

SOLIDWORKS CAM

OBJECTIVE

SOLIDWORKS® CAM is a 2.5-axis milling and turning solution powered by CAMWorks®. SOLIDWORKS CAM offers 3+2 machining as well as full support for configurations, parts, and assembly machining workflows. The foundation of SOLIDWORKS CAM is rules-based machining, where you can teach the system what standard machining strategies are important to them. These rules can then be automatically applied based on material type and geometry of the feature. Leveraging the SOLIDWORKS part and assembly interface allows you to easily and quickly learn SOLIDWORKS CAM and take advantage of rules-based machining with minimal effort.

SOLIDWORKS CAM delivers an easy-to-use and fully functional programming experience based on 19 + years of development as a Gold Level Solution Partner. With the programming data stored inside the SOLIDWORKS part or assembly, you can easily manage files and references just like they do today. Leveraging the SOLIDWORKS part and assembly interface enables you to easily and quickly learn SOLIDWORKS CAM and take advantage of rules-based machining with minimal effort.

OVERVIEW

SOLIDWORKS CAM uses rules-based machining to enhance the programming process in the same way rules-based design helped to speed up the drawing and design process. Rules-based machining leverages tolerances that have been assigned to each component, allowing users to focus on the critical areas of making a part rather than touching every feature that needs to be machined.

With SOLIDWORKS CAM, you will be able to enhance your manufacturing process by capturing company standards, speed up quoting, and test manufacturability of a design earlier in the process. By using rules-based machining, your business can automatically apply standard strategies to determine how long it will take to make a part and if it could easily be machined. This automation will allow you to make decisions faster and with more confidence.

SOLIDWORKS CAM combines design and manufacturing in one application with an easy-to-use interface. The result is an intuitive rules-based system that can be leveraged to save time and money while capturing company standards. Assigning machining strategies based on design tolerances, reduces errors and improve quality throughout the machining process.

BENEFITS

- Simplifies collaboration—single design and programming environment allows easier transition to CAM.
- Utilizes concurrent development process allowing companies to perform tasks sooner and find problems earlier, making changes less costly.
- Rules-based machining allows new users to adapt to a company's machining process readily.
- Tolerance-Based Machining allows for the best machining strategy and for quickly making adjustments as designs, materials, and tolerances change.
- Feature Recognition gives you to full control to define machinable features within the CAD/CAM environment.
- High-Speed Machining creates toolpaths that lead to shorter cycle times while extending tool life and lowering machine wear.
- The NC Editor makes verification of G-code simple quick. Users can also backplot the G-code for review and send the file directly to the CNC control using the DNC capabilities.
- Communication between programming and setup is easy with toolpaths output inside of eDrawings®. Operators can view the 3D model with the associated toolpaths to understand the machining order.
- Fixtures and tooling are easily visualized using SOLIDWORKS assemblies. Once the fixtures are designed, SOLIDWORKS CAM can automatically adjust toolpaths to avoid collisions with the designed components.
- Toolpath Simulation within SOLIDWORKS CAM allows you to verify the correct machining strategies and setup information on each component produced.

CAPABILITIES

SOLIDWORKS CAM Standard

SOLIDWORKS CAM Standard allows users to quickly program individual parts and configurations without leaving the SOLIDWORKS 3D CAD environment. You have full access to defining rules within SOLIDWORKS CAM to create and build to company standards. The use of knowledge-based machining and tolerance-based machining allows for:

- Recognizing any updated geometry as the model changes through feature updates or newly imported parts.
- Assigning machining strategies based on features that are recognized.
- Updating machining strategies if a design's tolerance changes.
- Rules-based machining enables designers and engineers to:
 - Catch design errors and new part setups through Automatic Feature Recognition.
 - Quote components quickly using company standards captured as rules.

SOLIDWORKS CAM Professional

SOLIDWORKS CAM Professional builds on the capabilities of SOLIDWORKS CAM Standard to increase programming capabilities. SOLIDWORKS CAM Professional adds the following functionality:

- Assembly Machining: Users who are looking to design fixturing or machine a group of parts can create the tables, vises, clamps, or any other holding mechanisms using a SOLIDWORKS assembly. Once the design is complete, programmers can define what components are used for machining and which ones are fixtures. SOLIDWORKS CAM will automatically make adjustments to the toolpaths to avoid the fixtures. This level of automation allows the programmer to focus on the entire machining process quickly.

- Turning: SOLIDWORKS CAM will support single turret turning inside the SOLIDWORKS part environment. Similar to part milling, users can take advantage of Automatic Feature Recognition, Knowledge-Based Machining, and configurations. A predefined library of tools and machining strategies are loaded into the Technology Database. These can be customized at any time to enhance the programming process. Users have the ability to create custom tools and holders for specific machining operations. Simultaneous mill/turn or live tooling functionality is not available with SOLIDWORKS CAM.
- 3+2 Milling: Programmers can take advantage of 4- and 5-axis machining centers with SOLIDWORKS CAM Professional. These machines can have the fourth and fifth axis' pre-positioned into place before running 2.5-axis milling strategies. This allows users to create multiple fixtures and work holding to reduce setup time. This type of programming allows companies to produce large production runs with minimal operator interaction.
- High-Speed Machining: With the advancements in machine tools and tooling it is necessary to optimize machining toolpaths to get the most out of your equipment investment. SOLIDWORKS CAM Professional utilizes VoluMill 2.5-axis milling routines from Celeritive™ Technologies. The advantages of using these machining strategies are
 - Up to 75% savings on cutting tool costs.
 - 50% to 80% reductions in machining cycle times.
 - Easy-to-learn machining strategies
 - No special machine tools are required
 - Reduced wear and tear on your equipment due to smoother transitions in toolpaths



Learn more about SOLIDWORKS solutions at FISHERUNITECH.com or contact us at marketing@fisherunitech.com or 248-251-0399 to get a demo or quote.

Our 3DEXPERIENCE® platform powers our brand applications, serving 12 industries, and provides a rich portfolio of industry solution experiences.

Dassault Systèmes, the 3DEXPERIENCE® Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 220,000 customers of all sizes in all industries in more than 140 countries. For more information, visit www.3ds.com.

