1 What's new

New and redesigned machining strategies
New Pencil operation

Strategies

One pass
One pass generates a single pass along every inner corner.

Parallel passes
The Parallel passes strategy generate multiple passes along inner corners of the part. The passes represent offsets along the machining surfaces of the passes like the One pass strategy would generate. The number and the step-over between passes can be set. The machine by strokes feature allows separation of the toolpath into regions so that smoothly connected regions are machined separately from one another. The connection angle defines the angle between passes that is considered to be a smooth connection.

Job zone

Machining surfaces
The passes are generated only in the places where the tool contacts with the machining surfaces. If the machining surfaces are missing, the toolpath is generated against the whole part.

Job zone
Use the job zone to trim the passes outside the specified containment areas.

Restrict zone
Use restrict zones to easily create restriction geometry from curves and edges.

Machining slope

The rest machining operation generates passes along inner corners of the part.
Use the Machining slopes parameter to machine only steep or only shallow areas.
Use the steep/shallow split angle parameter to set the slope angle which separates the steep from the shallow.

**Bottom level**
A bottom level can be set for the passes

New Corners cleanup operation

The rest machining operation takes the diameter of the previous tool as a parameter and generates passes where the previous tool would leave unmachined material.

**Parameters**

Previous tool diameter
The diameter of a spherical mill which is used for the rest material calculation.

**Cut depth**

The maximum cut depth for a cut

**Step**

The maximum step-over between passes

**Strategies**

**Along**
The passes are generated along the corners.

Across

The passes are generated across corners

Combined

For shallow areas - along passes are generated, for steep areas - across passes are generated.
Job zone

Machining surfaces
Passes are generated only where the tool contacts the machining surfaces. If the machining surfaces are missing, the toolpath is generated against the whole part.

Bottom level
A bottom level can be set for the passes

New 5d surfacing operation

The finishing operation allows machining of surface models with a variety of strategies (parallel to plane, parallel to curve, morph and others) and tool axis orientation modes (fixed, normal to surface, to rotary axis, through point, through curve, etc).

Typical workflow
1. Create the operation
2. Add the Machining surfaces to the Job assignment
3. Select the toolpath strategy
4. Choose between the tool center and tool contact calculation modes.
5. If needed, change the step-over and/or the toolpath margins.
6. Select the tool axis orientation mode.
7. If required, turn on the roughing passes.
8. Calculate the toolpath.

Calculation based on tool center versus calculation based on tool contact point

1. Calculation based on tool contact point
In this mode the work passes are generated by calculating curves on the machining surfaces as the first step and then positioning and orienting the tool relative to the calculated curves in such a way that the point of contact of the tool with a machining surface stays the same. It is desirable, for example, when smooth surfaces are machining, or doing flank milling.

2. Calculation based on tool center

In this mode the work passes are generated in such a way that at first the machining surfaces are being offset either by the tool shape itself for 3 axis machining or by the tool radius along the surface normal for 5 axis machining and only then the sections of the offset surfaces are calculated. For example, for the Parallel to plane strategies it means that the generated work passes all lie on parallel planes. Another advantage of this mode is that not only the surfaces themselves but the edges between machining surfaces are taken into account.

Strategies

Parallel to plane
The passes are generated as a result of intersection of the machining surfaces and parallel planes surfaces. Three options are to choose from.

1. Parallel to vertical plane
The planes are parallel to the tool axis, as in the Plane toolpath. Additionally the angle of rotation of the planes around the tool axis can be specified.

2. Parallel to horizontal plane

The planes are perpendicular to the tool axis, as in the Waterline toolpaths.

3. Parallel to 3d plane

The planes can be freely oriented in space regardless and independently from the tool axis orientation.

Margins

It is possible to limit the generated passes by two points.
The passes are generated by finding points on the machining surfaces that lie on the same distance from the First Curve. Unlike with the Scallop toolpath the step-over between passes is not guaranteed to be constant.

Margins

Use the **Start margin** to set the starting offset for the first generated pass.

Use the **Zone Width** to limit the number of generated passes. You can either specify an exact value for the zone width or define the job zone with a point.
Across curve

The passes are generated by sectioning the machining surfaces with planes perpendicular to the First curve.

Morph between two curves

The passes are calculated by finding points on the machining surfaces which satisfy the criteria that the ratio of the distance from the given surface point to the First curve to the distance from the point to the second curve stays the same for a given pass.

Margins
Use the Start margin and the End margin to set the offsets from the generated passes to the First and the Second curves respectively.

Around rotary axis
The passes are calculated as sections of the machining surfaces with the series of cylinders around the rotary axis.

**Tool axis orientation modes**

**Fixed**

In this mode the tool axis orientation stays the same for the entire toolpath (unless the vertical clearance angle is specified). Basically what you get is a conventional 3 axis milling toolpath. The Vertical clearance angle feature automatically tilts the tool away from the wall surfaces in places where the slope of a surface is steep or negative (a surface normal looks down the tool axis). An additional clearance angle can be used. It allows simple machining of undercut areas.
Normal to surface

The tool is oriented by normal to machining surfaces. Additionally the lead and lean tool angles can be applied to further tilt the tool along or to the side from the cutting direction.

Flank

The tool contacts machining surfaces with the peripheral part (cylindrical part for the cylindrical mills). Additionally lead and lean angles can be applied. The strategy can be used for swarf milling.

Through point
The tool axis is oriented to the specified point.

Through curve

The tool axis is oriented to the nearest point of the specified Tilt curve.

To rotary axis
The tool axis is directed to the rotary axis, as in the rotary machining. Additionally the side angle to the rotary axis can be specified.

4 axis machining with the Rotary axis
The rotary axis feature allows to transform a 5 axis toolpath into a 4 axis toolpath by locking one of the components (X, Y, Z) of the tool axis direction.

Job assignment

Machining surfaces
The machining surfaces define where the toolpath will be calculated.

First curve
The First curve is used in the Parallel to curve strategy to define the curves parallel to which the passes are calculated and in the Morph between two curves strategy to define the first curve. You can select one or more not necessary connected curves or edges as the First curve.

Second curve
The Second curve is used in the Morph between two curves strategy. You can select one or more not necessary connected curves or edges as the second curve.

Tilt curve
The tilt curve is used for the Through curve tool axis orientation mode.

Job zone
Use the job zone to trim the passes outside the specified 2d containment areas. The plane of the containment area is defined by the initial tool orientation.

Restrict zone
Use restrict zones to easily create restriction geometry from curves and edges.

Gouge control
By default the Check part option is enabled. It means that the operation generates a gouge-free toolpath. Often when the geometry of the machining surfaces is simple and the minimal curvature of the machining surfaces is larger than the tool radius you can disable the Check part option to speed up the toolpath generation.
Roughing

Enabling the roughing option turns on the roughing passes. The number of layers and the step between layers can be set. The Check workpiece option can be used to eliminate air cutting. There are several different modes of calculation of the roughing passes.

1. By surface normal

The roughing passes are calculated as offsets from the finishing passes along the surface normal.

2. Along tool axis

The roughing passes are calculated by simple shift of the finishing passes along the tool axis.

3. In tool plane

The roughing passes are calculated as offsets from the finishing passes in the frontal tool direction. It works best for the parallel to vertical plane strategy and when machining surfaces with the front of the tool.
4. Perpendicular to tool axis

The roughing passes are calculated as offsets of the finishing passes in the plane perpendicular to the tool axis. The mode works better for the parallel to horizontal plane strategy, and when machining surfaces with the peripheral part of the tool.

New Helical operation

The operation generates continuous helical passes with the given vertical stepover between the
top and the bottom level.

**Parameters**

**Step**
The vertical distance between any two adjacent complete helix turns.

**Strategy**

Use the Start From parameter to choose between the top-down and bottom-up machining. Use the milling type parameter to choose between climb and conventional milling.

**Job zone**

The whole part is being machined. The Top and the Bottom levels of machining can be specified.

New Rotary roughing operation

Roughing rotary is a 4 axis toolpath that removes workpiece material layer by layer. It is similar to the Roughing Waterline except that the machining layers are not planes, but cylinders around the rotary axis.

**Rotary axis**
The rotary axis is defined by its origin and direction. You can easily set the desired parameters of the rotary axis both in the inspector and with the mouse in the graphic view.

### Job zone

The job zone is defined by:
- the minimal and maximal axial positions,
- the angular sector,
- top and bottom levels of machining.

### Machining parameters

The depth step can be set as an exact value as well as the number of layers. The machining step defines the maximal distance between the machining passes in a layer.

### Machining strategies

Currently three strategies are available.

1. Circular

   The circular strategy generates passes around the rotary axis that represent 4 axis arcs.

2. Linear
The linear strategy generates passes along the rotary axis that represent linear cuts. The angular step-over between passes on each layer is the same, what means that the real step-over gradually decreases when approaching the bottom layer of machining.

3. Spiral

The Spiral strategy generates helical passes. The pattern is well suited for machining parts like screws and impellers.

New Lathe machining operations
Lathe machining has undergone complete overhaul. Eleven new operations based on Lathe contouring have been added. The lathe contouring operation itself is no longer available. The old lathe operations are moved into the Legacy group.

The new lathe operations are easy to use, require much less effort to setup and allow interactive editing of job geometry. The operations are classified by the machining type into roughing and finishing, and by the tool orientation into OD, ID and face. Only applicable types of cycles can be added to the job assignment. In addition, a created operation already has the most appropriate cycle automatically selected for it. If any geometry is selected when creating an operation it is automatically used as the job assignment. Moreover, the most suitable free turret head position is automatically selected for the tool holder based on the type of the created operation (OD, ID, face).

Also the limitation of simulation of lathe operations after milling operations has been overcome. In previous versions the simulation didn't take the machining result of milling operations in succeeding lathe operations into account. Now it does.

In `<PRODUCTNAME>` lathe machining is grouped by operations:

- `<Lathe facing>`
What's new

- <OD Roughing, ID Roughing>

- <OD Finishing, ID Finishing>

- <Lathe hole machining>
What's new

- Lathe part-off

- OD Grooving, ID Grooving, Face grooving

- OD Threading, ID Threading
All operations, which were inherited from the previous version moved to the Legacy group. See documentation for the previous version of `<%PRODUCTNAME%>` to get more info about these operations.

Lathe grooving cycle enhancements

The Lathe grooving cycle has been completely redesigned to always produce correct toolpaths. The Roughing direction parameter has been added to allow machining of grooves not only from the groove center but also in forward and backward directions. The direction can be easily toggled with the blue arrow in the graphic view. The Finishing direction parameter allows the finishing direction to be different from the roughing direction. For the forward and backward grooving the overlap can be added in the bottom-up direction. The maximum size of such motions can be restricted with the Max scallop for overlap parameter.

The numerous compensation problems have been resolved. Now grooves can be easily machined in any compensation mode: both with computer, CNC side (with second corrector) and manual compensations.

The Multilayer machining has been improved. The check workpiece feature is implemented much more reliably and is enabled by default.

**Advanced grooving** element generates the complex tool path for the any kinds of the lathe grooves. It allows to machine the outer radial, inner radial, face or inclined grooves.
The cycle parameters can be defined in the properties window.

The groove cycle defines the bottom point of the groove contour and machines the both sides of the groove by the different tool tips. Every side is machined from the end point of the profile to the bottom point. The tool tips are defined in the Tool dialog that is shown below.
**Machined side** parameter allows to machine only one side of the groove. If **1st tool tip** is selected then the only side that is touched by the first tool tip will be machined.

Advanced grooving cycle generates rough and finish passes. **Tool path** parameter defines what passes must be generated.

If **Finish only** is selected then finish pass only will be generated and the dialog will show only the parameters for the finish path. If **Rough only** is selected then the rough path will be generated and the parameter for the rough pass only is shown in the dialog. If **Rough and finish** is selected then all parameters are shown.

**Safe distance** parameter defines the distance from the groove top to the level of the rapid motions. To edit the safe distance it is possible to drag the point in the graphical window or input the value in the dialog.
Parameters of the finish path
Radius compensation parameter is described here.
Insert width compensation defines the correctors using.

Computer mode generates the tool path for the first corrector only.
Off mode is not realized.
Use 2nd corrector mode generates the tool path for the first and second correctors.
If the option Feed/Compensation on/off cuts is enabled then the additional cuts for the compensation on/off is generated. The length of the cuts also defined.

Parameters of the rough path

Rough step defines the distance between the rough passes. It can be defined in the percents of
the tool width or in the units of the length. If option Adjust step is defined then <%PRODUCTNAME%> automatically changes the step for the equal force on every plunge. Max Step deviation defines the maximal deviation of the adjusted step from the defined step. This value can be defined in the length units or in the percents of the tool width. 
Canned cycle option allow to use or not use the canned cycles in the tool path. 
Rough stock defines the additional stock for the rough passes. 
Multilayer option is necessary to generate the rough passes in some layers. It is possible to define the layers count or the depth of the layer.

If Overlap option is enabled then the additional tool path is generated from the end of the rough cut to the end of the previous rough cut. Overlap option would be inactive if toolpath uses the canned cycles.

Chip breaking option can be enabled on the first rough cut or on every rough cut. It is possible to define the number of the breaks or the step for the chip breaking. The return distance for the chip breaking can be defined in the length units or in the % of the plunge step.

If Delay at the bottom option is enabled then the Delay command is generated in the end of the rough cut. The delay time can be defined in seconds or in the turns of the part.

Back off distance can be defined in the length units or the percents of the rough cut step.

New Lathe slotting cycle

The Lathe slotting cycle is similar to the Grooving cycle and is fine-tuned for machining of rectangular grooves. So now rectangular grooves can be efficiently cleared in one click.

Lathe roughing cycle enhancements

The check workpiece feature is now much more reliable and can be safely used in most of the cases. The Overlap feature now works even when the check workpiece is enabled.

The Compensation switching sequence has been changed. Now the radius compensation is switched on and off not at the engage and retract moves but before and after the rapid moves leading to those. It eliminates unexpected toolpath deformations at the beginning and in the end of a contour.

New Lathe part-off cycle
The cycle performs separation of the finished machined part from the remaining workpiece with simultaneous finishing of the back face of the part. The job assignment is generated automatically as the vertical line at the left or the right face of the part connecting the outer and the inner diameters of the workpiece.

It is possible to produce a chamfer or a rounding at the finished face. It is also possible to perform chip breaking during the parting off with a delay only at the bottom or at every peak. Return to the top level can be disabled in case if you want to use the current tool as a stopper for the bar ejected from the feeder.

The Insert width compensation automatically adds the insert width to the axial coordinate of the machined face. The actual left or right tool length corrector can be easily switched.

Lathe threading cycles job assignment enhancements

The screen graphics for the Lathe threading job assignment has undergone complete overhaul. Now the profile of the machined thread is displayed at place. Dimensions of the thread pitch, depth, top and bottom diameters are displayed and can be edited directly in the graphic view. The new threads library widget allows to easily find and select standard and custom threads. The list of threads is both easily customizable and extendable in an intuitive manner.

New Lathe hole machining operation
The new Lathe hole machining operation replaces the old Lathe drilling operation. It allows drilling of axial holes with the fixed tool. In fact it is the same with the milling Hole machining operation. The main difference is that the lathe hole machining uses the lathe simulation type. The Cycle format parameter defines the type of the CYCLE command the operation generates (EXTCYCLE equals to mill operation, CYCLE - old drilling cycle).

Module works 5X kernel updated

The new version of the Moduleworks kernel from December 2016 with numerous new features and bug fixes has been integrated.

New features of milling operations

New Adaptive feedrate feature in the roughing waterline operation

The feature automatically adjusts the toolpath feedrate based on the actual tool engagement. It reduces feedrate in tight corners where the tool engagement approaches 100% and increases feedrate as the tool engagement approaches 0% (the air cutting).

New Check Holder feature
The Check holder feature detects segments of the toolpath where the tool holder collides with the part and modifies those segments according to the specified strategy. Three strategies are available: Trim toolpath, Frontal tilting, Side tilting.

The operations that support the check holder feature (rotary, morph, 5D surfacing) have the "Check holder" parameter at the "Parameters" tab in the inspector.

New Advanced axes limits control feature in 5-axes operations

The option allows to avoid two types of problems in a 5-axis toolpath.

1. It tries to avoid overturns in the middle of a work pass when it's possible. Overturns are preferably performed on rapid motions.
2. It generates a smooth path in the singularity zones. A singularity zone is the machining position where one tool axis orientation can be achieved with an infinite number of machine axes positions. For example, if A-axis is equal zero, then C-axis can be any.

New toolpath multiplying preview
New interactive widgets along with the preview of the final result of multiplying is now displayed in the graphic view when the Multiply scheme/Multiply by axis parameters of an operation are selected in the inspector.

New Robots features and enhancements
Robots map window enhancements

On the basis of numerous requests from users, the robot axes map editing window was updated. The main improvements include.
What's new

- the spline control points are now snapped to horizontal and vertical lines when being dragged with the mouse;
- The rapid motion segments of the toolpath are now visualized as a reddish background in the map, so you can easily see where a work pass starts and ends;
- The spline/linear interpolation mode option is added: In the linear interpolation mode the axis control spline points are connected with straight lines, which significantly reduces the number of NC blocks;
- The Build automatically feature has been reimplemented to produce more optimal paths with fewer control points and respected safe distance to the collision zones.

New approach and return editing and automatic motion planning

Now you can easily modify the approach and return sections of the generated toolpath directly at the Simulation tab. Just select an Approach (Return) node in the tree at the left, and the Approach Edit Panel will appear in the graphic view (see the screenshot). With this toolbar you can make the following changes: Insert the current machine state into the approach/return sequence as a command (PhysicGOTO or MultiGOTO); Edit the selected command (select the tree command you want to change, press the Edit button, then start dragging the machine nodes either directly on the screen or with the Machine control panel); Delete the selected command; Clear the whole approach/return section; Calculate automatic path with Motion Planner.

The calculate automatic path feature is able to automatically generate a collision free transition using one of the algorithms for motion planning. The parameters of the motion planner such as the time limit for the path calculation can be set in the drop-down menu.

New Technology features

New customizable Create new operation window and pop-up menu
The New operation window has undergone complete overhaul. The new Unavailable button at the bottom, when pressed, reveals the complete list of all operations, including the operations currently unavailable for creation. Reasons of why a particular operation is unavailable for creation are outlined at the Information panel on the right.

The new Customize... button activates the operations list customization mode. In this mode you can create new operation groups, or delete existing ones, you can turn on and off visibility of operations, change the order of operations, change the captions of both operations and groups and much more. You can save the changes as a new configuration and then easily switch between different configurations. The New operation pop-up menu reflects all the changes you make in the operations list.

New operation status widget

When clicking on the operation status icon next to the operation name in the technology tree the new operation status panel appears. In this panel you can see the operation simulation status together with the operation statistics such as the machining time, the toolpath length, the number of NC Blocks.

Selective G-code generation in the G-code generation window
The new tab selector at the top left part of the G-code generation window allows fast switching between the postprocessors view and the operations view. When the Operations view is selected, the left panel displays the full list of the operations of the project. The checkboxes against the operations allow selective generation of G-code.

New Interactive splines and curves drawing feature

Now you can easily draw various types of curves such as e.g. splines directly at the Technology tab and use the created geometry to define Job assignment, Part, Workpiece, or Fixtures primitives. The geometry is interactive so you can easily change it at any moment by dragging the hot points of curves with the mouse.

New Tool reach inspector
The Tool reach inspector panel allows to visualize reach zones of the part for the given tool-holder and tool axis orientation. Moreover, the Find best angle feature allows to find the most optimal tool axis orientation at which the tool removes maximum volume of workpiece material. The inspector is activated by pressing the button on the ribbon.

Utilities setup window enhancements

The position and the visibility of any utility regardless of its type can now be changed, no matter whether it's a standard utility, or an external one.

The new type of a plug-in extension, the *.dll extension, has been added in addition to the already available executable extensions (*.exe, *bat) and SprutScript extensions (*.spr).

New Simulation features

New G-code based simulation
The new G-code simulation button activates the new simulation mode in which the actual postprocessed G-code is being simulated instead of the intermediate CLData. The toolpath graphics is also generated based on the interpreted G-code. So in this mode possible errors that might appear at the postprocessing stage can be caught.

If the mode is enabled <%PRODUCTNAME%> automatically generates G-code of an operation with the postprocessor specified in the machine settings every time the operation is being recalculated. After that the generated G-code is being interpreted according to the settings specified in the interpreter file (the path to this file is set in the machine settings window in the box below the the postprocessor file box).

Note. The feature is in an alpha stage, be careful while using it. Only Fanuc like ISO G-codes are supported at the moment.

Solid simulation method rewritten from scratch
(not included yet at the alpha stage)

Simulation control methods enhancements

<%PRODUCTNAME%> now automatically skips all auxiliary commands that do not involve any tool motion, (e.g. the feedrate command), when you press the step forward button.

Pressing and holding the step forward button starts the simulation at slow speed. Releasing the button stops the simulation.

Pressing and holding the play button starts the simulation at normal speed. Releasing the button stops the simulation.

Hitting or holding the [Space] key on the keyboard is equivalent to pressing/holding the step forward button.

A number of keyboard shortcuts for the simulation control has been added. Hover the mouse pointer over a button to see a hint window with the hot key.
Improved visualization speed of the machining result with part comparison

The frame rate of the machining result with part comparison feature has been substantially improved, especially for the big models and meshes and high tolerances. Moreover, the video memory consumption is now just a fraction of what it used to be.

New 3D Model and CAD addins features
New quick ways of selecting multiple objects
Press and hold the [Shift] key and move the mouse with the left button pressed to select multiple faces under the cursor.
Use [Ctrl]+[Shift]+Double click to select smoothly connected faces.

New Import of Coordinate systems from STEP files

New Toolbar for CADbro™
The toolbar allows export of geometric data from CADbro™ into <%PRODUCTNAME%>™.
Addin and toolbar for Kompas™ 15 enhancements

Import of curves, such as linear cuts, circular arcs, polygons, splines, splines by objects, curves by law, splines on the surface, compounds of curves, curves by 2 projections, extract of curve, equidistant of curve, contour, isoparametric curve, projection curve, the curve of intersection of surfaces, cylindrical spiral, conical spiral, rounding of curves, has been added.

Import from SOLIDWORKS™ enhancements

Import of 3d curves (not sketches) of the following types: parting line, flat curve, the combined curve, the curve through the XYZ points, the curve through the reference point, the spiral / flat spiral has been added.

SpaceClaim™ import changes

For SpaceClaim™ with activated Parasolid module support of unique identifiers of geometry entities has been added. By default it is disabled. The activation instruction can be found in the «How change export file extension.txt» document which is located at C:\ProgramData\Sprut Technology\<%PRODUCTNAME%>\Version 11\Addins\SpaceClaim

New Internal import of PLY files

PLY is a computer file format known as the Polygon File Format or the Stanford Triangle Format. It was principally designed to store three-dimensional data from 3D scanners.

New Internal import of AMF files

Additive Manufacturing File Format (AMF) is an open standard for describing objects for additive manufacturing processes such as 3D printing. The official ISO/ASTM 52915:2013 standard is an XML-based format designed to allow any computer-aided design software to describe the shape and composition of any 3D object to be fabricated on any 3D printer.

New licence management features and enhancements for users

New Automatic synchronization of licences with the server
Now it is possible to schedule automatic updates of client licences from server with customizable time intervals (year, month, week, etc.). Running of the application on a virtual machine is disabled by default. Now to be able to run <%PRODUCTNAME%> on a virtual machine you have to get a special licence.