S SprutCAM X

let`s go
MANU
FACTU
RING

sprutcam.com

SprutCAM X

cost-effective and powerful solution for CNC programming

CAD model

SprutCAM X

G-code

```
%
G18 G21 G40 G80
(Roughing waterline)
G54
T0202 (16mm EndMill)
G97 S200 M04
G00 X-216.452
Z-67.6
Z-92.6
G01 Z-93.6 F200
...
```

Streamline your CNC programming workflow with the short learning curve yet powerful CAD/CAM software — SprutCAM
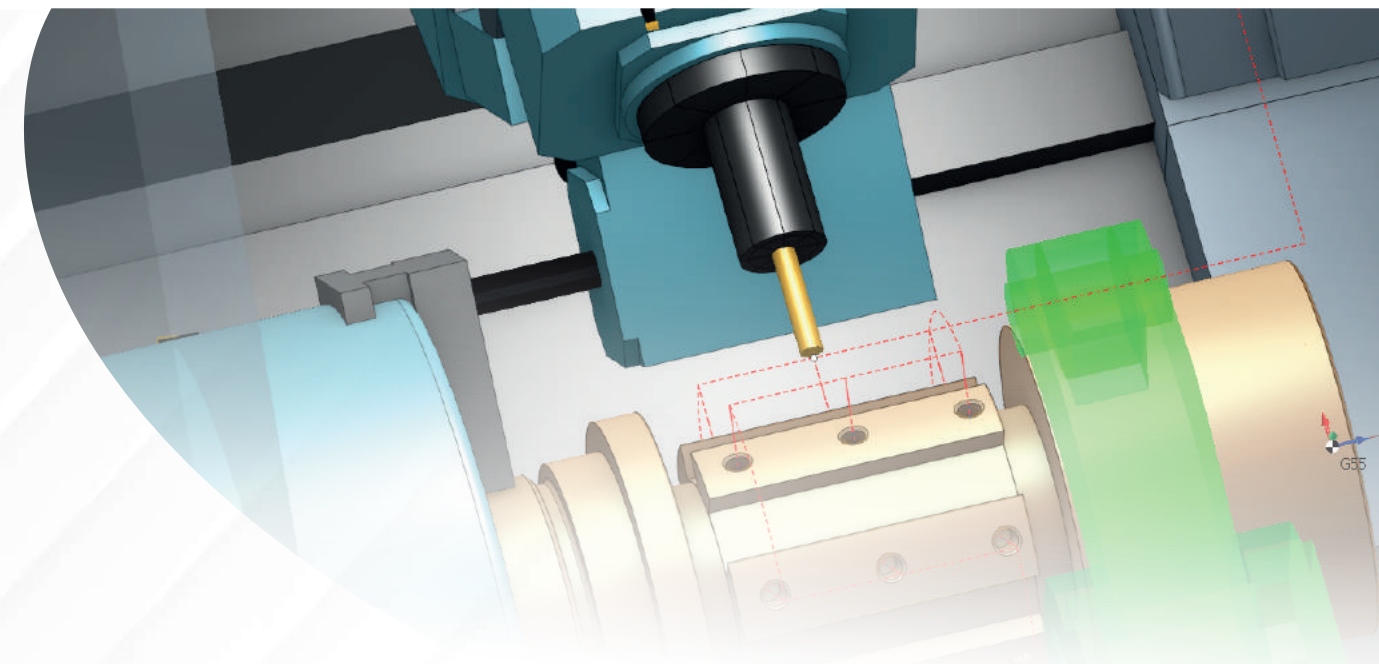
6000 users

CNC machinsts use SprutCAM worldwide

100 dealers in 54 countries

USA, Canada, Germany, Italy, Japan, Spain, Brazil, China, Eastern Europe and Asia

20 years

We are developing SprutCAM X for 20 years

Get the real-world toolpaths without switching to simulation and post-processing

Real-world toolpaths

In SprutCAM X the user is working with a full digital twin of the equipment. CNC machine or robot cell. SprutCAM takes into account the kinematic limitations during the toolpath calculation. Same about rest material during all machining stages. Which results in instant collision detection and reach zone limits consideration.
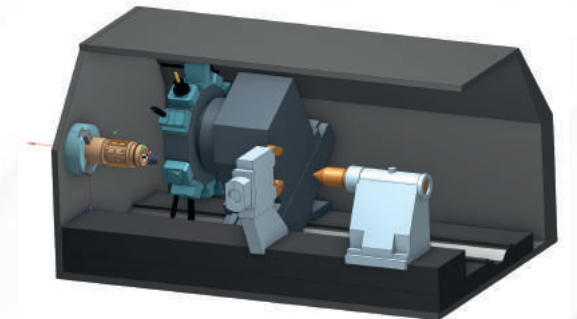

### Taking in account kinematics and rest material on every step

All the limitations of the machine kinematics and the actual rest material that remains after previous operations and the fixture geometry are used to calculate the toolpath. The result of the calculation is not just the Cartesian points in abstract space. It's the actual positions of the machine axes required to reach toolpath points. This means that the CNC machine is 100% able to execute the generated program.

Immediately after the toolpath is calculated, SprutCAM X performs the simulation in the background and shows the new state of the workpiece. Using this result, the user can easily decide how to adjust the parameters without waiting for full simulation or postprocessing.

Thanks to this, the iterative process of CNC programming becomes short and streamlined. And comparing to other CAD/CAM software the workflow itself is intuitive and clear.
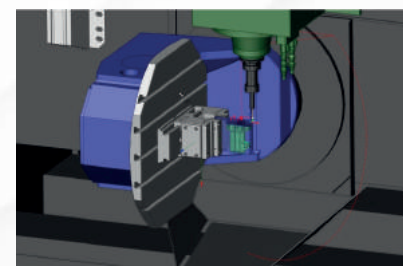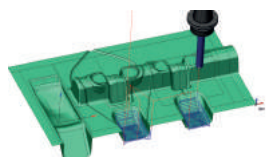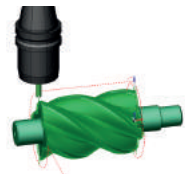
SprutCAM X
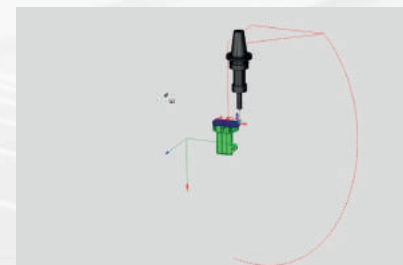
Other CAD/CAM

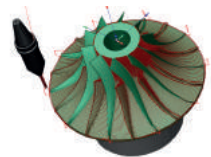# Extensive range

of CNC programming solutions

MILL

2.5 and 3 axis milling

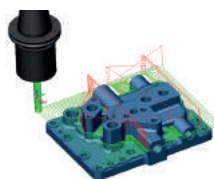Production of body parts, mold&dies and any other parts on a 3-axis milling machine.

4 axis milling programming

Rotary milling. Productions of screws, lades, gear wheels, balusters.
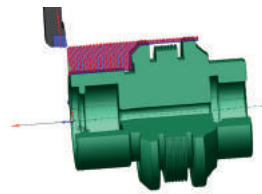
5 axis milling programming

Index (3+2) and simultaneous 5 axis milling. For turbine wheels, blades, ports milling

High speed machining (HSM)

High speed and adaptive toolpaths for quick material removal. Tool life considered.
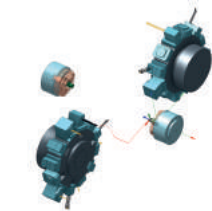
TURN

Turning programming

Turning and boring, threading, trimming, drilling, reaming

MILL-TURN

Mill-turn programming

Turn-mill machines programming. Turning and milling combined in one setup

Dual channel mill-turn with sync

Automatic sync points are set up to optimize machining time. Tool change optimization. Bar feeding and takeover with rotation sync support. Time scale simulation for multichannel control program verification.
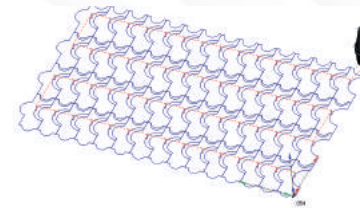
Swiss-type

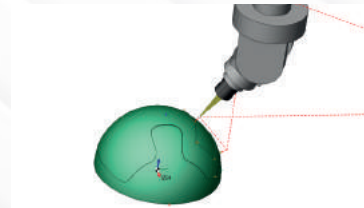Easy template-based swiss-type kinematics definition. User friendly workflow.

CUTTING

2D cutting machines programming

Programming of laser, waterjet, plasma, oxygen cutting and knife cutting on CNC machines
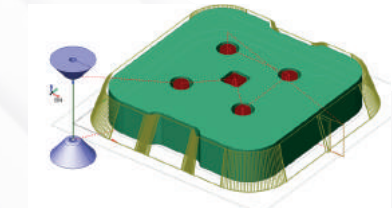
5D and 6D cutting

CNC programming of cutting, trimming, 6D knife cutting. Mill, laser, water, plasma, knife.
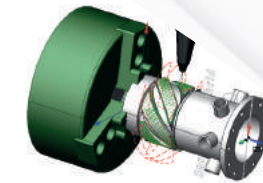
EDM

Wire EDM programming

2 - 4 axis strategies for wire EDM machinihg

HYBRID

Additive manufacturing and hybrid manufacturing programming

Cladding, hybrid technology on 5-axis, mill-turn machines and robots.

INDUSTRIAL ROBOTS

SprutCAM X Robot

We provide a streamlined industrial robots programming workflow for robot engineers all over the world SprutCAM Robot — all-in-one software for industrial robots offline programming

Programming industrial robots:

- Kuka
- Fanuc
- Motoman
- Mitsubishi
- Staubli
- ABB
- Toshiba

Why SprutCAM X

01

Streamlined CNC programming workflow

SprutCAM X calculates toolpath in the native kinematics of the CNC machine

What does this mean to the user? This means that toolpath calculation considers collisions of moving parts of the machine between themselves and with the workpiece and exits beyond the axis limits. All this happens while the toolpath is calculated, not in simulation. You do not need to postprocess the NC code to simulate the real-world result. All machine movements are visible immediately after the toolpath calcula‐ tion. CNC programming becomes visual, workflow is streamlined.
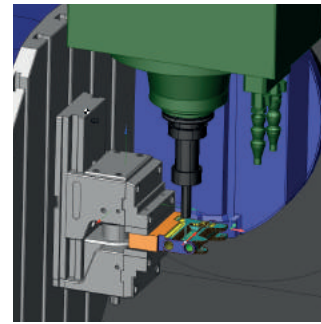
02

Advanced rest material handling:

watch how the workpiece is machined during every operation

In SprutCAM X simulation is processed in the background after every operation. So the workpiece for each subsequent operation is the result of a previous operation.
Thus, when calculating the toolpath, SprutCAM X considers the result of the previous operation.
Toolpath calculation based on the result of the previous operation gives three advantages at once:
visual programming: the result of processing is visible immediately after the toolpath calculation in each operation;
workpiece collision control;
reduced movements by air without material removal.
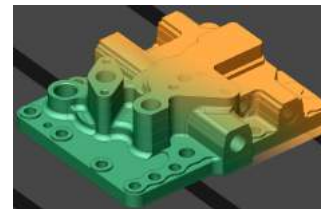
03

Equipment safety: collision control during toolpath calculation

SprutCAM X uses machine axis limits for initial toolpath calculation. There is no need for postprocessing and separate simulation for collision and axis limits control

Tool holder collision control

Tool holder collisions hadnling during toolpath calculation.

Moving parts collision control

Machine moving parts collisions with each other and with the workpiece handling during toolpath calculation.

Axis limits control

Axis limits are considered during toolpath calculation
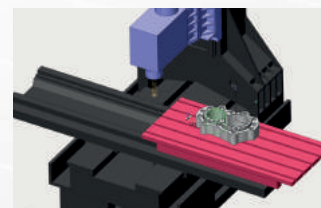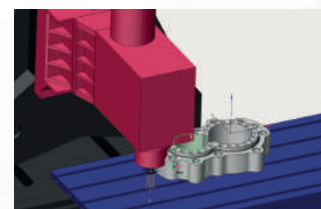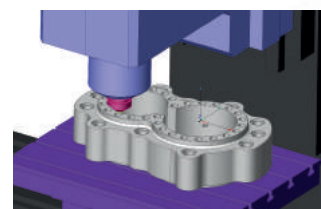
04

Reliable simulation: safety for the equipment

SprutCAM X simulation shows all machine movements during machining process. 5-axis and complex turn-mill machines are not exceptions.

In addition, SprutCAM X automatically marks NC program frames in which it detects collisions, part gauges and axis movements out of limits.

Reliable and detailed simulation is the guarantee to no surprises during physical machining process.
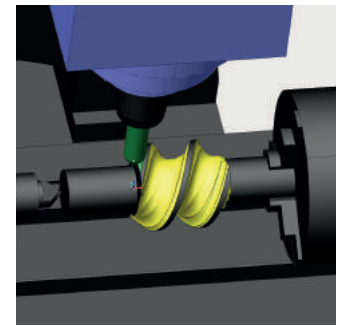
05

Reliable NC code: G-code simulation and verification

If you have SprutCAM X, then you do not need a separate software for G-code verification. G-code verification feature supports Fanuc, Heidenhein, and Siemens CNCs with canned cycles.

G-code verification reveals:

- possible bugs in the postprocessor; at what point of processing the technological commands are turned on and off;
- which movement causes gauge or collision;
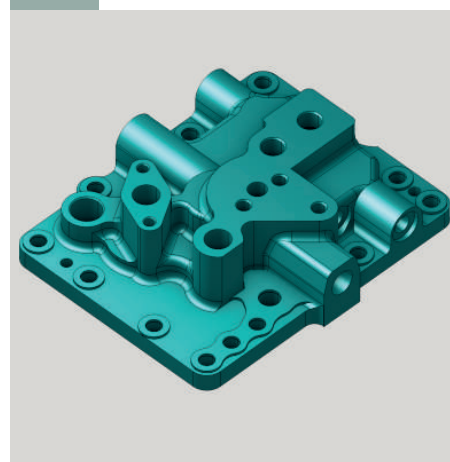- which reference points are used to retract the tool.
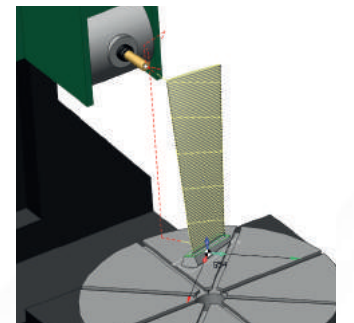
Mill machine programming in SprutCAM X

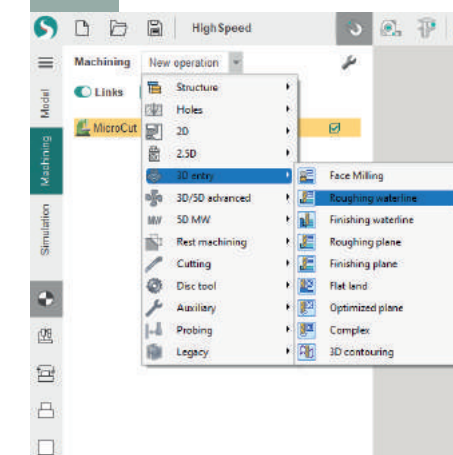## It's easy to get a toolpath in SprutCAM X

In SprutCAM X you get the first toolpath without setting any parameters at all. After that, you can play with parameters, visually controlling the result.
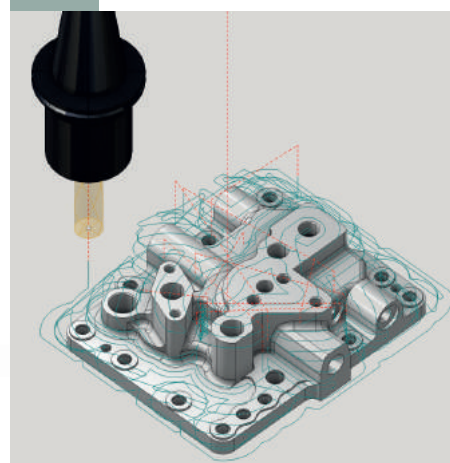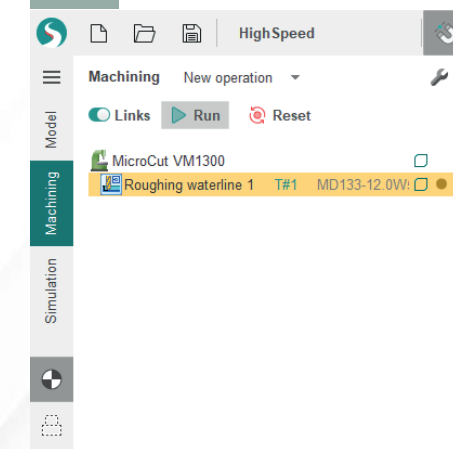
01

Import the model

02

Select the toolpath strategy

04

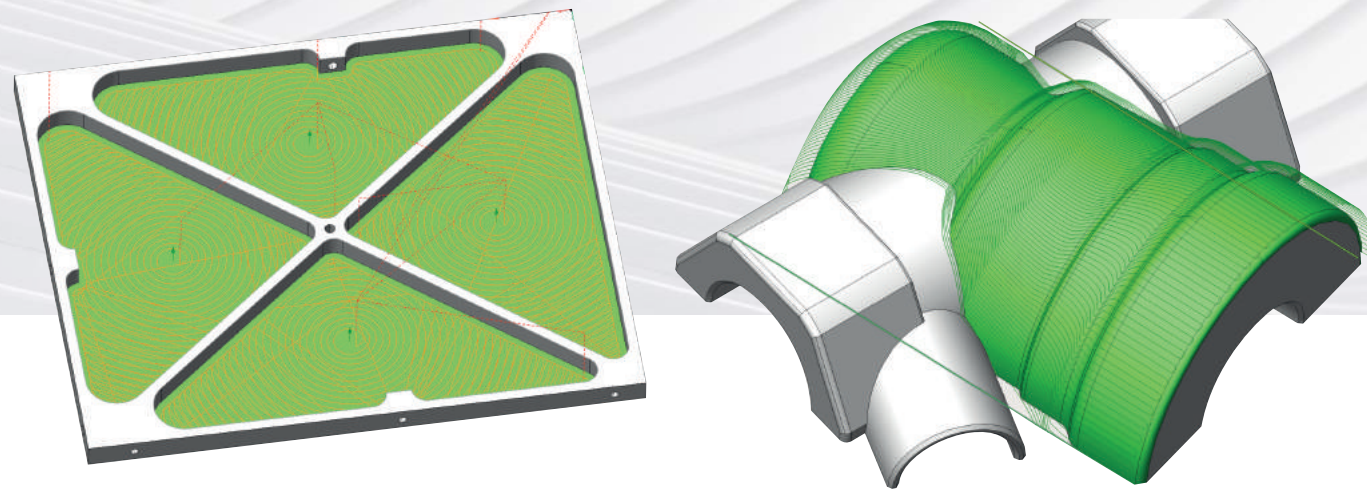Get the toolpath

03

Press "Run"

// 3 AXIS MILLING IN SPRUTCAM X

Remove material quickly

The new "Adaptive" strategy for high-speed sampling. Used for fast material removal. Rapid calculation and delicate with tool.

Get the required surface quality

**Scallop and morph toolpaths will help you to get desired surface finish.**

01
3 basic toolpath strategies for 3 axis machining in SprutCAM X

The most commonly used toolpath strategies for 3 axis machining in SprutCAM X. You will use them often, too.
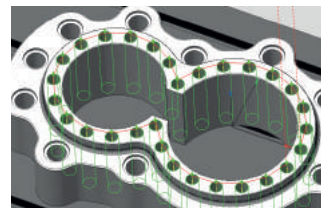
Roughing waterline

Industry standard toolpath. Roughing waterline in SprutCAM X is smart and fast. There are equidistant, parallel, adaptive and high-speed strategies.
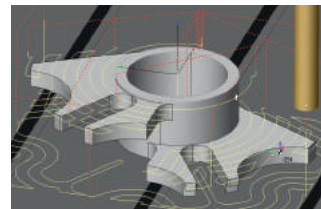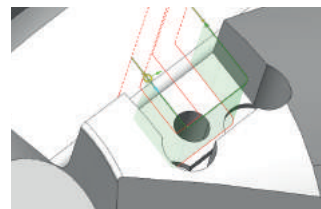
Hole machining operation

Drilling, threading, centering, countersinking, boring, milling of holes.

Along the contour (2D contour)

The most universal operation. You can set flat curves as a job assignment.

Full list of 3 axis machining strategies:

- Face milling
- Roughing waterline
- Roughing plane
- Hole machining
- 2D contour

- 2.5 D contour
- 3D contour
- Morph
- Scallop
- 3D helical
- Finishing waterline

- Flat land
- Finishing plane
- Complex
- Engarving
- Corners cleanup
- Pensil
- Chamfering

02
All 3 axis toolpaths can be used not only with 3 axis machines

But also with 4- and 5-axis milling and turn-milling machines
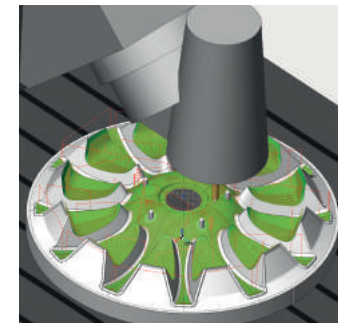
03
3 axis machining time-savers

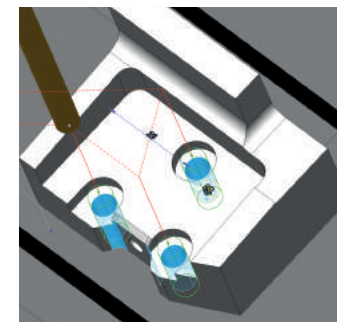Automatic flat faces detection, hole recognition, rest material removal.

Save time with automatic rest material machining

Rest machining removes material that a previous tool or operation could not remove. In SprutCAM X areas for rest machining are detected automatically. Thus SprutCAM X saves machinist's time for toolpath creation and the machine time. Primarily useful for freeform surfaces machining..
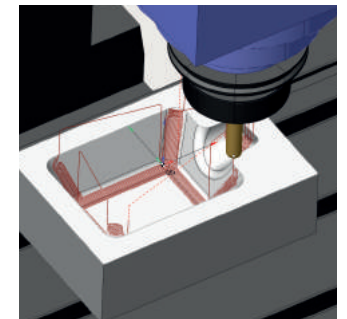
Save time with automatic "select same diameter" feature for holes machining

Select a set of holes with one click and run toolpath calculation. Huge time-saver for parts with large number of holes. For example part for heat exchangers can contain 2000 holes each. This automation saves hours for CNC machinist.

Save time with automatic flat faces detection

Automatic flat faces detection makes the additional passes for the all flat faces in the machining area. Additional toolpaths for the faces between the steps of roughing waterline are not required
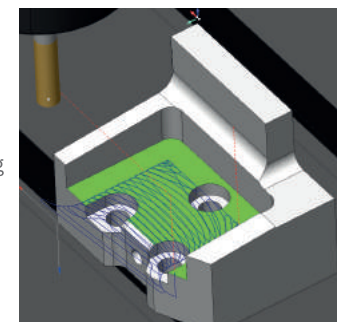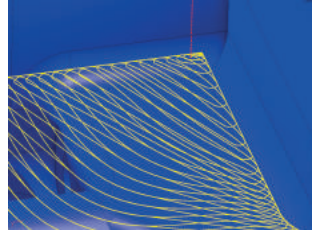
//
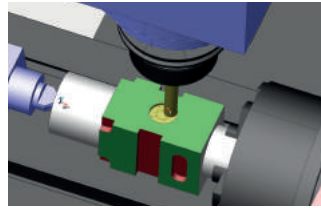# HIGH-SPEED MACHINING IN SPRUTCAM X

High speed machining

SprutCAM X contains its own branded Adaptive SC strategy for high-speed cutting conditions. Adaptive SC is providing constant tool load, optimized transitions between machining layers and lots of precise controls. Adaptive SC is also featuring very high calculation speed.

//
CONTINUOUS 5 AXIS MACHINING IN SPRUTCAM X

Comprehensive toolpath strategy pack for 5 axis machining.

8 toolpath strategies for tool tip trajectory:

- parallel to horizontal plane
- parallel to vertical plane
- parallel to 3D plane
- parallel to curve
- morph between two curves
- around rotary axis
- across curve
- morph between two faces

7 toolpath strategies for tool axis control:

- normal to surface
- flank
- fixed
- to rotary axis
- through the point
- through the curve
- perpendicular to the toolpath

8 toolpath strategies
for tool tip trajectory

7 toolpath strategies
for tool axis control

56 combinations
for the final toolpath strategy for any 5 axis machining task
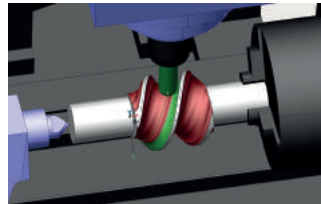
//
ROTARY MACHINING IN SPRUTCAM X

4 axis index machining

All 3-axis toolpath strategies are also available for 4 axis index machining

4 axis continuous machining

SprutCAM X has roughing toolpath for quick material removing and finishing toolpath for the required surface quality for the 4 axis continuous machining

SprutCAM X has a comprehensive set of 5 axis strategies. It is enough to solve any 5 axis machining task.

01

SprutCAM X advantages for 5 axis machining

Ultimate 5 axis machines programming workflow.

Rest material handling

SprutCAM X always keeps the workpiece updated live within the operations sequence to optimize the toolpath and avoid air-cutting.

Every subsequent operation detects the workpiece in the state it was left by previous operation. Thus the machine time is kept optimized. On the screenshot green color is the part itself and orange is the current state of the workpiece.
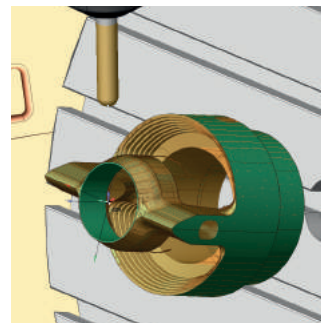
Machine kinematics consideration

SprutCAM X takes into account the machine
kinematics with all its limitations already while initial toolpath is calculated. The simulation goes in the background immediately after the toolpath is calculated. User can avoid all the collisions if happened and even does not need to switch to simulation. Thus the CNC programming workflow streamlines and takes much less time especially for 5 axis machines programming
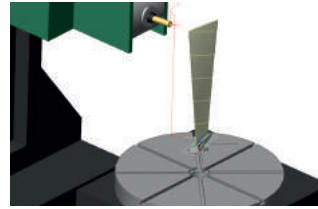
15
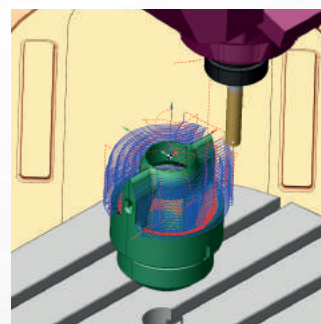
02

5 axis machining examples in SprutCAM X

Impeller wheels, blades, complex shaped ports, high walls milling with lean angle, tool flank machining.
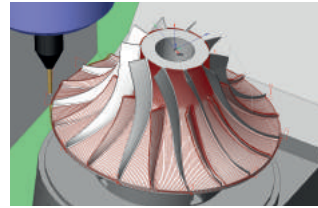
Blade machining

5 axis morphing between two curves operation, the tool axis is perpendicular to the machining surface.
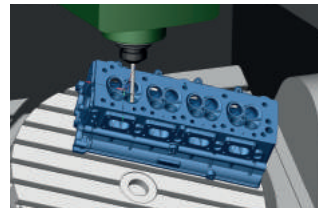
Turbine wheel machining

Toolpath strategies: rotation axial and normal to the guiding curves, with the adjustment of the lean angles strategies are used to set the 5 axis machining tool axis.
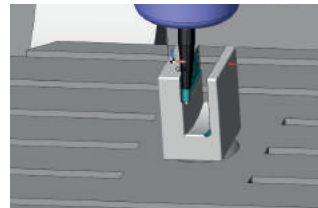
5 axis port machining in the cylinder head

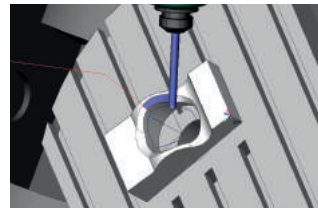Toolpath strategies: along the plane — morph, tool axis — through point.

High rising wall machining

Toolpath strategies: rotation axial and normal to the guiding curves, with the adjustment of the lean angles strategies are used to set the 5 axis machining tool axis.
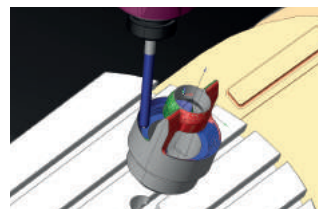
5 axis machining with tool axis control through the curve

Moving along the surface parallel to the curve, the tool axis - through the curve strategy is used.

5 axis machining with tool axis control through the curve

Moving along the surface parallel to the curve, the tool axis - through the curve strategy is used.

IN

NC for 4 (3+1)
5 (3+2) axis machining
are created in SprutCAM X.

is toolpath strategies are available
e index machining.

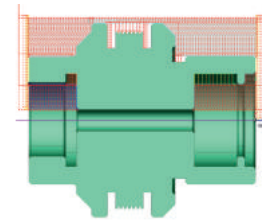NC code for 4 and 5 axis machining centers becomes easier.

Lathe machines programming

SprutCAM X for lathe machines: turning and boring of cylindrical, conical and shaped surfaces, threading, trimming, facing, drilling and reaming holes.

// LATHE FEATURES IN SPRUTCAM X

Streamlined CNC programming workflow

SprutCAM X calculates toolpath for lathe and all other machining processes in the native kinematics of the CNC machine. What does this mean to the user? This means that toolpath calculation considers collisions between moving parts of the machine between themselves and with the workpiece and exits beyond the axis limits. You do not need to postprocess the NC code to simulate the real-world result. All machine movements are visible immediately after the toolpath calculation. CNC programming becomes visual, workflow is streamlined. More than that, thanks to this feature user can save even more machine time by final tuning of the machine. For example minimize time consumption by optimizing tools location in the revolver.

All types of turning operations

Facing, ID and OD roughing, finishing, threading, grooving, face grooving, lathe hole machining, part-off.

Lathe canned cycles

According to user's desire, SprutCAM X can output G-code using lathe cycles or not.

01
Lathe programming time-savers

Toolpath preview, interactive approaches and retracts, drag&drop contour editing, threads database.
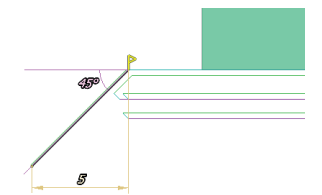
Save time with toolpath preview

SprutCAM X shows a toolpath sketch even before the toolpath calculation. So programming is getting faster.
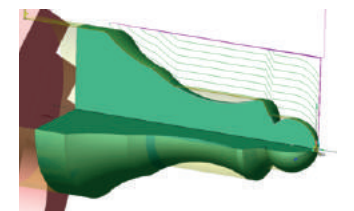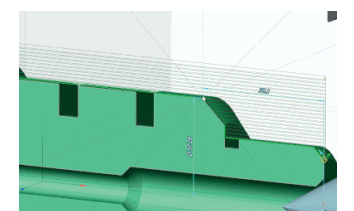
Setting the approach interactively

Useful for obtaining easy readable integers in the G-code. CNC machinists will appreciate it.

Drag&drop contour editing

Edit the machining contour directly in the interface. SprutCAM X will show the toolpath sketch with the changes.
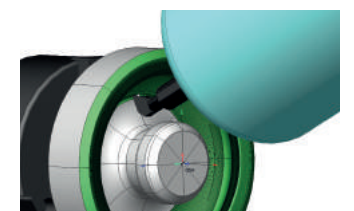
Threads database

Select the thread type and dimensions from the database. Step and depth will be filled automatically..

02
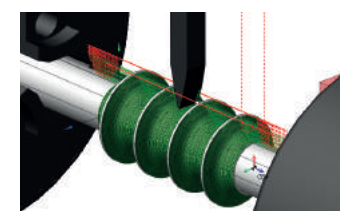## Advanced turning programming in SprutCAM X

B-axis support

You can use B-axis and machine more complex contours with one tool. Supporting the B axis reduces the number of used tools and allows to machine more sophisticated contours.

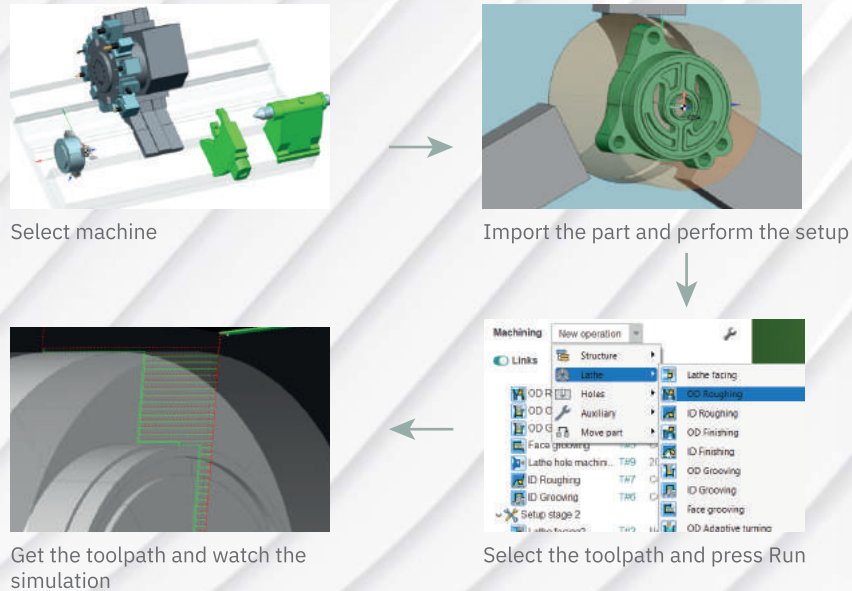### Screw parts with an arbitrary profile

Make a feed-screw with any profile or, for e.g., a buttress thread using this feature.
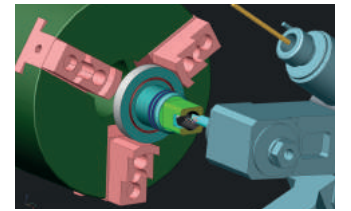
// MILL-TURN MACHINING IN SPRUTCAM X

Mill-turn programming workflow

Select machine

Import the part and perform the setup

Get the toolpath and watch the simulation

Select the toolpath and press Run

02

SprutCAM X features for mill-turn

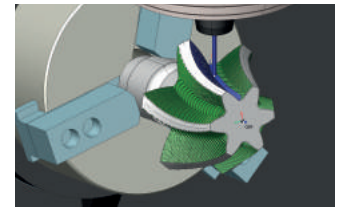**C and Y axes support**

Use C and Y for machining in the main spindle or counter-spindle, on the outer diameter and on the part face.
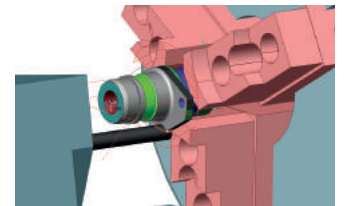

B-axis support

Index milling of the part in inclined planes and 5-axis machining using all machine axes.

Polar interpolation support

If the movement range along the Y-axis is very tight, or the Y-axis is absent, the polar interpolation option can be enabled in SprutCAM X. Then the displacement along Y will be replaced by the C-axis.


01

Turn-mill toolpath calculation in SprutCAM X

Workpiece and the machine scheme are considered while toolpath calculation in every operation

Observe the workpiece through the machining process at every moment

SprutCAM X can perform turning and milling toolpaths in any order. The workpiece for each next operation is the result of all the previous ones. You can see the current state of the workpiece at every moment. In the screenshot, green is a part, and orange the result of all previous operations by this stage. At the same time it is a workpiece for the next operation.
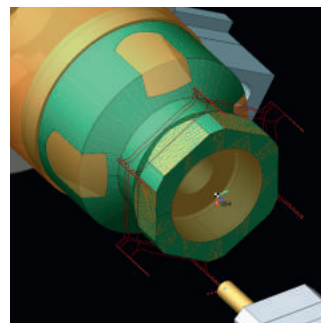
Streamline CNC programming workflow with SprutCAM

SprutCAM X calculates toolpath in the native kinematics of the CNC machine. What does this mean to the user? This means that toolpath calculation considers collisions between moving parts of the machine between themselves and with the workpiece and exits beyond the axis limits. You do not need to postprocess the NC code to simulate the real-world result. All machine movements are visible immediately after the toolpath calculation. CNC programming becomes visual, workflow is streamlined.
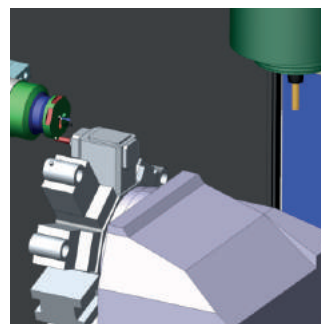
03

Auxiliary operations programming

Part takeover, long parts machining using two spindles, lunette, tailstock, bar feeders, parts catchers, workpiece loaders can be programmed in SprutCAM X.
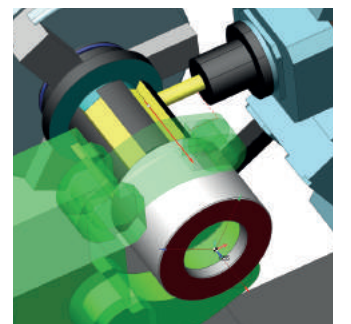
// ADVANCED MILL-TURN

01

Multichannel with synchronization (MTM)

Sync points setup

Each MTM node works with a separate controller, according to its program. Therefore, it is necessary to create NC code for each machine controller. These programs should be synchronized, so, waiting possibility is necessary.

SprutCAM X has the sync points setup feature for this purpose. Sync points can be put anywhere in the NC code.

Machine time optimization

Evaluation and load balancing on each of the control channels. Distribution of job between channels..

Toolpath calculation and simulation

Machine kinematic schemes with several nodes, several workpiece fixing places and possibility of axes distribution between several channels
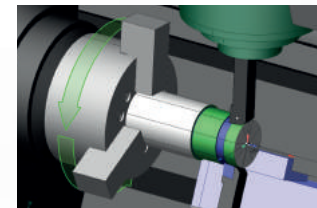
Auxiliary operations

Loading, unloading and takeover of the workpiece programming.

02

Multichannel machining examples

Spindle and counter-spindle synchronized machining with takeover
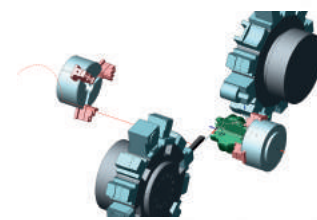
Synchronous 2-cutters rough turning

Automatic part loader programming

CNC lathe and industrial robot work together. The robot loads, turns and unloads the workpiece. Lathe works according to it's NC code. The machine and the robot are synchronized. The whole process is programmed in SprutCAM X.
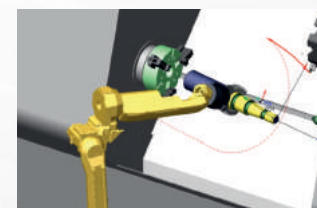
// ADVANCED MILL-TURN PROGRAMMING IN SPRUTCAM X

Swiss-type machines and MTM (multi-channel machining) with synchronization

Swiss-type toolpath calculation and machining simulation

The simulation show all complex moving parts with no exceptions and provider proper collision control to the user..

Double-spindle Swiss-type machining.

Multiaxis cutting programming
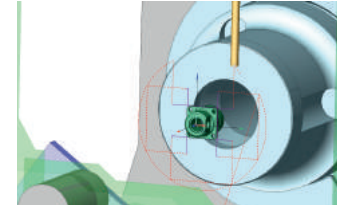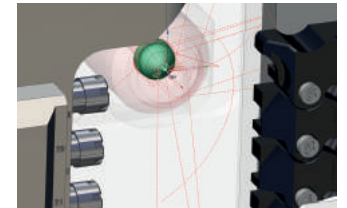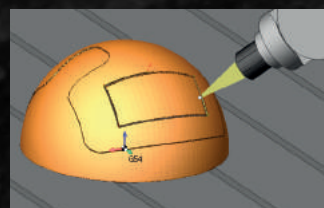## in SprutCAM X

// SPRUTCAM X SUPPORTS THESE TYPES OF 5D AND 6D CUTTING

Laser cutting

Plasma cutting

Knife cutting

Waterjet cutting

// 5D CUTTING PROGRAMMING IN SPRUTCAM X

5-axis cutting machines programming

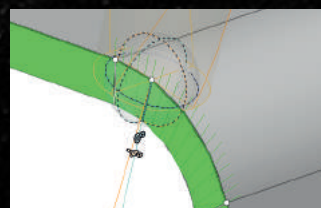### Get toolpath instantly

To get the toolpath:
1. Specify the edge on the model;
2. SprutCAM X will offer to choose one of the adjoining surfaces to control the tool axis;
3. Select the direction of the tool axis relative to the selected surface: tool tip or flank.
4. Press Run.
SprutCAM X works with both the solid model and the mesh model.

Tool axis control at each toolpath point: lead and lean angles

After calculating the toolpath, you can edit the tool vector at each point. It is possible to set the lean and lead angles at any point of the toolpath. Lean and lead angles can be set manually or automatically using the toolpath optimization map. It is useful to avoid collisions between machine and parts, and singularities.

Interactive toolpath editing after calculation

Often, the actual part geometry does not match the CAD model.

Therefore, in SprutCAM X, the toolpath is converted into a spline and can be edited directly in the interface.
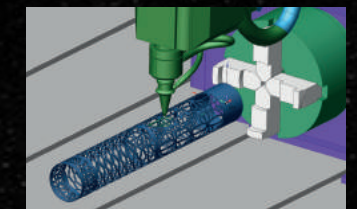

// 6D CUTTING PROGRAMMING IN SPRUTCAM X

Additional axis and knife cutting
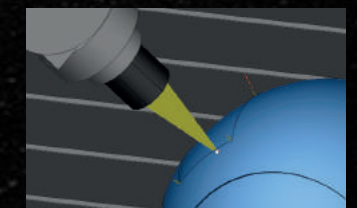
Additional axis support

SprutCAM X supports additional axes. For example, the 6th turning axis located on the machine table.

6D knife cutting

SprutCAM X has a special solution for knife cutting for the knife safety and blade direction control. What can be configured:
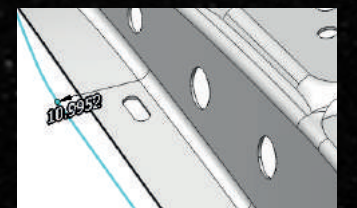
- toolpath curvature limit when knife turning in the material;
- the knife can be double-sided: you can cut by one side or by the other one; knife retraction and rotation in the air at sharp toolpath corners.

// SPRUTCAM X ADVANTAGES FOR MULTI-AXIS CUTTING

Streamlined workflow, toopath safety, NC programming at speed.

Taking into account machine kinematics

SprutCAM X takes into account the machine kinematics with all its limitations already while initial toolpath is calculated. The simulation is performed in the background immediately after the toolpath is calculated. User can avoid all the collisions if happened and even does not need to switch to simulation tab. Thus the CNC programming workflow streamlines and takes much less time especially for 5-axis machines programming.

Automatic singularity avoidance

SprutCAM X has special singularities controlling parameters which makes 5- and 6-axis machines programming much easier.
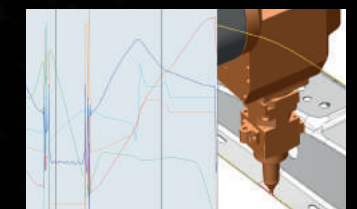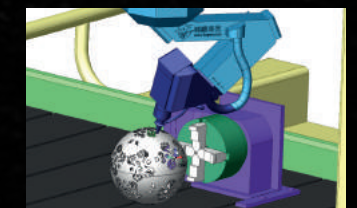
Additive and hybrid manufacturing programming
in SprutCAM X

What kinds of additive technologies are programmed in SprutCAM X

Metal cladding

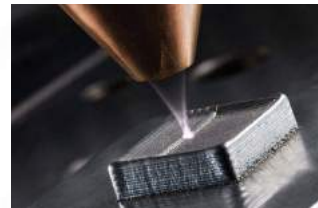**Plastic 3D printing**

Concrete 3D printing

Stem cells 3D printing

//
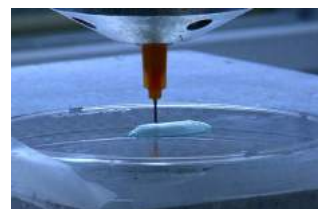
WHAT EQUIPMENT FOR ADDITIVE MANUFACTURING CAN BE PROGRAMMED

Whatever CNC machines or industrial robots, SprutCAM X can handle both.

CNC machines

3 — 5 axis and more, mill or turn-mill.

Industrial robots

6 axis industrial robots with additional axis.

//
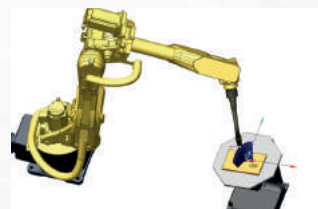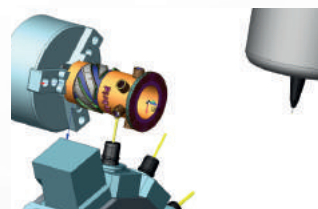
MULTIAXIS CLADDING PROGRAMMING

6 axis industrial robot with 2 axis turning table cladding. Programmed in SprutCAM X.

//

HYBRID MANUFACTURING PROGRAMMING

Cladding, mill and turn operations on one CNC machine. Whole process is programmed in SprutCAM X.

Wire EDM programming
in SprutCAM X

// 2 - 4 AXIS WIRE EDM MACHINIHG IN SPRUTCAM X

2&3 axis wire EDM

«Wire EDM 2D Contouring» operation is meant for generation of the wire path along the 2D contour (2-axis machining) as well as for the movement along the 2D contour with a certain tilt angle of the wire (conic or 3-axis machining).

Therefore, the resulting wire path is based on one contour in contrast to 4-axis machining, where the contours described by the wire in the lower and the upper planes are to be specified separately.
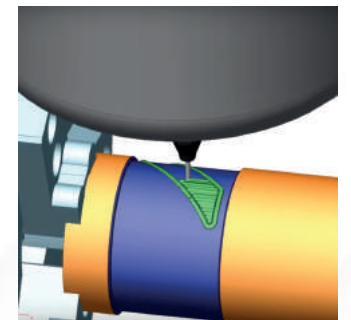
4 axis wire EDM

«4D electrical discharge machining of the contour» operation is intended for generation of the wire path along two 2D contours simultaneously. One of these contours defines the movement of the lower sliding guide, and the second contour specifies the movement of the upper sliding guide.

Therefore, in this operation in contrast to «2D machining», the upper and lower ends of the wire can describe an absolutely different path.

// INTERACTIVE INTERFACE FOR EDM PROGRAMMING

SprutCAM X allows generating different types of cuts: rough cuts, finish cuts, link cutting passes, cleanup cuts after the link cutting, etc.

You can also:

- set up machining sequence;
- optimize cut directions;
- specify the wire radius compensation technique;
- allow the output of process shutdown commands into the control program

Approach-withdrawal indicators are interactive, they can be moved in the graphic window and you can define precise dimensions relative to other elements as well as to the origin of coordinates.

Links for EDM programming

SprutCAM X provides a defined set of parameters, allowing to leave special unfinished areas, termed links, on the work piece. When the wire reaches such areas, the system will generate process shutdowns, after which some supplementary measures must be taken to fix certain parts of the work piece.

Then unfinished links can be automatically cut. You can define the parameters, determining the number of passes for the link cutting, number of cuts for cleaning the contour after the link cutting, and also parameters, determining the execution sequence of these cuts.

Cutting speeds

Modern CNC controllers for EDM allow loading of the table of predefined cutting speeds for each cut of the wire along the contour into the column. In the process of machining one of the table works is selected by specifying a special code of cutting speeds.

SprutCAM X support such representation of cutting conditions in its erosion operations. Depending on the thickness and the material of the work piece, the material and the diameter of the wire, the required surface smoothness of the part it is possible to select cutting speed, wire offset value (allowance, individual for each cut and taking into account the wire radius, burnouts, depending on the power of the electric spark, etc.) and other parameters separately for each cut.

The tables of machining conditions can be either selected from special cutting speeds libraries or filled out in each operation all over again.

2D cutting programming
in SprutCAM X

// WHAT TYPES OF 2D CUTTING ARE PROGRAMMED IN SPRUTCAM X?

Laser cutting

Plasma cutting

Waterjet cutting

Knife cutting

// AUTOMATION

Crosspieces function

In order for the cut out pieces not to fall out from the sheet, you can leave several crosspieces on their contours. After the cutting, the cross pieces are cut and the parts are taken out from the sheet. SprutCAM has the following modes for detecting crosspieces:

- machining without crosspieces;
- a definite number of crosspieces is set, hence we get an even number of crosspieces for all the curves;
- number of crosspieces for each curve is variable and depends on its length;
- number of crosspieces for each curve is variable and depends on its surface area.

Automatic schemes for angle passing

In order to avoid the bad angle cutting of the part, the system has several methods of forming of tool path in the angles. The passing of the angles can be set automatically or manually, depending on the angle.

Contour cutting sequence optimization according to the lengths of the passes, taking into consideration the inner contours

The contour cutting sequence is determined automatically, taking into consideration the inner contours. This has to be done in order to avoid the untimely cutting out of a part with uncut parts inside.
Besides that, the system provides the optimization of the machining sequence taking into account the length of the passes between parts.

Automatic detection of the optimal approach points to the workpiece contour with a simultaneous control of the approaching and withdrawal of the cut of the neighbouring parts

The system has an automatic creation of the positions for the tool penetration, depending on the approach chosen, avoiding a cutting of the neighbouring parts.

Avoiding passes over already machined zones

Often, the machined piece will bend because of the thermo treatment and it can break when the cutter head is moved above it. In order to avoid such situations, the passes between cutting in points can be done above the unmachined zones of the sheet. When the function is turned off, the passes will be made following the shortest distance between the points. When the function is turned on, the passes are made in such a way, that the cutter head avoided passing above the already cut out contours.

3D cutting simulation

The NC code is checked in the machining simulation mode. The cutting simulation assures visual control of the quality of the NC code.
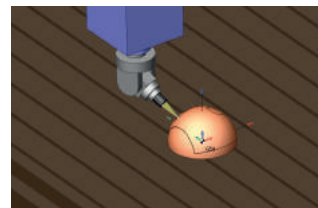
Routers programming
in SprutCAM X

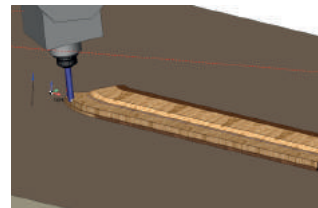## // WHICH TYPES OF MACHINING FOR ROUTERS ARE PROGAMMED IN SPRUTCAM X

Cutting
From simple 2D to complex multiaxis cutting. Plasma, laser, water-jet, mill and knife are supported.

Milling 2D - 5D
All the power of professional algorithms which are used for industrial machining centers programming is available for routers.
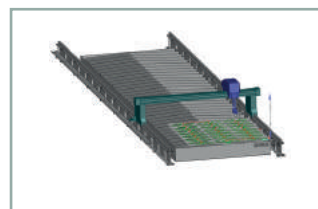
Engraving
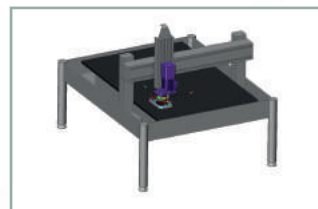Engraving on flat and curved surfaces.
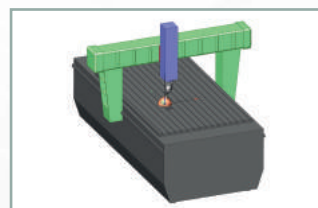

// EQUIPMENT SUPPORT

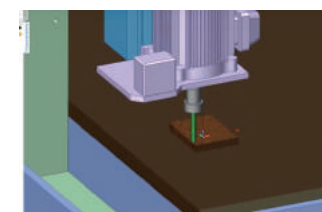**2 axis cutting machines**

3- and 4-axis milling machines

5-axis machines

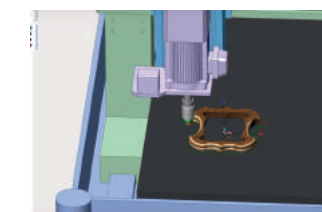# Programming of wood machining
in SprutCAM X

// Embossed and relief surface milling
Machinig simulation example

// Shaped cutter milling
Simulation example in SprutCAM X

// Making of balusters
Simulation example in SprutCAM X with 4-axis machine

// Plywood/chipboard/fiberboard cutting
Simulation example in SprutCAM X

// 5-axis wood machining
Multiaxis machinig simulation example

SprutCAM X

SprutCAM Tech — CAD / CAM / CAE developer

The SprutCAM Tech team has been working on CAM software development since 1987. During this long period we have gained extensive experience and expertise in computer-aided manufacturing. Our team consists of experts and practitioners in the technologies required to power the tools to make CNC programming easier, faster, and more effective for our customers. Algorithms of toolpath strategies and reliable simulation are developed by the SprutCAM Tech team to create the foundation for our software.

SprutCAM Tech reinforces its commitment to technical excellence by dedicating nearly 25 percent of its annual revenue to ongoing research and product development. This long-term focus has produced powerful technological innovations that have placed SprutCAM in an industry-leading position since its market launch in 1997.

Sales and support operations are located in Asia, Africa, North and South America, Europe, and Australia.

Today we have thousands of implementations of our products all over the world in the USA, Canada, Brazil, UK, Netherlands, Italy, Germany, Poland, Spain, China, and many more.

SprutCAM Tech Ltd.
9, Aiolou and Panagioti
Diomidous
3020 Limassol
Cyprus

email: info@sprutcamtech.io
#sprutcam