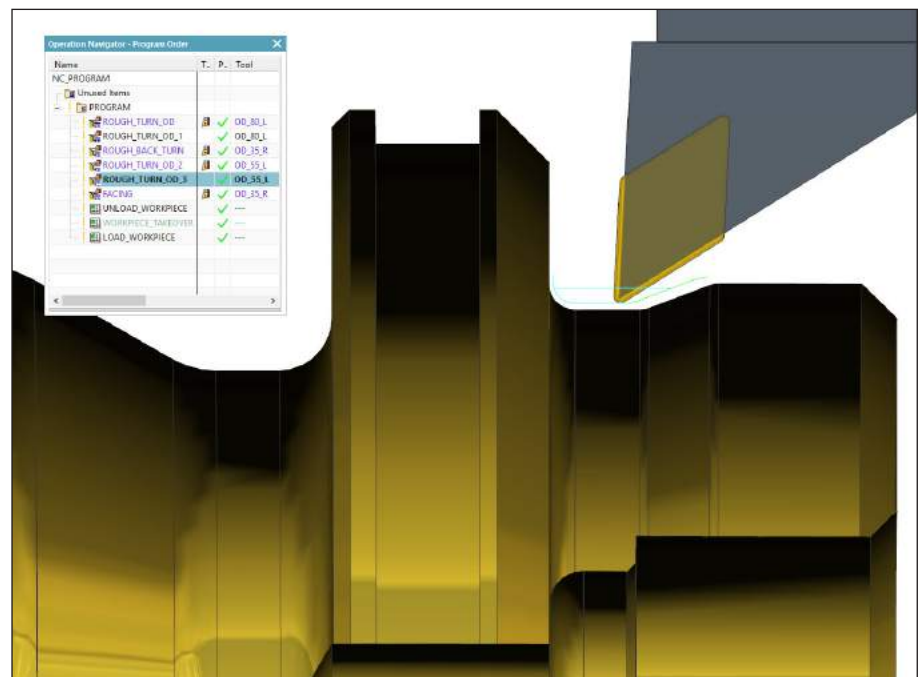
You can produce HTML setup documents with the included shop doc capability.

## Feed rate optimization

NX CAM helps you to easily maximize material removal rates and minimize tool wear. The ability to adjust feed rates to achieve a specified chip volume eliminates overloading as well as under utilization of the cutting tool.

## Standard model editing functions

NX CAM provides design tools to support common model editing functions that make NC programming more efficient. The standard range of functions provided are effective for making the kinds of changes to existing models that are often needed in the NC programming context, such as altering a basic stock material shape. Using the standard model editing functions, you can perform important model cleanup tasks with key editing capabilities for:



**Associative geometry**

You can associate copies of master geometry, enabling model editing without affecting the original design model.

**3D wireframe construction for boundaries**

Wireframe elements can be constructed from scratch or derived from part edges. Boundaries can be used directly for certain 2D operation types or used indirectly as containment boundaries to control toolpath coverage.

**Basic solid shapes**

Solid blocks and cylinders can be created for use as blank models or fixture components.

**Surface extensions and patches**

You can patch over features that don't need to be machined. Toolpaths will move smoothly across the patch, rather than trying to dig into the feature. You can extend faces to help guide tool tips around 3D edges if necessary.

**Surface and edge extraction**

Surfaces and edges can be extracted, divided, projected and used in various ways to more precisely support the needs of NC operations. For example, a divided surface provides an additional edge that can be used to drive or define toolpaths. Likewise, trim, sew and join functions enable surface and edges to be modified to support NC operations.

**Synchronous modeling**

Synchronous modeling, a powerful method for directly editing solid faces, is also available in the NC environment. Developing as-cast models or other derivatives from the existing parts is easy since the programmer simply needs to push and pull on the existing faces.

**Assembly modeling**

NX CAM includes important fixture components in the programming session and facilitates intelligent collision avoidance. Our master model concept keeps the assemblies and toolpaths associative to your source model. NX assembly

design supports "top-down" and "bottom-up" approaches and provides rapid navigation of the assembly hierarchy with direct access to the design model of any component or subassembly.

**Extended CAD content (included with the CAD/CAM package)**

The standard computer-aided manufacturing (CAM) version of the CAD/CAM package provides geometry tools to support common model editing functions. A set of more extensive CAD functions (described below) are available in the CAD/CAM version of this package that enable you to create and edit complex 3D part models and produce drawings. The combination of CAD and CAM in one system provides one of the most fully integrated solid modeling systems available today.

**Drafting**

You can create engineering drawings that remain associative to your part models, including automatic view creation from solids, hidden line

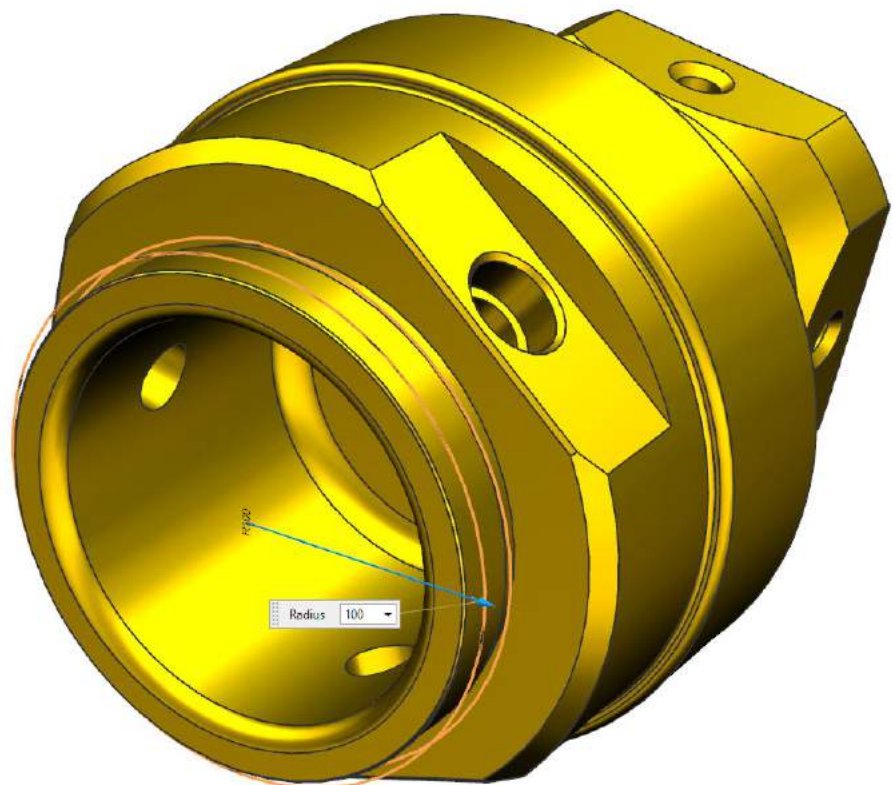
processing and automated parts lists. You can use templates to automatically create drawings for borders and view layouts.

**Solid modeling**

Using the turning foundation, you can develop 3D models with a full range of synchronous modeling and parametric modeling tools, including wireframe, primitives, swept and revolved bodies. Boolean operations and associative editing are supported. You can provide details such as blends, tapers and thin-wall features. NX modeling is powered by synchronous technology, Siemens Digital Industries Software's unique approach to 3D design that combines the power of parametric feature-based modeling with the freedom of history-free modeling.

**User-defined features**

The turning foundation lets you capture and store families of features for easy retrieval and editing, starting with existing associative solid models. You can



establish relationships between parameters, define feature variables, set default values and decide the form the feature will take when started.

### Associativity

This CAD/CAM package allows you to quickly respond to design changes through a fully integrated CAD/CAM system that keeps machining operations associative to the part model.

### Available add-ons

You can configure the perfect CAM seat by adding the toolpath creation options that your programs require. You can choose from:

- Wire electrical discharge machining (EDM)

- 2.5-axis milling
- 3-axis milling
- 5-axis milling
- Turbomachinery milling
- Machine simulation
- Feature-based machining author

### Available Teamcenter integration

You can integrate NX CAM with Teamcenter® software by including the NX CAM Teamcenter Client add-on. This integration provides a managed environment for your NX data. Additional Teamcenter functionality is available separately.

Siemens Digital Industries Software  
[siemens.com/software](https://www.siemens.com/software)

Americas +1 314 264 8499  
Europe +44 (0) 1276 413200  
Asia-Pacific +852 2230 3333