Lathe Machining

CATIA V5 Training
Foils

Lathe Machining

Version 5 Release 19
January 2009
EDU_CAT_EN_LMG_FF_V5R19
About this course

Objectives of the course
Upon completion of this course you will be able to:
- Identify and use the Lathe Machining workbench tools
- Define Lathe Machining operations
- Manage Tools and Tool Assemblies
- Use different methodologies for Lathe Machining

Targeted audience
NC Programmers

Prerequisites
Students attending this course should have knowledge of CATIA V5 Fundamentals and Numerical Control Infrastructure workbench

8 hours
# Table of Contents (1/5)

- **Introduction to Lathe Machining**
  - Workbench User Interface 9
  - General Process for Lathe Machining 10
  - Defining a Machine-Tool: General Process 11
  - Creating the Reference Machining Axis System 12
  - Stock Management in Lathe product 13
  - Maximum Turning Revolution Envelope Computation 14
  - B-Axis Management on Turning Operations 15
  - C-Axis Management on Turning Operations 16

- **Lathe Operations**
  - How to Create a 2-axis Lathe Operation 17
  - Basic Lathe Operations 18

- **Rough Turning Operation**
  - Rough Turning Operation: Introduction 19
  - Rough Turning Operation: General Process 20
  - Rough Turning Operation: Strategy 21
  - Rough Turning Operation: Geometry 22
  - Rough Turning Operation: Tools 23
# Table of Contents (2/5)

- **Rough Turning Operation**: Feeds and Speeds 32
- **NC Macro Definition** 33

**Groove Turning Operation** 37
- **Groove Turning Operation**: Introduction 38
- **Groove Turning Operation**: General Process 39
- **Groove Turning Operation**: Strategy 40
- **Groove Turning Operation**: Geometry 45
- **Groove Turning Operation**: Macro 46

**Recess Turning Operation** 47
- **Recess Turning Operation**: Introduction 48
- **Recess Turning Operation**: General Process 49
- **Recess Turning Operation**: Strategy 50
- **Recess Turning Operation**: Geometry 54

**Groove Finish Turning Operation** 55
- **Groove Finish Turning Operation**: Introduction 56
- **Groove Finish Turning Operation**: General Process 57
- **Groove Finish Turning Operation**: Strategy 58
- **Groove Finish Turning Operation**: Geometry 62
## Table of Contents (3/5)

- **Profile Finish Turning Operation**
  - Profile Finish Turning Operation: Introduction 65
  - Profile Finish Turning Operation: General Process 66
  - Profile Finish Turning Operation: Strategy 67
  - Profile Finish Turning Operation: Geometry 72

- **Thread Turning Operation**
  - Thread Turning Operation: Introduction 75
  - Thread Turning Operation: General Process 76
  - Thread Turning Operation: Strategy 77
  - Thread Turning Operation: Geometry 80

- **Sequential Turning Operation**
  - Sequential Turning Operation: Introduction 82
  - Sequential Turning Operation: General Process 83
  - Sequential Operation: Strategy - Management 84
  - How to use Sequential Operation: Standard Go to or Go Go 86
  - How to use Sequential Operation: Go Delta 90
  - How to use Sequential Operation: Go to in Given Direction 91
  - How to use Sequential Operation: Go to Along a Curve 93
# Table of Contents (4/5)

- **Ramp Rough Turning Operation**  
  - Ramp Rough Turning Operation: Introduction  
  - Ramp Rough Turning Operation: General Process  
  - Ramp Rough Turning Operation: Strategy  
  - Ramp Rough Turning Operation: Geometry  
  
- **Ramp Recess Turning Operation**  
  - Ramp Recess Turning Operation: Introduction  
  - Ramp Recess Turning Operation: General Process  
  - Ramp Rough Turning Operation: Strategy  
  - Ramp Recess Turning Operation: Geometry  
  
- **Tools and Tool Assemblies Management**  
  - Lathe Tool Assembly: Overview  
  - Lathe Tool Assembly: Presentation of the Tools  
  - Selecting a Tool from a Tools Catalog  
  - Creating a New Tool  
  - Tool Assembly  
  - Tool and Insert Technology  
  - How to Define a Tool Compensation
# Table of Contents (5/5)

- Lathe Tool Assembly Convention .............................................. 122
- Insert-Holder ........................................................................... 123
- Output Point ............................................................................ 124
- Summary of Conventions ......................................................... 125
- Importing Tools: General Process ............................................. 126
- Methodologies for Lathe Machining ......................................... 127
  - Frontal and Back Groove Turning .......................................... 128
  - Frontal Groove Turning Operation Definition ......................... 129
  - Back Groove Turning Operation Definition ............................ 130
  - Turning on a Milling Center with Facing Head: illustration ..... 131
  - Facing Head ........................................................................ 132
How to Use This Course

To assist in the presentation and learning process, the course has been structured as follows:

Lessons:
Lessons provide the key concepts, methodologies, and basic skill practice exercises. The goal of each lesson is to present the necessary knowledge and skills to master a basic level of understanding for a given topic.

A Master Exercise:
A Master Exercise provides a project where an industry type part is used to assist you in applying the key knowledge and skills acquired in the individual lessons as they apply to real world scenarios. The master exercise also highlights the process and steps for completing industry parts.

Added Exercises:
Added Exercises are provided after execution of Master Exercise Steps.

Note: According to preference, the Master Exercise individual steps may be completed after an individual lesson containing its key concepts.
Introduction to Lathe Machining

In this lesson, you will learn fundamentals of Lathe Machining.

- Workbench User Interface
- General Process for Lathe Machining
- Defining a Machine Tool
- Creating the Reference Machining Axis System
- Stock Management in Lathe Product
- Maximum Turning Revolution Envelope Computation
Lathe Machining

Workbench User Interface

- Part Operation Creation
- Manufacturing Program Creation
- Rough Turning Operation Creation
- Groove Turning Operation Creation
- Recess Turning Operation Creation
- Profile Finish Turning Operation Creation
- Groove Finish Turning Operation Creation
- Thread Turning Operation Creation
- Sequential Turning Operation Creation
- Ramp Rough Turning Operation Creation
- Ramp Recess Turning Operation Creation
- Drilling Operation Creation
- External Tool Change Creation
- Tool Change Creation
- Machine Rotation Creation
- Machining Axis Change
- Post-Processor Instructions Creation
- Copy Operator Instruction
- Tool Path Replay & Simulation
- Generate NC Code
- Manage Batch Queue
- Generate Documentation
- Screen Capture

Lathe Machining Workbench

- Batch mode or Interactively
- Copy / Tracut, Copy Transformation & Opposite Hand
- Auxiliary Operations

Student Notes:
General Process for Lathe Machining

1. Part designed using 3D Wireframe or Solid geometry

2. Create Wireframe elements necessary for manufacturing (profiles, safety planes, axis, points, etc)

3. Define Part Operations necessary to machine the whole part

4. If needed, Re-order the operations

5. Create Machining Operations and simulate them

6. Generate Auxiliary Operations

7. Generate NC data

Associative links
Defining a Machine-Tool: General Process (1/2)

Horizontal/Vertical Lathe
Assign a Machine-tool from a file or directly from PPR selection
Numerical Control Data:
Name of the PPWords Table
NC Data type (APT, Cifile, NC Code)
NC Data format (Point, Axis)
The machine is optional. If the machine is not defined, the PPWords Table ‘PPTableSample’ is used by default.
Advanced Options on 2D, 3D & Helical Interpolation
Allow to define Maximum Machine feedrate and the value of the Rapid feedrate and define the mode of transition path between operations
Tooling Data:
Default Tools Catalog
Radius Compensation availability
Defining a Machine-Tool: General Process (2/2)

Turret Data:
- Coordinates of the Home point
- Initial Axis orientation

Spindle Data:
- Spindle and radial axis
- Coordinates of the Center point
- Initial angular position
- Rotary angle
- Rotary Direction
- Rotary Type
- Rotabl Output in multi-axis

Compensation:
- None
- Contact
- Tip & Contact

All the coordinates are given according to the reference Machining Axis System.
Creating the Reference Machining Axis System

By default, the plan XZ must contain the profile.

If you change the Radial Axis in the Machine definition, don’t forget to verify whether the profile is in the right plane.

All the output coordinates generated in the Apt, CLFile or NC Code are computed according to the current Machining Axis System.
Stock Management in Lathe Product

On the Part Operation, you have the capability to define the Stock. That will be used for:
- Material removal simulation
- Automatically compute the current stock for each turning and drilling cycles (if the option ‘Automatic Stock Selection for Turning Operations’ is activated)

If this option is not activated, the user will have to select manually the stock for all Rough, Groove and Recess Turning operations and to use the ‘Update Input Stock’ icon to compute the stock resulting from the previous operations.

As the stock will be automatically updated operation by operation, it cannot be a CGR model (CATIA Graphic Representation).

Information regarding the status of the stock is displayed in the PPR tree and in the Geometry Tab page:
- Stock up to date
- Stock to update (resulting from a stock modification, Copy/Cut/Delete or modification of an operation)

You can compute the final stock after the last operation in the Program has been performed using ‘Compute Final Stock’ in the contextual menu on the program.
Maximum Turning Revolution Envelope Computation

You can generate automatically the max turning envelope of a selected part. The result is a Body and a sketch that can be used to define geometry (part, stock or limit element) of a machining operation.

Maximum Turning Envelope can be computed by:
Menu > Insert > Machining Features > Maximum Turning Envelope or Click

Selection of the part where the ‘maximum turning envelope’ solid and sketch are to be created
Selection of the part for which envelop is to be created
Selection of the turning axis for revolution
Specification of tolerance (length) value to compute the revolution envelope

The resulting solid and sketch are created. The sketch can be used to enable selection of 2D wireframe for machining operations.

You can edit or delete the turning envelope through Manufacturing View (by Machining Features).
B-Axis Management on Turning Operations

You can define a tool axis on turning operations. You can define the tool axis orientation on turret by managing the B-Axis and using the same tool assembly.

- The functionality is available on a turning operation having turret set as a ‘Mill Turret’.
- The tool change activity is not required.
- In case of multi-spindle machines, the tool axis is defined on the main spindle axis and the tool path computation is done according to the main spindle axis system.
- The specified tool axis is displayed on the tool path replay and is generated in the APT output.
- Tool axis is defined by geometry selection or manually.

OR

B-Axis angle = 0deg
B-Axis angle = 30deg
C-Axis Management on Turning Operations

You can machine different areas (front and back areas of a part, or part on main spindle and part on a counter-spindle) using the same tool assembly by managing C-Axis. Only the tool inversion is different for different operations.

- The functionality is available on a turning operation having turret set as a ‘Mill Turret’.
- You can rotate the C axis (0/180 degrees) without performing the tool change activity.
- You can define the C-Axis orientation on the Machining operation editor itself.
- The option ‘Tool axis flip’ is available in Strategy tab.

Frontal and Back Groove Turning

Tool axis flip option not selected

Tool axis flip option selected
Lathe Operations

In this lesson, you will learn the general process for creating a 2-Axis Lathe Operation and the basic Lathe Operations.

- How to Create a 2-axis Lathe Operation
- Basic Lathe Operations
How to Create a 2-Axis Lathe Operation

1. Click the icon of any 2-axis Lathe Operation

2. The new Operation is created after the current one

3. The Operation dialog box appears to edit it. Define the Operation geometry and parameters in the dialog box

4. Replay the Tool Path

5. Confirm Operation creation

The Operation is created in the PPR tree with a default tool. This capability can be removed by customizing the NC Manufacturing options.
Basic Lathe Operations

In the lessons ahead, you will learn in detail about the following basic Lathe Operations and how to define them.

- Rough Turning Operation
- Groove Turning Operation
- Recess Turning Operation
- Profile Finish Turning Operation
- Groove Finish Turning Operation
- Thread Turning Operation
- Sequential Turning Operation
- Ramp Rough Turning Operation
- Ramp Recess Turning Operation
Rough Turning Operation

*In this lesson, you will learn how to create a Rough Turning Operation.*

- Introduction
- General Process
- Strategy
- Geometry
- Tool Assembly
- Feeds and Speeds
- NC Macro Definition
**Rough Turning Operation: Introduction**

A Rough Turning Operation machines a part roughly keeping stock as per part offset on it. The machined part does not match with the required design part. This operation must be followed by Profile Finish turning Operation.

A Rough Turning Operation for External, Internal and Frontal machining:
- Includes Conventional and Parallel Contour strategies.
- Provides Approach and Retract macros (Direct, Axial-Radial, Radial-Axial).
- Offers full flexibility for defining individual motions within cycle (lead-in, retract at each individual pass).
Rough Turning Operation: General Process

1. Type the Name of the Operation. (optional because a default name is given by the system ‘Type_Of_Operation.X’)

2. Type a line of comment (optional)

3. Define operation parameters using the 5 tab pages:
   - Strategy tab page
   - Geometry tab page
   - Assembly tab page
   - Feeds & Speeds tab page
   - Macros tab page

4. Replay and/or Simulate the operation tool path
Rough Turning Operation: Strategy (1/4)

Roughing Mode:
- Longitudinal: Machining part by horizontal paths
- Face: Machining part by vertical paths
- Parallel Contour: Machining part by following the part profile

Orientation:
- External: Machining an external contour
- Internal: Machining an internal contour
- Frontal: Machining a frontal contour

You can define the vertical and horizontal offsets directly on the picture.
Rough Turning Operation: Strategy (2/4)

Location:

Front Location to machine toward the chuck jaws:

Back Location to machine from the chuck jaws:

Machining direction:
Allows to select between:

- From spindle and to spindle for Frontal Roughing mode (A)
- From head Stock and to head stock for Parallel contour (B)

Recess machining:
Only available if Part Contouring is set to Each path or Last path only. The recess machining paths are performed after the last Part Contouring path.

Under spindle axis machining capability for Face roughing and Parallel contour roughing to move tool under spindle axis machine

Part Contouring:
Specify a contouring path in order to clear the part profile

- No: Option not used
- Each path: Profile following at each roughing pass
- Last path only: Profile following at last roughing pass only

Tool compensation:
Number of the tool compensation used if this one is already defined on the tool.
Rough Turning Operation: Strategy (3/4)

Lead-in distance and Attack distance for Entry path  
Lift-off distance for exit path  
Lead-in angle only available if “Lead-in” option set to “Each path” or “Last path only” otherwise the “lead-in distance” is parallel to the machining direction

Double-click on the value to modify it

Min machining radius
Max machining radius

Axial limit for chuck jaws:  
(for external or frontal machining): Offset defined from the machining axis system.

Leading and Trailing safety angles:  
Clearance used to reduce the slope defined on the Insert or on the Tool.
Rough Turning Operation: Strategy (4/4)

Insert-holder constraints:

**Ignore:** Software is not checking the following technological attributes on the insert-holder
- Max Recessing Depth
- Max Cut Depth
- Max Boring Depth
- Trailing Angle
- Leading Angle
- Gouging Angle

**Apply:** These attributes are taking into account and can reduce the machined area

**Machining tolerance** is the maximum distance between approximated curve and tool path segments.
Rough Turning Operation: Geometry

This Tab Page includes a sensitive icon dialog box which allows the selection of:

A. Part elements (only profile)
B. Stock elements (only profile)
C. End Relimiting element
D. MB3 on value to define the position of the tool in, on, or out with respect to the End Relimiting Element.

Stock Extension distance: Allows you to extend the stock in machining direction. Stock extension is considered in tool path computation when End limit is defined (In/On/Out).

Offsets (Global or Axial/Radial) can be applied on the Part (Double-click on the value)
- Stock offset: perpendicular to the stock profile
- End limit offset: perpendicular to the clearance element

You can generate the profile corresponding to the previous operation and reuse these elements to define the new geometry.

All the created geometry is stored in the IPM CATPart.
Rough Turning Operation: Tools (1/2)

To carry out a Rough Turning Operation, a Tool Assembly is required. The Assembly consists of a External Tool or holder and a Tool Insert.

You can select the Tool Assembly from catalog and also you can modify the holder and/or Tool Insert.

- Select or modify Assembly (A)
- Select or modify External tool (B)
- Select or modify Tool insert (C)

For all information concerning Assembly tab page you can see the dedicated chapter named Tool Management.

Use the 2D Viewer to modify the parameters of the tool. The 2D Viewer is updated with the new values.

Click More to expand the dialog box to access all tool's parameters such as Geometry, Technology and Compensation.
Rough Turning Operation: Tools (2/2)

Groove Insert Positioning:
Groove insert positioning can be managed, with respect to the IN/ON/OUT setting on the Start/End Limits taking into account the tool compensation number (P9/P9R) and the insert geometry.

IN ON OUT

Tool movement direction

IN/ON/OUT Setting on Start/End Limits

The tool compensation number on the insert geometry

P9 with IN setting on the Start/End Limits

P9r with IN setting on the Start/End Limits
Rough Turning Operation: Feeds and Speeds

A. Define the Feedrate values (Angular mm/turn) or (Linear mm/mn) for:
   - **Lead-in**: This feedrate is used by default during Lead-in and Attack distances
   - **Plunge**: This feedrate is used during Plunge motion during recess machining
   - **Machining**: This feedrate is used during Machining motion
   - **Lift off**: This feedrate is used as during Lift-off motion
   - **Finishing**: This feedrate is used by default during Finishing paths
   - Light loading feedrate (ramping operation only)
   - Number of feed steps (ramping operation only)
   - **Transition**: This feedrate is used during transition. In the drop down you will find options like machining, lead in, lift off, rapid, local, finishing and air cutting.
   - **Air cutting feedrate**: by default = rapid, possible to define a value for synchronization calculation

B. Define the Spindle Speed value according the unit Linear (m/mn) or Angular (turn/mn)

C. Dwell Options: Dwell time or number of Dwell revolutions at the end of the path

D. You can automatically compute the machining feedrate according to the speeds you have define on your tool for roughing and finishing
NC Macro Definition (1/4)

You will learn how to create NC Macros.
Three types of Macros are available: Approach, Retract and Linking

- **Approach Macro**
- **Retract Macro**
- **Linking Macro**

Five interrupt modes for Linking Macros (Only for Rough, Recess, Groove Turning Operations to control or change the tool):
- **No**: No interrupt macro
- **Start of Path**: at the start of the path where the life time is reached
- **End of path**: at the end of the path where the life time is reached
- **On time**: as soon as the life time of the insert is reached
- **Number of path**: at the end of the path specified by a number
NC Macro Definition (2/4)

Select a predefined motion for each macro:
- None (no macro)
- Build by the user (User definition of the macro)
- Direct (direct line from the selected start point)
- Axial Radial (first axial motion then radial motion)
- Radial Axial (first radial motion then axial motion)

To modify a local feedrate in the macro, right-click on an element and select the type

Depending on the feedrate type selected, the element takes a different color:
- Yellow: Approach
- White: Local & Finish
- Green: Machining
- Blue: Retract
- Red: Rapid

All these predefined Colors for Feedrates can be customized in CATIA V5 Settings.
NC Macro Definition (3/4)

Macro build manually by the user: You can create an infinite number of different macros by selecting each of these basic trajectories in various order.

- Tangent or Normal trajectory to the previous motion
- Circular
- Motion to a point
- Motion perpendicular to a plane
- Distance along a line
- Erase the macro or only the selected motion
- Copy Approach (or Retract) macro on approach (or retract) motion of the Linking macro
- Insert a PP word on a point of the macro. Cross symbols localize the possible points to insert the PP word.

To insert a PP word, you can also press right mouse button on the cross and select « PP word list »

Possibility to select Major/Minor words and pre-defined syntaxes
NC Macro Definition (4/4)

Macros can have the following status:

- (): Not Activated
- Green Light: Already defined
- Yellow Light: Need to be replayed
- Red Light: Missing information

Store/Retrieve Macros in/from catalog

Read a macro from a predefined catalog

Store a macro in an already existing catalog or in a new catalog
Groove Turning Operation

In this lesson, you will learn how to create a Groove Turning Operation.

- Introduction
- General Process
- Strategy
- Geometry
Groove Turning Operation: Introduction

A Groove Turning Operation is used when the groove depth exceeds width. A groove is machined by a series of plunging cuts.

Groove Turning Operation:
- Valid for External, Internal, Frontal and Inclined machining.
- Includes several strategies to control first plunge.
- Provides Approach and Retract macros (Direct, Axial-Radial, Radial-Axial).
- Offers full flexibility for defining individual motions within cycle (lead-in, retract at each individual pass).
Groove Turning Operation: General Process

1. Type the Name of the Operation. (optional because a default name is given by the system ‘Type_Of_Operation.X’)

2. Type text of comment (optional)

3. Define operation parameters using the 5 tab pages
   - Strategy tab page
   - Geometry tab page
   - Assembly tab page
   - Feeds & Speeds tab page
   - Macros tab page

4. Replay and/or Simulate the operation tool path
Groove Turning Operation: Strategy (1/5)

Select the Orientation of the Groove:

- **External**
- **Internal**
- **Other**
- **Frontal**

Double-click on the value to modify it.

Select the First plunge position to define the start point of the operation:

- **Left** is the nearest groove side from head stock (A)
  You can machine from the head stock
- **Center** is the middle of the groove (B)
- **Right** is the farthest groove side from head stock (C)
  You can machine to the head stock
- **Up** is the farthest groove side from spindle axis
  You can machine to the spindle axis
- **Down** is the nearest groove side from the spindle axis
  You can machine from the spindle axis
- **Automatic**: You can machine from the spindle axis
Groove Turning Operation: Strategy (2/5)

Next plunges position:
Active only with 'Center' as 'First plunge position'

Allows selection between:
- 'From spindle' and 'To spindle' for Frontal Orientation.
- 'From head stock' and 'To head stock' for External and internal Orientations.
- 'Right of groove' and 'Left of groove' for other Orientation.

Left of groove: 'left' means from the head stock or from spindle axis.
Right of groove: 'right' means to the head stock or to spindle axis.

Single Plunge: Allows to machine a narrow groove by plunging co-axially and leaving less material on both sides.

For using this functionality, the First plunge position must be Center only.

Grooving by level and Chip Break options are respected by Single Plunge.

Part contouring is not possible with Single Plunge.
Groove Turning Operation: Strategy (3/5)

**Grooving by level:**
Allows machining a groove progressively by using a level strategy to remove a constant depth of material.

**Max approach distance:**
This is the distance from part profile up to which the tool returns to the first pass of a new grooving level at Rapid or Air-cutting feedrate. Then the feedrate reduces to lead-in feedrate for this approach distance. Thus reduces machining cycle time.

**Part contouring:**
Allows to add a contouring path.

**Under spindle axis machining:**
- Active only with 'Frontal' or 'Other' as 'Orientation'.
- Allows to move tool axis under spindle axis machine.
Groove Turning Operation: Strategy (4/5)

Tool compensation:
Number of tool compensation is used if it is already defined on the tool. By default, the output point corresponding to type P9 will be used.

Change output point:
An option for changing the tool point is available. The tool output point is automatically changed during the operation according to the profile geometry to be machined. The Tool Output Point at the end of operation is the same as at the start of operation. It is dependent on the machining axis system.

You must define the tool compensation. (for example P4, P9, P9R or P3, for a external groove)
Groove Turning Operation: Strategy (5/5)

Selecting Chip break allows you access to the different parameters of this strategy:
- Plunge distance (in white)
- Retract distance (in blue)
- Clear distance (in yellow)

Gouging safety angle: This angle allows to reduce further the maximum slope on which the tool can machine.

Unmachined zone

Leadin distance and Attack distance for Entry path

Lift-off distance and angle for exit path

Machining tolerance is the maximum distance between approximated curve and tool path segments.
Groove Turning Operation: Geometry

This Tab Page includes a sensitive icon dialog box which allows the selection of:

- **Part Elements (only profile)**
- **Stock Elements (only profile)**
- Offsets can be applied on the Part and the stock (Double-click on the value)
  - Stock Offset: Offset defined perpendicular to the stock profile
  - Radial and Axial offset on part
  - Part Offset: Offset defined perpendicular to the part profile
- You can generate the profile corresponding to the previous operation and reuse these elements to define the new geometry

All the created geometry is stored in the IPM CATPart
Groove Turning Operation: Macro

For Groove Turning Operation, you can use following Approach and Retract macros that define tool motions before and after machining:
Direct, Axial radial, Radial axial, and Build by user.

The Linking macro consists of approach and retract motion, can be used.

Interrupt modes for Linking macros are:
No, Start of path, On time, End of path, Number of path, and By Level.

You can interrupt approach and retract motions of linking macros at specified levels, when ‘Grooving by level’ and ‘By Level’ options are selected.

If the Linking macro is interrupted at the end of a specified level, the operation continues from the first pass of the level next to the specified level.
Recess Turning Operation

In this lesson, you will learn how to create a Recess Turning Operation.

- Introduction
- General Process
- Strategy
- Geometry
Recess Turning Operation: Introduction

A Recess Turning Operation is used to machine a groove where width exceeds depth.

A Recess Turning Operation:
- Valid for External, Internal, Frontal and Inclined machining.
- Includes Zig-Zag, one way and Parallel Contour strategies.
- Provides Approach and Retract macros (Direct, Axial-Radial, Radial-Axial).
- Offers full flexibility for defining individual motions within cycle (lead-in, retract at each individual pass).
Recess Turning Operation: General Process

1. Type the Name of the Operation. (optional because a default name is given by the system ‘Type_Of_Operation.X’)

2. Type text of comment (optional)

3. Define operation parameters using the 5 tab pages
   - Strategy tab page
   - Geometry tab page
   - Assembly tab page
   - Feeds & Speeds tab page
   - Macros tab page

4. Replay and/or Simulate the operation tool path
Recess Turning Operation: Strategy (1/4)

Three Recessing Modes are available as given below:

- **Zig - zag:**
  The tool alternatively machines in one direction then in opposite direction.

- **Parallel Contour:**
  The tool follows the part profile. The levels are machined in the same direction and are defined by a axial and radial depth of cut.

- **One Way:**
  The tool machines always in the same direction.

Four Orientations are available as shown:

- They are defined by axial parameters like Max depth of cut or Axial depth of cut and Radial depth of cut.
- For Other orientation we can modify the angle of incline of the tool.
Recess Turning Operation: Strategy (2/4)

Machining direction:
Allows to select between:
- ‘From spindle’ and ‘to spindle’ for Frontal Orientation.
- ‘From head stock’ and ‘to head stock’ for External and Internal Orientations.
- ‘Right of recess’ and ‘left of recess’ for other Orientations.

Change output point:
An option for changing the tool point is available.
The tool output point is automatically changed during the operation according to the profile geometry to be machined.
The Tool Output Point at the end of operation is the same as at the start of operation.
It is dependent on the machining axis system.

You must define the tool compensation.
(for example P9, P9R or P3,P4)
Recess Turning Operation: Strategy (3/4)

**Tool compensation:**
Number of the tool compensation used if it is already defined on the tool.

**Part contouring:**
Allows to add a contouring path.

**Under spindle axis machining:**
Active only with “Frontal” or “Other” as “Orientation”.
Allows to move tool axis under spindle axis machine.
Recess Turning Operation: Strategy (4/4)

Lead-in distance and Attack distance for Entry path
Lift-off distance and Angle for exit path
Distance and Angle before plunge for intermediate path

Lead-in angle only available if “Lead-in” option set to “Each path” or “Last path only” otherwise the “lead-in direction” is parallel to the machining direction

Machining tolerance is the maximum distance between approximated curve and tool path segments

Leading and Trailing safety angles:
Clearance used to reduce the slope defined on the Insert or on the Tool.

Gouging safety angle:
This angle allows to reduce further the maximum slope on which the tool can machine
Recess Turning Operation: Geometry

This Tab Page includes a sensitive icon dialog box which allows the selection of:

A. Part Elements (only profile)
B. Stock Elements (only profile)
C. Offsets can be applied on the Part and the stock (Double-click on the value)
   - Stock Offset: Offset defined perpendicular to the stock profile
   - Radial and Axial offset on part
   - Part Offset: Offset defined perpendicular to the part profile
D. You can generate the profile corresponding to the previous operation and reuse these elements to define the new geometry.

All the created geometry is stored in the IPM CATPart
Groove Finish Turning Operation

In this lesson, you will learn how to create a Groove Finish Turning Operation.

- Introduction
- General Process
- Strategy
- Geometry
Groove Finish Turning Operation: Introduction

A Groove Finish Turning Operation is a finishing operation. You can finish the pre-machined groove.

A Groove Finish turning Operation to finish a external, internal, frontal or inclined groove:
- Includes several strategies such as start/end relimitation, corner processing options (cornering, chamfering).
- Provides Approach and Retract macros (Direct, Axial-Radial, Radial/Axial).
Groove Finish Turning Operation: General Process

1. Type the Name of the Operation. (optional because a default name is given by the system ‘Type_Of_Operation.X’)

2. Type text of comment (optional)

3. Define operation parameters using the 5 tab pages
   - Strategy tab page
   - Geometry tab page
   - Assembly tab page
   - Feeds & Speeds tab page
   - Macros tab page

4. Replay and/or Simulate the operation tool path
Groove Finish Turning Operation: Strategy (1/4)

**Orientation:**
You can choose between, External, frontal and internal

- **External**
- **Frontal**
- **Internal**
- **Other**

**Under spindle axis machining**
The capability is for Face roughing and Parallel contour roughing. Allows to move tool axis under spindle axis machine.

**Tool compensation**
Number of tool compensation used if it is already defined on the tool.

**Contouring for outside corners**
Allows you to define whether angular or circular contouring is to be applied to corners of the profile (only if corner processing is set to rounded or chamfer)

**CUTCOM:**

- **CUTCOM/RIGHT or left** is generated in 2
- **CUTCOM/OFF** is generated in 4
- **CUTCOM/RIGHT or left** is generated in 1
- **CUTCOM/OFF** is generated in 5

During computation, CUTCOM order is generated for Approach and retract paths if this option is set to ON or REVERSE (before linear trajectory).

ON: CUTCOM/RIGHT instruction is generated if tool is to the right of the tool path.

REVERSE: CUTCOM/LEFT instruction is generated if tool is to the right of tool path.
Groove Finish Turning Operation: Strategy (2/4)

**Machining direction:**
Allows to select between:
- From spindle and to spindle for Frontal Orientation.
- From head stock and to head stock for External and internal Orientations.
- Right of recess and left of recess for other Orientation.

**Right of recess:** 'right' means to the head stock or to spindle axis
**Left of recess:** 'left' means from the head stock or from spindle axis

**Change output point:**
An option for changing the tool point is available.
The tool output point is automatically changed during the operation according to the profile geometry to be machined.
The Tool Output Point at the end of operation is the same as at the start of operation.
It is dependent on the machining axis system.

**Copyright DASSAULT SYSTEMES**

You must define the tool compensation.
(for example P9, P9R or P3,P4)
Groove Finish Turning Operation: Strategy (3/4)

This tab pages allows to manage Lead-in and Lift-off motion
You can decide to make a Linear or Circular as lead-in and
Lift-off motions (1)

Lead-in Angle and distance or Radius for first, last and
intermediate path (2)

Lift-off Angle and distance or Radius for exit path

Linear motion is defined by a distance and an angle
Circular motion is defined by a radius and an angle

Machining tolerance is the maximum distance between
approximated curve and tool path segments

Overlap corresponding to the superposition (distance) between
first path (in red) and last path (in blue)
Clearance is the distance to the opposite flank where the tool
must be retracted
Groove Finish Turning Operation: Strategy (4/4)

This tab pages allows you to manage Corner Processing.

You can decide to make a chamfer or a corner radius on Entry, Exit and Other Corner

If you select

None: The tool path on the corner is not modified (3)

Chamfer: modify the tool path by adding a chamfer defined by its length (4)

Corner Radius: modify tool path by adding a corner radius defined by its radius and distance (5)
Groove Finish Turning Operation: Geometry (1/2)

This Tab Page includes a sensitive icon dialog box which allows the selection of:

A. Part Elements (only profile)
B. Start Limit Elements (Only profile)
C. End Limit Element
D. MB3 on value to define the position of the tool in, on, or out of the Limit Elements.

E. Thickness and Offsets can be applied on the Part (Double-click on the value)
   - Start/End Limit Offsets: Offsets defined perpendicular to the limit clearance elements
   - Part Offset: Offset defined perpendicular to the part
   - Axial and Radial offset

F. You can generate the profile corresponding to the previous operation and reuse these elements to define the new geometry

All the created geometry is stored in the IPM CATPart
Groove Finish Turning Operation: Geometry (2/2)

Add Local Feeds and Thickness

We can have different strategies for each part (segment) of the profile. You can associate to a segment the property of another segment. The Copy/Paste Functionality allows to define the local information:
- Feed Rate
- Thickness
- Offset (Radial, axial)
- Possibility to invert the way of machining on this element

Allows to see all profile elements and its properties

List of all your local information:
- Select a line and the element on the part is highlighted
- You can Manage these local information:
  - using double-click or
  - using right panel which allows to copy partially the local information on another element
Profile Finish Turning Operation

In this lesson, you will learn how to create a Profile Finish Turning Operation.

- Introduction
- General Process
- Strategy
- Geometry
Profile Finish Turning Operation: Introduction

A Profile Finish Turning Operation is a finishing operation. You can machine the stock remained after performing the rough turning operation. The machined part matches the design part for profile.

A Profile Finish Turning Operation consists to finish a Longitudinal, Face or Contour Parallel Rough Turning operation:
- Includes several strategies such as start/end relimitation, corner processing options (cornering, chamfering).
- Provides Approach and Retract macros (Direct, Axial-Radial, Radial/Axial).
Profile Finish Turning Operation: General Process

1. Type the Name of the Operation.
   (optional because a default name is given by the system ‘Type_Of_Operation.X’)

2. Type text of comment (optional)

3. Define operation parameters using the 5 tab pages
   - Strategy tab page
   - Geometry tab page
   - Assembly tab page
   - Feeds & Speeds tab page
   - Macros tab page

4. Replay and/or Simulate the operation tool path
Profile Finish Turning Operation: Strategy (1/5)

Orientation:
You can select among
External, Frontal and Internal

Location:
Front: Location to machine toward the chuck jaws
Back: Location to machine from the chuck jaws

Recess machining:
The recess machining path is performed after the Profile finish path

Machining direction:
Only available for Frontal Orientation
Allows selecting between “From spindle” and “to spindle” for Frontal Orientation
Profile Finish Turning Operation: Strategy (2/5)

Contouring for outside corners:

It allows you to define whether angular or circular contouring is to be applied to corners of the profile (only if corner processing is set to rounded or chamfer).

Under spindle axis machining:
Allows to move the tool under spindle axis machine

Tool compensation:
Number of the tool compensation used if it is already defined on the tool

CUTCOM:
During computation, CUTCOM order is generated for Approach and retract paths if this option is set to ON or REVERSE (before linear trajectory)

ON: CUTCOM/RIGHT instruction is generated if tool is to the right of the tool path

REVERSE: CUTCOM/LEFT instruction is generated if tool is to the right of tool path

CUTCOM/RIGHT or left is generated in 2
CUTCOM/OFF is generated in 4
CUTCOM/RIGHT or left is generated in 1
CUTCOM/OFF is generated in 5
Profile Finish Turning Operation: Strategy (3/5)

Lead-in Angle and distance or Radius for Entry path
Lift-off Angle and distance or Radius for exit path

Lead-in and Lift-off type allows to select the type of entry and exit motion between Linear (A) and Circular (B). Linear motion is defined by a distance and an angle. Circular motion is defined by a radius and an angle.

Machining tolerance is the maximum distance between approximated curve and tool path segments.

Leading and Trailing safety angles:
Clearance used to reduce the slope defined on the Insert or on the Tool.
Profile Finish Turning Operation: Strategy (4/5)

This tab page allows you to manage Corner Processing.

You can decide to make a chamfer or a corner radius on Entry, Exit and Other Corner.

If you select None: The tool path on the corner is not modified

Chamfer: modify the tool path by adding a chamfer defined by its length (A)

Corner Radius: modify tool path by adding a corner radius defined by its radius and its angle (B)
Profile Finish Turning Operation: Strategy (5/5)

This tab page allows you to manage Local Invert:

**Invert strategy:** There are following three cases
- None: No Invert Strategy
- Overlap: defined by a clearance and a overlap parameter (A)
- Thickness: defined by a clearance and a thickness parameter (B)

**Machine inverted element first:**
The first path, you machine inverted elements and after the other elements

Invert Lead-in Angle and distance or Radius for Entry path
Invert Lift-off Angle and distance or Radius for exit path

Invert Lift-off type allows you to select the type of entry and exit motion between Linear and Circular.
Profile Finish Turning Operation: Geometry (1/2)

This Tab Page includes a sensitive icon dialog box which allows the selection of:

- **A** Part Elements (only profile)
- **B** Start Limit Elements (only profile)
- **C** End Limit Element
- **D** Right-click on value to define the position of the tool in, on, or out of the Limit Elements.

**E** Thickness and Offsets can be applied on the Part (Double-click on the value)
- Start/End Limit Offsets: Offsets defined perpendicular to the limit clearance elements
- Part Offset: Offset defined perpendicular to the part
- Axial and Radial offset

**F** You can generate the profile corresponding to the previous operation and reuse these elements to define the new geometry.
Add Local Feeds and Thickness

We can have different strategies for each part (segment) of the profile. You can associate to a segment the property of another segment using Copy/Paste Functionality.

Allows to define the local information:
- Feed Rate
- Thickness
- Offset (Radial, axial)
- Possibility to invert way of machining on this element

Allows to see all profile elements and its properties

List of all your local information:
- Select a line and the element on the part is highlighted
- You can manage these local information:
  - Using double-click or
  - Using right panel which allows to copy partially the local information on another element
Thread Turning Operation

In this lesson, you will learn how to create a Thread Turning Operation.

- Introduction
- General Process
- Strategy
- Geometry
There are two basic requirements for Thread Turning operation. A proper tool is needed because thread turning is a form-cutting operation. The resulting thread profile is determined by the shape of the tool and its position relative to the workpiece. The second requirement is that the tool must move longitudinally in a specific relationship to the rotation of the workpiece, because this determines the lead of the thread. This requirement is met through the use of the lead screw and the split unit, which provide positive motion of the carriage relative to the rotation of the spindle.
Thread Turning Operation: General Process

1. Type the Name of the Operation. (optional because a default name is given by the system ‘Type_Of_Operation.X’)

2. Type text of comment (optional)

3. Define operation parameters using the 5 tab pages
   - Strategy tab page
   - Geometry tab page
   - Assembly tab page
   - Feeds & Speeds tab page
   - Macros tab page

4. Replay and/or Simulate the operation tool path
Thread Turning Operation: Strategy (1/3)

Five different Profiles:

- UNC
- ISO
- Trapezoidal
- Other
- Gas

Two different Orientations:

- External
- Internal

Location:

- Front: Location to machine toward the chuck jaws
- Back: Location to machine from the chuck jaws

Thread Unit: Only available for Other Profile. Allows selecting ‘Pitch’ or ‘Thread per inch’ as ‘Thread Unit’. ‘Pitch’ value can be modified on the picture. To define the number of Threads per inch, use the appropriate zone.

You can also define the Number of Threads.
Thread Turning Operation: Strategy (2/3)

Thread type:
- **Constant depth of cut**: Same depth of cut at each pass. When it is activated, you can define the Max depth of cut.
- **Constant section of cut**: Decreasing the depth of cut at each pass. When it is activated you can define the Number of passes. After that, the section of cut is automatically set.

Thread penetration:
- **Entry Type**
  - Straight
  - Oblique
  - Alternate

If you select Oblique as Thread penetration, you need to define the Penetration angle.

Manage the first passes: activate ‘First passes’
- Select the number of passes you want to define as First passes
- **Section rate**: After the Rate was defined, the section of cut is automatically computed.

Manage the Last passes: activate ‘Last passes’
- Select the number of passes you want to define as Last passes
- **Depth of cut**: Constant depth material removal.

Spring passes:
the last pass is done again with a null depth of cut.
Thread Turning Operation: Strategy (3/3)

Lead-in distance for entry path

Lift-off Angle and distance for exit path

Clearance on crest diam. for link path:
It is the perpendicular offset between crest and link path.

Tool compensation:
Number of the tool compensation used if it is already defined on the tool.

Output CYCLE Syntax:
Allows you to generate automatically the drilling cycle syntax corresponding to the selected machine.

Edit Cycle:
Allows you to modify the standard syntax.
Thread Turning Operation: Geometry

This Tab Page includes a sensitive icon dialog box which allows the selection of:

- **Part Elements (only profile)**
- **Start Limit Elements**
- **End Limit Element**
- **MB3 on value to define the position of the tool in, on, or out of the Limit Elements.**

**Length**: length of the thread
**Start/End Limit Offsets**: Offsets defined perpendicular to the Limit Elements

You can generate the profile corresponding to the previous operation and reuse these elements to define the new geometry.

All the created geometry is stored in the IPM CATPart.
Sequential Turning Operation

*In this lesson, you will learn how to create a Sequential Turning Operation.*

- Introduction
- General Process
- Strategy-management
- Standard Go to or Go Go
- Go Delta
- Go to in Given Direction
- Go to Along a Curve
Sequential Turning Operation: Introduction

Sequential Turning Operation has ability to define individual tool motions within a single Machining Operation.

Sequential Turning Operation:
- Provides full control on individual Tool Motions
  - Allows to easily deal with complex machining situations
  - Allows to finely tune tool motions for optimized paths
- Targeted Customer Needs:
  - High Cost / High Precision Parts (one or few units)
    - Detailed control of individual motions is required
  - Serial Production of precision parts
    - Production optimization requires highly optimized tool path
Sequential Turning Operation: General Process

1. **Type the Name of the Operation.** (optional because a default name is given by the system ‘Type_Of_Operation.X’)

2. **Type text of comment (optional)**

3. **Define operation parameters using the 4 tab pages**
   - Strategy tab page
   - Assembly tab page
   - Feeds & Speeds tab page
   - Macros tab page

4. **Sequential operations list with management button**

5. **Replay and/or Simulate the operation tool path**
Sequential Operation: Strategy– Management (1/2)

- Edit the selected Tool motion
- Erase the selected Tool motion
- Move Up the selected Tool motion
- Move Down the selected Tool motion
- Create a Standard « Go to or Go Go » Motion
- Create a « Go Delta » Motion
- Create a « Go to in given direction » Motion
- Create a « Go to by following element » Motion
- Insert a PP word instruction

3 possibilities:
- NC = Non-computable
- ! = Geometry is missing
- = Good
Sequential Operation: Strategy– Management (2/2)

Column Order: allows you to modify the sequence of the Parameter in the table using up and down button.

Column Filter: allows you to customize your table by only selecting the parameter you want to see.

Properties: edit the operation
Delete – Cut – Copy – Past: Delete or duplicate selected Motion.
How to use Sequential Operation: “Standard Go to or Go Go” (1/4)

Geometry Tab page:
- Allows to create « Go to » motion by defining check elements
  - Name your motion (A)
  - Keys to navigate between the different motions (B)

- Geometry Tab page:
  - Define the 2 check elements by clicking on the lines (C)
    If you select a point as first check you don’t need to define a second check. In this case the Tool compensation point will be on the selected point.

- Define the position of the tool (MB3 on Check Mode) (D)
  - To: Tool Insert tangent to the check element and in
  - On: compensation point on the check element
  - Past: Tool Insert tangent to the check element and out
How to use Sequential Operation: “Standard Go to or Go Go” (2/4)

Define the Check limit Mode (E)
- Extended
- Actual

Reference Point (F)
You can specify tool positioning of the first motion with respect to TO/ON/PAST according to the reference point.
- Select: Click on a point in 3D window
- Initialise From Previous: Automatic selection from previous operation or tool change
- Remove: Cancels the selection of point
- Analyze: Visualizes the selected reference point in 3D window

Define the Offset on the two check elements
- Offset
- Axial: Axial offset
- Radial: Radial offset
How to use Sequential Operation: “Standard Go to or Go Go” (3/4)

Strategy Tab page:

This tab page is the same for all the four sequential motions, except for « Go Delta » motion where you cannot define Collision and Guiding point parameters.

- Name your motion (A)
- Keys to navigate between the different operations (B)

Feed and Speed:

Define the feedrate mode (C)

- For Lead-in / Lift off / Machining / Rapid, you can customize the corresponding values in General Feed & Speed Tab page.
- If you select Other value, you can define the local Feedrate value and the Feedrate unit (Angular or linear).

Change spindle speed (D)

- Tag Local spindle mode
- Key the new value of Machining Spindle
- Select the Spindle unit (Angular – turn/mn or linear – m/mn)

Copyright DASSAULT SYSTEMES
How to use Sequential Operation: “Standard Go to or Go Go” (4/4)

Strategy Tab page:

Collision (E)
- Possibility to add an angle to avoid collision with the tool

Others (F)
- Machining tolerance:
  It is the maximum distance between approximated curve and tool path segments
- Select the Compensation point:
  Number of the tool compensation used if it is already defined on the tool
- Define Guiding Point:
  Select the guiding point (left or right) for Groove insert
How to use Sequential Operation: “Go Delta”

- Allows to create motion by defining Axial/Radial delta, distance or distance + Angle
  - Name your motion (A)
  - Keys to navigate between the different operations (B)

- Define the geometry:
  - Select Delta check mode (C)

among:

- Select a Point
- Select a Line
- Key value for distance angle
- Key value for Axial delta
- Key value for Radial delta

- Select the geometry:
  - Associate the geometry for drive element (D)

- Key your parameter values (E)
How to use Sequential Operation: “Go to in Given Direction” (1/2)

- Allows to create «Go to a given direction» motion by defining check element and a direction
  - Name your motion (A)
  - Keys to navigate between the different operations (B)

- Geometry Tab page:
  - Define the Drive element type: line or angle (C)

- Select the geometry (D)
  - Associate the geometry for check and drive elements

- Define the check mode: To / On / Past (E)
  - To: Tool Insert tangent to the check element and in
  - On: compensation point on the check element
  - Past: Tool Insert tangent to the check element and out
How to use Sequential Operation: “Go to in Given Direction” (2/2)

- Define the Check limit Mode (F)
  - Extended
  - Actual

- Define the Drive direction (G)
  - Inverted
  - Same

- Define the Offset on the two check elements
  - Offset
  - Axial: Axial offset
  - Radial: Radial offset
How to use Sequential Operation: “Go to Along a Curve”

- Allows you to create « Go to along a curve» motion by defining a check element. The drive curve is the first check element of the previous motion.
  - Name your motion (A)
  - Keys to navigate between the different operations (B)

Define the Geometry
- Associate the check element (C)

Define the check mode: To / On / Past (D)
- To: Tool Insert tangent to the check element and in
- On: compensation point on the check element
- Past: Tool Insert tangent to the check element and out

Define the check limit mode
- Extended
- Actual

Define the Offset on the two check elements
- Offset
  - Axial: Axial offset
  - Radial: Radial offset

Define the drive direction
Ramp Rough Turning Operation

In this lesson, you will learn how to create a Ramp Rough Turning Operation.

- Introduction
- General Process
- Strategy
- Geometry
Ramp Rough Turning Operation: Introduction

A Ramp Rough Turning Operation is a roughing operation suitable for machining hard materials using round ceramic inserts, thereby minimizing wear and cutting stress.

A Ramp Rough Turning Operation for External, Internal and Frontal machining:

- Provides Approach, Retract and Linking macros (support interrupt macro for insert control or change).
- Offers full flexibility for defining individual motions within cycle (lead-in, retract at each individual pass).
Ramp Rough Turning Operation: General Process

1. Type the Name of the Operation. (optional because a default name is given by the system ‘Type_Of_Operation.X’)

2. Type text of comment (optional)

3. Define operation parameters using the 5 tab pages
   - Strategy tab page
   - Geometry tab page
   - Assembly tab page
   - Feeds & Speeds tab page
   - Macros tab page

4. Replay and/or Simulate the operation tool path
Ramp Rough Turning Operation: Strategy (1/4)

Roughing Strategy:

**Longitudinal:**
Machines part by horizontal paths

**Face:**
Machines part by vertical paths

**Orientation:**

**External:**
Machines an external contour

**Internal:**
Machines an internal contour

**Frontal:**
Machines a frontal contour

You can define directly on the picture, the vertical and horizontal offsets.
Ramp Rough Turning Operation: Strategy (2/4)

Location:
- Front: Location to machine toward the chuck jaws.
- Back: Location to machine from the chuck jaws.

Machining direction:
- It allows you to select between: From spindle and To spindle for Face Roughing strategy with Frontal orientation.

Under spindle axis machining:
- Allows you to move the tool under spindle axis machine.

Tool compensation:
- Number of the tool compensation used if this one is already defined on the tool.
Ramp Rough Turning Operation: Strategy (3/4)

MO Leading angle:
leave material on vertical area according an angle
(similar to draft angle in prismatic machining)

Clearance on 1st Cut:
Clearance between the part with the leading angle and
the end of the ramping path

Clearance on 2nd cut:
Clearance between the part with the leading angle and
the end of the horizontal path

Minimum machining radius

Maximum machining radius

Machining tolerance is the maximum distance between
approximated curve and tool path segments.

Axial limit for chuck jaws
(for External or Frontal machining):
Offset defined from the machining axis system.
Ramp Rough Turning Operation: Strategy (4/4)

Distance and angle before Rework Plunge for rework approach

Lift-off Distance & Angle at the end of the path before the rework

Clearance on 2nd cut
Clearance between the part with the leading angle and the end of the horizontal path

Lead-in distance: Distance before attack in material

Start Pass Mode:
None: no special start pass
Chamfer: add a 45deg chamfer at the beginning of each pass (activate Attack chamfer length parameter)
Rounded: add a corner radius at the beginning of each pass (activate Corner radius & corner angle parameters)
Ramp Rough Turning Operation: Geometry

This Tab Page includes a sensitive icon dialog box which allows the selection of:

A. Part elements (only profile)
B. Stock elements (only profile)
C. End Relimiting element
D. MB3 on value to define the position of the tool in, on, or out with respect to the End Relimiting Element.

E. Stock Extension distance: Allows you to extend the stock in machining direction. Stock extension is considered in tool path computation when End limit is defined (In/On/Out).
F. Offsets (Global or Axial/Radial) can be applied on the Part (Double-click on the value)
   - Stock offset: perpendicular to the stock profile
   - End limit offset: perpendicular to the clearance element
G. You can generate the profile corresponding to the previous operation and reuse these elements to define the new geometry.
Ramp Recess Turning Operation

In this lesson, you will learn how to create a Ramp Recess Turning Operation.

- Introduction
- General Process
- Strategy
- Geometry
Ramp Recess Turning Operation: Introduction

A Ramp Recess Turning Operation is a semi-finishing operation uses round insert tool to machine hard materials thereby minimizing wear and cutting stress.

A Ramp Recess Turning Operation is used to machine a groove where width exceeds depth:

- Valid for External, Internal, Frontal and Inclined machining.
- Includes Zig-Zag, one way and Parallel Contour strategies.
- Provides Approach and Retract macros (Direct, Axial-Radial, Radial-Axial).
- Offers full flexibility for defining individual motions within cycle (lead-in, retract at each individual pass).
Ramp Recess Turning Operation: General Process

1. Type the Name of the Operation. (optional because a default name is given by the system ‘Type_Of_Operation.X’)

2. Type text of comment (optional)

3. Define operation parameters using the 5 tab pages
   - Strategy tab page
   - Geometry tab page
   - Assembly tab page
   - Feeds & Speeds tab page
   - Macros tab page

4. Replay and/or Simulate the operation tool path
Ramp Recess Turning Operation: Strategy (1/4)

Recessing Strategy:

Zig-zag: The tool alternatively machines in one direction then in opposite direction.

One Way: The tool machines always in the same direction.

Orientations:
They are defined by axial parameters like ‘Max depth of cut’ or ‘Axial depth of cut’ and ‘Radial depth of cut’. For Other orientation we can modify the ‘angle of incline’ of the tool.

Part contouring:
Allows to add a contouring path

Under spindle axis machining:
Active only with ‘Frontal’ or ‘Other’ as Orientation. Allows to move tool axis under spindle axis machine

Tool compensation:
Number of the tool compensation used if it is already defined on the tool
Ramp Recess Turning Operation: Strategy (2/4)

Machining Direction:

It allows you to select among:
- ‘From spindle’ and ‘To spindle’ for Frontal Orientation.
- ‘From head stock’ and ‘To head stock’ for External and Internal Orientations.
- ‘Right of recess’ and ‘Left of recess’ for other Orientation.

Right of recess: ‘right’ means to the head stock or to spindle axis
Left of recess: ‘left’ means from the head stock or from spindle axis

Change output point:

An option for changing the tool point is available. The tool output point is automatically changed during the operation according to the profile geometry to be machined.

The Tool Output Point at the end of operation is the same as at the start of operation. It is dependent on the machining axis system.

You must define the tool compensation. (for example P9, P9R or P3,P4)
Ramp Recess Turning Operation: Strategy (3/4)

For One Way strategy only:
Entry Flank Angle & Exit Flank Angle: The insert geometry is taken into account to avoid collision by reducing the maximum slope on which machining can be done. Defining Entry and Exit Flank Angles on the operation allow you to further reduce the area to machine.

Angle and distance before plunge for intermediate path

Clearance on 1st Cut: Clearance between the part with the leading angle and the end of the ramping path

Plunge Distance for 1st Flank: Plunge distance for ramping path
Plunge distance for 2nd Flank: Plunge distance for horizontal path

Machining tolerance is the maximum distance between approximated curve and tool path segments

Double-click on the value to modify it
Ramp Recess Turning Operation: Strategy (4/4)

Distance and angle before Rework Plunge for rework approach

Lift-off Distance & Angle at the end of the path before the rework
Ramp Recess Turning Operation: Geometry

This Tab Page includes a sensitive icon dialog box which allows the selection of:

A. Part Elements (only profile)
B. Stock Elements (only profile)
C. Offsets can be applied on the Part and the stock. (Double-click on the value)
   - Stock Offset: Offset defined perpendicular to the stock profile.
   - Radial and Axial offset on part
   - Part Offset: Offset defined perpendicular to the part profile.
D. You can generate the profile corresponding to the previous operation and reuse these elements to define the new geometry.

All the created geometry is stored in the IPM CATPart.
Tools and Tool Assemblies Management

In this lesson, you will learn how to manage Tools and Tool Assemblies.

- Lathe Tool Assembly: Overview
- Presentation of the Tools
- Selecting Tool or Assembly from a Catalog
- Creating a New Tool
- Tool Assembly, Tool, Insert, Tool Compensation
- Lathe Tool Assembly Convention
- Insert-Holder
- Output Point
- Summary of Conventions
- Importing Tools: General Process
Lathe Tool Assembly: Overview

In the Tool Tab page there are three different sub-tab pages:
A. Assembly
B. Insert Holder
C. Insert

In each of these 3 tab pages, you can:
D. Change the Name (attribute)
E. Add a Comment (attribute)

Using More button, you can access to all tool and insert parameters.
Lathe Tool Assembly: Presentation of the Tools (1/2)

Accessible tools in different Machining Operations:

The different types of tool you can use for the current operation (A)

- Rough, Profile Finish Turning
- Groove, Groove Finish, Recess Turning
- Ramp Rough, Ramp Recess Turning
- Thread Turning
- Sequential Turning

Select a tool already used in the document (B)
Select a tool from a catalog, external database (C)
Lathe Tool Assembly: Presentation of the Tools (2/2)

There are five different types of Tool bodies:

There are eight different types of Inserts:

You can use 2D viewer for editing tool characteristics by double-clicking on the values.
Selecting a Tool from a Tools Catalog (1/2)

- Different types of Query: (A)
  - Simple
  - Advanced
- You can select a tool from a predefined catalog (B)
  (the catalogs location is specified in the NC Manufacturing settings)
- The different types of tool you can use for the current operation (C)
Selecting a Tool from a Tools Catalog (2/2)

Simple query: specify the name of the tool (or a part of it)

Advanced query: create criteria for search via Attribute-condition-value settings
To appear in the result list, a tool must meet all criteria

Via contextual menu you can:
- Reorder or filter the list of attributes
- Look for a tool by a character string

The same functionality (Simple, Advanced queries) is available to search for a full tool assembly in a Tool Assembly catalog rather a single tool.
Creating a New Tool (1/3)

Now you will see how to create a new Tool.

- In Tool tab page:
  - Select the type of tool you are creating (external, internal or frontal)
  - Type a new name for your tool
  - Type a comment (optional)
  - Click « More » to access tool parameters:
    - Geometry
    - Technology
    - Compensation
Creating a New Tool (2/3)

- Select the type of Insert you are creating
- Type a new name for your insert
- Type a comment if needed
- Click « More » to access insert parameters:
  - Geometry
  - Technology
  - Feeds & Speeds
Creating a New Tool (3/3)

In Assembly tab page:

- Type a new name for your assembly
- Type a comment if needed
- Number of the tool in the catalog (attribute)
- Modify parameters if needed:
  - Geometry
  - Technology

You will see now in more detail different important points
Tool Assembly

Select a tool assembly already used in the document

Search for a tool assembly in a Tool Assembly catalog

Number of the tool in the turret

This is the angle between tool axis and perpendicular direction to spindle axis.

Specifies the X,Y,Z components of the total set length of the tool assembly

Allows to inverting the tool.

Defines the compensation point for which gauges values can be defined.

Defines the Number of components of the assembly, it may include the tool, the holder and the adapters

Define the Output point for each component
Tool and Insert Technology

Insert:

Geometry Technology Feeds & Speeds
Life time: 0s
Machining quality: Either
Material: Ceramics

You can choose for the machining quality between: Rough quality, finish quality or Either

Insert material, you can select one of the following: Coated Carbide, Cermets, Ceramics, Carbide Boron Nitride and Other.

Tool:

Geometry Technology Compensation
Max machining length: 0mm
Max life time: 0s
Coolant syntax:
Weight syntax:
Min diameter: 0mm
Max boring depth: 1000mm
Max recessing depth: 1000mm
Leading angle: 90deg
Trailing angle: 90deg

Specify the coolant supply syntax
Gives the weight of the component
Only used for internal tools. Defines the minimum diameter of the hole which can be cut
Defines the maximum recess/boring depth

If they are set to 0, the angle of the insert is used for the trailing and leading angles.
How to Define a Tool Compensation

1. Select Tool tab and Holder sub-tab
2. Select the More button
3. Click Compensation Tab Page
4. MB3 on Compensation site to edit
5. Modify the Compensation parameters
   - Corrector Number
Lathe Tool Assembly Convention

- Machining Axis is always a Direct Axis system:
  If ZX considered (with Z being the Spindle Axis, X the radial axis), it defines a direct axis system.

- Selected Geometry (e.g. Finish Profile) can be selected either in X+ or X-:
  - System will figure out whether machining is in X+ or X- based on Insert-Holder orientation (as defined using Set-Up Angle)
  - Depending on the options of a Machining Operation, several possibilities might exist, (e.g. Frontal machining), User option is then available (X+/X-) to specify what the system must do.

Note for Horizontal Lathe Machine-Tool
Convention for spindle rotary direction:
Looking in Z direction, Chuck and Jaws to your back
Insert-Holder

In mixing three parameters (Setup Angle, Tool Inverted, Hand Style), we can obtain different configurations for positioning the tool.

Left Hand (A): Insert is visible and points to left
- Left Hand Inverted
  « Tool inverted » tagged (B)

Right Hand (A): Insert is visible and points to right
- Right Hand Inverted
  « Tool inverted » tagged (B)

e.g. Setup Angle = -20°
Output Point

Reference is made by the Spindle/Radial Axis (usually Z/X)

Output Point definition is to be considered according X+/Z+ quadrant
Summary of Conventions

- **« Right» Tool**
  - Set-Up Angle = 0°
  - Inversion = NO

- **« Left» Tool**
  - Set-Up Angle = 0°
  - Inversion = YES

- **« Left» Tool; Set-Up Angle = -90°**
  - Inversion = NO
  - Inversion = YES

- **« Right» Tool**
  - Set-Up Angle = -90°; Inversion = NO

- **« Right» Tool**
  - Set-Up Angle = 180°
  - Inversion = YES

- **« Left» Tool; Set-Up Angle = 180°**
  - Inversion = NO

- **« Left» Tool**
  - Set-Up Angle = 180°; Inversion = NO
  - Inversion = YES

Student Notes:

- « Left » Tool Set-Up Angle = 0°
  - Inversion = YES

- « Right » Tool Set-Up Angle = 0°
  - Inversion = YES

- « Left » Tool Set-Up Angle = -90°
  - Inversion = NO

- « Right » Tool Set-Up Angle = -90°
  - Inversion = NO
Importing Tools: General Process

1. Click Import Tools Icon

2. The Search Tool dialog box is displayed
   Select Tools Catalog via Look In Item

3. Select your Tools in the list

The selected tools are automatically added in the Resources List and available now for queries in the Document
Methodologies for Lathe Machining

In this lesson, you will learn methodologies for specific cases of Lathe Machining.

- Frontal and Back Groove Turning
- Frontal Groove Turning Operation Definition
- Back Groove Turning Operation Definition
- Turning on a Milling Center with Facing Head
- Facing Head
Frontal and Back Groove Turning

You will see which is the right methodology to machine Frontal and Back grooves.

An optional exercise is available on this topic: AddEx04 – Groove Turning
Frontal Groove Turning Operation Definition

Strategy Parameters:

Tool Parameters:

Use an Internal Groove Insert-Holder with Left Hand style and a Setup Angle = 90°
Back Groove Turning Operation Definition

Strategy Parameters:

Tools Parameters:

- Use an Internal Groove Insert-Holder with Right Hand style and a Setup Angle = 90°
- Select ‘Other’ as Orientation (Back doesn’t exist)
- Define an ‘Angle of incline’ of 180 degrees
- Use the same tool as previously but change the Hand Style from Left to Right
Turning on a Milling Center with Facing Head: illustration

This task illustrates how to create turning operations on a milling center machine equipped with a facing head (Andreas Head) and rotary table.

An optional added exercise is available on this topic: AddEx05 – Facing Head
Facing Head

Machine Selection: Milling Machine

Milling Machine tool selection on part operation allow you to specify a local machining axis system for the Lathe Operation.

Tool Path Definition and Simulation:

In a single part operation you can mix Milling, Drilling and Lathe operations according to several local machining axis system. Reduce number of set-ups and machines!