Feature-Based Machining
What’s New

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Feature-based Machining
What does it include

Machining Feature identification
- Re-use from CAD, Recognition, Tagging, …

Manufacturing Process Planning
- Suggest the best set of operations for a feature
- Suggest the best cutting tool for each operation
- Suggest the best process conditions for each operation
- Calculate the operation cycle time
- Group the operations into setups
- Optimized operation sequence

NC Programming
- Generate the tool path for each operation
- Create and validate NC programs
Feature-based Machining
Why should you be interested

**Process Automation**
- Significantly reduce the time needed to create NC programs
- Productivity improvements of 10x have been documented

**Process Quality**
- Reduce the amount of mistakes in “simple” and “repetitive” NC programming tasks
- Spend more time on critical / non-standard tasks

**Process Standardization**
- Ensure that the “standard process” is used by default
- Support the standardization of cutting tools
Feature-based Machining
Where can it be applied today?

2½D Prismatic Machining
- Standard features appear in many different parts like
  - Mold bases
  - Machinery parts
- Predominantly holes, pockets and slots
NX 6 Plans

Make it work for the majority of the NX CAM users without having to go through a long and costly customization project first

- Increase Feature Recognition scope and robustness (milling features, intersections/interruptions, compounds, PMI, …)
- Provide out-of-the-box solution without the need for extensive customer specific implementation
- Simplify the customization process (both for the features and the operation selection rules)
Machining Features
1. Add the *Machining Line Planner™ (MLP)* Feature Recognition technology to NX CAM
2. Enhance the Recognition with OOTB support for PMI
   - Upper and lower dimension tolerances, Fits and Limits
   - Thread and thread tolerances
   - Surface finish
   - Face attributes & colors
3. Support for new NX 6 AHF features
4. Configurable Feature Mapping
Machining Feature Recognition Projects

1.1 Offers additional 2½D milling Feature Types

Proven functionality (> 10 years of industry experience)
Large number of extra OOTB 2½D milling features
### Machining Feature Recognition Projects

#### 1.2 More Robust Recognition

<table>
<thead>
<tr>
<th>Issue</th>
<th>Description</th>
<th>MLP component</th>
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</thead>
<tbody>
<tr>
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<td>Open features</td>
<td>🧻</td>
</tr>
<tr>
<td><img src="image" alt="Intersecting features" /></td>
<td>Intersecting features</td>
<td>🧻</td>
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<tr>
<td><img src="image" alt="Blended features" /></td>
<td>Blended features</td>
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</tr>
<tr>
<td><img src="image" alt="Partially intersecting features" /></td>
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</tr>
<tr>
<td><img src="image" alt="Split faces (non NX geometry)" /></td>
<td>Split faces (non NX geometry)</td>
<td>🧻</td>
</tr>
</tbody>
</table>
Machining Feature Recognition Projects
NX 6 User Interface and Functionality

- Replaced NX5 Identification & Recognition commands with a single **Find Features** command
- All identification and recognition modes are now available from within the **Find Features** dialog
- Block based UI that adjusts to the “Type” of Recognition / Identification
Machining Feature Recognition Projects
NX 6 User Interface and Functionality

Five ways to Find Feature:
- CAD Feature Identification
- Parametric (new) Feature Recognition
- Legacy Hole Recognition
- Legacy Face & Pocket Recognition
- Manual Feature Definition

Additional enhancements:
- No double recognition
- Sheet and Solid body support
- Persistent feature type selection

Configurable Feature Mapping available for all Identification and Recognition methods

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New standard capabilities:

- Bodies to Search
  - Workpiece
  - All Bodies
  - Specify
- Machining Access Direction
  - Find only features from a specific direction
- Limit Search Area
  - Find only features that contain any of the selected faces
- Feature Settings
  - Assign Color Attribute

Customer Defaults are available to configure the preferred default behavior
Machining Feature Recognition Projects
NX 6 User Interface and Functionality

- Use Customer Defaults to pre-set UI defaults
  - Find Features (Type)
  - Bodies to Search
  - Use Feature Name as Type
  - Assign Color Attribute
  - Map Features
Feature Settings for Feature Identification:

- Use Feature Name as Type
  → Optionally identify and create machining features by either their design type or name (if i.e. renamed)
What is PMI?

**Product and Manufacturing Information**

- Any associated annotation that can be placed in 3D space with the model, or directly on the model.

- PMI is supported by national and international standards

**Directions**

- Capture and communicate design intent in the context of a single 3D “master” definition
- Facilitate downstream reuse of design information
- Automated documentation tools
- Automatic Standards conformance for all annotation
- Reduce or eliminate redundant, tedious effort required for 2D drawing creation
NX PMI Basic Functionality Overview

Product and Manufacturing Information (PMI) Directly on 3D Model
- Dimensions w/Tolerances
- Datums and Targets
- Feature Control Frames
- Associative Notes / URL

Model Views
- PMI Created in Each View
- Filters Control Visibility
- 3D Section Views
- Query Associated Objects

PMI on Parts and/or Assemblies

Transfers Directly to 2D Drafting
PMI flow down to CAM
How does PMI impact Machining?

1. Select the right manufacturing / machining process
   - NC Machining (Milling, Drilling), Honing, Grinding, Heat treatment, …
2. Define the appropriate setup datum scheme and/or perform key operations in the same setups
   - Form tolerances (perpendicularity, parallelism, flatness, concentricity, etc.)
3. Select the right processes & resources per machining feature
   - Dimension tolerance (±0.01)
   - Fits and limits (e.g. H7)
   - Surface finish value
   - Thread information (type, pitch, thread length, …)
   - User defined attributes and colors
4. Machine individual operations at the correct location using the right dimensions
   - Mid tolerance machining on non-symmetrical position & dimension tolerances
   - Minimum / Maximum material conditions
Machining Feature Recognition Projects
Recognizing PMI data
Dimension Tolerance Recognition
Machining Feature Recognition Projects
Recognizing PMI data
Checked Limits & Fits Tolerances

[Image of software interface showing tolerance settings and feature recognition]

SIEMENS
Machining Feature Recognition Projects
Recognizing PMI data
Surface Finish Recognition
Machining Feature Recognition Projects
Recognizing PMI data
Thread Tolerance Recognition
Machining Feature Recognition Projects
Recognizing PMI data
Feature Attribute and Color Recognition
Machining Feature Identification Projects
New NX AHF hole feature support

What is it?
- New NX hole features (preview available with NX5.0.2)
  - General Hole
    - Simple, Counterbored, Countersunk
  - Screw Clearance Holes
  - Threaded Holes
  - Hole Series (as individuals)

NX 6 FBM implementation:
- Feature Identification
- Feature mapping onto new parametric feature types
- Standard operations & tools
Machining Feature Recognition Projects

3. Configurable Machining Feature Mapping

- **NX Part**
  - NX Recognition/Identification
    - NX Features: UDF, SIMPLE_HOLE, RECTANGULAR_POCKET
  - MLP Feature Recognition
    - Intermediate Machining Features
      - MLP Features: STEP1HOLE, CORNER_NOTCH_STRAIGHT
  - Configurable Feature Mapping
    - Final Machining Features as presented to the user
Machining Feature Recognition Projects
3. Configurable Machining Feature Mapping

SCREW_CLR_COUNTER_BORE_HOLE

STEP2HOLE
Left: SCREW_CLR_COUNTER_BORE_HOLE
Right: STEP2HOLE
Machining Feature Recognition Projects
3. Configurable Machining Feature Mapping

Allows customers that use their private best machining practice to benefit from the improved machining feature recognition
- By mapping recognized features to customer specific UDF features
- So existing automatic process selection can be applied to components that were not designed using customer specific UDF features

Allows customers using UDF based design to work with Siemens PLM NX 6 best machining practices even though that does not reference those UDF types
- By mapping customer UDF features to standard NX/MLP features
Automatic Machining Process & Tool Selection
1. Add a new **Machining Knowledge Editor** application
   - Provide a dedicated User Interface for code-less process customization
     - Select feature, operation and tool **classes** from pull-down menus
     - Select feature, operation and tool **attributes** from pull-down menu’s
     - Full syntax checking
   - Does not require programming
   - Much easier to learn and use
   - Will significantly reduce the implementation time

2. Deliver OOTB **Process Content** for selected domain(s)
   - Features will get a set of “standard” processes
   - Customers can use the Machining Knowledge Editor to modify the standard content and further reduce the deployment time
Machining Knowledge Editor
Machining Best Practice definition using the Machining Knowledge Editor

What is it?
- Go forward strategy for defining rules in FBM
- Helps you create and modify the rule libraries which define the best practice operations and tools required to machine features

Benefits
- Use the Machining Knowledge Editor to modify the standard machining knowledge supplied with NX or to define your company’s best practices
Why are we adding the Machining Knowledge Editor?

Need to speed up the NX CAM automation deployment
- Single application to define, organize and document the rules
- User Interface to guide and support the user
- Knowledge management and distribution using Teamcenter
- Single set of criteria for metric and inch knowledge (no duplication)
- Excellent performance
- Training and online documentation material available

Offers a series of additional technical capabilities that will be integrated into future NX CAM versions:
- In-process features
- Compound operations for multiple features
- Creation (instead of selection) of new tools
- Supports rule teaching
Machining Knowledge Definition
How does it work? - Concept

Subject Matter Expert; configures the best practice machining processes
(Machining Knowledge Editor application)

NC Programmer (NX CAM)

Machining Knowledge Library

Create Geometry… command

Features
Tool Database
Teamcenter Resource Manager or ASCII

Template Database
Teamcenter or native

NX CAM loads the appropriate Machining Knowledge Library

Operations
Managing of the knowledge data files can be done in Teamcenter.
Machining Knowledge Definition

How does it work? - What is a unique operation?

Example machining processes for a hole

Through Drilling Hole (4)
  Spot Drill → Drill 1 → Drill 2 → Chamfer

Through Boring Hole (4)
  Spot Drill → Drill → Chamfer → Boring (Reaming)

Through Gun Drilling Hole (3)
  Guide Hole → Gun Drill → Chamfer

Through Tapping Hole (4)
  Spot Drill → Drill → Chamfer → Tapping

4 alternative sets of operations

15 operations overall

9 unique operations:
  → Spot Drill
  Spot Drill → Drill
  Drill → Drill
  Drill → Chamfer
  Chamfer → Boring, etc.

A unique operation can be used in different alternative sets of operations if it meets the criteria
Machining Knowledge Definition
How does it work? - What is a unique operation

**Mandatory Criteria:**
- When can an rule be used
  - Feature dimensions
  - Tolerances, etc.
- Which tool should be used
  - Type
  - Attributes
- Which operation template to use
  - Possibility to overrule template settings
  - Possibility to overrule Program and Method parents in the Operation Navigator

**Optional Criteria:**
- Workpiece Material
- Machine Tool type
Machining Knowledge Definition
Input Feature (lwf) / Output Feature (mwf)

<table>
<thead>
<tr>
<th>Operation</th>
<th>Input Feature</th>
<th>Output Feature</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spot_Drill</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Drill_S1H</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Chamfer_S1H</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tapping_S1H</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
The Machining Knowledge Editor is loading the resources required for rule definitions based on the selected configuration file.
(i.e. Feature, Operation and Tool Types from the templates and library definitions)
Machining Knowledge Editor
Create new rule → F8 → Select Output Feature (mwf)

The available features types (UDF, NX design, AHF, MW, PDW, Identification, Recognition, Manual Definition, etc.) will be shown in the Machining Knowledge Editor.
The available features types (UDF, NX design, MLP, MW, PDW, etc.) will be shown in the Machining Knowledge Editor.
The tool classes from the configured tool library will be shown in the Machining Knowledge Editor.

Tools from Resource Manager are fully supported.

The available Tools are retrieved from LIBRARY_TOOL in the CAM Configuration.
The default generic template contains:
- List of template operation (subtypes) that can be instantiated by the process generation
- Optional program parents for the template operation
- Default machining methods

Customer templates are fully supported

The available template operations are retrieved from TEMPLATE_OPERATION in the CAM Configuration
Machining Knowledge Editor
Set priority - highest priority = preferred (least expensive) process

![Machining Knowledge Editor Screenshot](image)

### Machining Knowledge Editor

**Set priority - highest priority = preferred (least expensive) process**

The generic template contains:

- List of Operation Subtypes that can be instantiated by the process generation
- Optional Program Parents for the Operation Subtypes
- No tools
- Default Machining Methods

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Machining Knowledge Editor
Define the conditions of the machining rules

The definition of Operation Navigator parents is optional:
oper.METHOD_GROUP = “DRILL_METHOD”
oper.PROGRAM_ORDER_GROUP = “DRILL”

Free text editing (manual typing, delete, insert, copy, paste, etc.)
Standard windows shortcuts (Cut/Copy/Paste, Undo/Redo, Find/Replace/F3, etc.)
Machining Knowledge Editor
Use constants for transparency

Replace 0.5 with `constant.through_offset`

Constants have 2 values:
- One for metric parts
- One for inch parts
Machining Knowledge Editor

Rules can be valid for a subset of materials

Rules can be valid for a subset of materials.
Machining Knowledge Editor

Rules can be valid for a subset of machines
Simplify knowledge maintenance and enhance process transparency by documenting conditions and dependencies.
Machining Knowledge Editor
Provide Rule Image

Input Feature

Output Feature

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Siemens PLM Software
Machining Knowledge Editor
Use Table View to find rules

### Excel Style User Interface

<table>
<thead>
<tr>
<th>Classif</th>
<th>class</th>
<th>name</th>
<th>Priority</th>
<th>OperationClass</th>
<th>InputFeatures</th>
<th>OutputFeatures</th>
<th>Resources</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td>Boring</td>
<td></td>
<td>1</td>
<td>1 STEPHOLE</td>
<td>1</td>
<td>1 BORE</td>
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<tr>
<td>2</td>
<td></td>
<td>Center_Drill</td>
<td>100</td>
<td>CENTER_DRILL</td>
<td>1 BLANK</td>
<td>1 POCKET_ROUND_TAPERED</td>
<td>1 CENTER_DRILL</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>Spot_Drill</td>
<td>1</td>
<td>SPOT_DRILLING</td>
<td>1 STEPHOLE</td>
<td>1 STEPHOLE</td>
<td>1 COUNTER SINKING</td>
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<tr>
<td>4</td>
<td></td>
<td>Flaten_PROFILE</td>
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<tr>
<td>6</td>
<td></td>
<td>Countertum_CSP</td>
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<td>COUNTERBORING</td>
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<td>1 COUNTER SINKING</td>
<td>1 COUNTER_BORE</td>
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<tr>
<td>7</td>
<td></td>
<td>Countertum_CSP_D01</td>
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<td>COUNTERBORING</td>
<td>1 STEPPOCKET</td>
<td>1 COUNTER SINKING</td>
<td>1 COUNTER_BORE</td>
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<td>1 COUNTER_BORE</td>
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<tr>
<td>10</td>
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<td>Drill_center_channel_STH</td>
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<td>1 POCKET_ROUND_TAPERED</td>
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</tr>
<tr>
<td>14</td>
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<td>1 TWIST DRILL</td>
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<tr>
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<td></td>
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<td>1 STEPHOLE</td>
<td>1 TWIST DRILL</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Machining Knowledge Editor
Use Table View to find rules

<table>
<thead>
<tr>
<th>Classif</th>
<th>class</th>
<th>name</th>
<th>Priority</th>
<th>OperationClass</th>
<th>InputFeatures</th>
<th>OutputFeatures</th>
<th>Resources</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
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<td>Rough_POCKET_RECT_STR</td>
<td>1</td>
<td>FACE_MILLING_AREA</td>
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<td></td>
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<td></td>
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<td>1</td>
<td>FACE_MILLING_AREA</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td></td>
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<td></td>
<td></td>
</tr>
</tbody>
</table>

Total

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Machining Knowledge Editor

MB3 → Build on Rule Library node

Is building the knowledge library (dll) at UGII_CAM_MACHINING_KNOWLEDGE_DIR (NX native mode)

Is building the knowledge library (dll) and saves it back into Teamcenter (TC mode)
Rule Based Operations for Features
(Operation selection component)

Inference Engine
Operation selection component (Inference Engine)

What is it?
- Rule–based operation selection is a proven technology seamlessly integrated into NX 6 that helps you automatically create operations such as milling, drilling and tapping from a generic template.
- Lets you select features such as holes, slots, and pockets from any source, including features that are User Defined, identified, recognized or tagged.
- Apply best practice machining rules on the features while taking into account any defined PMI.

Benefits
- Standardize on best practice machining knowledge. The software finds the best solution for machining task within a company’s environment.
- Save time with process automation.
Machining Knowledge Definition
How does it work? - Concept

Subject Matter Expert; configures the best practice machining processes (Machining Knowledge Editor application)
How does the Operation Selection work?

Example

Drill_S1H

---

STEP1HOLE Ø12H7

BLANK

STEP1HOLE

The lwf (Required Input Feature) would be a Blank

Diameter tolerance (H7) can NOT be reached with this drilling process

**List of alternative rules that can produce a STEP1HOLE feature (Output Feature)**

<table>
<thead>
<tr>
<th>name</th>
<th>Priority</th>
<th>OperationClass</th>
<th>InputFeatures</th>
<th>OutputFeatures</th>
<th>Resources</th>
</tr>
</thead>
<tbody>
<tr>
<td>(All)</td>
<td>(All)</td>
<td>(All)</td>
<td>(All)</td>
<td>STEP1HOLE</td>
<td>(All)</td>
</tr>
<tr>
<td>Gun_Drill_S1H</td>
<td>10</td>
<td>DRILL</td>
<td>1 STEP1POCKET</td>
<td>1 STEP1HOLE</td>
<td>1 GUN_DRILL</td>
</tr>
<tr>
<td>Bora_S1H</td>
<td>20</td>
<td>BORING</td>
<td>1 STEP1HOLE</td>
<td>1 STEP1HOLE</td>
<td>1 BORE</td>
</tr>
<tr>
<td>Ream_S1H</td>
<td>30</td>
<td>REAM</td>
<td>1 STEP1HOLE</td>
<td>1 STEP1HOLE</td>
<td>1 TAPER_REAMER</td>
</tr>
<tr>
<td>Chamfer_S1H_Mill</td>
<td>50</td>
<td>PLANAR_PROFILE</td>
<td>1 STEP1HOLE</td>
<td>1 STEP1HOLE</td>
<td>1 COUNTER_SINKING</td>
</tr>
<tr>
<td>Chamfer_S1H_Drill</td>
<td>60</td>
<td>SPOT_DRILLING</td>
<td>1 STEP1HOLE</td>
<td>1 STEP1HOLE</td>
<td>1 COUNTER_SINKING</td>
</tr>
<tr>
<td>Drillup_S1H (optional)</td>
<td>80</td>
<td>DRILL</td>
<td>1 STEP1HOLE</td>
<td>1 STEP1HOLE</td>
<td>1 TWIST_DRILL</td>
</tr>
<tr>
<td>Drill_in_center_chamfer_S1H</td>
<td>90</td>
<td>DRILL</td>
<td>1PCOCKET_ROUND_TAPERED</td>
<td>1 STEP1HOLE</td>
<td>1 TWIST_DRILL</td>
</tr>
<tr>
<td>Drill_in_center_S1H</td>
<td>80</td>
<td>DRILL</td>
<td>1PCOCKET_ROUND_TAPERED</td>
<td>1 STEP1HOLE</td>
<td>1 TWIST_DRILL</td>
</tr>
<tr>
<td>Drill_S1H</td>
<td>100</td>
<td>DRILL</td>
<td>1 BLANK</td>
<td>1 STEP1HOLE</td>
<td>1 TWIST_DRILL</td>
</tr>
</tbody>
</table>

Start with highest priority rule = preferred (least expensive) process
How does the Operation Selection work?

Example

The lwf (Required Input Feature) would be POCKET_ROUND_TAPERED

Diameter tolerance (H7) can also NOT be reached with this drilling process

Try next higher priority rule
How does the Operation Selection work?

Example

The Reaming process is suitable (all conditions are TRUE) to produce a tolerated STEP1HOLE

...Keep climbing up the priority ladder
How does the Operation Selection work?

**Example**

STEP1HOLE Ø12H7

Ream_S1H

Drill_in_center S1H

POCKET ROUND TAPERED

This process requires POCKET_ROUND_TAPER as input feature (lwf)

Drill_in_center_S1H would be a valid candidate to produce a STEP1HOLE.

<table>
<thead>
<tr>
<th>name</th>
<th>Priority</th>
<th>OperationClass</th>
<th>InputFeatures</th>
<th>OutputFeatures</th>
<th>Resources</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drill-in-center_S1H</td>
<td>30</td>
<td>DRILL</td>
<td>POCKET_ROUND_TAPER</td>
<td>STEP1HOLE</td>
<td>TWIST_DRILL</td>
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<tr>
<td>DRIL_S1H</td>
<td>100</td>
<td>DRILL</td>
<td>BLANK</td>
<td>STEP1HOLE</td>
<td>TWIST_DRILL</td>
</tr>
</tbody>
</table>

Since the previous Ream_S1H operation requires a STEP1HOLE to be resolved, the engine is now again trying to find the best suitable process to machine a STEP1HOLE.
How does the Operation Selection work?

Example

This process requires a BLANK as input feature (lwf)

Spot_Drill would be a valid candidate to produce a POCKET_ROUND_TAPERED.

In order to complete the process, the engine is now again trying to find the best suitable process to machine the POCKET_ROUND_TAPERED which is requested to be resolved by the previous drilling operation.
How does the Operation Selection work?

Example

The engine has successfully found a suitable set of rules to completely machine this feature.

The resulting process (in this simplified example): Spot_Drill => Drill => Ream.

It will now create the operations in the Operation Navigator.
How does the Operation Selection work?
Advanced Concepts

**Single process selection**
- Ranges
  - $6 \leq \text{Tool Diameter} < 12$
  - Tool Length $> 30$

**Tool selection is optimized for all features in a run**
- Consolidate tools across operations
  - Operation 1: Diameter 6-12
  - Operation 2: Diameter 10-14
  - Use Tool with Diameter = 12 for both operations
Automatic generation of Tool Queries based on tool conditions from Machining Knowledge Editor

Enables quick selection of alternative tools
How does the Operation Selection work?

Example log file – Optional Information Window

```plaintext
Methods for features: 1 (STEP1HOLE_1)

# AF : 1 (STEP1HOLE_1)
# 2 : Chamfer_S1H_Drill [AF_1]
  [2] Condition 1: This is a small diameter so the chamfer can be done by the centering operation. (IF mwf.DEPTH / mwf.
  [3] 3 : Drill_S1H [AF_2]
  [3] Condition 2: Constant has value = 1 (Use spot drilling) (bool_use_spot_drill.Yes1.No0 = 0 )
  [4] 4 : wall_u[1][1]_S1H [AF_3]
  [5] 5 : Real_S1H [AF_4]
  [5] Condition 1: This is a small diameter so the chamfer can be done by the centering operation. ( IF mwf.DEPT
  [6] 6 : Chamfer_S1H_Drill [F_12]
  [6] Condition 1: This is a small diameter so the chamfer can be done by the centering operation. ( IF mwf.DEPT
  [7] 7 : Drill_S1H [F_12]
  [7] Condition 2: Constant has value = 1 (use spot drilling) (bool_use_spot_drill.Yes1.No0 = 0 )
  [8] 8 : Gun_Drill_S1H [F_12]
  [8] Condition 2: IS FALSE( mwf.DEPTH / mwf.DIAMETER_1 >= DepthDiaRatio_limit )
  [9] 9 : Real_S1H [F_12]
  [9] Condition 2: roughness can be reached by drilling ( mwf.SIZE_ROUGHNESS_1 < max_Roughness_Drilling )
  [10] 10 : Bore_S1H [F_12]
  [10] Condition 2: Diameter tolerance can be made by drilling. ( IT_class_ISO { mwf.DIAMETER_1, mwf.DIAMETER_1
  [12] 12 : Drill-up_S1H (optional) [F_12]
  [12] Condition 2: ( mwf.DIAMETER_1 > pre_drillLimit )
  [14] 14 : Chamfer_S1H_Drill [F_14]
  [14] Condition 2: Chamfering already applied or feature was not chamfered. ( mwf.DEPTH_TOP_CHAMFER > S)
  [15] Condition 1: Application condition ( mwf.Machining_Rule = "TWIST_DRILL" )
  [16] 16 : Drill-up_S1H (optional) [F_14]
  [16] Condition 1: Application condition ( mwf.Machining_Rule = "TWIST_DRILL" OR mwf.Machining_Rule =
  No tools available for following parameters ( tool_type : END_MILL_INDEXABLE )
  Diameter < 10.000000
  [18] 18 : Drill_S1H [F_14]
  [18] Condition 1: Application condition ( mwf.Machining_Rule = "TWIST_DRILL_GUIDE" )
  [20] 20 : Spot_S1H [F_16]
```

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Siemens PLM Software
Knowledge content supplied with NX CAM
Machining knowledge content supplied with NX

What is it?
- Provides machining best practices for the NX machining feature types (holes, pockets, slots, etc.)
- Decision criteria (e.g. when to drill and when to ream) can be configured
- Content can be modified using the new Machining Knowledge Editor
- Customers can exchange methods

Benefits
- Customers can use FBM without up-front customization
- Reduces the FBM customization effort (it is much easier to modify existing content than to define content from scratch)
- Content will be maintained and updated with subsequent releases
Machining Knowledge Content

Concept

Subject Matter Expert; configures the best practice machining processes
(Machining Knowledge Editor application)

Machining Knowledge Library

NX CAM loads the appropriate Machining Knowledge Library

NC Programmer (NX CAM)

Create Geometry… command

Features

Tool Database
Teamcenter Resource Manager or ASCII

Template Database
Teamcenter or native

Operations
Machining knowledge content supplied with NX

Content is defined in:
UGII_CAM_MACHINING_KNOWLEDGE_DIR\machining_knowledge.xml
Edit with Machining Knowledge Editor application
Current status of Content Implementation (1/3)

~150 unique operation rules defined for
STEP1-3HOLE/POCKET
STEP1-3HOLE/POCKET_THREAD
POCKETS, CORNER NOTCH, etc.
Current status of Content Implementation (2/3)

Behavior can easily be configured using global CONSTANTS:
- BOOL_Always_Chamfer_Features (Y/N)
- BOOL_Check_PointAngle_Twist_Drill (Y/N)
- etc.

Change threshold values as a 1st level of customization
- Pre_Drill_Limit (12)
- BEST_IT_Class_Drilling (6)
- BEST_IT_Class_Milling (10)
- etc.

<table>
<thead>
<tr>
<th>Conditions</th>
<th>Constants</th>
<th>Materials</th>
<th>Machines</th>
<th>Explanation</th>
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<td>12.5 double</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Current status of Content Implementation (3/3)

Basis for content:
- JAD partner input (best practice)
- Threshold values based on IT (International Tolerance System)
- Threshold values based on IT (International Tolerance System)
Additional NX 6 FBM enhancements

What is it?
- Enhancements to Machining Feature Navigator filters
  - Safe filter in part file or in registry / indicate what filter is applied
  - Define filter on common attributes across multiple feature types
  - Filter editing
  - Additionally apply MCS filter to list only features that are parallel to the tool axis defined in the MCS
- Change propagation (indicate feature changes)
- Safe clearance in hole making
- Tool path status/indication in the Operation Navigator

Benefits
- Faster, improved feature handling
- Immediate feedback when features have changed
- Secure tool path
Additional FBM enhancements
Feature Navigator Filter Enhancements

- Safe filter in part file (new) or in registry (existing) and indicate current filter
- Retain filters in part file
- Define part specific filters for reuse
- Makes it easier for users to share filters
- Indicates what filter is currently in use
Additional FBM enhancements
Feature Navigator Filter Enhancements

- Define filter for common attributes across multiple feature types
- Select multiple feature types and use common attributes for criteria definition
- Reduce the amount of filters (i.e. want to define filter for features of various types that have a common tolerance parameter)
Additional FBM enhancements
Feature Navigator Filter Enhancements

- Filter editing
  - Change filter content in saved filters instead of deleting/creating new filters
  - Add / remove criteria
Additional FBM enhancements
Feature Navigator Filter Enhancements

- Apply permanent MCS filter to list only features that are aligned to the tool-axis definition of the MCS
- Improves feature handling for multi-side programming
Additional FBM enhancements
Feature Change Propagation (1/3)

- Change propagation (indicate feature changes)
Additional FBM enhancements
Feature Change Propagation (2/3)

- Change propagation (indicate feature changes)
  - Automatically re-runs the recognition to update feature data
  - Indicate changes in Feature Navigator immediately after design change
  - Current support for geometry and attribute change
  - Indication of non geometrical changes (i.e. PMI) is planned
  - Indicates change status with individual icon in front of the feature name
    - Feature is up-to-date (initial situation)
    - Feature is changed-and-updated
    - Feature is invalid (user should remove it)

Pull Face

**UPDATED Features**

- [ ] STEP1POCKET_1
- [ ] STEP1POCKET_2
- [x] STEP1POCKET_3
- [x] STEP1POCKET_4
Additional FBM enhancements
Feature Change Propagation (3/3)

- Change propagation (indicate feature changes)
  - MB3 → Approve Change command will reset the status to up-to-date (✓)
Additional FBM enhancements
Safe Clearance parameter in Hole making

What is it?
- Safe Clearance parameter in hole making

Benefits
- Avoid collision with a part, such as excess material on a casting, that is not represented by the solid model
Additional FBM enhancements
Safe Clearance Parameter in Hole making

- Clearance zone is applied to all horizontal transition moves
- Avoid collisions or near collisions when transitioning between holes
- Retracts to Vertical Clearance before transition if zone violates the part

No Safe Clearance → Direct Transition

Safe Clearance → Retract to “Safe Level” before horizontal transition
Additional FBM enhancements

Tool Path Generation

What is it?

- Avoid tool path generation if a previous operation has failed to generate the feature (i.e. due to collision check)
- Indicate in the Operation Navigator that the tool path was not generated (is empty) because a previous operation has failed to machine the feature

Benefits

- Avoid postprocessing invalid tool path

---

Warning messages displayed during tool path generation:

---

Diagnostic Information

Object name: DRILL_SIMPLE_HOLE_3

One or more features were not generated because a previous operation failed to generate these features.
Additional FBM enhancements
Tool Path Status indication in Operation Navigator

What is it?
- Indicate Suspect or Empty tool path in the Operation Navigator if the operation fails to generate the tool path for one or more features

Benefits
- See what operation failed to generate so that you can take action
Additional FBM enhancements
Highlight Features and Tool Path

What is it?
- Highlighting features and tool path when selecting objects in the Operation Navigator
  - Highlight features when selecting feature groups
  - Highlight features and tool path when selecting optimized groups
  - Highlight tool path when selecting operations

Benefits
- Directly see tool path results and associated features without editing the objects
Summary
What is available with NX 6 CAM - Features

Feature & PMI Recognition
• New Feature library with **50 additional feature types**
  • Focus on prismatic machining (drilling and 2½D milling)
  • Holes, Pockets, Slots, Grooves, Notches
• Support for **PMI**
  • Dimension tolerances
  • Surface finish
  • Thread & Thread tolerances
  • Colors & Attributes

Feature Identification
• Support for NX 6 AHF features
Summary
What is available with NX 6 CAM - Operations

Automatic best practice process and tool selection
• New Machining Knowledge Editor application
• Example best practice rules (~ 150) for the new feature types
  • Mold bases
  • Machinery parts