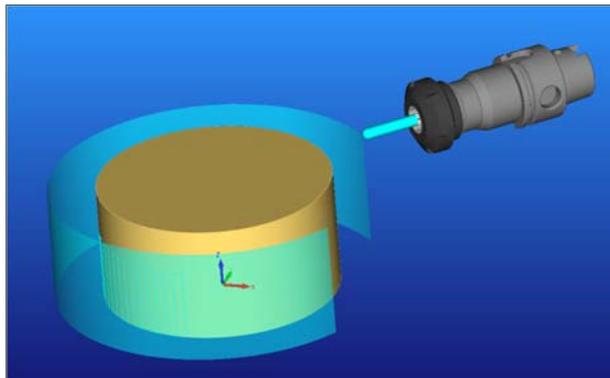


alphacam

2017 R1

新機能紹介



Alphacam 2017 R1 の新しい機能を紹介します

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重要事項

対応 OS

Alphacam 2017 R1 が対応しているOSを以下に示します。 **OSは64ビットの必要があります。**

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 2017 R1 をご使用になるには、保守終了日が **2016 年 10 月** 以降の必要があります。

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。

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 2017 R1 のインストール

Alphacam 2017 R1 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。 **OS は 64 ビットの必要があります。**

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 R2 のリボンバー設定読み込み

Alphacam 2017 R1 は 2016 R2 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.



ライセンス

CLS ネットワークライセンスプロファイル

ネットワークライセンスプロファイルを使用すると、使用するシステム、モジュール設定を保存することができ、以前に保存した設定を簡単に復元することができます。

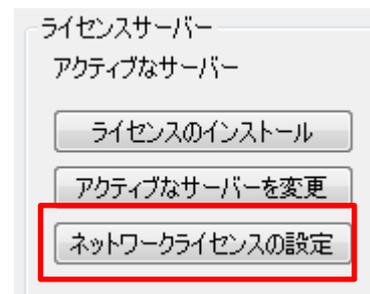
- 全ネットワークユーザーは自分用のプロファイルを作成できます。また保存したプロファイルをコピーして、他のユーザーが同じプロファイルを利用することもできます。

ライセンスプロファイルはネットワークライセンスの設定画面で使用することができます。

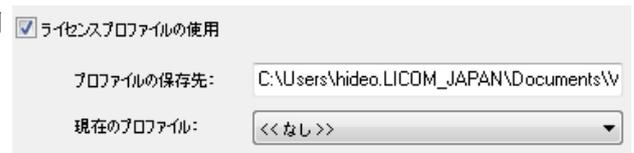
ライセンスマネージャダイアログ

ネットワークライセンスの設定 ボタンが追加されました。

スタンドアロンライセンスのときは使用できません。

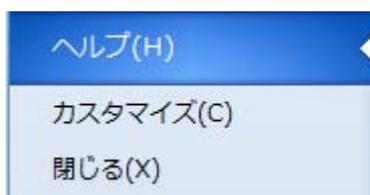


ネットワークライセンスの設定画面にはプロファイルの使用を選択できるチェックボックスが追加されました



サポートツール

ファイルタブのヘルプにチームビューの起動コマンドが追加されました。Alphacam からチームビューを起動して、ただちに弊社の技術スタッフのサポートを受けられます。インターネット接続が必要です。セキュリティの厳しいネットワークでは接続できないことがあります。



サポートスタッフと画面を共有するにはリモートサポートソフト

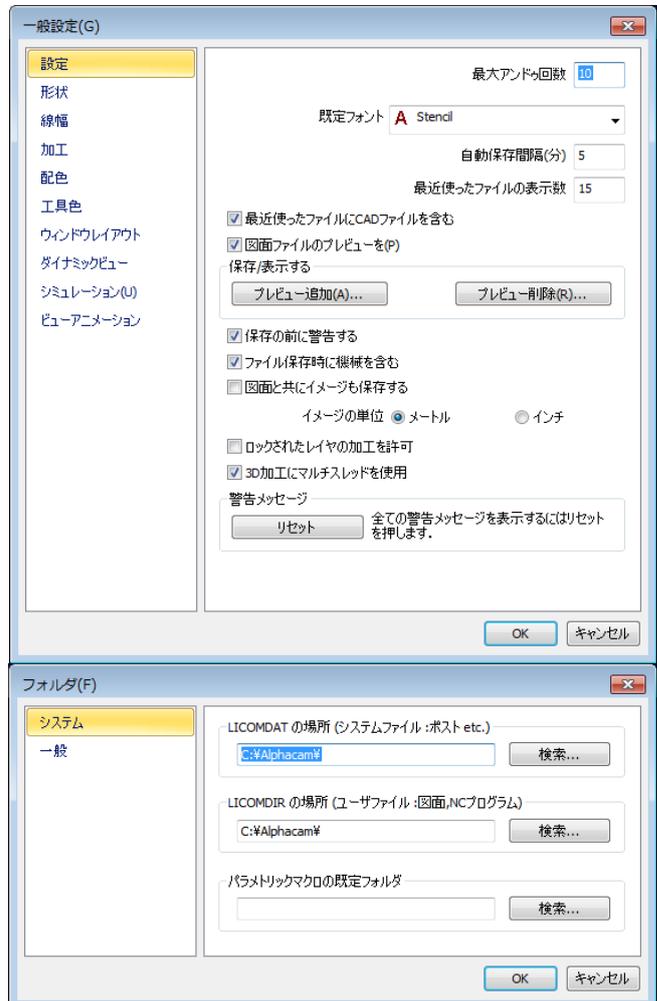
インターフェース

環境設定 – 一般設定ダイアログなど

- 環境設定 – 一般設定
- 環境設定 – フォルダ

のダイアログが更新され、設定しやすくなりました。

左のタブによって切り替えを行います。

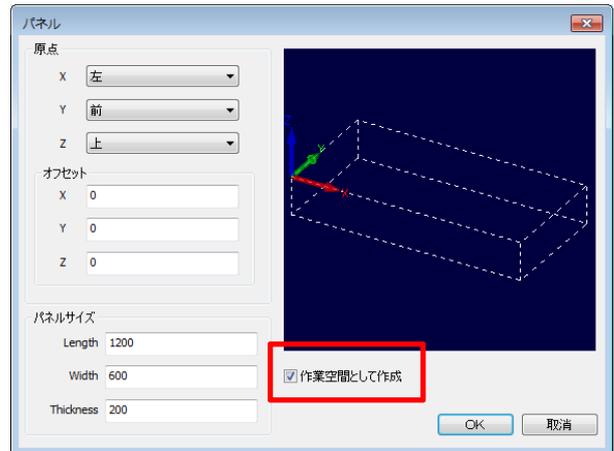


パラメトリックスケッチャー – 作業空間としてパネルを作成

パネル作成のダイアログに、「作業空間として作成」オプションが追加されました。

注記: 作業空間としてパネルを作成した場合、既存の作業空間は削除されます。

プロジェクトマネージャの挿入ページでパラメトリックスケッチャーを挿入する場合は、既存の作業空間が優先され、パラメトリックスケッチャーの作業空間は作成されません。



作業平面ページ – 作業平面プロパティの簡易編集コマンド

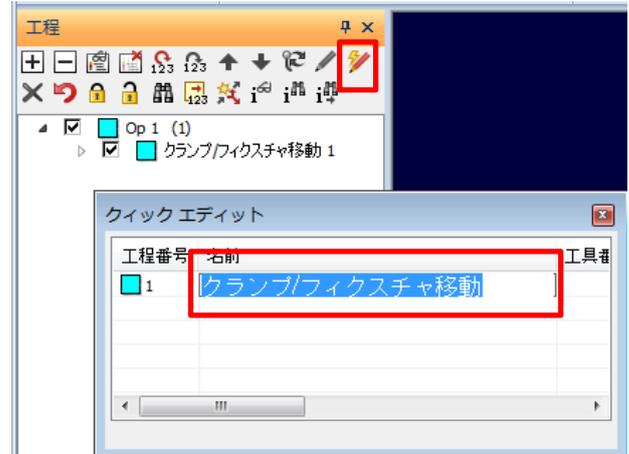
プロジェクトマネージャの作業平面のページに「作業平面のプロパティ」コマンドが追加されました。

作業平面の名前とオフセットが一覧形式で表示され、この画面で編集することができます。



工程ページ-クランプ/フィクスチャ移動の名前変更

クイックエディット画面で、クランプ/フィクスチャ移動の工程名を変更することができます。

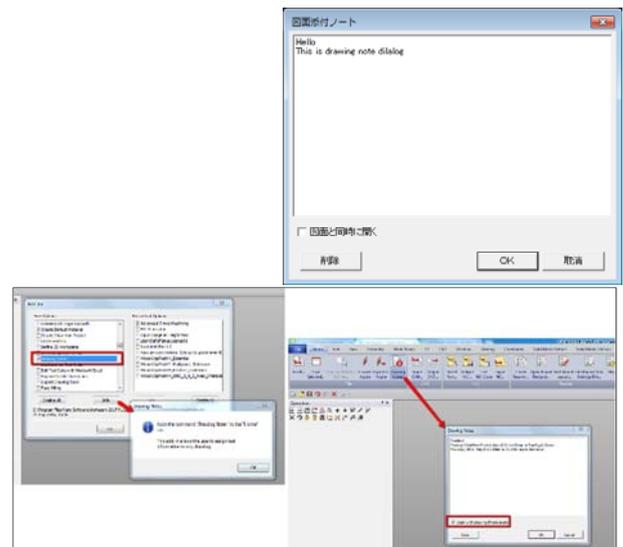


図面添付ノート - 新オプション - 図面と同時に開く

新オプション - 図面と同時に開く

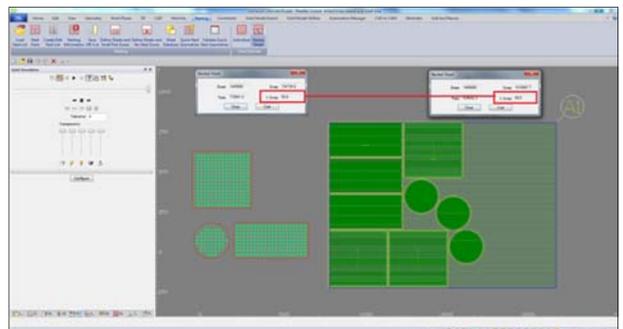
このオプションをチェックすると、開いたファイルが図面添付ノートを含んでいる場合に、図面添付ノート画面が表示されます。

注記: 図面添付ノートは無償アドインなので、アドイン画面で使用する/しないを切り替えることができます



面積見積もり - ネスト済みシート - 穴あけをスクラップ計算に加算

面積見積もりは、穴あけをスクラップとして計算するようになりました。



加工

円筒面沿い走査線 – 切削の幅の指定方法の改善

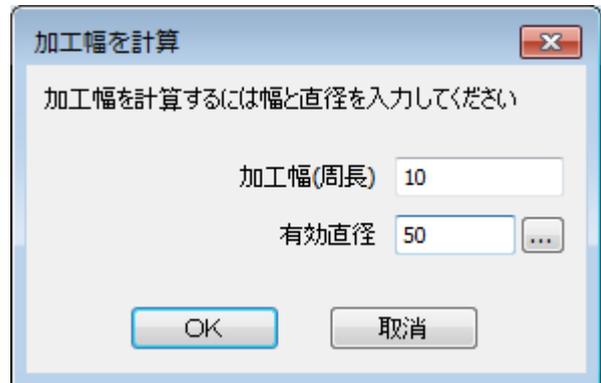
切削の幅の角度の指定方法に、より簡単な方法が追加されました。

切削の幅の指定の横の  ボタンをクリックすると、新しい設定ダイアログが表示されます。

この画面では円周でのピッチ長さを指定し、Alphacam が自動的に角度に変換します。

この画面では有効直径を取得するための  ボタンが用意されています。このボタンをクリックし、円周上と見なす点をクリックすると、自動的に有効直径が入力されます。

この画面で **OK** をクリックすると、元の画面の角度には計算結果が入ります。



加工ダイアログ – 3D 加工コマンド

加工ダイアログは今後も継続的に改善されていきますが、その一段階として 3D 加工コマンドのダイアログが改善されました。

加工の軸数と加工方法をドロップダウンのリストから選択します。

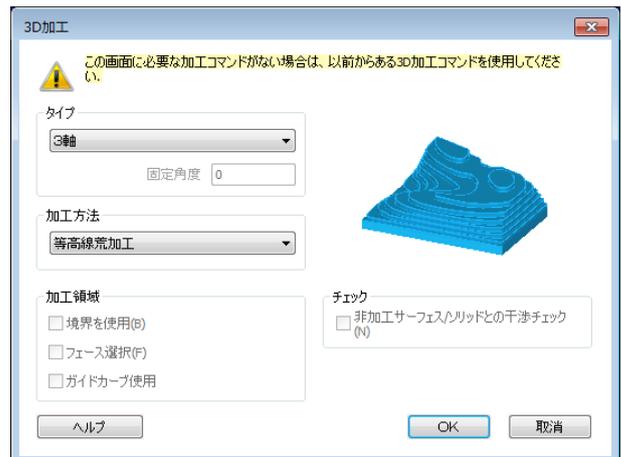
選択された加工方法に応じてアニメーションが連動します。

以下の加工コマンドが 3D 加工コマンドで使用できます。

- 交差線沿い
- カस्पー一定
- 円筒面沿い走査線
- 円筒面回り仕上げ
- ドライブカーブ
- 等高線荒加工

そのほかの加工コマンドは 3D サーフェス加工、3D ソリッド加工コマンドから作成してください。

3D サーフェス加工、3D ソリッド加工は将来的に 3D 加工コマンドに統合されます。



円筒面沿い走査線 - ダイナミックプレビュー

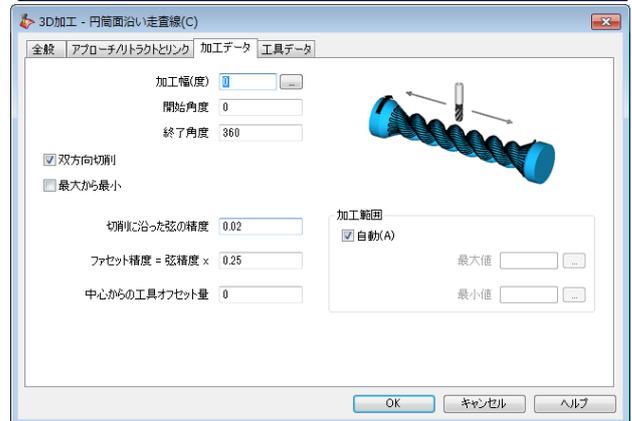
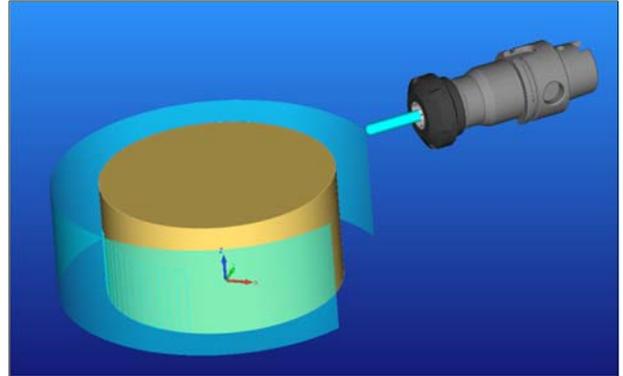
円筒面沿い走査線のダイアログで、加工範囲を設定する際に、数値設定に連動して加工範囲のプレビューが表示されます。

数値範囲は製品の回りに半透明で表示されます。

注記: ダイナミックプレビューはデフォルトで表示する設定ですが、表示させないこともできます

設定変更箇所

ホーム | 設定 | 環境設定 | 一般設定 | 加工タブ
「加工範囲のダイナミックプレビューを使用」

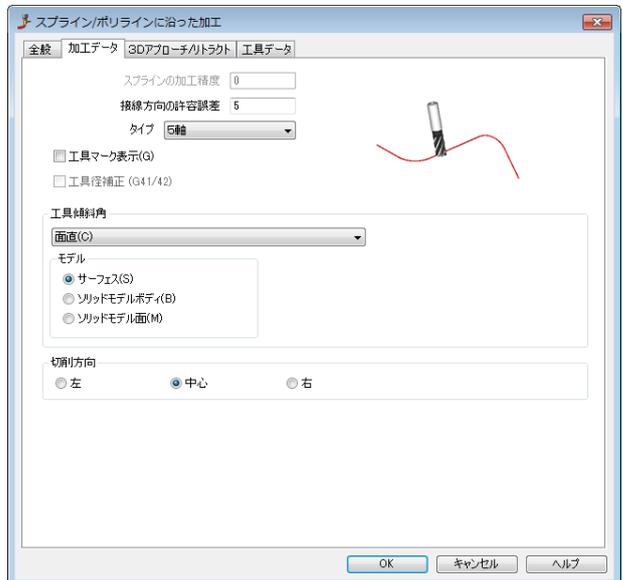


スプライン/ポリライン沿い加工ダイアログの更新

スプライン/ポリライン沿い加工のダイアログが更新され、タブ切り替え方式になり、イメージやツールチップが追加されました。

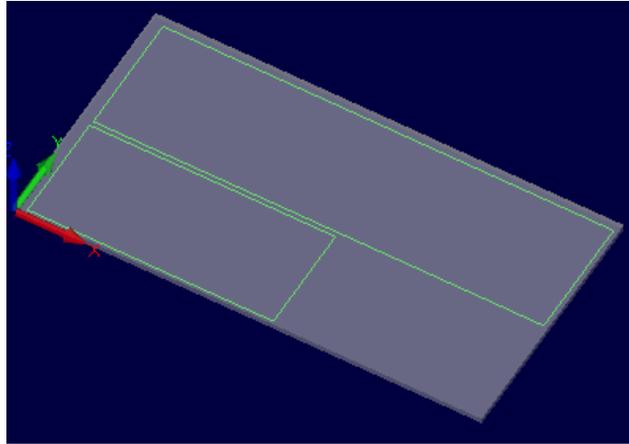
注記: 加工コマンドから工程を作成するときは、ダイアログは2回に分かれて表示されます。

工程を編集するときは全ダイアログのページがタブ切り替えで表示されます。

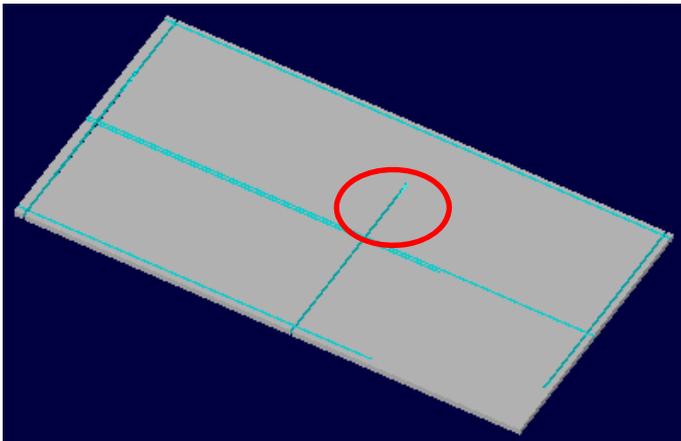


ソーカット - 干渉の検出と回避

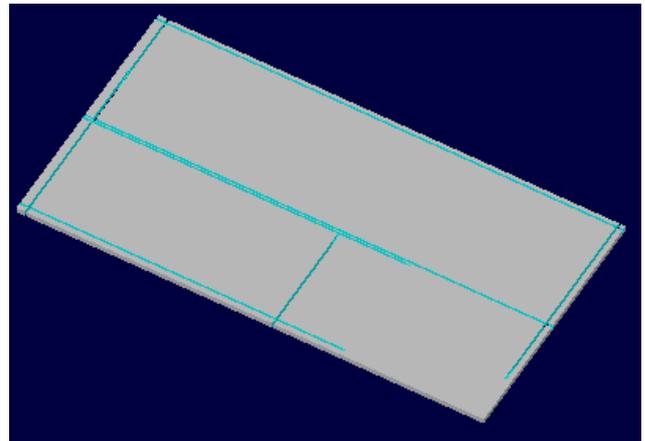
ソーカットを行う製品が接近して配置してある場合、ある製品の加工で隣の製品を切り込むことがあります。新しいソーカットコマンドは工程作成に選択した形状に対する干渉を検出し、隣の製品を切り込む場合は工具経路をトリムし、干渉をなくします。



製品形状



Alphacam2016R2



Alphacam2017R1

ドリル/タップ 自動 Z 加工深さの詳細な指定

自動 Z を使用したドリル/タップコマンドに詳細な加工深さ指定ができるようになりました。

特にフィーチャー抽出された止まり穴に対し、工具の先端角度と穴の角度が異なる場合に、「自動」を選択すると穴底の形を壊さない方を選択します。

この設定は次のサイクルで使用できます:

- ドリル
- ペッキング
- ボーリング

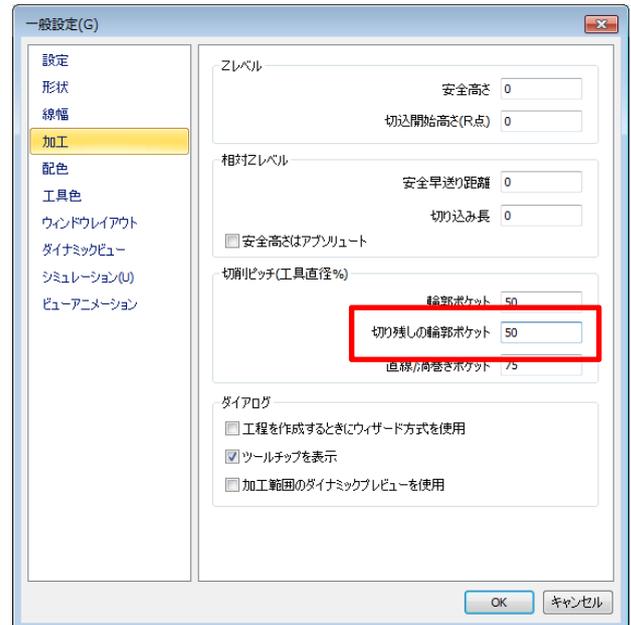
穴の深さ - 自動Z
<input checked="" type="radio"/> ドリル先端
<input type="radio"/> ドリルの肩
穴の深さ - フィーチャーの止まり穴
<input checked="" type="radio"/> 自動
<input type="radio"/> ドリルの肩
<input type="radio"/> ドリル先端
穴の深さ - フィーチャーの貫通穴
<input checked="" type="radio"/> ドリルの肩
<input type="radio"/> ドリル先端

環境設定 - 一般設定 - 加工 - 切削ピッチ 切り残しの輪郭ポケット

切り残しの輪郭ポケットの切削幅のデフォルト%値が追加されました。

ポケット加工での「切り残し」オプションを使用したときのデフォルトの切削幅としてこの設定が使用されます。

注記: この設定のデフォルトは 50 です

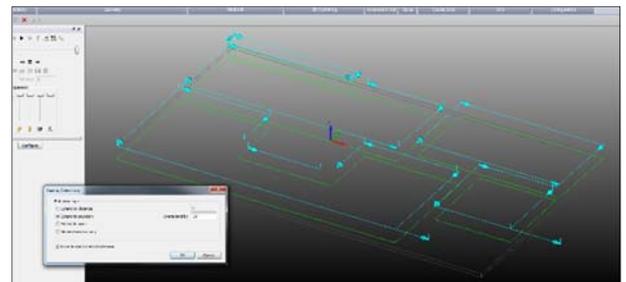


ソーカット - 加工編集 - ノコギリ工程延長

ノコギリ工程を手動で編集するコマンドが加工 | 加工編集 | ノコギリ工程延長 が追加されました

距離や境界まで延長させることや、進行方向を反転させることができます。

注記: 工程編集を行うと手動での編集は全て破棄され、オリジナルの設定に従って再計算をします。



2D加工 – 荒加工サイクル

インストール後にアドイン設定が必要になります。

【インストール】 DVD\ExtraAdd-Ins\荒加工サイクル\setup.exe

【アドイン設定】 アドイン > アドイン > 無償オプション >
 荒加工サイクル

【輪郭追込み加工】

複数形状に対しての一括輪郭追込み加工

【開ポケット加工】

開形状に対する境界からの進入スタイルでのポケット加工

【輪郭荒加工】

閉形状に対する外側境界からの進入スタイルでのポケット加工

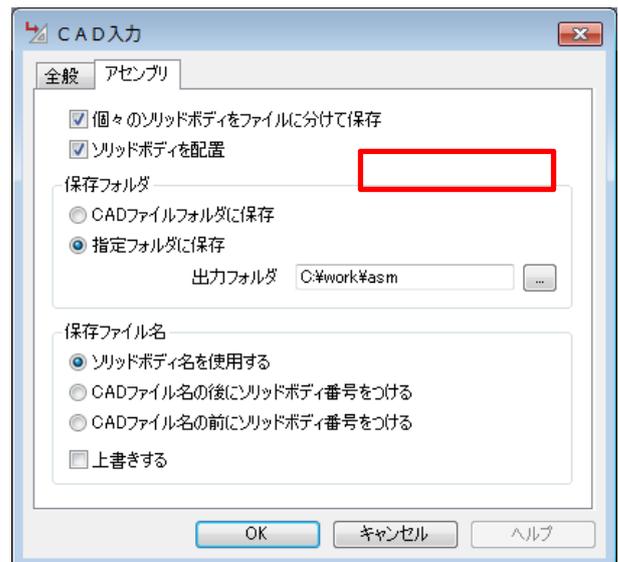
【隅加工】

指定半径以下のコーナ部を自動認識して複数回の小径工具によるコーナ追込み加工



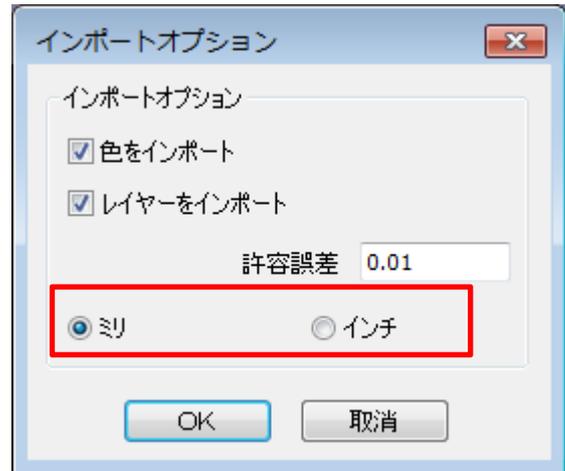
CAD データ入力 – ソリッドモデルアセンブリ – ファイルに分割して保存

CAD データ入力には、新しく「アセンブリ」タブが追加されました。このタブでは読み込んだアセンブリを指定フォルダに個別にファイル保存する設定や保存ファイル名の指定方法の設定ができます。



CAD データ入力 – Postscript, PDF , AI ファイルの単位指定

Postscript, AI, PDF ファイルを読み込むとき、単位の指定ができるようになりました。これにより読み込み後のスケーリング操作が不要になります。

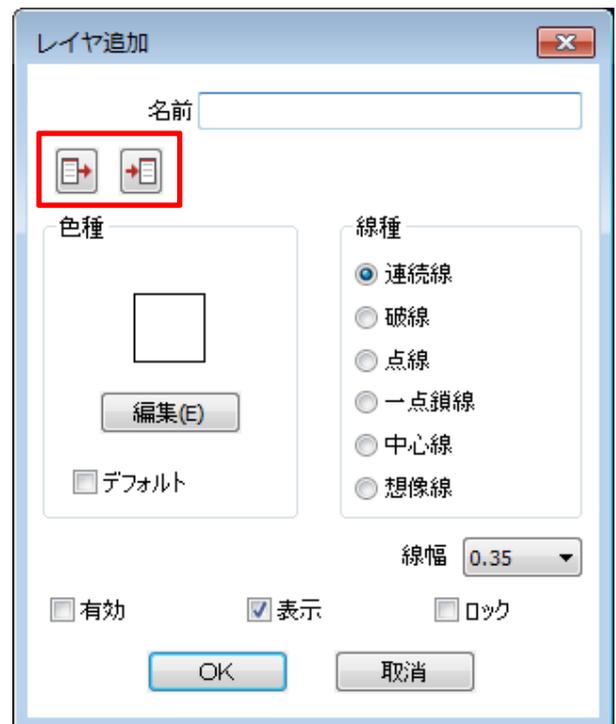


レイヤーデフォルト設定の保存と取得

レイヤーの追加、編集ダイアログにはデフォルトの設定と取得ボタンが追加されました。

デフォルトとして扱われる項目は、色、線種、線幅、有効状態、表示状態、ロック状態です。

注記: デフォルトの設定と保存は素材、機械設定レイヤーには使用されません。



ソリッドワークス – アセンブリファイルのサポート

これまでのバージョンの Alphacam はソリッドワークスのアセンブリからエクスポートする CadLink アドインが必要でした。

2017R1 バージョンの Alphacam はソリッドワークスのアセンブリを直接読み込むことができます。アセンブリから参照されるサブアセンブリやパートファイルにはアクセスできることが前提です。

注記: ソリッドワークスアセンブリローダーは CADLink が使える場合はそれを使います。CADLink が使えない場合に直接読み込みを行います。

ソリッドエッジ – CADlinks を使用してのアセンブリのサポート

ソリッドエッジアセンブリは CADLinks アドインを利用してインポートできるように改善しました。

ソリッドエッジと Alphacam が同じ PC にインストールされている場合は、CADLinks アドインをインストールし、CADlinks ボタンにより Alphacam に直接アセンブリをエクスポートできます。

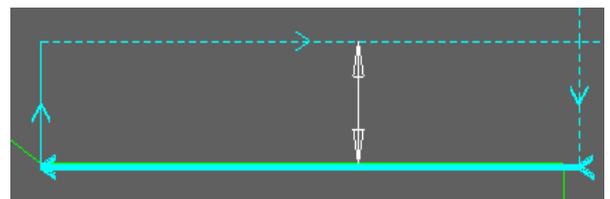
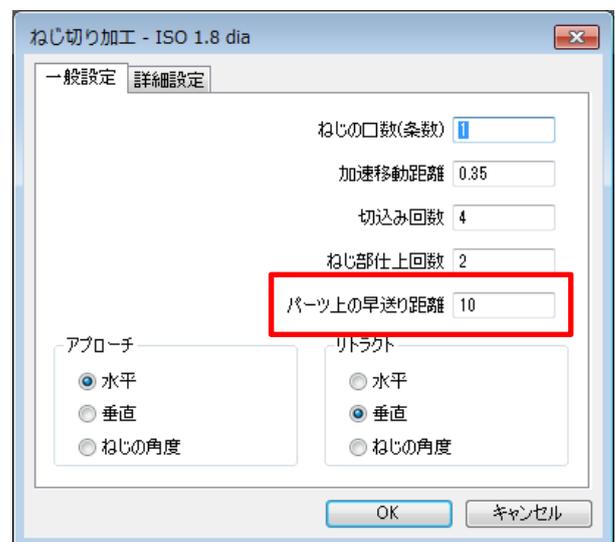
ソリッドエッジが Alphacam と別の PC にインストールされている場合も CADLinks アドインを使用し、Alphacam が読み込めるファイルを生成します。ファイルは .asm ファイルと同じフォルダに生成されるので、Alphacam がインストールされている PC にコピーして読み込ませます。

旋盤

ねじ切り – 安全距離の指定

パーツ上の早送り距離の設定が可能になりました。

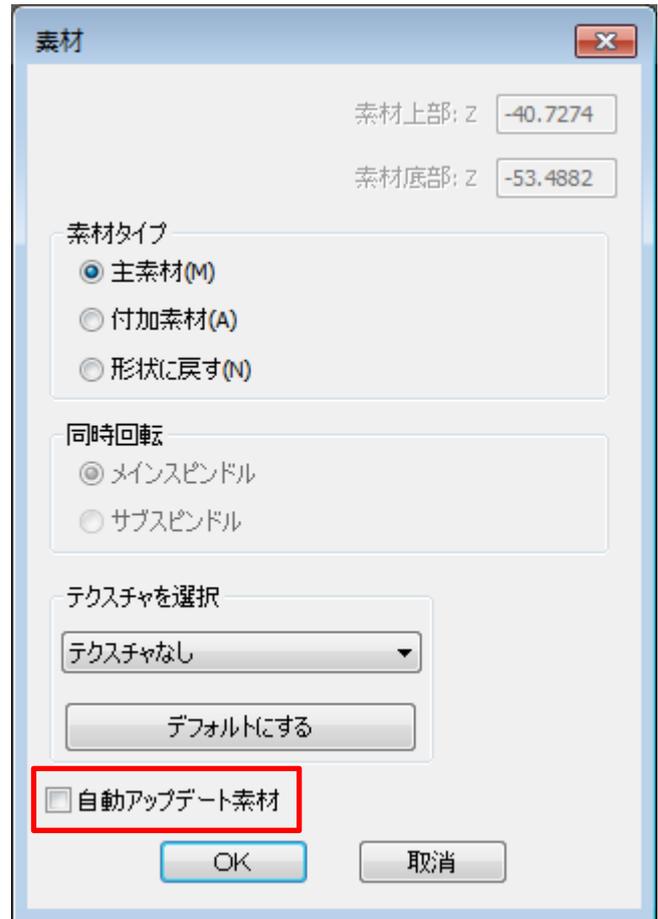
この設定により各ねじ切り切削を接続する早送りの、パーツからの距離を指定できるようになりました。



旋盤 – ミーリング素材設定 – 自動アップデート素材

ミーリング素材設定に自動アップデート素材の指定ができるようになりました。

この指定がされた素材は、3D 等高線荒加工コマンドで前工程までの加工を考慮します。



ルーター

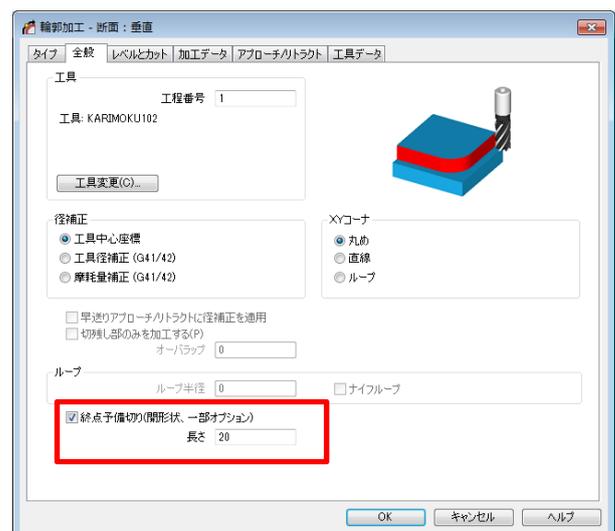
輪郭加工 – 終点予備切りオプションの追加

終点予備切りオプションが追加されました（開形状、または一部オプション）。

このオプションは先に加工終点側を指定された距離だけ切削方向とは反対の方向に加工します。

加工終点を工具が通過する際に、製品の割れなどが発生した場合に有効です。

注記: 閉じた形状には作用しません。



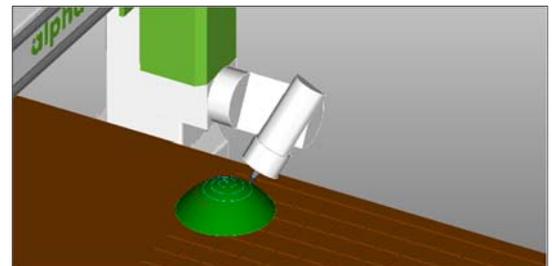
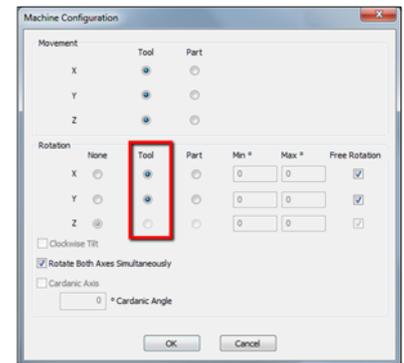
シミュレーション

5 軸パスの転送の高速化

Alphacam から外部シミュレータに 5 軸パスを転送する速度が向上しました。

シミュレーション – 機械設定 – 主軸側に AB 軸(X 軸回り + Y 軸回り)がある機械のサポート

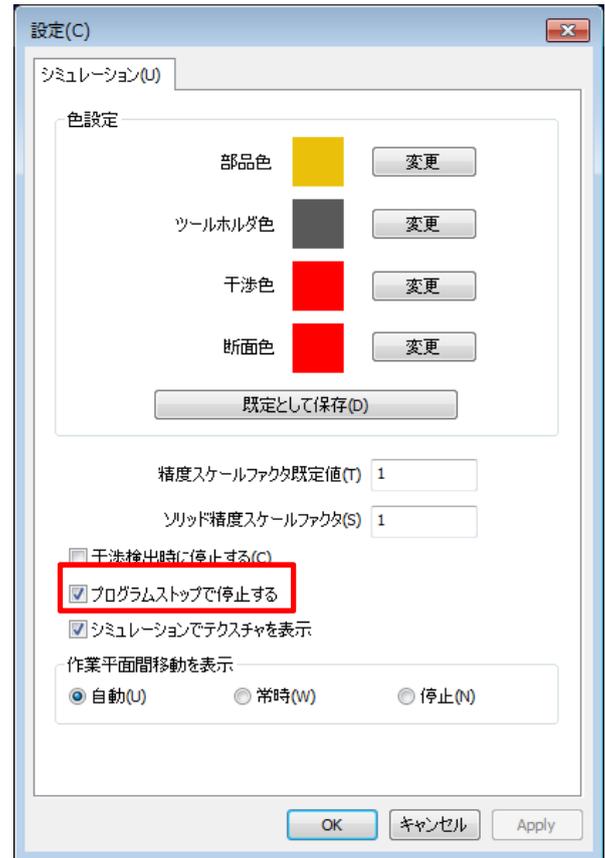
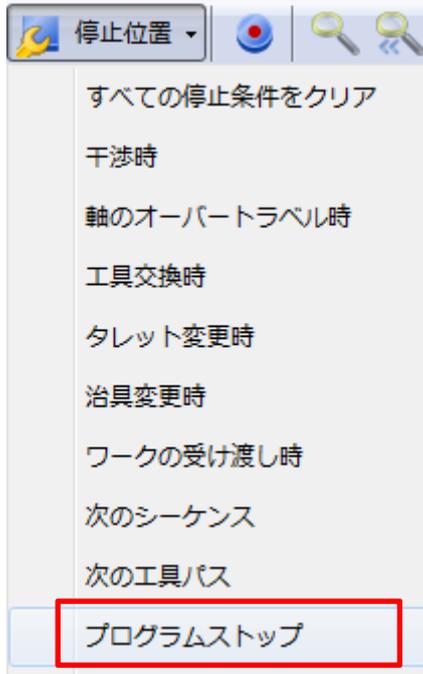
シミュレーションで XY 軸回りの回転がヘッド側、テーブル側でサポートされました。



プログラムストップ - シミュレーションの一時停止設定

プログラムストップでシミュレーションを一時停止にする設定が追加されました。

外部シミュレータでは、ツールバーのドロップダウンリストに設定があります。



シミュレーション - クランプの移動速度の指定

シェーディングシミュレーションで、クランプの移動速度はポストプロセッサで定義された早送り速度をデフォルトとして使用しますが、クランプ定義画面で指定できるようになりました。

The screenshot shows the 'クランプ定義' (Clamp Definition) dialog box. The '送り速度' (Feed Rate) field is highlighted with a red box and set to 200. Other fields include '名前' (Name) set to 'クランプ 1', '番号' (Number) set to 1, 'ソリッドタイプ' (Solid Type) set to '押し出し' (Extrusion), '上部Z' (Top Z) set to 0, and '底部Z' (Bottom Z) set to -20. The '移動可能' (Moveable) section has 'X', 'Y', and 'Z' checked. The 'ポップアップ可能' (Pop-up Possible) section has 'ポップアップ可能(U)' checked. The '送り速度' (Feed Rate) field is highlighted with a red box and set to 200. The 'OK' and '取消' (Cancel) buttons are at the bottom.

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)	ESA U	ESA U	ESA U	ESA U	SA	AU
IGES (バージョンなし)	ESA U	ESA U	ESA U	ESA U	SA	AU
Rhino 5.0	SA U	SA U	SA U	SA U	SA	AU
STL (バージョンなし)	SA U	SA U	SA U	SA U	SA	AU
SketchUp 2016	SA U	SA U	SA U	SA U	SA	AU
PostScript	ESA U	ESA U	U	ESA U	SA	AU
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 – 2017 1.0	A U	A U	A U	A U	A	U
Parasolid 9.0 – 29.0.137	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
保守期間限定(保守契約終了で使用できなくなります)						
Adobe PDF	ESA U	ESA U	U	ESA U	SA	AU
Adobe Illustrator	ESA U	ESA U	U	ESA U	SA	AU
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](http://Autodesk.com) 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 2017 R1** で行われました

ID	Description	Closing Notes
1234	Reports - Windows 8 / 8.1 / 10 - Reports are failing in multiple ways: Crashing, white screen, incorrect parts shown, incomplete reports of sheets.	Fixed.
1419	Splines - Create Spline, Convert Splines to Lines/Arcs - Geometry moves after saving and re-opening a file when a work plane is used with Reverse Current work plane checked and the Undo command is used. Unexpected behaviour and Alphacam crashes under certain conditions.	Fixed.
1941	Stone - Cut with Disk - Horizontal - Roughing - Incorrect profile achieved.	Fixed. Dialog and cycle enhanced.
1994	Sheet Database - With the Reverse Side Nesting add-in turned on, there is no XY prompt when inserting a sheet into the drawing.	Fixed.
2231	Automation Manager - When nesting using sheets of multiple sizes within any one material, an incomplete nest list is produced.	Fixed.
2488	Operations Manager - Clamp move - Unable to move operations above or below the clamp move (if present).	Fixed.
2527	Stone - Cut Shape - Dialog box refers to rough finish; not cut shape.	Fixed.
2627	Automation Manager - Material types are not saved when Alphacam is re-opened after processing.	Fixed.
2634	Automatic Extraction - Improved contour extraction on parts with curved edges.	Enhancement.
2873	Lathe - Solid Simulation - Simulation freezes on specific operation. An error message is displayed when using dialogs, even when no changes are made.	Fixed.
2875	Machine Configuration - Clamps/Fixtures: <ul style="list-style-type: none"> Edit Clamps/Fixtures - The radial menu is displayed when right-clicking, cancelling the command. Edit Clamps/Fixtures - clicking a clamp or right-clicking and selecting Show Clamp in Project Manager does not work for clamps in the drawing. Edit Clamps/Fixtures and Find commands are unreliable at finding clamps in the machine configuration if parts of the tree are collapsed. 	All Fixed.
2891	Automation Manager - Layer Mapping Setup - Feature extract layers are not translated.	Fixed.
2961	Project manager - Edit Lead in/Lead out - If the user presses enter, tab order is set to get default. Change this to Apply.	Fixed by adding an 'Apply' button on the Dialog.
3023	Simulation - Tools imported from partner software TDM are flat.	Fixed.
3130	Machine - Rough/Finish - Helical - Auto Z - When the depth of cut value is reduced, processing takes longer than expected.	Fixed.
3174	Automation Manager - Nest list - The previous nest list is remembered on the next individual part job.	Fixed.
3202	Text - Thai characters do not work when using Thai TTFs.	Fixed.
3220	Automation Manager - Files not found when path directs to LICOMDAT or LICOMDIR folders.	Fixed.

3255	Save solid bodies - Automatically align each body - Solid parts are always aligned if Align On Import is selected on Feature Configurations.	Fixed.
3269	Machine - Pocketing - Take account of previous machining - If width of cut is set to over 50%, the cutter pockets everything.	Fixed. *The toolpath must be updated to apply this fix.
3270	Lathe - Simulation - Program Stop - When program stop is used, it is affecting simulation.	Fixed.
3366	Drill / Machine Holes - Spiral - Machine holes cannot be edited and crash if start point is centre.	Fixed.
3393	Machine - Rough or Finish - If a user-defined tool is used and the number of cuts is changed, the cut order changes.	Fixed.
3397	Insert Parametric Sketch - Preview not updated when editing a variable.	Fixed.
3411	Machine - Rough or Finish - Links - Looped links do not work with open elements.	Fixed.
3419	API - Drawing.MoveNestedSheetAndPartsToDrawing does not move nested sheet bubble geometry to new drawing.	Fixed.
3464	Lathe - Part-Off - If the value for "Groove Side Clearance" is set to 0, the toolpath changes. When the operation is edited again however, the field is populated with a default value.	Fixed.
3512	Reports addin - COM objects are not being released.	Enhanced memory efficiency.
3515	AcamAddins - Common Collections and AcamExtensions.	Enhanced memory efficiency.
3516	AcamAddins - Common ExtensionMethods.	Enhanced processing efficiency.
3517	AcamAddins - Common - Enums, Helpers, Math, Files into Addins and Common folders.	Enhanced processing efficiency.
3527	Automation Manager - Addins not releasing COM objects.	Enhanced memory efficiency.
3604	Define Tool - Advanced 5 Axis - Cylinder and Cylinder+Cone options do not change the tool definition.	Fixed.
3611	Lathe - Diameter Roughing - Collision detection is failing when a small depth of cut is used. The final cut added causes the tool to collide with the part.	Fixed.
3613	Rough or Finish - Take Account of Previous Machining - Toolpaths created are in inconsistent directions.	Fixed.
3617	Automation Manager - CSV import - Special characters are not displayed correctly.	Fixed. *Exception - CSV files saved with ANSI encoding (default system encoding), in a different language, being opened on an English system.
3619	Lathe - Select Tool - If a C-Y axis tool is selected, the feedrate is now set as per MIN by default instead of per REV.	Enhancement.
3620	Cut Spline/Polyline - Toolpaths created in 2016 R2 produce unexpected results compared with the same toolpath in previous releases.	Fixed. New 'Angle Control Setting' added.
3621	API - OpenTempDxfFile - Solids on active drawing are included in temp drawing.	Fixed.
3630	Panelling - Documentation - Help button in the dialog not working correctly in Stone, Lathe and Mill modules.	Fixed.
3656	Nesting - Re-Multidrill - Machines the first sheet multiple times.	Fixed.

3661	Advanced 5 Axis - Multi Axis Roughing crashes before the MW dialog is displayed on some non-UK systems.	Fixed.
3669	Pocketing - 3D Project - If a pocket cycle is created using Z Contour Roughing and projected onto a surface, there is no feed move retracting from the surface.	Fixed.
3674	Automatic Extraction - Feature Extraction - Drillable Holes - Alphacam takes longer to process a part than expected. An incorrect result is produced.	Optimised processing efficiency, enhanced progress feedback.
3689	Z Contour Roughing - Associate material for auto-update - If an operation after the initial one is edited, all subsequent operations are deleted.	Fixed.
3691	Cylindrical Parallel - If Start Angle is not equal to End Angle, cut amount is less than expected. One cut fewer than the expected number of cuts occurs.	Fixed.
3696	Send to Simulator - Duplicate arc in NC code causes a circular movement in the simulator.	Fixed.
3704	Send to Simulator - Time taken to send drawing files to simulator, with rapids manager turned on, is longer than expected.	Enhanced efficiency to reduce processing time.
3709	Cylindrical Parallel - Max and Min extents take all solids and surfaces in the active drawing into account.	Enhanced accuracy - Only the parts being machined are taken into account.
3724	Automation Manager - Sample data sheets are not retained when Alphacam is closed.	Fixed.
3725	'Cannot Find Offset Path' When applying toolpath.	Fixed.
3738	Rough or Finish - Take Account of Previous Machining - If this option is checked, previous machining operations are not taken into account.	Fixed.
3739	Pocketing - Take Account of Previous Machining - If stock is changed on a previous operation, the following operation is not highlighted for updating.	Fixed.
3741	Rough or Finish - Profile - When Bi-Directional is used, unwanted rapid moves are produced on second slope.	Fixed.
3812	Geometry - Circle - If the user tries to turn off already drawn geometry to draw a circle, existing geometry is not hidden.	Fixed.
3815	Nesting - True Shape - Toolpaths overlap.	Fixed. Note: Operations for the nest parts must be regenerated before the nest is retried.
3816	Copy Part - By Tool - When Copy Part By Tool is used, not all operations are copied.	Fixed.
3822	Automation Manager - Some controls have larger and over-saturated icons.	Fixed.
3823	Automation Manager - Support File - If an auto query is used in Automation Manager, a support file cannot be created.	Fixed.
3825	Pocketing - Incorrect result with set / unset open elements.	Fixed.
3826	Dialogs - Re-sizing the Alphacam window may cause certain dialogs to display with the title bar above the top of the screen, making re-positioning them impossible.	Fixed.
3839	Pocketing - Take Account of Previous Machining - Incorrect result when Type is set to Contour.	Fixed.
3840	Machining - Update/Edit Operations - Alphacam crashes when operations are edited or updated.	Fixed.

3862	Router - Essential - Pocketing - When the Z contour roughing option is checked, the related options are not available until pocketing is restarted. If the Z contour roughing option is unchecked, not all of the standard options are available.	Fixed.
3863	Pocketing - Use Z Contour Roughing - Radii are created as lines instead of arcs.	Fixed. Prismatic option activated.
3888	Tool Axis Conversion - Through Point - Unexpected behaviour with arcs.	Fixed by quadrant limiting arcs.
3889	Input CAD - STL files with an extremely high facet count are not imported. Drawing files containing these models cannot be re-loaded after saving.	Fixed.
4092	Edit Machining - Toolpath data - The Machining Data tab crashes in the Rough or Finish dialog. This occurs with 32-bit Alphacam installations on systems using Windows 8 or Windows 10.	Fixed.
4106	Constraints - Insert Constrained Drawing - Inserting some files using this command displays an error message saying that the referenced parameter does not exist, constrained data is fatally corrupted and will be removed from the drawing.	Fixed.
4136	Create default Material - Texture and Associate For Auto Update are ignored if user selects 'Manually Select'.	Fixed.
4137	Add-Ins - Create Default Material - Geometry selection not updated when using Manually Selected.	Fixed.
4151	Cut With Disk - Vertical Finishing / Vertical with Disk Side - Cutting operations start on the incorrect side of the geometry.	Fixed.
4170	Queries - Colour RGB fails if layer is in a User Layer.	Fixed.
4172	Translation - Input CAD - Postscript Options dialog. Cancel button is not translated.	Fixed.
4210	Constraints - Insert Constrained Drawing - Parametric Rules - When inserting two constrained drawings into Alphacam, not all rules are displayed.	Fixed.
4211	Shop Floor - Ribbon Bar - Automatically updating the Ribbon Bar displays command buttons that should not be present.	Fixed.
4216	Pocketing - Z Contour Roughing - Spiral stock removal option is displayed in incorrect product levels. It is only available at Standard level or higher.	Fixed.
4231	Nesting - Sheet Database - User has to re-select the correct material and add the thickness again.	Fixed.
4250	Automation Manager - Database update fails if the value entered is 0.00001 or smaller.	Fixed.
4260	Lathe - Circles - Not all Circles are output as segments through post.	Fixed.
4261	Aggregate - Tool Vectors. Tool vectors appear different when a tool is in an aggregate head.	Fixed - Toolpaths need updating.
4269	3D Machining - Parameter Lines - Incorrect toolpath, area un-machined.	Fixed.
4271	Add-ins - Alternate File Extension - With a 64 bit Install on Win 8 or 10, this add in crashes Alphacam	Fixed.
4313	Rough or Finish - Bi-Directional - If multiple cuts are used, the rapid move between them goes through the job.	Fixed.
4439	Machining - Sawing - Alphacam crashes when trying to edit operation.	Fixed.
4457	Rough or Finish - Spiral - Bad toolpath on open geometries and partial selection.	Open or Partial not supported - User warned.

4464	Constraints - Auto Update Toolpaths - Not working for Partial Rough or Finish toolpaths.	Fixed.
4483	Shop Floor Lathe - Materials and Solids are not imported.	Fixed.
4486	NC Output - Rough or Finish - Incorrect lead in/out start and endpoint.	Foul Check and warning added.
4677	Lathe - Balanced Machining - If an operation is edited, it explodes some operations and duplicates others.	Fixed.
4683	Cut Spline/Polyline - 3D Lead In/Out - Load settings disables 'Apply Auto Lead In/Out' checkbox.	Fixed.
4790	GUI - Change Font - Cancel button on Change Font prompt translate to German.	Fixed.
4804	Automatic Feature Extraction - Parts over 3m are giving strange results when extracted.	Fixed.
4989	Automatic Extraction - Inaccurate result (Elliptical Edges).	Fixed.
5054	Automatic Extraction - Single hole features are not named or grouped.	Fixed.

Appendix A - Alphacam 2017 R1 での API 改造

Application – Get Alphacam File Name with specified initial directory

This method invokes the Windows File Open/Save dialog box to prompt the user to pick a filename for an Alphacam file type. Identical to `GetAlphacamFileName` except this method allows the caller to specify an initial folder to display in the File Explorer dialog.

App.GetAlphacamFileNameEx(Title As String, Type As AcamFileType, Action As AcamFileAction, StartFolder As String, FullName As String, FileName As String) As Boolean

Title As String

Caption for the File Explorer dialog.

Type As AcamFileType

The type of file. One of `acamFileTypeDRAWING`, `acamFileTypeFONT`, `acamFileTypeNC`, `acamFileTypePOST`, or `acamFileTypeTOOL`.

Action As AcamFileAction

Specify if the file is to be opened or saved which changes the File Explorer dialog slightly. Note: This command does not open or save any files – it simply gets a filename from the user.

StartFolder As String

Specify the folder that will initially appear in the File Explorer dialog.

FullName As String

If the function returns **True**, this will contain the complete path to the chosen file.

FileName As String

If the function returns **True**, this will contain the filename (no path or extension) of the chosen file.

Returns **True** if the user chose a file, **False** otherwise.

Example

```
Public Sub GetAlphacamFileName()  
  
    'Prompt user for an Alphacam Drawing to load  
    Dim FilePath As String  
    Dim FileName As String  
  
    'Set the initial folder to D:\TestFiles  
    Dim InitDir As String  
    InitDir = "D:\TestFiles"  
  
    If App.GetAlphacamFileNameEx("Please choose an Alphacam Drawing to load", acamFileTypeDRAWING, _  
        acamFileActionOPEN, InitDir, FilePath, FileName) = True Then  
        App.OpenDrawing FilePath  
    End If  
  
End Sub
```

Auto Z Drilling Of Feature Found Holes - Advanced Depth Settings

Two API methods have been added in order to give user further depth settings on feature found holes. These methods reflect the options available through the user interface.

MillData. DrillAutoZBlindHoles As DrillAutoZBlindHoles

On Blind Holes, defines if drill's tip/shoulder will match with feature extracted hole's tip/shoulder.

MillData. DrillAutoZThroughHoles As AcamDrillAutoZThroughHoles

On Through Holes, defines if drill's tip/shoulder will match with feature extracted hole's tip/shoulder.

Example

```
Public Sub DrillUsingDrillTipOrShould()
Dim Drw As Drawing
Set Drw = App.ActiveDrawing
' Let the user select some
Dim Geos As Paths
Set Geos = Drw.UserSelectMultiGeosCollection("Select drilling geometries created through feature
Extraction", 0)
If Geos.Count = 0 Then
Exit Sub
End If
' Select the circles
Geos.Selected = True
' Select a drill
App.SelectTool App.LicomdatPath & "LICOMDAT\MTOOLS.ALP\Drill - 10mm.amt"
' Setup the machining data
Dim MD As MillData
Set MD = App.CreateMillData
MD.DrillType = acamDRILL
MD.AutoZ = True
MD.SafeRapidLevel = 20
MD.RapidDownTo = 1
' For Blind Holes, match the tool's tip to the tip of the cone at the bottom of the hole Feature.
MD.DrillAutoZBlindHoles = acamDrillBlindDrillTip
' For Through Holes, use tool's shoulder to drill to the bottom of the hole
MD.DrillAutoZThroughHoles = acamDrillThroughShoulder
' Create the tool paths
MD.DrillTap
End Sub
```

BeforeSaveSolidPart Event

When loading a CAD file, if the user has enabled the “Save each Solid Body to a separate file” in the Assemblies tab of the Input CAD dialog, this event will be fired immediately before saving each Solid Part. The event enables an add-in to control which Solid Parts are saved and to optionally override the output location.

Function BeforeSaveSolidPart(Solid As SolidPart, Filename As String) As String

The SolidPart object about to be saved. The add-in can query the SolidPart to determine how to proceed.

Filename As String

The filename that will be used to save the Solid Part.

Return value As String

An add-in should return one of the following: -

“0” – Instructs Alphacam to continue as normal.

“2” – Instructs Alphacam to skip this file and continue on to the next Solid Part.

“3” – Instructs Alphacam to abort the operation – no more Solid Parts will be saved.

“<filename>” – Instructs Alphacam to save the Solid Part using the given filename. The filename should contain a complete path and include an extension.

Example

```
Public Function BeforeSaveSolidPart(S As SolidPart, Filename As String) As String

    MsgBox "BeforeSaveSolidPart - Saving " & S.Name & " to " & Filename

    Dim ret As String
    'If the Solid Body Name is "SolidWorks 1 My Assembly Cylinder" change
    'the output location, otherwise continue as normal

    If S.Name = "SolidWorks 1 My Assembly Cylinder" Then
        ret = "D:\Assemblies\Cylinder.ard"
    Else
        ret = "0"
    End If

    BeforeSaveSolidPart = ret

End Function
```

CadInputSettings – Import DXF / DWG Model Space only

A new property, ModelSpaceOnly, can be used to restrict the entities imported from DXF and DWG files to just those in “model space”. When set to true, entities in “paper space” are ignored.

CadInputSettings.ModelSpaceOnly As Boolean

When this property is set to true, the DXF and DWG importer will only import entities in “model space”. The default value is false, to preserve backwards compatibility.

Example

```
Public Sub ImportDXFTest()

    Dim cadSettings As CadInputSettings
    Set cadSettings = App.CadInputSettings

    'Import Model-Space entities only
    cadSettings.ModelSpaceOnly = True

    App.OpenDxfFile "D:\TestFiles\Sample.dxf", True

End Sub
```

Drawing – SaveSolidParts

Automatically save each Solid Part in the Drawing to separate Alphacam Drawing files. Each Solid Part can optionally be aligned (excludes STL models) before saving.

Drawing.SaveSolidParts(OutputDirectory As String, BaseFilename As String, FilenameModifier As AcamSolidPartFilenameModifier, Align As Boolean, OverwriteExisting As Boolean, Filenames As String) As Long

OutputDirectory As String

Set the output folder to receive each Alphacam Drawing file.

BaseFilename As String

Provide a partial filename (no path or extension) that will be combined with the FilenameModifier to create unique names for the drawing files. This name is ignored if FilenameModifier is set to acamSolidPartName.

FilenameModifier As AcamSolidPartFilenameModifier

acamSolidPartName – Use the name of the Solid Part as it appears in the Layers Project Manager. Illegal filename characters will be replaced with the underscore character.

acamAppendSolidPartIndex – Append the BaseFilename with the index of the Solid Part.

acamPrependSolidPartIndex – Prepend the BaseFilename with the index of the Solid Part.

acamAppendSolidPartCounter – Append the BaseFilename with a counter that is incremented for each successfully saved part.

acamPrependSolidPartCounter – Prepend the BaseFilename with a counter that is incremented for each successfully saved part.

Align As Boolean

Set to true to align the Solid Part before saving. The alignment settings used are the ones in the Alignment tab of the Feature Extraction Configuration dialog.

OverwriteExisting As Boolean

Set to true to overwrite any existing files. If set to false and a file already exists with the same name as the Solid Part about to be saved, that file will not be saved, but the operation will continue to the next Solid Part.

Filenames As String

On successful completion of the command this string will received a “*” delimited list of all the filenames that were written. An example string might be “Part_001.ard*Part_002.ard*Part_003.ard”.

Return value As Long

The number of Solid Parts that were saved.

Example

```
Public Sub SaveSolidParts()

'Load an assembly
App.OpenExternalFile "D:\Assemblies\My Assembly.SLDASM", "SOLIDWORKS", True

'Save aligned Solid Parts to D:\Assemblies, with the names Part_001.ard,
'Part_002.ard, Part_003.ard etc..
Dim n, i As Integer
Dim SavedFilenames As String
Dim FilenamesArray() As String

n = App.ActiveDrawing.SaveSolidParts("D:\Assemblies", "Part", acamAppendSolidPartCounter, True, True,
SavedFilenames)

MsgBox "Saved " & n & " Solid Parts"

If (n > 0) Then
'Split the SavedFilenames into an array of filenames
FilenamesArray = Split(SavedFilenames, "*")

'Do something with the filenames
For i = LBound(FilenamesArray) To UBound(FilenamesArray)
Debug.Print FilenamesArray(i)
Next
End If

End Sub
```

Image Preview – Save preview image with no margin

Use the following method to create a drawing preview image which uses the maximum available space. This method will reduce the gap between Alphacam entities and the image margin.

SavePreviewImageNoMargin(FileName As String, ImageType As AcamPreviewImageType, Width As Long, Height As Long, View As AcamView, Shading As Boolean, Quality As Long, DrawAxes As Boolean)

As SavePreviewImageEx but with no margin.

Example

```
Sub CreatePreviewNoGap()  
  
    Dim drw As Drawing  
    Set drw = App.ActiveDrawing  
  
    Dim strFileName As String  
    strFileName = App.LicomdirPath & "\PreviewNoBorder.png"  
  
    drw.SavePreviewImageNoMargin strFileName, acamImagePNG, 700, 500, acamViewXY, True, 10, False  
  
End Sub
```

Nesting - Suppress Message Dialogs

The following method allows the suppression of warnings and error messages while nesting. If this method is set to true, then messages will be suppressed. Otherwise messages will be normally displayed.

Nesting.SuppressDialogs As bool

Enables or disables suppression of all Nesting dialogs, for automation.

Example

```
Sub TestSuppressDialogs()  
    Dim N As Nesting  
    Set N = App.Nesting  
  
    ' Set SuppressDialogs to true, so no messages are displayed  
    Nesting.SuppressDialogs = True  
  
    ' Your nesting code here  
  
    ' Set SuppressDialogs to False, so messages are normally displayed again  
    Nesting.SuppressDialogs = False  
  
End Sub
```

Path - Get Paths Inside or Intersecting Other Path in Collection

This method can be used to easily get paths inside or intersecting another path. A different set of flags can be used in order to modify this function return.

AcamPathsInsideOrIntersecting valid flags are acamIncludeItemNumber, acamIncludeSheetBubble, acamIncludeVisibleOnly and acamPerformSheetCheck.

acamIncludeItemNumber: Include text converted to geometry. Geometry must have the attribute LicomUKsab_item_number_text set.

acamIncludeSheetBubble: Include nested sheet bubble.

acamIncludeVisibleOnly: If set hidden paths will be ignored

acamPerformSheetCheck: Check if given path is a sheet. If so, ignore other sheet geometries.

Path.GetInsideOrIntersecting(PathsToCheck As Paths, Flags As Long) As Paths

Get paths inside or intersecting this Path inside given collection. If path is open return only intersecting paths.

Example

```
Sub GetPathsInsideOrIntersecting()
    Dim Drw As Drawing
    Dim MyPath As Path
    Dim PathsInsideOrIntersecting As Paths
    Dim Flags As Long
    Set Drw = App.ActiveDrawing
    ' Select a geometry to get other geometries inside or intersecting it.
    Set MyPath = Drw.UserSelectOnePath("Select a geometry with other geometries intersecting or inside it")
    ' Select Flags
    Flags = AcamPathsInsideOrIntersecting.acamIncludeSheetBubble +
    AcamPathsInsideOrIntersecting.acamIncludeVisibleOnly +
    AcamPathsInsideOrIntersecting.acamPerformSheetCheck
    ' Get Paths inside or intersecting
    Set PathsInsideOrIntersecting = MyPath.GetInsideOrIntersecting(Drw.Geometries, Flags)
    ' Change color of all geometries inside or intersecting MyPath to Red
    Dim TmpPath As Path
    For Each TmpPath In PathsInsideOrIntersecting
        TmpPath.ColorRGB = RGB(255, 0, 0)
    Next TmpPath
    Drw.RedrawShadedViews
End Sub
```

SubOperation – Get Partial Geometries

Some operations such as Rough or Finish can use partial geometries. These geometries are not returned by the SubOperation.Geometries property, but can now be retrieved using this new property.

SubOperation.PartialGeometries As Paths

Returns a path collection containing the partial geometries associated with this sub-operation. If the sub-operation does not use partial geometries then the property returns 'Nothing' (or NULL).

Example

```
'Get the Partial Geometries associated with the first operation
Public Sub PartialGeosTest()
    Dim ops As Operations
    Set ops = App.ActiveDrawing.Operations
    If (ops.Count > 0) Then
        Dim subOps As SubOperations
        Set subOps = ops.Item(1).SubOperations
        If (subOps.Count > 0) Then
            Dim partialGeos As Paths
            Set partialGeos = subOps.Item(1).PartialGeometries
            If Not (partialGeos Is Nothing) Then
                MsgBox "Found " & partialGeos.Count & " partial geos"
            End If
        End If
    End If
End Sub
```

Toolpaths - Get toolpath time and feed distance

Use the following methods to obtain the individual toolpath machining time, in seconds, and the feed distance.

Path.TimeForToolPath As Double

Time, in Seconds, if this is a tool path.

Path.FeedDistanceForToolPath As Double

Feed distance if this is a tool path.

Example

```
Sub GetToolPathTimeAndFeedDistance()

' Select a toolpath
Dim p As Path
Set p = App.ActiveDrawing.UserSelectOneToolPath("Pick a toolpath")

' Display toolpath time and feed distance via MessageBox
MsgBox p.Name & vbCrLf & _
    "Machining Time: " & p.TimeForToolPath & vbCrLf & _
    "Feed Distance: " & p.FeedDistanceForToolPath

End Sub
```

Wire - Get/Set Sub Operation WireData

Use the following methods to get or set a SubOperation WireData. SetWireData will automatically update the SubOperation using the new WireData provided.

SubOperation.GetWireData() As WireData

Get WireData from SubOperation

SubOperation.SetWireData(WD As WireData)

Set SubOperation with given WireData

Example

```
Public Sub ModifyClearAreaWireData()
    Dim WD As WireData
    Dim Op As Operation
    Dim SubOp As SubOperation
    ' Get First Operation in Active Drawing
    Set Op = App.ActiveDrawing.Operations(1)
    ' Get First subOperation in Op
    Set SubOp = Op.SubOperations(1)
    ' Get Mill Data
    Set WD = SubOp.getwiredata
    ' Change WireData Values
    WD.Stock(1) = 0.2
    WD.ClearWidthOfCut = 0.5
    WD.ClearLeadInLength = 0.5
    WD.ClearStartPointX = 1
    WD.ClearStartPointY = 2
    ' Update SubOperation WireData
    SubOp.SetWireData WD
End Sub
```

WheelCutData – New operation parameters

To improve ease of use, several new parameters have been added to the “Cut with Disk” operation. Many of these are most useful for the user to modify with the benefit of seeing a dynamic preview in the active drawing.

WheelCutData.ReverseGeo As Boolean

Default: False. When True, reverses the direction of the geometry to be machined (for the operation only).

WheelCutData.ReverseProfile As Boolean

Default: False. This property will only affect finishing strategies. For finishing strategies the cutting will begin at the profile's start-point and continue to the profile's end point. If this property is set to True, cutting will begin at the profile's end point and continue to the profile's start point.

WheelCutData.ChangeGeometryToolSide As Boolean

Default: False. When True, changes the tool side of the geometry to be machined (for the operation only).

WheelCutData.AttachProfileAtStartPoint As Boolean

Default: True. This property will only affect profiles that are in flat-land or in the same work-plane as the geometry to be machined. When the profile is put into a work-plane, this property controls whether the profile's start or end point is attached to the start of the geometry to be machined.

WheelCutData.FlipProfile As Boolean

Default: False. This property will only affect profiles that are in flat-land or in the same work-plane as the geometry to be machined. When the profile is put into a work-plane, this property controls whether the profile is reflected in a local work-plane axis or not. For Vertical strategies the reflection is done in the local X-Axis, for Horizontal and Normal to Profile, the reflection is done in the local Y-Axis.

Example 1 - Horizontal Roughing

```
Public Sub HorizontalRough()
    Dim MD As MillData
    Dim MT As MillTool
    Dim WCD As WheelCutData
    Dim PS As Paths

    Set MT = App.SelectTool(App.LicomdatPath & "LICOMDAT\stools.alp\Disk - 150mm dia 5mm thick 1mm
rad.ast")
    If (MT Is Nothing) Then Exit Sub

    Set MD = App.CreateMillData
    Set WCD = MD.WheelCutData

    MD.SafeRapidLevel = 50
    MD.DownFeed = 125
    MD.WidthOfCut = 6
    MD.Stock = 0.5

    'New - Geometries(1) is in flat-land
    WCD.SetSideProfile App.ActiveDrawing.Geometries(1)
    WCD.SetGeometry App.ActiveDrawing.Geometries(2)

    WCD.Method = acamWheelMethodHorizontal
    WCD.DepthOfCut = 15
    WCD.WidthOfFirstCut = 2
    WCD.LeadLength = 15
    WCD.FinishPassOnly = False
    WCD.Bidirectional = True
    WCD.CleanerCutDepth = 1.5
    WCD.SafeXYDistance = 5

    'New - attach the profile at its end point
    WCD.AttachProfileAtStartPoint = False

    'New - flip the profile
    WCD.FlipProfile = True

    'New - Reverse the Geometry
    WCD.ReverseGeometry = True

    Set PS = WCD.CutWithDisk

End Sub
```

Example 2 – Horizontal Finishing

```
Public Sub HorizontalFinish()  
  
    Dim MD As MillData  
    Dim MT As MillTool  
    Dim WCD As WheelCutData  
    Dim PS As Paths  
  
    Set MT = App.SelectTool(App.LicomdatPath & "LICOMDAT\stools.alp\Disk - 150mm dia 5mm thick 1mm  
rad.ast")  
    If (MT Is Nothing) Then Exit Sub  
  
    Set MD = App.CreateMillData  
    Set WCD = MD.WheelCutData  
  
    MD.SafeRapidLevel = 50  
    MD.DownFeed = 125  
  
    WCD.SetSideProfile App.ActiveDrawing.Geometries(1)  
    WCD.SetGeometry App.ActiveDrawing.Geometries(2)  
  
    WCD.Method = acamWheelMethodHorizontal  
    WCD.LeadLength = 10  
    WCD.FinishPassOnly = True  
    WCD.Bidirectional = True  
    WCD.MaxErrorOnProfile = 0.8  
    WCD.SafeXYDistance = 5  
  
    'New - Reverse the profile to change if cutting starts at the top or the bottom of the profile  
    WCD.ReverseProfile = True  
  
    'New - Change the tool-side of the geometry to be machined  
    WCD.ChangeGeometryToolSide = True  
  
    Set PS = WCD.CutWithDisk  
  
End Sub
```

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.