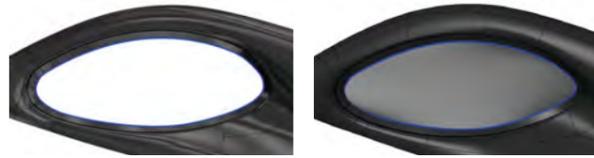


DESIGN

Power Surfaces

With the new power-surfaces function, surfaces can easily be created from curves and adjacent surfaces.



Layer Structures

Mastercam 2019 allows you to import and export layer names and structures. Some advantages are: Less input, saving time and similar file structures.

Drill construction

Solid construction now includes a feature for parametric drill hole construction. Drill forms are associative and can be edited via the legend anytime.



More Possibilities



Mastercam Port Expert

Creates toolpaths on solid or surface files for the precise manufacture of ports in cylinder heads for engines, while minimizing unnecessary tool movement.



Mastercam Blade Expert

Highly efficient tools for impellers and similar parts to create high quality surface finishes of blades, splitters, faces and fillets.



Mastercam ProDrill

Automatic recognition, grouping and application of drill strategies to a multitude of drill holes, and automatic creation of tool definitions from existing geometry.



5-axis drill and 5-axis swarf

Add-ons, that enhance Mastercam programming with purposive and efficient multi-axis features. Can easily be upgraded to the full multi-axis product for complete 4- and 5-axis manufacturing.



Mastercam Productivity+™

Adds process-internal Renishaw features for measuring and verification to Mastercam. Process-internal adjustments, where applicable, are easy to set up, verify and apply.



Data interfaces

Includes the following: IGES, Parasolid®, SAT (ACIS solids), AutoCAD® (DXF, DWG), Autodesk® Inventor® (IPT, IAM), SOLIDWORKS® (SLDPRT, SLDASM; SLDDRW), Solid Edge®, STEP, EPS, CADL, STL, VDA and ASCII. Optional direct interfaces: CATIA® V4, V5, V6, Siemens NX, JT, ProE/Creo (incl. tree style structure with model information like form and place tolerances)



Mastercam Art

Bring your 2D sketches, images, pictures and CAD files to life by editing them on the screen and producing them with simple, specialised tools.

Mastercam® 2019

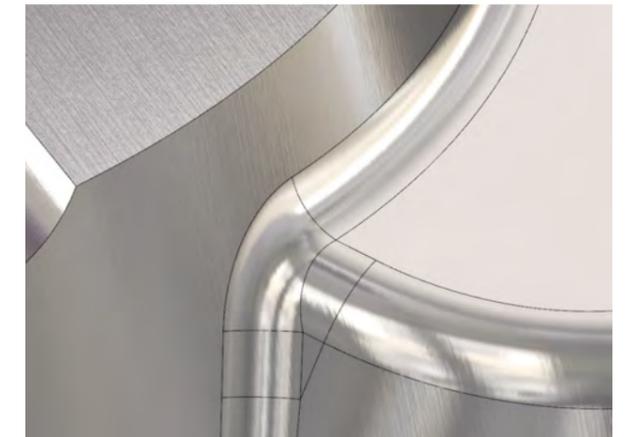
SHAPING THE FUTURE OF MANUFACTURING

The newest version of our CAD/CAM software offers many new features, advancements and groundbreaking innovations.

Since different aspects are important for individual users, depending on their field of application, this summary merely presents a section of all the novelties. Detailed and relevant information are, as usual, readily available at you Mastercam reseller. Just ask for the current "What's New"!

Display

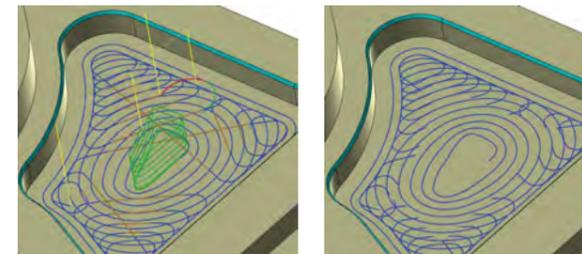
Within shading, several new textures have been added to material mode. You decide the sheen and surface quality for the perfect impression.



NEW FEATURES

Extended toolpath display

Individual sections can be modified in color and line thickness. Entry and exit, as well as rapid and micro lift movements can now be toggled and viewed separately.

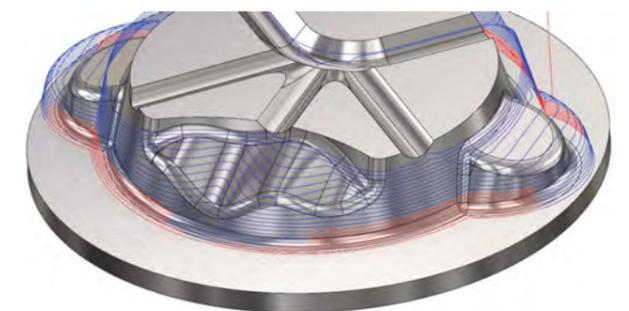


MILL

Check holder

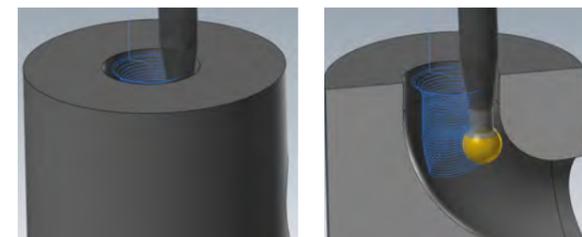
The check holder function has been reworked and integrated completely into the Mastercam interface.

This function allows for quick and reliable checking of tool stickout lengths, which can be adjusted automatically. The collision lines can be highlighted (here in red).



Cross-section view

With just one click, any view can become a cross-section view. A very helpful function for part analysis or viewing toolpaths within parts, which are hard to see.



Mastercam
35 YEARS

Your Mastercam Reseller:

Producer:

CNC Software, Inc.
671 Old Post Road
Tolland, CT 06084 - USA
Tel.: +1 860 875 5006
<http://www.mastercam.com>
mcinfo@mastercam.com

German Distributor:

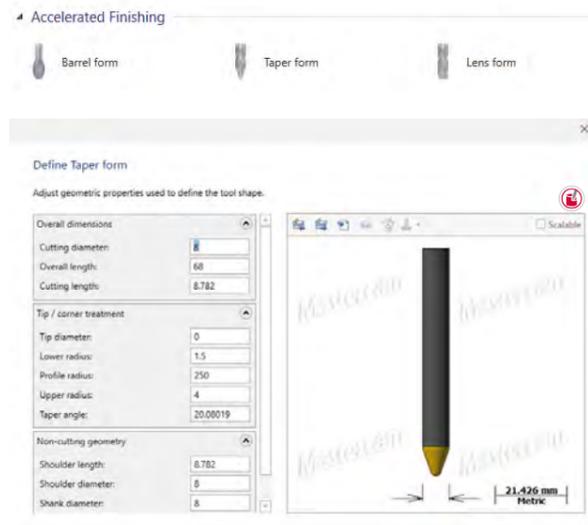
InterCAM-Deutschland GmbH
Am Vorderflöß 24a
33175 Bad Lippspringe
Tel.: (05252) 98 999 0
<http://www.mastercam.de>
info@mastercam.de

Mastercam® is a registered trademark of CNC Software, Inc. © 1983-2018. All rights reserved. All other registered trademarks are property of the respective copyright holders. © 2018 InterCAM-Deutschland

MILL

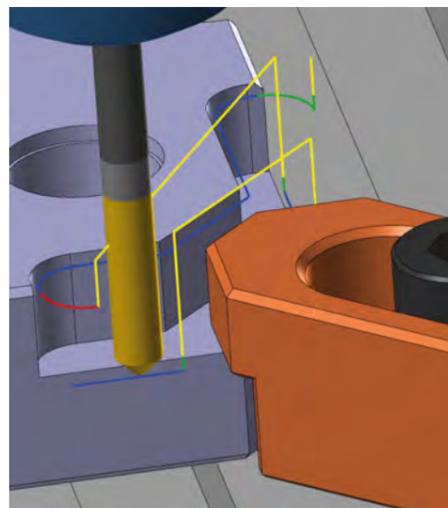
New circle segment tools

Following the success of Mastercam's "Accelerated Finishing" technology, the definition of these tools has been further optimized. The tool types were categorized as Barrel Form, Taper Form and Lens Form. Existing libraries from Emuge, Fraisa, Hoffmann and Voha are readily available for all users.



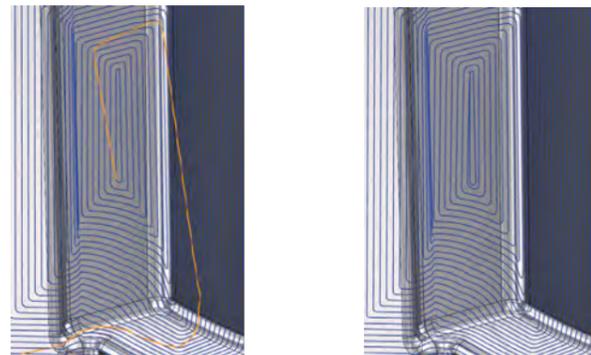
New 2D toolpath: Model Chamfer

By choosing any body edges, chamfers are created automatically while considering clearance. Entry and exit moves, as well as contour cuts and extensions are determined by the system. Contours of fixtures are recognized and protected from collision. A complex, prismatic part can be chamfered completely, with just a few clicks.



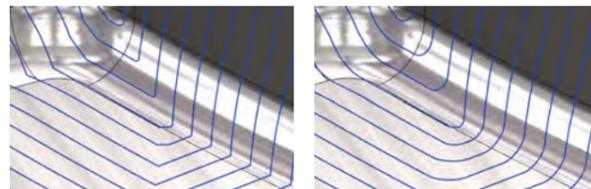
New 3D toolpath: Equal Scallop

The new spiral structure of 3D contours eliminates the need for stepovers in between offset paths (highlighted in orange). The machine executes a fluid movement, resulting in higher optical surface finish quality.



New 3D toolpath: 3D Contour smoothing

The new parameter "smoothing" filters out sudden changes in direction. Reversing points are reduced and the machine can more easily keep the feed rate.



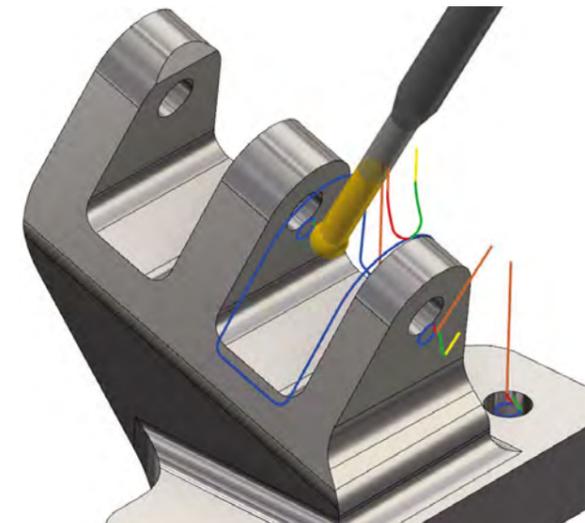
Multithreading (multi-core CPU support)

The classic 5-axis strategies like contour, swarf, flowline and multi-surface now support multithreading. Testing showed time savings of up to 90% (24 sec. instead of 240 sec). CPU load increased by 20%. A big advantage of multithreading is that during calculations, work within the system can continue.



New toolpath: Deburr

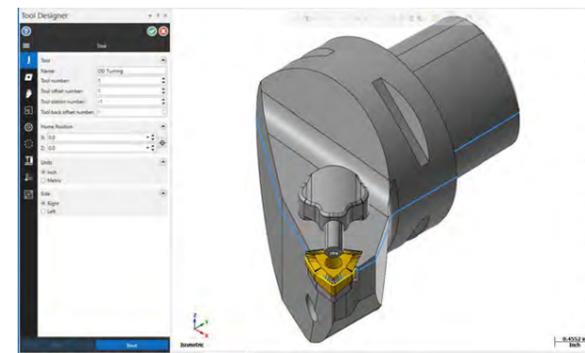
Mastercam multi-axis offers a new toolpath for breaking edges in 3D- and up to 5-axis manufacturing. This new strategy supports ball end and lollipop mills. Advantage: breaking edges within complex parts as well as machining geometry with undercuts.



TURNING

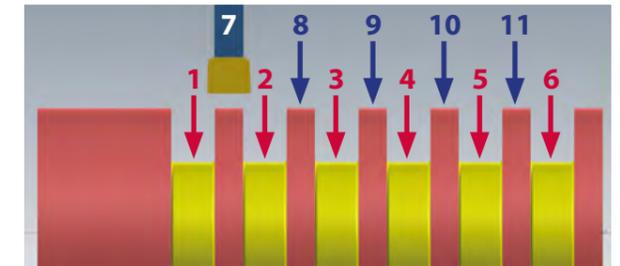
Support of 3D tools

Mastercam 2019 allows for the integration of turning toolpaths via 3D-STEP data within Mastercam. The new Tool Designer systematically leads you through the definition of the turning tools, simplifying the creation of new tools.



Multiple Plunge

The Multiple Plunge option is now available within Groove toolpaths. It minimizes tool deflection during the cut and offers full control over the machining process. This option is suitable for deep and wide grooves. While using the XX option, machining is done with 30% to 50% higher feed rates.



Prime Turning™

The PrimeTurning™ toolpath, an add-on to Mastercam 2018, has been fully integrated into Mastercam 2019.

As opposed to conventional turning, PrimeTurning™ allows for machining in all directions. This leads to a significant increase in efficiency and productivity.

New machining strategies for the biggest innovation in turning!

