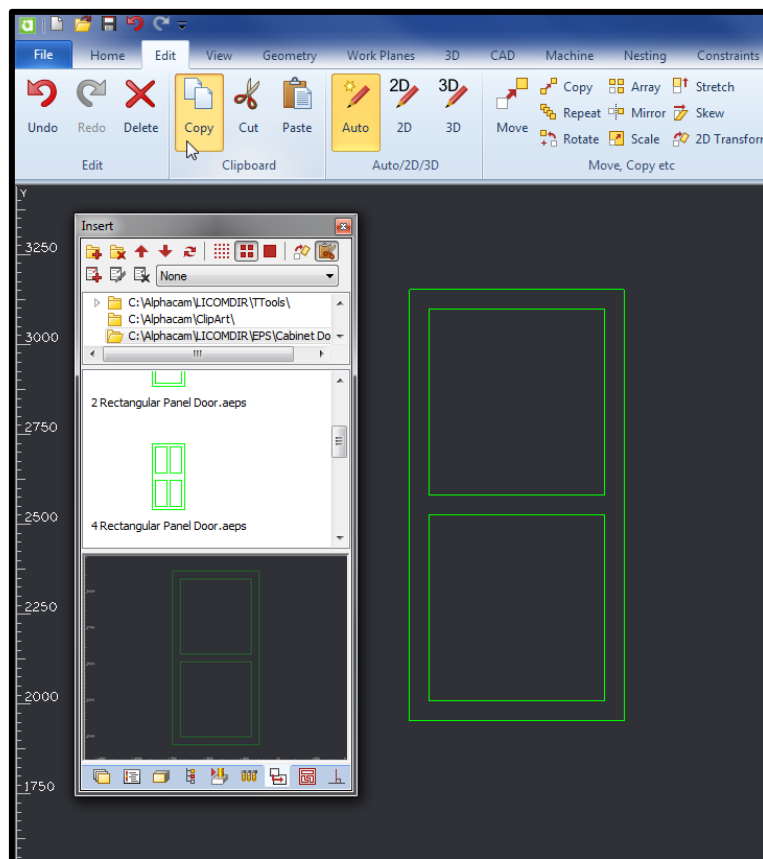


alphacam

2016 R2

新機能紹介



Alphacam 2016 R2 の新しい機能を紹介します

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重要事項

対応 OS

Alphacam 2016 R2 が対応しているOSを以下に示します。

Windows 7 - Professional, Enterprise, Ultimate

Windows 8 - Professional, Enterprise

Windows 10 - Professional, Enterprise

詳細な情報はインストールマニュアルまたは <http://www.alphacam.com> をご覧ください。

ライセンス要件

Alphacam 2016 R2 をご使用になるには、保守終了日が **2016 年 3 月** 以降の必要があります。

保守契約が終了すると以下の機能制限が発生します。

レポートの新規作成、編集ができなくなります。既存のレポートを起動することはできます。
パラメトリックスケッチャの新規作成、編集ができなくなります。既存のパラメトリックスケッチャを起動することはできます。
PDF、イラストレータファイルの読み込みができなくなります。

メニュー+ツールバー形式のインターフェースは 2016R2 で最後となります

Alphacam 2017 R1 からはメニュー形式のインターフェースはなくなりリボン形式のみとなります。

32 ビット Alphacam は 2016R2 で最後となります

Alphacam 2017 R1 からは 32 ビット版がなくなり 64 ビット版のみとなります。インストールできる OS は 64 ビットの必要があります。

注記: 32 ビット版でしか動作しないマクロやポストは 64 ビット対応にする必要があります。ライコムシステムズ社製以外のマクロやポストをご使用の場合、開発元にお問い合わせください。

Visual Basic 7.0

Alphacam の 64 ビット対応のため Microsoft Visual Basic for Applications 7.0 (VBA 7) にアップグレードしました、VBA7 は 32/64 ビット両方の Alphacam, AlphaEdit に搭載されています。

Alphacam 2015 R2 以降にインストールされる標準 VBA アドインは VBA7 にアップグレードされ 32/64 の両方に使用できます。

独自に開発された VBA を 64 ビット Alphacam で動作させるには修正の必要があることがあります。 [Appendix B](#) を参照してください。

インストール

Alphacam 2016 R2 のインストール

Alphacam 2016 R2 をインストールする際には 32/64 ビットのどちらをインストールするか指定できます。32 ビット Alphacam は 32/64 ビット OS のどちらにもインストールできます。64 ビット Alphacam は 64 ビット OS にしかインストールできません。

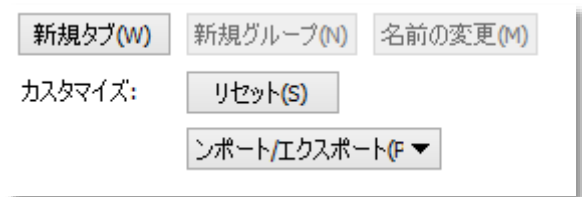
64 ビット Alphacam は大容量のメモリーを使用できる利点があります。

2016 R1 のリボンバー設定の読み込み

Alphacam 2016 R2 は 2016R1 のリボンバー設定を自動的に読み込みます

リボンバー設定の読み込み/保存のデフォルトフォルダは以下になっています。

...\LICOMDIR\Configurations.



インターフェース

Alphacam クリップボード

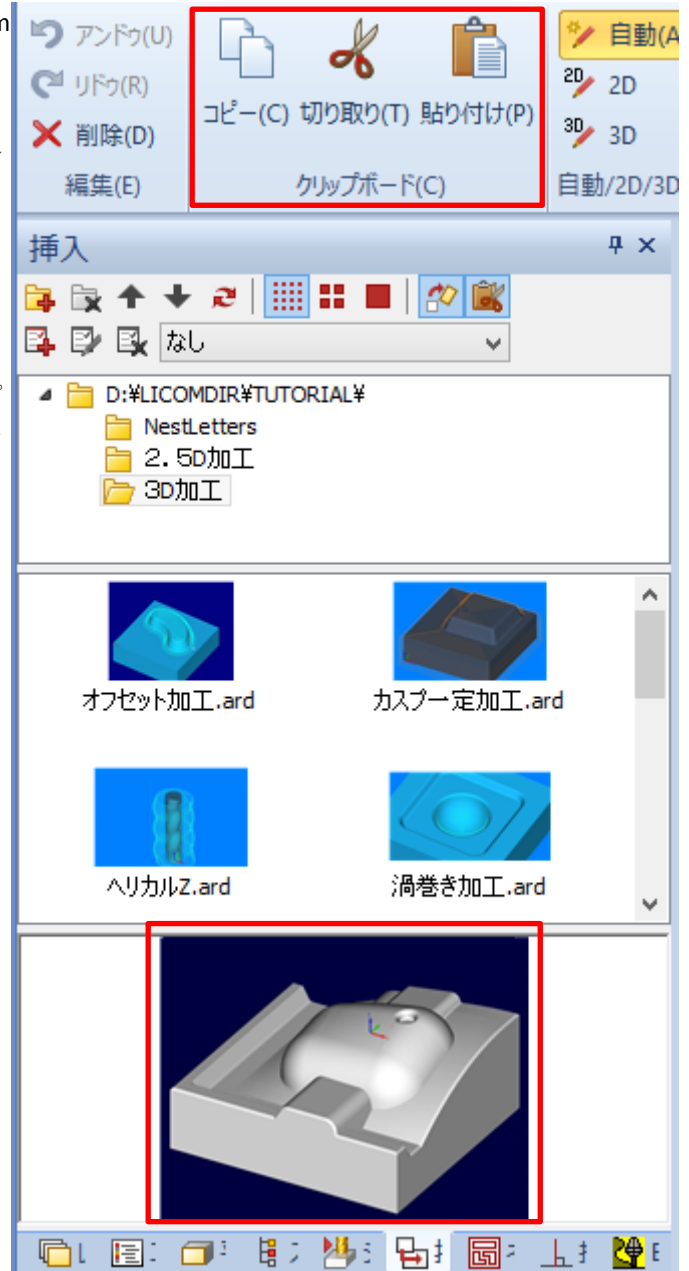
クリップボードがサポートされ、コピー／切り取り／貼り付けを行うことができます。別に起動した Alphacam や別モジュールの Alphacam 間でデータをやりとりすることができます。

リボンバーの編集タブにクリップボード用の 3 コマンドが追加されました。

- コピー
- 切り取り
- 貼り付け

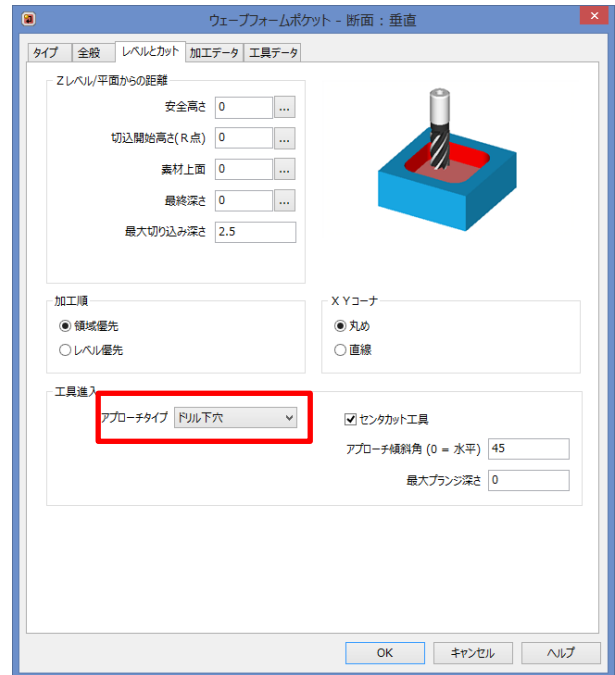
プロジェクトマネージャの挿入ページには、クリップボードに保存された内容を表示する領域が追加されました。

プレビュー領域からドラッグ&ドロップで貼り付けることができます。



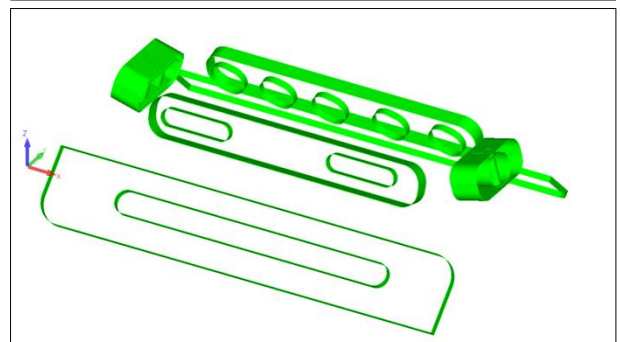
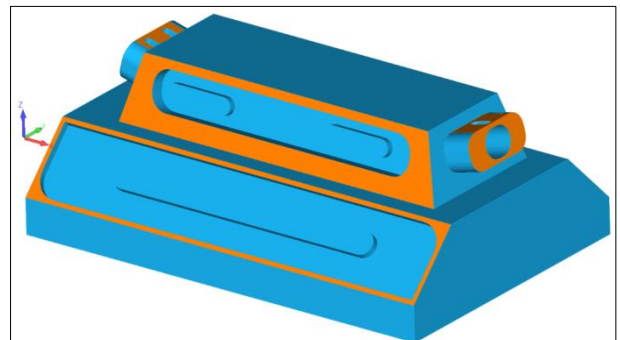
ウェーブフォームポケットのオプション追加

ウェーブフォームポケットの工具進入オプションにドリル下穴が追加されました。



フィーチャー抽出 - 選択されたフェースに対する輪郭フィーチャーの抽出

選択された様々な角度のフェースから、一度に各フェースに垂直な方向で輪郭を抽出できるようになりました。



アプローチ/リトラクトタブの追加

いくつかの加工コマンドにアプローチ/リトラクトタブが追加されました。

アプローチ/リトラクト設定をデフォルトとして保存、取り出しする機能に加え、個別に保存することが可能となりました。

これまでアプローチ設定をリトラクトにコピーすることができましたが、リトラクトからアプローチに設定をコピーするボタンが追加されました。

プロジェクトマネージャの工程ページの工程を右クリックすると表示される「アプローチを編集」のダイアログでは「適用」ボタンがあり、ダイアログを開いたままでアプローチの変化を確認することができます。

注記: アプローチ/リトラクト設定ファイルの拡張子

- アプローチ/リトラクト設定ファイル: *.alio.
- 3D アプローチ/リトラクト設定ファイル: *.a3dlio.

保存されるデフォルトフォルダは以下のフォルダです。

...\LICOMDAT\LeadData.



挿入ページ - 自動配置

プロジェクトマネージャの挿入ページに、配置設定に従って自動的に複数個の配置を行うコマンドが追加されました。

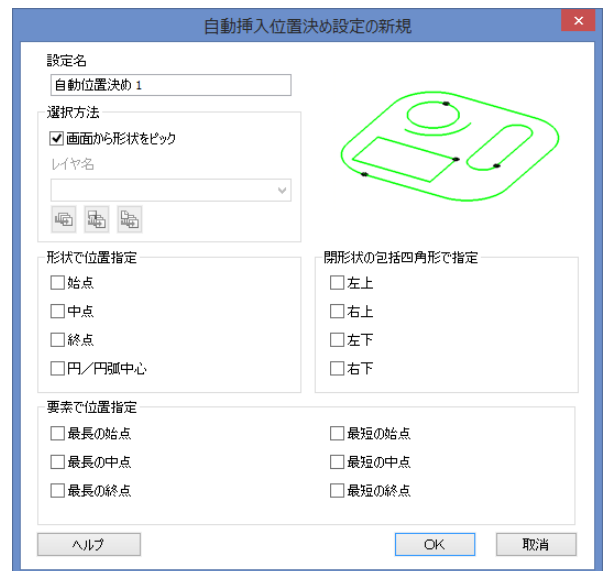
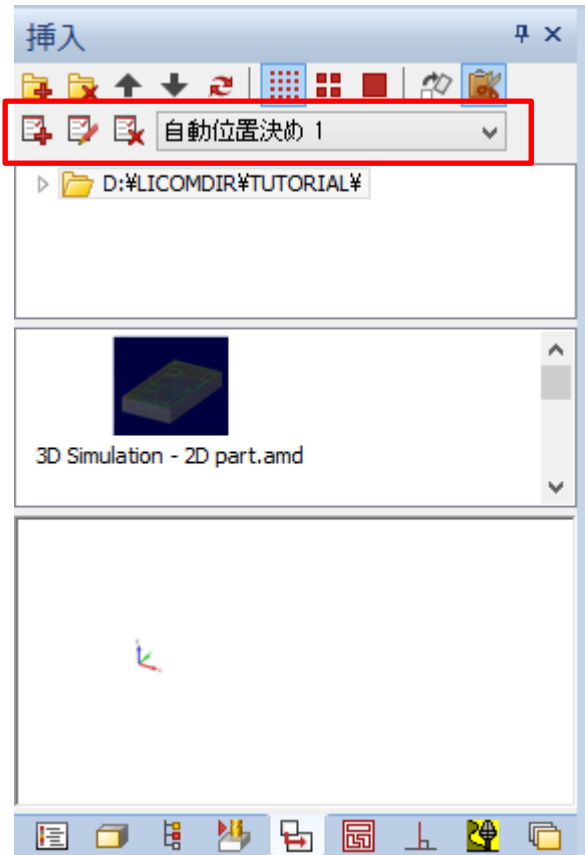
新しいコマンド:

- 配置設定の新規作成
- 配置設定の編集
- 配置設定の削除
- 作成された配置設定を選択するドロップダウンリスト

配置設定の作成、編集をするためのダイアログが追加されました。

この機能は、加工部位の形状をあらかじめ作成し、挿入によって一度に複数個配置する場合に非常に便利です。

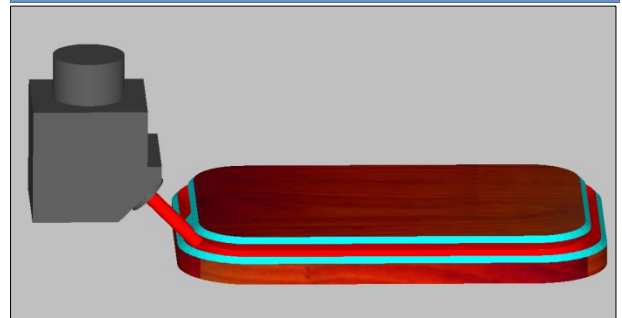
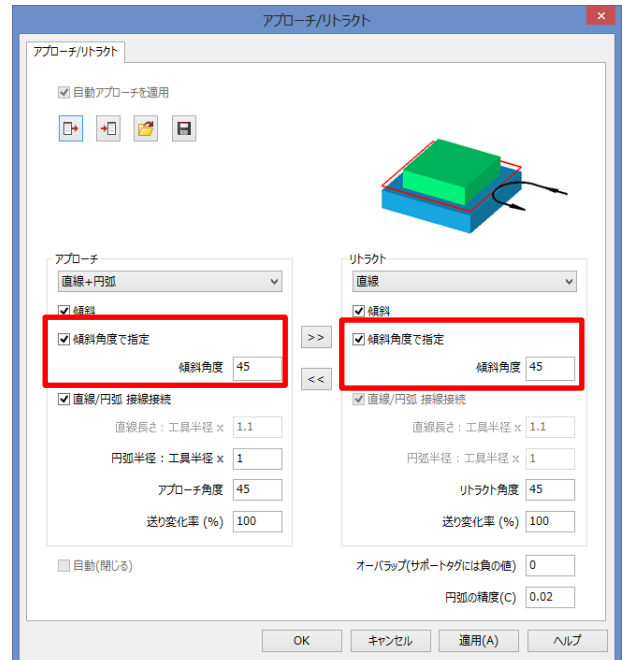
注記: 自動配置が選択されたときは 2D トランスフォームを使用できなくなります。



傾斜アプローチの角度指定オプション

アプローチ/リトラクトの傾斜アプローチで傾斜角度を指定するオプションが追加されました。

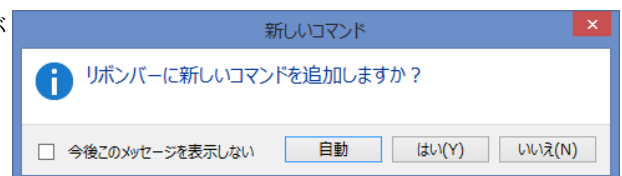
アングルヘッドの傾斜角度を手動で行う場合、機械的に角度が決まっています。このようなヘッドでは傾斜アプローチを角度で指定すると工具の方向と一致したアプローチ/リトラクトが作成され安全な移動を簡単に作成できます。



新規コマンドボタンをデフォルト位置に登録

Alphacam2016R2 で追加されたコマンドのボタンをリボンバーのデフォルト位置に自動登録できるようになりました。自分で任意の位置に登録することもできます。

初回起動から右のメッセージが表示されるようになります。

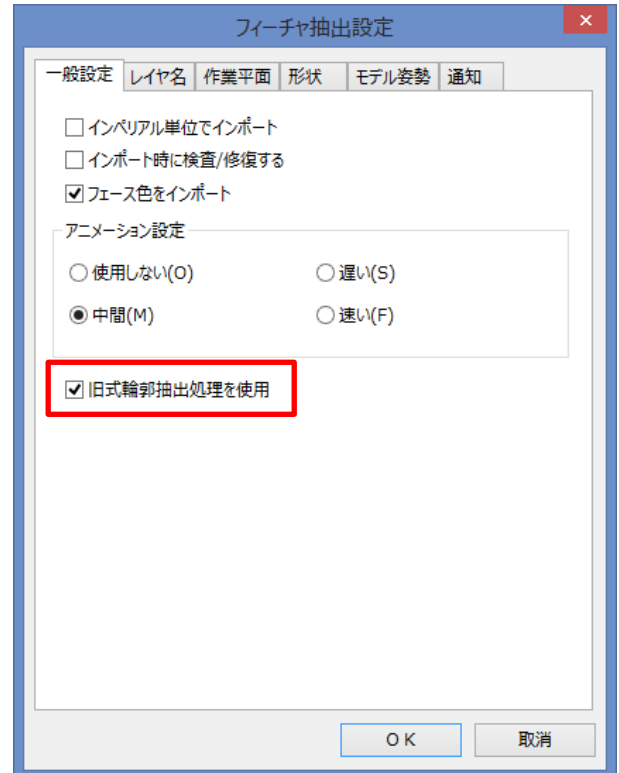


注記: いいえを選択すると新しいコマンドはリボンバーには登録されませんが、リボンバーの設定で登録することができます。

フィーチャー抽出 パラソリッドカーネルの輪郭抽出

フィーチャー抽出の輪郭抽出にはパラソリッドカーネルから提供される処理を使用するように変更されました。

デフォルトでは新しい処理が使用されますが、設定を行うことで従来処理の輪郭抽出を行うことができます。



アドイン 自動素材作成

無償オプションに自動素材作成が追加されました。選択した形状より少し大きめの素材を簡単に作成できます。

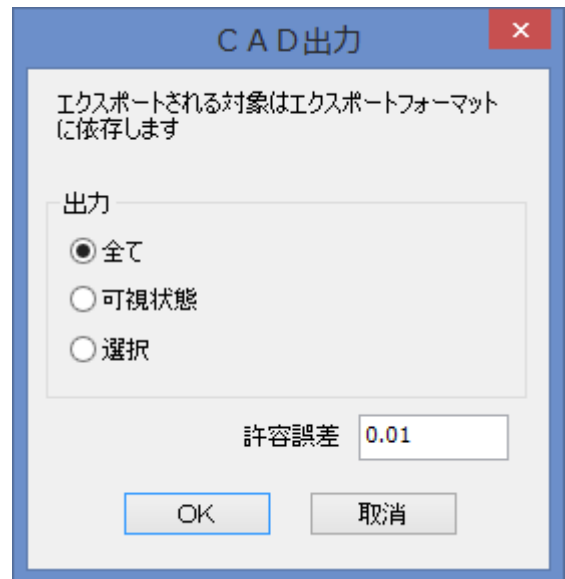
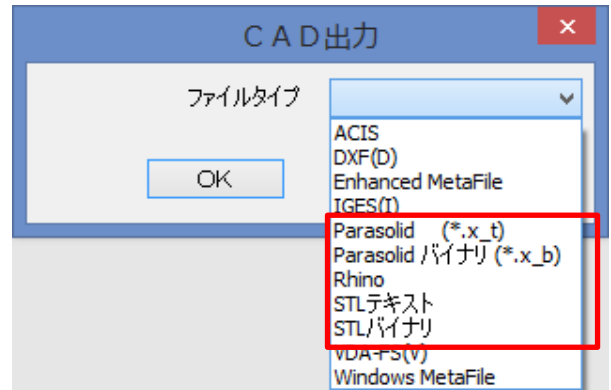


CAD データ出力 出力対象が選択可能

以下の CAD データ出力が更新されました:

- ACIS.
- Parasolid (*.x_t).
- Parasolid バイナリ (*.x_b).
- Rhino.
- STL バイナリ.
- STL テキスト.

これらの CAD 出力を行うとダイアログが表示され、全て、可視状態、選択のいずれかを選択することができます。



ポストプロセッサ ノコギリ加工固定サイクル用アトリビュート

以下のアトリビュートがノコギリ加工の固定サイクル用に追加されました。:

- _LicomUKCRHSawStock
加工データタブで指定された切り残し量
- _LicomUKCRHSawClimbConventional
+1 : ダウンカット, -1 : アップカット

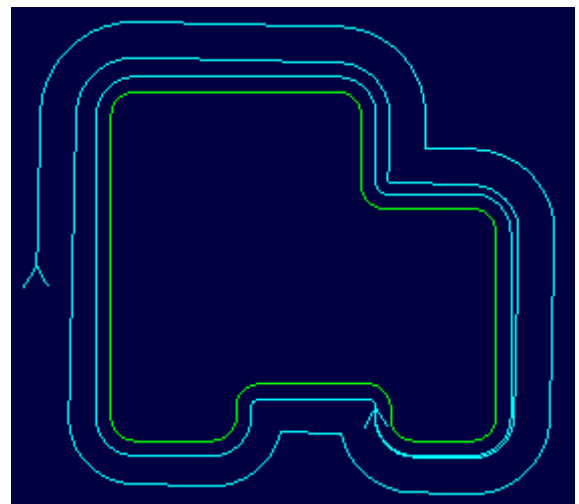
また ポスト変数 SCA はノコギリの切断加工の角度が形状指定、ダイアログ指定のどちらでも保持されるようになりました。

輪郭加工にスパイラル追い込みオプション追加

輪郭加工と穴加工コマンドにスパイラル追い込みオプションが追加されました。

外側（または内側）から徐々に仕上げ形状に追い込んでいく工具経路が生成されます。

※Standard 以上から搭載



ポストプロセッサ 4/5 Axis 軸用設定 - Maximum Step Length が追加

ポストプロセッサに“Maximum Step Length”設定が追加されました。

先端点制御機能のない機械では、1要素の同時4/5軸の工具先端の動きが大きいと、実際の機械では工具先端の動きが直線になりません。

この設定を使用して、1回の移動量を小さくして実機の工具先端の動きをより直線に近づけることができます。

注記: これはポストプロセッサの設定で、4/5軸の工具経路に有効です。

CAD データ読み込み

SpaceClaim 2016

Alphacam 2016 R2 は SpaceClaim 2016. のパートモデルをサポートしました。

ポストスクリプト読み込みの更新

Alphacam 2016 R2 は更新されたポストスクリプトローダーを搭載しました。

対応 CAD 形式の追加 Adobe PDF , Adobe イラストレータ

Alphacam 2016 R2 は Adobe PDF とイラストレータのファイルを読み込めるようになりました。

Note: 保守契約が終了するとこの機能は使えなくなります。

形状

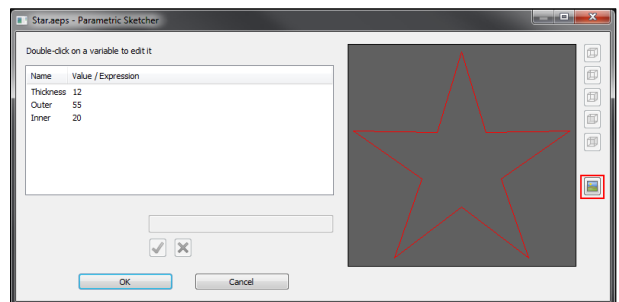
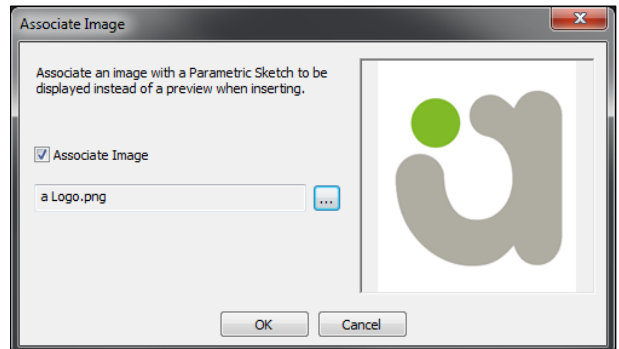
パラメトリックスケッチャ – 画像の連動

パラメトリックスケッチャダイアログのリボンバーにイメージの関連づけコマンドが追加されました。イメージはパラメトリックスケッチャの設定全体と個々の変数に割り当てることができます。

関連づけされた画像は以下に表示されます。

- プロジェクトマネージャの挿入ページ
- パラメトリックスケッチャのデータを開くダイアログのプレビュー
- パラメトリックスケッチャの起動

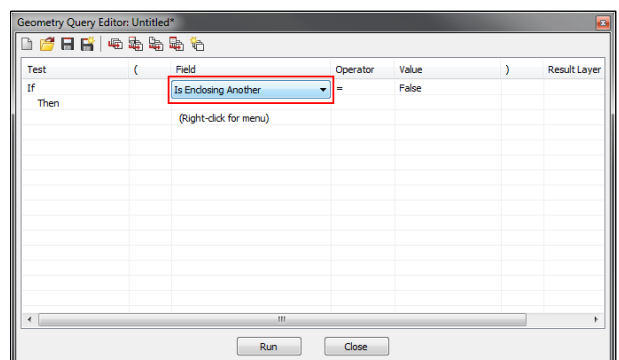
注記: パラメトリックスケッチャを起動するダイアログはサイズ変更可能になりました。またイメージと従来のプレビューを切り替えるボタンが追加されました。



形状のレイヤ分離 – 新規テスト関数: “囲まれている”

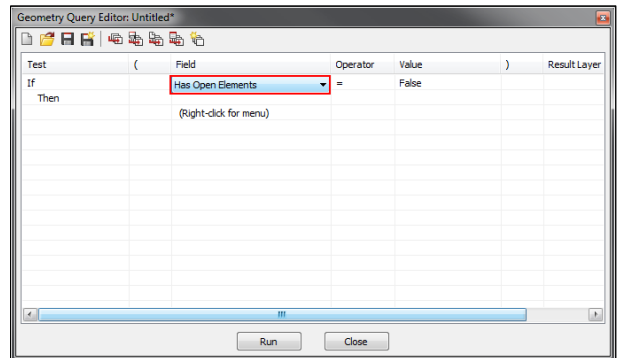
新しいテスト関数 “**囲まれている**” が追加され、True または False を返します

この関数はある形状が別の形状の内側に含まれているか調べます。ある形状が最も外側であるか調べるときに便利です。



形状のレイヤ分離 – 新規テスト関数: “開要素を持つ”

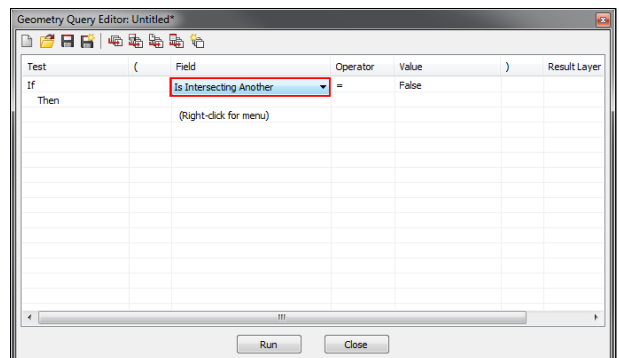
新しいテスト関数 “開要素を持つ”が追加され、True または False を返します。



形状のレイヤ分離 – 新規テスト関数 “交差している”

新しいテスト関数 “交差している”が追加され、True または False を返します。

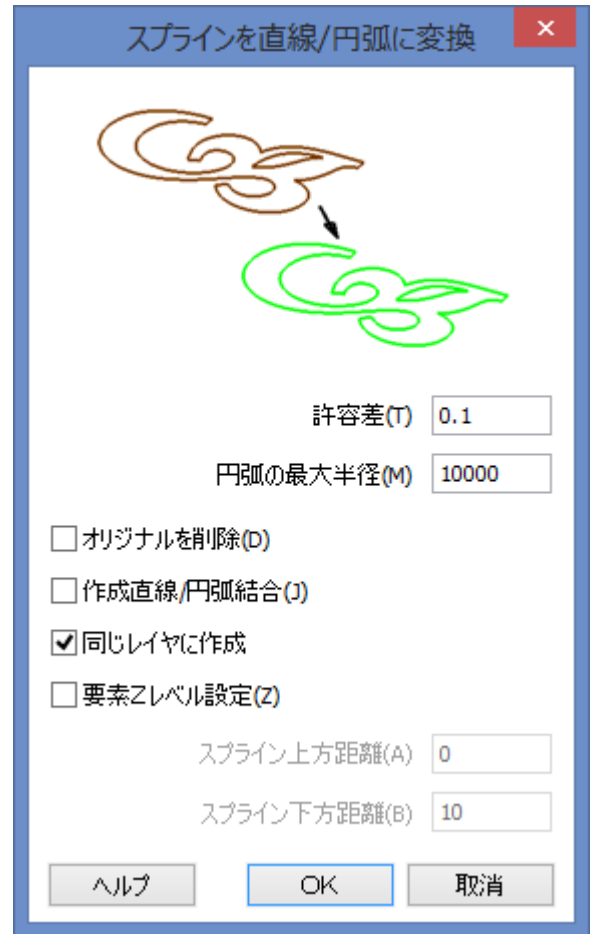
注記: 交差形状は同じ作業平面上であり、補助線ではいけません。



スプライン変換 - 同じレイヤに作成オプション

スプラインを直線/円弧に変換 コマンドダイアログが更新され、イメージとツールチップがあります。

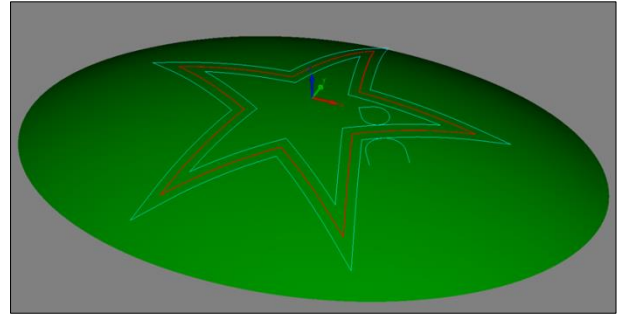
“同じレイヤに作成”オプションが追加されました。チェックされていると、生成される形状のレイヤはスプラインと同じレイヤになります。



スプライン/ポリライン沿い加工 – 角の処理の改善

スプライン/ポリライン沿い加工が改善されました。

この改造により内回り/外回りの角が共に食い込みを起こさないようになりました。



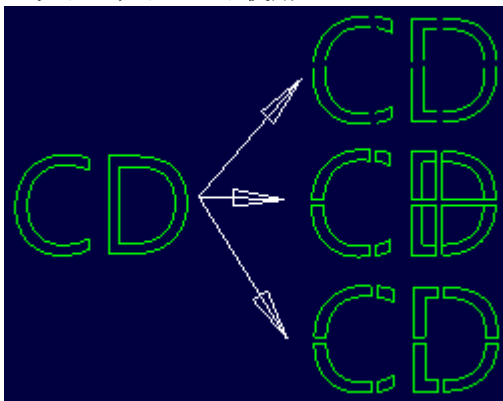
形状取り出し – パネリング

形状取り出し – パネリングのコマンドダイアログが更新され、イメージとツールチップがあります。

以下のオプションが追加されました。

- **パネルエッジの形状と閉じる** パネル境界で分断された形状を自動的に閉じます。
- **カットアウトモードを使用** 内側と外側の形状がある場合に有効です。

以下の結果は、上から
チェックなし
パネルエッジの形状を閉じる
カットアウトモード使用

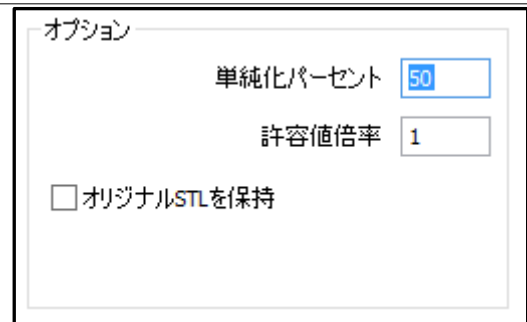
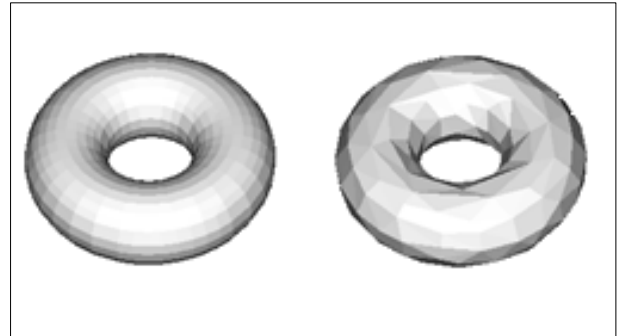


STL の単純化コマンド

STL の単純化コマンドを使用して STL のファセット数を減らすことができます。コマンドは三次元-STL ツールに追加されました。

このコマンドは荒加工を行う際に、非常に詳細な STL である必要がないときに有効で、計算時間の短縮を行うことができます。

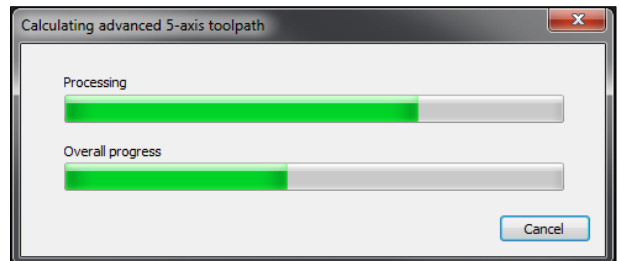
単純化はオリジナル STL のファセット数の何パーセントにするかで指定します。



アドバンスド 5 軸加工

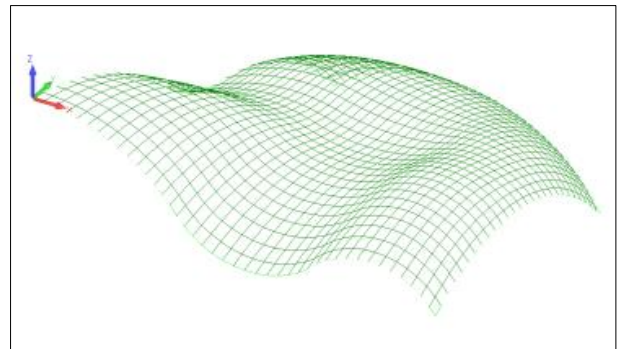
プログレスバーの改善

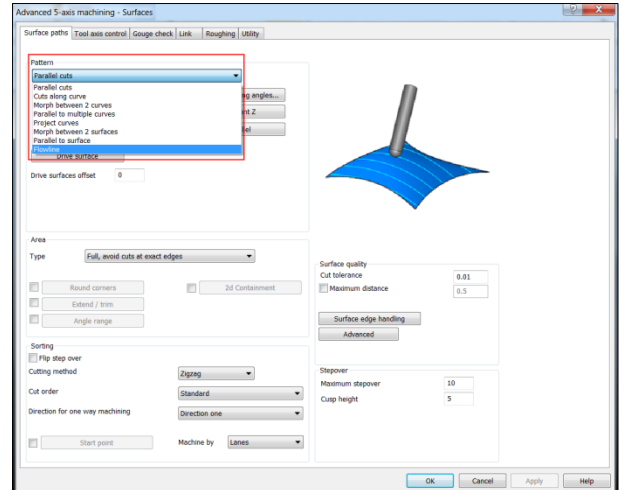
長時間の計算を必要とする場合の、利用者に対する進行状況の通知が改善されました。



フローライン加工の追加

サーフェスのパラメータ線を使用したフローライン加工が追加されました。





API

C# アドイン DLLs

Alphacam 2016 R2 から C# で開発した DLL のサポートを開始しました。

これまでは C# の DLL は VBA から呼び出す必要がありました。今回の改造で VBA は必要なくなります。イベントもサポートされました。

注記: 詳細な情報は以下を参照してください。

- [Appendix C.](#)
- The 2016 R2 DVD. Microsoft Visual Studio 2013 C# プロジェクトの 3 サンプルが含まれています。
\\ExampleFiles\API\DotNetAddIns フォルダに詳細な文書も含まれています

API - SetAttributeForNextElement アトリビュートを格納します

MillManualToolPath.SetAttributeForNextElement 関数は次のエレメントに複数のアトリビュートを格納する予約をします。


シミュレーション

クランプ、フィクスチャ -Z 方向の移動で素材と干渉チェックを行います

クランプ、フィクスチャを Z 方向に移動させるとき、自動的に素材との干渉チェックを行います。

CAD データ入力 対応バージョン

E = エッセンシャル S = スタンダード A = アドバンスド U = アルティメイト

CAD データ形式	ルータ	ストーン	旋盤	ミル	ワイヤ	レーザー
DXF/DWG (AutoCAD 2015)	E S A U	E S A U	E S A U	E S A U	S A	A U
IGES (バージョンなし)	E S A U	E S A U	E S A U	E S A U	S A	A U
Rhino 5.0	S A U	S A U	S A U	S A U	S A	A U
STL (バージョンなし)	S A U	S A U	S A U	S A U	S A	A U
SketchUp 2015	S A U	S A U	S A U	S A U	S A	A U
Adobe PDF	E S A U	E S A U	U	E S A U	S A	A U
Adobe Illustrator	E S A U	E S A U	U	E S A U	S A	A U
PostScript	E S A U	E S A U	U	E S A U	S A	A U
STEP AP214/AP203 (preferred Schemas)	A U	A U	A U	A U	A	U
Part Modeler (Alphacam と同一バージョン)	A U	A U	A U	A U	A	U
VISI Version 20	A U	A U	A U	A U	A	U
Autodesk Inventor 2017 (current, IPT & IAM files) 	A U	A U	A U	A U	A	U
ACIS R1 – 2016 1.0	A U	A U	A U	A U	A	U
Parasolid 9.0 – 28.0.159	A U	A U	A U	A U	A	U
SOLIDWORKS 2016 (SLDPRT ファイルのみ)	A U	A U	A U	A U	A	U
Solid Edge ST8 (PAR & PSM ファイル)	A U	A U	A U	A U	A	U
Spaceclaim 2016	A U	A U	A U	A U	A	U
有償オプション						
Catia V4	A U	A U	A U	A U	A	U
CATIA V5 V5R8 – V5R25 and V5-6R2012 – V5-6R2015	A U	A U	A U	A U	A	U
Creo Parametrics 2.0 (formerly Pro/E Wildfire)	A U	A U	A U	A U	A	U
NX9 (formerly Unigraphics NX)	A U	A U	A U	A U	A	U



Autodesk Inventor ファイルの読み込みには files, Autodesk Inventor® または Inventor® View™ がインストールされている必要があります。Inventor® View™ は Autodesk からダウンロードできます。

メンテナンスレポート

以下の不具合修正、改善が **Alphacam 2016 R2** で行われました

ID	Description	Closing Notes
218	Undo - Solids are not removed after the Home File Insert command has been used.	Fixed.
985	Input CAD - DXF/DWG - Geometries on angled planes are imported into the wrong position.	Fixed.
1006	3D / Space Mouse - 64-bit - Poor performance. Low resolution movement and cross hair traces.	Fixed.
1007	Clamps and Fixtures - Assigned colour is lost after saving and re-opening a drawing.	Fixed.
1021	3D Machining - Z (Enhanced Undercuts) - Gouging. Bad toolpath created on some models.	Fixed.
1128	APM - 'Object variable or Width block variable not set' when copying and pasting a Rule.	Fixed.
1136	CAD Import - Solid Edge ST8 files do not import.	Fixed.
1151	Automation Manager - Unhandled exception - No valid paths in file: C.	Fixed.
1211	Advanced 5 Axis - Alphacam crashes when using Surface Machining in Italian Language.	Fixed.
1215	Drill/Tap Holes Auto Z - Levels and Cycle Data tab - There is a pick button for the Rapid Down To distance which is unnecessary. The safe rapid level pick button should only be enabled if the "rapid is absolute" option is checked.	Fixed.
1224	Previews in various Open / browser dialogs are solid black rectangles.	Previews for files with no graphic display now contain a white box with the Alphacam logo.
1229	Automation Manager - splash screen and dialog boxes are displayed incorrectly with a DPI setting greater than 100%.	Fixed.
1233	Simulation - Solid Simulation - When the Machine Configuration is set for part rotation, the simulation is incorrect and the part is gouged.	Fixed.
1360	Automation Manager - keyboard shortcuts override typing and return to the Drawing Area.	Fixed.
1382	Homework mode - List NC causes Alphacam to crash.	Fixed.
1392	ECO Option IDs missing	Fixed.
1400	NC Output Manager - The file extension specified by the user has no effect. Always outputs .anc.	Fixed.
1420	Reports - CDM - Nesting - Incorrect sheet numbers produced.	Fixed.
1485	Nesting - True Shape - Poor sheet usage compared to Quick Nest Geometries (True Shape) add-in and competitor. Regression from 2015 R1.	Fixed.
1510	Lathe - Waveform Pocket - If a value is entered for Width of Cut for Final Pass, two operations are created.	Fixed.

1512	3D Machining - Z Contour Roughing - Take Account of Previous Machining does not machine final depth when the first operation has Z stock.	If "Take Account of Previous Machining" is checked, the "XY stock to be Left" and "Z Stock to be Left" fields are both disabled.
1525	File inserter - does not prompt you to save the drawing after dragging and dropping files into the Drawing Area and selecting File New .	Fixed.
1546	Canned Cycle Sawing - Saw shown in wrong orientation.	Fixed.
1627	Nesting - LicomUKsab_outer_path attribute is not always set on the outer geometry of a part.	Fixed.
1643	Save As Tool Holder dialog has a corrupted file type.	Fixed.
1656	Multidrill - Use Legacy Multidrill - when two or more holes are concentric: "This key is already associated with an element of this collection".	Fixed.
1662	Stone - Cut Surfaces / Solids With Disk - Clearance plane value entered is different in drawing.	Fixed.
1664	Automation Manager - NC Output - Tool Ordering is not ordering tools. NC is output before tool ordering.	Fixed.
1669	Under certain conditions, surfaces are not rendered and cannot be converted to STL or machined.	Fixed.
1734	Machining - The addition of C axis positions in vertical work planes.	Fixed.
1735	Nesting - Cut small parts first - does nothing.	Fixed.
1742	Automation Manager - Saving nested sheet drawings taking too long when there are a large number of nested sheets.	Fixed.
1750	Rough / Finish and Z Contour Roughing - Auto Z - When applying a toolpath to multiple geometries of different levels, no toolpath is applied under certain circumstances.	Fixed.
1752	Automation Manager - Different result from Automatic Extraction. Not all features are extracted.	Fixed.
1781	Single cut internal arc sawing - Leads only on first arc.	Fixed.
1788	Automation Manager - Configuration - Disable Screen updates has no effect when selected.	Fixed.
1789	Error creating solid object for solid simulation.	Fixed.
1792	Nesting - Minimum Gap between Paths - Gap incorrect when a helical toolpath is inside another toolpath.	Fixed.
1876	Feature Progress Dialog box is too short.	Fixed.
1902	Lathe - Simulation - Enhanced simulation of sub-spindle 3D toolpaths.	Moves between work planes are now visible in simulation.
1909	Automation Manager - Import Settings - CSV import Field is duplicated in field list after processing.	Fixed.
1926	Automation Manager - If a post file name contains a comma, no NC is output.	Fixed.
1928	Define Tool - User-Defined Tool - Diameter and Length not picked up Correctly.	Fixed.
1932	Add-Ins - Reverse Side Nesting - Nesting Information - When using Nesting Information on a drawing with reverse side nesting multiple sheets are created.	Fixed.
1942	Automation Manager - 'Runtime Error. R6052 - pure virtual function call' when path to styles differs from the setting in Home Settings Configure ► Folders .	Fixed.

1954	Automation Manager - In the output drawings tools are sometimes incorrectly ordered.	Tool Ordering tab superseded by Machining Order tab.
1960	Part Modeler - CAD Import -Export - Alphacam link buttons not working.	Fixed.
1966	Lathe - Solid Simulation - Turret orientation is rotated by 90 degrees.	Fixed.
1967	Rough/Finish - Incorrect output when specified cuts are used along with rapid being below material top.	Fixed.
1969	Automation Manager - Onepost - Stops working and crashes with app.selectpost error if outputting with a onepost.	Fixed.
1985	Material - Associate for Auto update - When material is set for auto update, only half the part is cut.	Fixed.
1988	Nesting - Text - Nesting fails when common lines are found in text of parts.	Fixed.
2127	Machine - Clamps/Fixtures - Select all on Pop Up/Down Clamps doesn't work.	Fixed.
2133	Rough or Finish / Cut Shape - Oscillation - NC Output - Loop created in NC code.	Fixed.
2140	Lathe - Balanced Finishing - Second number in 3 digit sequence is incorrect.	Fixed.
2155	Lathe - Finishing - When using the same tool as roughing, a toolpath cannot be applied.	Fixed.
2157	Automation Manager - Nesting - Special functions settings retained.	Fixed.
2158	Automation Manager - Unhandled exception - Tool Sorting - Procedure not found or invalid name.	Fixed.
2166	Automation Manager - Layer mapping setup - Crashes when expanding an invalid/corrupted or a newer version machining style.	Fixed.
2182	Rough or Finish - Helical Interpolation - XY stock - On Danish systems -/+ is reversed. Related to decimal separator in region settings.	Fixed.
2187	Auto-Align Part - Has no effect on some parts.	Fixed.
2188	Automation Manager - Unhandled Exception related to Auto Align.	Fixed.
2193	Simulation - Collision of tool shank is shown as material colour. This is inconsistent with the behaviour in previous releases.	Fixed.
2194	Automation Manager - Tool Ordering - Changes have no effect until Automation Manager is restarted.	Fixed.
2211	Radial Menu - Alphacam stops working / crashes.	Fixed.
2236	Reports - No scrap reported on nested sheet.	Fixed.
2238	Reports - Drilling operation causing incorrect scrap value, sometimes negative.	Fixed.
2239	Reports - Incorrect scrap and time calculation related to manually nested parts.	Fixed.
2240	Reports - Scrap is incorrectly calculated by tool centre.	Fixed.
2241	Simulation - Error creating solid object for simulation. Certain geometries fail to create a solid for simulation.	Fixed.
2250	Machine - Insert Program Stop - Extra operation created when program stop is moved to an operation with multiple sub operations.	Fixed.
2251	Rough or Finish - Automatic Support Tags - Negative stock is causing rapids to safe Z at the tags.	Fixed.
2255	Reports - Cycle Time is incorrect with Auto Z. 100% Scrap reported.	Fixed.
2266	Automation Manager - Feature Extraction with Query has a different result from Automatic Extraction.	Fixed.

2427	Simulation - Single Step – Lead -in is ignored when a single step is used.	Fixed.
2434	Panelling - Incorrect Trimming of Geometries.	Fixed.
2439	Lathe - Z Contour Roughing - NC Output - Down Feed is at Cut Feed speed.	Fixed.
2441	Stone - Cut Surfaces/Solids with Disk - 'Join Paths With Arcs' option cannot be disabled.	Fixed.
2462	Styles - When a style contains Auto Z drilling, there is an error when editing. 'Which Holes' selection changes.	Fixed.
2489	Stone - Simulation - Guard is simulated with incorrect orientation.	Saw guard will no longer be displayed when clicking toolpath elements. It will be displayed in simulator as before.
2492	Stone - Cut Surfaces/Solids With Disk - The links created using "Specify moves between cuts" are incorrect shape.	Fixed.
2494	Advanced 5 Axis - Global co-ordinates are not used when using a cylindrical retraction plane.	Fixed.
2505	Automatic Extraction / Extract Solid Body Outlines - Regression from 2015 R1. Incorrect geometry shape caused by Parasolid error.	Fixed.
2509	Program Stop - OPN variable is not correct when using the STOP operation.	Fixed.
2517	Stone - Cut Surfaces/Solids with Disk - Roughing - Bi-directional selection doesn't have any effect.	Fixed.
2619	Solid model properties - Under certain circumstances these are not all displayed or are duplicated. There is no efficient method of copying all properties to the clipboard.	Fixed.
2629	Automation Manager - Nesting assumes there are toolpaths being nested.	Fixed.
2663	Automation Manager - Multidrill - Error 'Multidrill Add-in not loaded'.	Fixed.
2669	Lathe - Solid Simulation - User-defined tools are not displayed correctly.	Fixed.
2672	Lathe - Display Options - Material Cut - When complex user-defined tools are used, incorrect material is shown.	Fixed.
2712	Nesting - 'The following parts are invalid. If you continue they will not be nested'. Related to internal geometry elements touching external geometry / common lines.	Fixed.
2715	Multidrill - Not using slave drills. All holes are drilled using only one tool.	Fixed.
2720	Multidrill - Not saving user settings (for example, machining parameters, tolerances) to the database on 64-bit version.	Fixed.
2844	Rough/Finish - Using a small overlap on lead-in or lead-out moves results in an incorrect toolpath.	Fixed.
2845	Clipboard - API Drawing.SaveAs - Error "DISK FULL! Change disk or save to a different drive" is displayed when saving a big drawing.	Fixed.
2854	API - Reports - Set objFiles = rptJob.ExportReports returns nothing.	Fixed.
2864	Convex tip tool doesn't convert to Alphacam correctly.	Fixed.
2914	Automation Manager - Custom macro fails when LICOMDIR is located on a network drive.	Fixed.
2932	Automation Manager - "What to Extract values" is ignored. Contour and Drillable Holes are always extracted.	Fixed.
3106	Automation Manager - Incorrect output when nesting with multiple sheets in one job. Only one NC file and one drawing is output when nesting onto multiple sheets.	Fixed.

3115	Automation Manager - Setting of Part Origin produces a poor result when combined with Extract Face options.	Fixed.
3244	Panelling - Support for cut-outs.	New checkbox added to the Panelling dialog.

Appendix A - 2016 R2 での API 改造

Expose Nesting Text Placement

Example:

```
Public Sub PlaceTextInPathSample()
' To use the "PlaceTextInPath" command, you must have a reference
' to "Alphacam Geometry Utilities" - this is done through the
' Tools | References
' menu of the VBA Editor

Dim geoUtils As AlphaCAMUtilities.GeoUtilities
Dim geo2D As geo2D
Dim pthTest As Path
Dim txtInsert As Text

' Create a simple geometry - this must be a closed profile
Set geo2D = ActiveDrawing.Create2DGeometry(0, 0)

With geo2D
.AddLine 100, 0
.AddArcPointRadius 100, 10, 5, False, False
.AddLine 0, 10
.AddArcPointRadius 0, 0, 40, False, True

Set pthTest = .Finish

End With

' Get the Geometry Utilities interface from the current drawing
Set geoUtils = ActiveDrawing.GeoUtilitiesInterface

' Place the text inside the geometry
Set txtInsert = geoUtils.PlaceTextInPath("Text", pthTest)

' Returned "txtInsert" is an Alphacam Text object which can be
' manipulated if needed

End Sub
```

Feature API calls for “From Faces” Orientation type

SolidExtraction2 Object

Read/Write Property: FromFacesSelection As FromFacesSelectionTypeEnum (one of FromFacesAllFaces, FromFacesAllPlanarFaces or FromFacesSelectedFaces)

When using the “From Faces” orientation type, find contours on all faces, all planar faces, or selected faces (see below) respectively.

Read/Write Property: FromFacesExtractSolidBodyOutline As Boolean

When using the “From Faces” orientation type, this property controls whether the Solid Body outline is extracted or not (in addition to the contours on faces).

Read/Write Property: FromFacesOptimisePlanes As Boolean

When using the “From Faces” orientation type, if this property is TRUE, parallel contours are put on a single work-plane with adjusted Z-levels, if it is FALSE, each contour is put on a separate work-plane such that its top Z-level is 0.

Method: SetFromFacesSelectedFaces(SolidSelector)

When using the “From Faces” orientation type with FromFacesSelection set to FromFacesSelectedFaces, use the faces in the given SolidSelector object.

Examples

Extract contours using 'FromFaces' orientation and 'AllPlanarFaces'

```
Public Sub ExtractContoursFromAllPlanarFaces()
    Dim SF As SolidFeatures
    Set SF = App.ActiveDrawing.SolidInterface
    Dim SE As SolidExtraction2
    Set SE = SF.SolidExtraction2
    SE.FindContours = True
    SE.FindDrillHoles = False
    SE.Source = FeatureAllBodiesExtracted
    SE.FeatureExtractionMode = FeatureExtractionModeContour
    SE.Tolerance = 0.001
    SE.Step = 0
    SE.OpenAirPocket = False
    SE.MaxDrill = 100

    SE.Orientation = FeatureFromFaces2
    SE.FromFacesExtractSolidBodyOutline = False
    SE.FromFacesOptimisePlanes = True
    SE.FromFacesSelection = FromFacesAllPlanarFaces
    SF.AutoExtract2 SE
End Sub
```

Extract contours using 'FromFaces' orientation and 'SelectedFaces'

```
Public Sub ExtractContoursFromSelectedFaces()
    Dim SF As SolidFeatures
    Set SF = App.ActiveDrawing.SolidInterface
    Dim SE As SolidExtraction2
    Set SE = SF.SolidExtraction2
    'Ask user for faces to get contours from
    Dim Selector As SolidSelector
    Set Selector = SF.Selector
    Selector.What = FeatureSelectFace
    Selector.Select "Please select some faces"
    SE.FindContours = True
    SE.FindDrillHoles = False
    SE.Source = FeatureAllBodiesExtracted
    SE.FeatureExtractionMode = FeatureExtractionModeContour
    SE.Tolerance = 0.001
    SE.Step = 0
    SE.OpenAirPocket = False
    SE.MaxDrill = 100
    SE.Orientation = FeatureFromFaces2
    SE.FromFacesExtractSolidBodyOutline = False
    SE.FromFacesOptimisePlanes = True
    SE.FromFacesSelection = FromFacesSelectedFaces
    SE.SetFromFacesSelectedFaces Selector
    SF.AutoExtract2 SE
End Sub
```

Feature API – AutoExtract and AutoExtract2 Update

Previously when configuring parameters for the AutoExtract or AutoExtract2 methods, if the Source property was set to “FeatureSelectedBodiesExtracted” or “FeatureSelectedFacesExtracted” the user would be prompted to select entities but would not be able to do so. Now, if either of these Source types are used, then Alphacam will look at the “MainSelector” object to determine which Bodies/Faces to use. This enables the API user to prompt the user for bodies/faces to use and then pass them to the AutoExtract methods.

Example

```
'Extract contours in flat-land from selected bodies
Public Sub ExtractContoursOnSelectedBodies()
    Dim SF As SolidFeatures
    Set SF = App.ActiveDrawing.SolidInterface
    Dim SE As SolidExtraction2
    Set SE = SF.SolidExtraction2
    'Ask user for Solid Bodies to use
    Dim Selector As SolidSelector
    Set Selector = SF.Selector
    Selector.What = FeatureSelectBody
    Selector.Select "Please select some solids"
    'Copy the selection to the 'MainSelection'
    Selector.CopyToMainSelection
    SE.FindContours = True
    SE.FindDrillHoles = False
    SE.FeatureExtractionMode = FeatureExtractionModeContour
    SE.Orientation = FeatureFlatLandOrientation2
    SE.Source = FeatureSelectedBodiesExtracted
    SE.Tolerance = 0.001
    SE.Step = 0
    SE.OpenAirPocket = False
    SE.MaxDrill = 100
    ' Auto extract the contours on the bodies selected by the user
    SF.AutoExtract2 SE
End Sub
```

Drawing Object

Method: SimplifySTL(double SimplifyPercentage, double ToleranceScaleFactor, Boolean KeepOriginal)

Simplifies the selected STL models in the Active Drawing. SimplifyPercentage must be between 0 and 100 exclusive and ToleranceScaleFactor must be greater than 0. The SimplifyPercentage is approximate because it will depend on the model and the ToleranceScaleFactor. Setting it to 50 should give a model with roughly half the number of faces as before. If KeepOriginal is True, then the simplified model is added to the drawing (the original STL model is unmodified), if it is False, then the faces on the existing model will be replaced with the simplified faces.

Examples

```
Public Sub SimplifySelected()
    ' Ask user to pick some STL models
    App.ActiveDrawing.UserSelectMultiAddinObjects2 "Please select STL models to be simplified", 0,
    "InputSTL"
    ' Simplify the selected models by 50% replacing the original models
    App.ActiveDrawing.SimplifySTL 50#, 1#, False

    App.ActiveDrawing.RedrawShadedViews
End Sub
```

```

Public Sub SimplifyAll()
    ' Simplify all STL models in the drawing.
    Dim s As STL
    Set s = App.ActiveDrawing.STLInterface
    Dim part As stlpart
    For Each part In s.Parts
        part.Selected = True
    Next
    App.Frame.ProjectBarUpdating = False
    App.ActiveDrawing.ScreenUpdating = False
    ' Simplify the selected models and keep the originals.
    ' The simplified STL models will be added to the drawing.
    App.ActiveDrawing.SimplifySTL 25#, 1#, True

    App.Frame.ProjectBarUpdating = True
    App.ActiveDrawing.ScreenUpdating = True
End Sub

```

Drawing.HasPanelingSheets and Drawing.MoveNestedSheetAndPartsToDrawing

Drawing.HasPanelingSheets

Returns true if there are any sheets created using the Edit|Panelling command. Otherwise false.

Drawing.MoveNestedSheetAndPartsToDrawing (SheetPath As Path, NewDrawing As Drawing, CopyBeforeMove As Boolean)

This function moves a nested sheet and the parts inside it to a temp drawing.

Example

```

Sub MoveNestedSheetToDrawing()
    Dim drw As Drawing
    Set drw = App.ActiveDrawing

    ' Check if drawing has panelling sheets on it
    If Not drw.HasPanelingSheets Then

        Dim path As path

        ' Ask user to pick a nested sheet geometry
        Set path = drw.UserSelectOneGeo("Please, select a nested sheet geometry")

        ' Move sheet and parts to a temp drawing
        Dim TempDrw As Drawing
        Set TempDrw = App.CreateTempDrawing
        drw.MoveNestedSheetAndPartsToDrawing path, TempDrw, False

        TempDrw.SaveAs ("C:\Temp.ard")
    End If
End Sub

```

Nesting Sheet Database - Command to Insert Sheet at Point

Example

```

Public Sub Test()

    Dim N As Nesting
    Set N = Nesting

    Dim sdb As SheetDatabase
    Set sdb = N.SheetDatabase

    Dim s As DatabaseSheet
    Set s = sdb.FindSheet("Holey")

    Dim p As Paths
    Set p = s.InsertInActiveDrawingAtPoint(10, 100)

End Sub

```

API – Ability to Edit Styles

New API - MillSubStyle.ShowDialogBox()

Example

```
Sub EditStyleUsingDialogBox()

Dim NewStyle As MillStyle
Dim StyleToEdit As MillStyle
Dim SubStyle As MillSubStyle
Dim SaveNewStyle As Boolean

SaveNewStyle = False

' Start a new Style
Set NewStyle = App.CreateMillStyle

' Get a style to edit
Set StyleToEdit = App.MillMachiningStyles(1)

For Each SubStyle In StyleToEdit.MillSubStyles

' Show dialog box to edit the style
If SubStyle.ShowDialogBox Then

'User Pressed OK, add modified substyle to the new style
NewStyle.AddOp SubStyle.GetMillData, SubStyle.Tool, SubStyle.GetLeadData
SaveNewStyle = True
End If

Next SubStyle

' Save Modified Style
If SaveNewStyle Then
NewStyle.SaveAs App.LicomdirPath & "\LICOMDIR\Styles\ModifiedStyle.ary"
End If

End Sub
```

API - For SetAttributeForNextElement to store more than one attribute

Example

```
Public Sub ApplyMultipleAttributes()
' Create and Set MillData
App.SelectTool "Flat - 12 mm + holder.art"
Dim MD As MillData
Set MD = App.CreateMillData
MD.SafeRapidLevel = 50
MD.RapidDownTo = 5
MD.MaterialTop = 0
MD.FinalDepth = -10
' Create a manual toolpath
Dim MTP As MillManualToolPath
Set MTP = MD.ManualToolPath(100, 100, 0)
' Create element without attributes
MTP.Add2DLine 200, 100
' set multiple attributes for next element
MTP.SetAttributeForNextElement "MultiAttribute1", 1
MTP.SetAttributeForNextElement "MultiAttribute2", 2
MTP.SetAttributeForNextElement "MultiAttribute3", 3
' Create a new element. All attributes will be assigned to it
MTP.Add2DArcPointCenter 200, 200, 200, 150, False
' New element, create a new set of attributes
MTP.SetAttributeForNextElement "MultiAttribute4", 4
MTP.SetAttributeForNextElement "MultiAttribute5", 5
MTP.SetAttributeForNextElement "MultiAttribute6", 6
' Create a new element. All new attributes will be assigned to it
MTP.Add2DLine 100, 200
Dim PS As Paths
MTP.Finish
End Sub
```


New Methods to Help Optimise Add-Ins

Drawing Object

Method: GetGeosFrom(long Index)

Returns: Paths Collection containing all the paths in the Drawing from Index onwards.

If an add-in creates or imports geometries and needs to apply some processing on the geometries most recently added, then this new API call can be a more efficient method than getting a collection of all geometries in the drawing, especially if the Drawing contains a large number of geometries.

Example

```
` Add 10 random rectangles to the Drawing
Public Sub CreateGeos(drw As Drawing)
    Dim i As Integer
    Dim x As Integer
    Dim y As Integer
    ' Create some random rectangles
    For i = 1 To 10
        x = CInt(Int((100 * Rnd()) + 1))
        y = CInt(Int((100 * Rnd()) + 1))
        drw.CreateRectangle x, y, x + 100, y + 100
    Next i
End Sub

Public Sub OriginalMethod()
    Dim count As Integer
    Dim i As Integer
    Dim drw As Drawing
    Dim Start, Finish As Double
    Set drw = App.ActiveDrawing
    drw.Clear True, True, True, True, True, True, True, True
    drw.ScreenUpdating = False
    Start = Timer
    For i = 0 To 1000
        ' Get count of pre-existing geometries
        count = drw.GetGeoCount
        ' Call some method(s) to create / import some new geometries
        Call CreateGeos(drw)
        ' Get a collection of all geometries
        Dim AllPaths As Paths
        Dim APath As Path
        Dim j As Integer
        Dim total As Integer

        Set AllPaths = App.ActiveDrawing.Geometries
        total = AllPaths.count
        For j = (count + 1) To total
            Set APath = AllPaths.Item(j)
            APath.ColorRGB = 255
        Next
    Next
    drw.ScreenUpdating = True
    drw.RedrawShadedViews
    Finish = Timer
    MsgBox "Took " & (Finish - Start) & " seconds"
End Sub
```

```

Public Sub Optimized()
    Dim count As Integer
    Dim i As Integer
    Dim Start, Finish As Double
    Dim drw As Drawing
    Set drw = App.ActiveDrawing
    drw.Clear True, True, True, True, True, True, True, True
    drw.ScreenUpdating = False
    Start = Timer
    For i = 0 To 1000
        ' Get count of pre-existing geometries
        count = drw.GetGeoCount
        ' Call some method(s) to create / import some new geometries
        Call CreateGeos(drw)
        ' Get a collection of the new geometries
        Dim newPath As Paths
        Dim NewPath As Path
        Set newPath = App.ActiveDrawing.GetGeosFrom(count + 1)
        For Each NewPath In newPath
            NewPath.ColorRGB = 255
        Next
    Next
    drw.ScreenUpdating = True
    drw.RedrawShadedViews
    Finish = Timer
    MsgBox "Took " & (Finish - Start) & " seconds"
End Sub

```

In the Optimized() method, the new call 'GetGeosFrom' will create a collection of just the Paths we need compared to OriginalMethod () method's use of drw.Geometries that returns a collection of everything – which could be thousands of geometries. In OriginalMethod(), the loop will take longer and longer to execute as the number of geometries in the Drawing increases.

Method: GetToolPathsFrom(long Index)

Returns: Paths Collection containing all the tool paths in the Drawing from Index onwards.

If an add-in needs to apply some processing on the tool paths most recently added, then this new API call can be a more efficient method than getting a collection of all tool paths in the drawing, especially if the Drawing contains a large number of tool paths.

See GetGeosFrom for an example.

Method: GetSplinesFrom(long Index)

Returns: Splines Collection containing all the splines in the Drawing from Index onwards.

If an add-in needs to apply some processing on the splines most recently added, then this new API call can be a more efficient method than getting a collection of all splines in the drawing, especially if the Drawing contains a large number of splines.

See GetGeosFrom for an example.

Appendix B - 64-bit VBA Macros

Introduction

With the introduction of the 64-bit version of Alphacam, it was necessary to upgrade to Microsoft Visual Basic for Applications 7.0 (VBA7). VBA 7 is now used in the 32-bit and 64-bit Alphacam and AlphaEdit applications.

All Alphacam add-ins installed with Alphacam 2015 R2 have been updated to work with Visual Basic 7.0 in both 32-bit and 64-bit Alphacam (with the exception of APM and CDM).

If you develop your own add-ins and want them to work with 64-bit Alphacam they must be tested because they may not work without modification. Specifically if your add-in uses any of the Windows API functionality it will require some updates before it can run.

The main reason why VBA code needs to be updated is because many of the Windows API function parameter types are pointers (memory location addresses) and these need to be stored as 64-bit values when running in a 64-bit environment, and 32-bit values in a 32-bit environment. Lots of existing VBA 6 code uses 32-bit values to store pointers which won't work in a 64-bit environment.

In VBA 7, you must update all Windows Application Programming Interface (API) statements (Declare statements) to work with the 64-bit version. You may also need to update the definitions of the user-defined types that are used by these statements as well as code that calls these functions.

Microsoft have a download which includes Windows API Declarations and Constants for Visual Basic which has been updated for 64-bit here: <http://www.microsoft.com/en-us/download/details.aspx?id=9970>

After you download and install it then you can refer to the file here: "C:\Office 2010 DeveloperResources\Documents\Office2010Win32API_PtrSafe\Win32API_PtrSafe.TXT". It contains a lot of useful declarations you can copy and paste from.

Another useful article can be found here: [https://msdn.microsoft.com/en-us/library/ee691831\(office.14\).aspx](https://msdn.microsoft.com/en-us/library/ee691831(office.14).aspx)

Fixing Declare Statements

The first thing you are likely to have to do is to update Declare statements with the **PtrSafe** keyword. This tells VBA 7 that you believe the function call is safe to call in a 64bit environment (it doesn't mean that it necessarily is though!)

In an existing VBA 6 macro you may have a statement like this...

```
Private Declare Function GetUserDefaultLCID Lib"kernel32" () As Long
```

It declares that we want to use a function called GetUserDefaultLCID inside Kernel32.dll that takes no parameters and returns a Long. This function doesn't use any pointer parameters so it can be simply updated to include the PtrSafe keyword...

```
Private Declare PtrSafe Function GetUserDefaultLCID Lib "kernel32" () As Long
```

No other changes are required (unless you need to make your macro work with VBA 6 and VBA 7 - see later).

Fixing functions with pointer types

Here is another example from a VBA 6 macro...

```
Private Declare Function RegOpenKeyEx Lib"advapi32.dll" Alias "RegOpenKeyExA"
(ByVal hKey As Long,ByVal lpSubKey As String, ByVal ulOptions As Long, ByVal
samDesired As Long,phkResult As Long) As Long
```

This function is used to open a registry key. Referring to the Win32API_PtrSafe.txt we can update this to the following...

```
Private Declare PtrSafe Function RegOpenKeyEx Lib "advapi32.dll"
Alias"RegOpenKeyExA" (ByVal hKey As LongPtr,ByVal lpSubKey As String, ByVal
ulOptions As Long, ByVal samDesired As Long,phkResult As LongPtr) As Long
```

In this case, in addition to the **PtrSafe** keyword, two of the parameters are pointers and need to change type from 'Long' to 'LongPtr'. 'LongPtr' is a special type that will automatically be replaced by a type large enough to hold a pointer value for the current run-time environment. In other words if you are running 64bit Alphacam, a 'LongPtr' will be a 64bit value, and if you are running 32bit Alphacam it will be a 32bit value.

The code that uses this function needs to be updated too. In VBA 6 the original code surrounding this function was...

```
Public Function gb_ExportRegKey(ByVal lKeyRoot As KeyRoot,ByVal sKeyPath As String,
ByVal sFileName As String) As Boolean
Dim lngHKey As Long
Dim lngRet As Long
lngRet = RegOpenKeyEx(lKeyRoot, sKeyPath, 0&,KEY_ALL_ACCESS, lngHKey)
```

In VBA 7 it should be...

```
Public Function gb_ExportRegKey(ByVal lKeyRoot As KeyRoot,ByVal sKeyPath As String,
ByVal sFileName As String) As Boolean
Dim lngHKey As LongPtr
Dim lngRet As Long
lngRet = RegOpenKeyEx(lKeyRoot, sKeyPath, 0&,KEY_ALL_ACCESS, lngHKey)
```

In this case, just the type of the HKEY returned by the RegOpenKeyEx function needs to be updated from a Long to a LongPtr.

VBA 6 Backwards compatibility

As mentioned above there is some additional work to do if you wish to maintain backwards compatibility with VBA 6. If you install a macro in the \LicomDir\VBMacros folder for example, it may be used by Alphacam 2015R1 (or older) as well as Alphacam 2015 R2. Unfortunately the new keywords 'PtrSafe' and 'LongPtr' were introduced in VBA 7, and VBA 6 doesn't know what they mean.

To allow your VBA code to work in both VBA 6 and VBA 7 it is necessary to use a Conditional Compilation test for VBA 7 as follows: -

```
#If VBA7 Then
Private Declare PtrSafe Function GetUserDefaultLCID Lib "kernel32" () As Long
Private Declare PtrSafe Function RegOpenKeyEx Lib "advapi32.dll" Alias
"RegOpenKeyExA" (ByVal hKey As LongPtr, ByVal lpSubKey As String, ByVal ulOptions
As Long,ByVal samDesired As Long, phkResult As LongPtr) As Long
#Else
Private Declare Function GetUserDefaultLCID Lib"kernel32" () As Long
```

```
Private Declare Function RegOpenKeyEx Lib"advapi32.dll" Alias "RegOpenKeyExA"  
(ByVal hKey As Long,ByVal lpSubKey As String, ByVal ulOptions As Long, ByVal  
samDesired As Long,phkResult As Long) As Long  
#EndIf
```

It is also necessary to 'wrap' any instances of 'LongPtr' in the code. In the case of the gb_ExportRegKey function above it would need to be modified as follows:

```
Public Function gb_ExportRegKey(ByVal lKeyRoot As KeyRoot,ByVal sKeyPath As String,  
ByVal sFileName As String) As Boolean  
#If VBA7 Then  
Dim lngHKey As LongPtr  
#Else  
Dim lngHKey As Long  
#End If  
Dim lngRet As Long  
lngRet = RegOpenKeyEx(lKeyRoot, sKeyPath, 0&,KEY_ALL_ACCESS, lngHKey)
```

Testing Macros

Attempting to load macros into Alphacam that are not compatible with 64bit VBA 7 will result in one or more warning dialog boxes appearing. The last of these dialog boxes contains a summary of all the macros that failed to compile. You can copy this information to the clipboard and optionally choose to disable these macros.

You can use the VBA Editor Debug | Compile <Macro name> command to help you find the code and declarations that need to be updated.

Active X Components

If you have used any of Microsoft's Active X objects to add advanced User Interface controls to your VBA Macro you will unfortunately run into additional difficulties. Microsoft has not made 64 bit versions of these components (see [https://msdn.microsoft.com/en-us/library/office/ee691831\(v=office.14\).aspx#odc_office2010_Compatibility32bit64bit_ActiveXControlCOMAddinCompatibility](https://msdn.microsoft.com/en-us/library/office/ee691831(v=office.14).aspx#odc_office2010_Compatibility32bit64bit_ActiveXControlCOMAddinCompatibility))...

"This includes the common controls of MSComCtl (TabStrip, Toolbar, StatusBar, ProgressBar, TreeView, ListViews, ImageList, Slider, ImageComboBox) and the controls of MSComCtl2 (Animation, UpDown, MonthView, DateTimePicker, FlatScrollBar)"

Unfortunately there aren't many options ... either removing the component (and simplifying the interface) or seeking an alternative solution. Regarding alternatives ... some of the VBA components inside Alphacam use Codejock's Xtreme Suite Pro Active X components, which are available in 32 and 64bit. This is a commercial product so there is cost involved. It is recommended that you contact support if you would like more information.

Appendix C – C# Add-Ins

Alphacam will search for C# DLLs in two specific folders; “Dot NetAdd-Ins” in the same folder as the Acam.exe and “...\LicomDir\DotNetAdd-Ins”. Each C# add-in must include a text file with the extension “.acamaddin” which Alphacam will read to determine information about the C# add-in, including how and when it should be loaded.

A typical .acamaddin file contains the following: -

\$1 ' Name of .Net DLL to load, no path or extension.

ExampleAddIn

\$2 ' 1 to always load, 0 to show in Utils | Add-Ins dialog box

0

\$3 ' NoDefaultLoad, = 1 don't load if user registry entry not set. (Only used if \$2 = 0)

1

\$4 ' 1 if this is an Extra Cost Option

0

\$10 ' Load for Mill, 1 = yes, 0 = no

1

\$20 ' Load for Router, 1 = yes, 0 = no

1

\$30 ' Load for Stone, 1 = yes, 0 = no

1

\$40 ' Load for Lathe, 1 = yes, 0 = no

1

\$50 ' Load for Profiling, 1 = yes, 0 = no

1

\$60 ' Load for Wire, 1 = yes, 0 = no

1

The entry under \$1 must be the name of the C# DLL without extension. The DLL must be in the same folder as the .acamaddin file itself. (The files can be in sub-folders of one of the “DotNetAdd-Ins” directories).

Each C# add-in must include a reference to “Interop.AlphaCAMMill.dll” (which is installed in the same folder as acam.exe), and implement a public class called “AlphacamEvents” with a constructor defined as taking a single IAlphaCamApp parameter. Alphacam will call this constructor when the add-in is loaded.

From the IAlphaCamApp interface the add-in can retrieve the Frame interface and use the method “CreateAddInInterface” to create an AddInInterfaceClass which can be used to add event handlers such as InitAlphacamAddIn, BeforeInputCad, AfterMachining, etc. The InitAlphacamAddIn event handler can be used to add new commands to the Alphacam interface. In addition to supporting to events, an add-in can respond to various notifications such as when different kinds of entities (geometries, splines, surfaces, etc.) are added or modified. The “CreateAddInNotifications” method on the Frame interface can be used to create various notification event handlers.