

BobCAD-CAM V24 Release Notes

BobCAD-CAM V24 Build 546

Release Date: March 2, 2012

Corrections:

CAM

- Instances of Z-Level Rough toolpath not cutting a BobART model have been fixed
- Errors in the Weave texture in BobART have been corrected
- Corrections have been made to the post scripting engine to handle memory issues
- Advanced Posting page variables not getting initialized in certain situations has been fixed
- Spindle speeds getting reconverted with metric units has been fixed
- Corrections have been made to loading machining features from previous versions
- Instances of Bottom of Job value in the 3D toolpaths not functioning properly has been fixed
- Angle of Approach for High Speed Pocketing feature not working has been corrected
- Drill depths inside Reamer cycle have been corrected
- Instances of the finish tool not going to clearance inside a Pocket feature has been corrected
- Problems with large tool numbers have been corrected
- Corrections have been made to cleanup cutting in the V-Carve toolpath
- Instance of Trim Modal not working properly has been corrected
- Default peck depth not being displayed properly has been corrected
- Depths not transferring from geometry page in some drilling features has been corrected
- Minor corrections to the 2 Rail Sweep feature have been made
- Errors being produced from computing a toolpath while still in selection mode of the Profile Start Point have been corrected
- Extra feed move within the lathe Threading cycle has been removed
- Instances of lathe Roughing toolpath displaying diagonal moves have been corrected
- Tap selection has been moved to Geometry page to aid in quicker editing of Tap features in Mill

BobCAD-CAM V24 Build 376

Release Date: September 12, 2011

The new BobART Pro adds enhanced capabilities to make designing artistic models easier and faster.

BobART Pro Features:

- **2-Rail Sweep Feature**
Create embossed models by sweeping one or more cross sections along two rails.
- **Sculpting Feature**

Modify an embossed model in three different ways. You can deposit, remove, or smooth the material of the model using the mouse (brush).

- **Smoothing Feature**
Smooth the material for an entire embossed model, without the need for a mouse (brush). You can smooth the whole model, or you can select geometry as a boundary to contain the smoothing to that area.
- **Material/Texture Selection**
You can now add a material or texture appearance to an embossed model, which can be merged with a color. Bitmap images (.bmp) can be added to the BobCAD-CAM Data\BobCAD-CAM V24\Images folder for additional material textures.
- **Load Vector Images**
Vector files can be opened from the BobART Manager tree item. This opens files using .ai, .eps, and .pdf file extensions.

Corrections:

General

- Tab order in Stock Edit dialog has been corrected

CAM

- Counter Bore diameter and depth options have been added to the geometry selection page, allowing the user to define these values before generating the machining features.
- Thread Type field was added to the geometry selection page for Tap features, allowing the user to define their tap sizes before generating the machining features.
- Slice Planar and Slice Radial Bottom of Job errors when using boundary geometry has been corrected.
- Lathe System Compensation has been corrected to give more accurate coordinate output when no gouges occur.
- Errors with Part in Part shapes being overlaid in nesting, in certain scenarios has been corrected.
- Slice Radial Cut Interval corrections have been made.

BobCAD-CAM V24 Build 208

Release Date: March 28, 2011

Corrections:

General

- Added the Visual Studio 2008 Redistributable SP1 to the installer for use with SolidWorks 2011 file reading.
- Corrected the software freezing with a white screen during startup on some systems.
- General fixes to the SolidWorks file reader to correct reported software crashes.

- Delete, Copy, and Paste icons have been updated to resolve the buttons from always appearing enabled.
- What's New section has been added to the BobCAD-CAM V24 Help System. This allows users to track what has changed between each software update.

CAM

- Arcs that share the same center point will be filtered out now during geometry selection for Hole making features.
- Tap feed per revolution changing during manual edits has been corrected.
- Drill depths not updating when a manual tool was selected has been corrected.
- Corner types for the Profile toolpath using incorrect values has been corrected.
- Reported issues of tools being omitted from the posted program have been corrected.
- Contour ramping will now work inside of the Nesting feature.
- Manual RPM settings being ignored has been corrected.
- Changed the *Feed per Tooth* text label to *Feed per Revolution* inside of all drilling features.
- Corrected all reported dialog issues with the *Laser/Plasma/Waterjet* tool pages.
- Feed rates not being output in *Laser/Plasma/Waterjet* code has been corrected.
- V-Carve dialog refresh issue when using the *Use Endmill* option has been corrected.
- Chamfer Mill icon inside of the Machining Wizards has been updated.

BobCAD-CAM V24 Build 169

Release Date: February 18, 2011

In this release, a new Registration Wizard has been added to the software to allow for an easier registration experience for new users. Now when the software is launched, if the user has not registered the software, they will be presented with a dialog informing them of their license status as well as provide them with options for registering their purchased products.

Registration Wizard Features:

- Trial Mode/Demo Mode dialog has been added to display when the software is initially opened without a license.
- Enhanced Online Activation dialog to improve user experience.
- New License Status dialog to allow for easier understanding of licensed modules.

Corrections:

General

- General fixes to the SolidWorks file reader to correct reported software crashes.
- Delete, Copy, and Paste icons have been updated to resolve the buttons from always appearing enabled.

- What's New section has been added to the BobCAD-CAM V24 Help System. This allows users to track what has changed between each software update.

CAM

- Reported issues of tools being omitted from the posted program have been corrected.
- Contour ramping will now work inside of the Nesting feature.
- Manual RPM settings being ignored has been corrected.
- Changed the *Feed per Tooth* text label to *Feed per Revolution* inside of all drilling features.
- Corrected all reported dialog issues with the *Laser/Plasma/Waterjet* tool pages.
- Feed rates not being output in *Laser/Plasma/Waterjet* code has been corrected.
- V-Carve dialog refresh issue when using the *Use Endmill* option has been corrected.
- Chamfer Mill icon inside of the Machining Wizards has been updated.

BobCAD-CAM V24 Build 133

Release Date: January 13, 2011

Corrections:

- Fixed 3-axis engrave output when working with 3D arc segments.
- Fixed the incorrect Chamfer mill icon being displayed within the Chamfer feature and Pocketing feature.
- Fixed the lathe roughing tool path not reaching the end of the selected chain in certain situations.
- Fixed the Machine Sequence page not being displayed when in Laser/Plasma/Waterjet mode for the profile feature.
- Fixed the chamfer depth not being retained inside of the pocket feature
- Now a tool path with 0 depth is allowed to be calculated. Previously entering a depth of 0.000 would result in no tool path.
- Added the Machine Sequence to the first page of the edit dialog for all drilling operations
- Altered the lead out in contour ramping to lead out at the bottom of the chain.
- Cross hatching can now be applied on a user defined UCS
- Fixed instances of contour ramping not cutting the entire chain when an open chain is selected.
- Fixed the Adv Posting page not being displayed for 3 axis features.
- Fixed the ramping options being improperly displayed in the finish pass of the pocket feature.
- Fixed the counterbore parameters page from displaying incorrectly when loading a previously saved file.
- Fixed improperly displayed feeds and speeds fields in several drilling operations.
- Fixed the crash that resulted in clicking on the Tap tool page in a tap feature.
- Fixed the No Post X on the top level of the feature from not being displayed in previously created part files.
- Fixed radio buttons on Profile parameters page from not storing previously selected value.
- Fixed issues with the Thread parameters page not storing their data.
- Fixed the geometry selection for the Thread and Facing cycle to use the first selected point.

- Added the Hole Pattern to the entity types that can be selected for a drilling feature.
- Fixed the Top of Job and Bottom of Job settings not being displayed in the Equidistant Offset feature.

BobCAD-CAM V24 Pro Release 112

Release Date: December 22, 2010

For the first update for BobCAD-CAM V24 we have added 3 new machining strategies to our Mill Pro module. Enhancements and functionality has also been added to the Profile toolpath for all Mill users.

Mill Pro Features:

- **Advanced Roughing**
 - Optimized toolpath for open areas and core roughing
 - Rest Roughing strategy for optimized removal of material in areas larger tools could not enter.
 - Intermediate slices can be made to remove large “stair step” effect to decrease amount of material that needs removed by semi-finishing and finishing operations.
- **Flatlands Machining**
 - This toolpath automatically locates all flat areas on the model and creates an optimized toolpath for semi-finishing or finishing of the flat floors.
- **Equidistant Offset**
 - This toolpath strategy creates a constant cusp along the model, regardless of surface complexity.
- **Pencil Milling**
 - This strategy can be used to trace around internal corners and fillets to improve surface finish.

General CAM Enhancements:

- **Profile Feature Enhancements**
 - Ramp option has been added to the Lead types for the Profile feature. This allows the users to perform a zig zag ramp at a specified length and angle into the material on profile cuts. The ramp will be applied along the profile toolpath contour at the selected start location.
 - Contour Ramp pattern has also been added allowing the user to generate a continuous ramping motion around the perimeter of the part. The user can enter an angle of decent, or a depth per pass to determine the number of rotations around the part.
 - Side Roughing has been enhanced to work with open shapes allowing better roughing strategies for the Profile Feature.
 - Reverse Direction item has been added to allow the users to easily reverse all chain directions for the geometry associated to a Profile Feature.

- **Tool Library**
 - Ability to sort by specific columns have been added in to the tool library allowing for easier searching.

What's New in BobCAD-CAM V24 Build 82

Release Date: November 24, 2010

The focus for BobCAD-CAM V24 was based completely off of user feedback. The toolpath creation was redesigned giving the user an even easier machining process designed to walk the user step by step through setting up each machining feature, while giving the user more control over each element of each machining operation. The user interface has been improved by grouping commonly used tools together, making it easier and faster to get to the most commonly used functions in the software.

Hundreds of new descriptive images have been added to the software giving the user more confidence over controlling each aspect of the BobCAD-CAM toolpath strategies.

Features:

- **Machining Wizards**
 - Completely redesigned machining process setup focused on giving the user more control over all aspects of BobCAD-CAM's multi-operation machining features, while still maintaining BobCAD's automation of the entire machining process. The Drilling, 2 Axis, and 3 Axis Machining Wizards make cutting parts easier and faster, while giving more control to the user.
- **Drilling Wizard**
 - Automatically creates separate drilling features for each individual hole size.
 - Automatically assigns appropriate drilling tools for each operation inside the drilling features.
 - Allows for editing of several machining features at one time.
- **CAM Enhancements**
 - Independent control over system compensation and machine compensation for all roughing and finishing profiling toolpaths.
 - On the fly start point and direction editing for Profile, Chamfer, and Engraving machining features, eliminating the need to create extra geometry.
 - Redesigned Tool Library to make creating and editing tools faster and easier.
 - Redesigned tool numbering scheme allowing the user to view and edit all tools from within any machining feature.
 - Improved manual feed and speed calculator allowing users to perform SFM calculations on the fly.

- Independent flood, air, mist, and oil coolant control for each tool.
 - Posting blocks:
 - 706. Coolant Mist code? "M07"
 - 707. Coolant Air code? "M07"
 - 708. Coolant Oil code? "M07"
 - Diameter and Height Offset register editing on the fly.
 - Editing of either effective depth or overall depth of drilling tools has been added.
- **CAD Enhancements**
 - Quick Trim tool to simplify the geometry cleanup process. While in this function, all geometry will be broken at the intersections allowing users to simply click on entities they wish to remove. This tool essentially eliminates the need to use the Break Many function followed by the Cleanup and Optimize functions to simply delete portions of geometry.
 - Users can open *.SLDPRT files from SolidWorks 2011
- **User Interface Enhancements**
 - Redesigned toolbars that make accessing the most commonly used functions faster and easier, while taking up less valuable screen area. Toolbar buttons have been grouped in a more logical fashion reducing the guess work and increasing speed and efficiency in the programming process.
 - Every icon has been redesigned, making finding needed functions easier than ever, while keeping consistency for current users.
 - New default keyboard shortcuts further improving the speed of designing and machining parts.
 - Post Yes/No icon has been moved to top level of the machining feature making it easier to locate disabled machining features.
 - Blank and Unblank for all toolpaths has been added to making moving between machining and design even easier.
 - Post Yes/No for all features makes modifying your machining process even easier.