extensive CAD interfaces

comprehensive tool library

adaptive roughing technology

operation rest roughing

combined finishing strategies

steep / shallow corner rest milling

full tool & holder gouge protection

high speed optimized toolpath movements

full kinematic simulation

customizable post processors

reliable & efficient nc code

multi threading processor support

: visi machining 3D

fast and intelligent toolpaths

VISI Machining 3D creates intelligent toolpaths on the most complex 3D parts. Dedicated high speed milling techniques and built in smoothing algorithms create highly ef cient NC code. Intelligent toolpaths will reduce cycle times on your machine, improve productivity and continuously produce high quality components.

Extensive range of CAD interfaces.

VISI can work directly with Parasolid, IGES, CATIA v4 & v5, Pro-E, UG, STEP, Solid Works, Solid Edge, ACIS, DXF, DWG, STL and VDA les. The extensive range of translators ensures that users can work with data from almost any supplier. Companies working with complex designs will benefit from the simplicity with which their customer's CAD data can be manipulated. VISI can work directly with wireframe, solid, surface and mesh data or a combination of all four, providing the user with tools to work with any CAD data or to quickly re-model parts ready for machining, fully utilizing the power of true hybrid surface and solid modeling.

Intuitive interface.

A simple tree structure makes it easy to navigate around the machining operations. Machining parameters for depth of cut, step over, ramp angle etc. are input using a highly graphical interface. Most commonly used values, can be stored as default settings enabling the operator to use a company standard consistent method of machining. On line context sensitive help will guide the programmer through the available machining options.

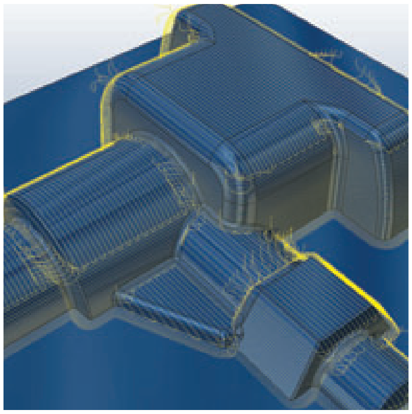
Comprehensive tool library.

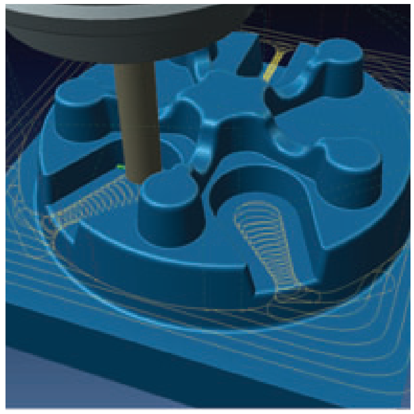
Catalogs of tools, holders, extensions, adapters, storing speeds, feeds, optimal cutting depth, stepover values and tool offsets along with tool and gauge lengths can be selected from user definable libraries. For longer machining cycles, VISI will keep track of the amount of machining completed. When the specified tool life has been reached the system will automatically call for a sister tool, minimizing the risk of damage to the part being machined, by worn or broken tooling.

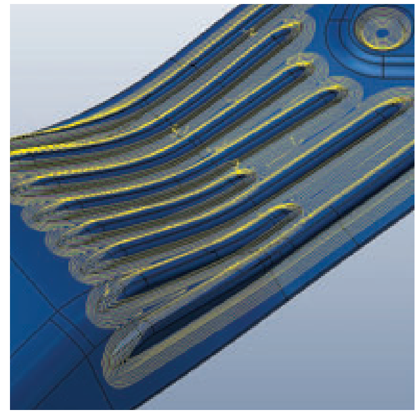
Multiple roughing toolpaths.

A combination of constant Z roughing, adaptive roughing, core roughing and rest roughing, combined with intelligent ramp, helical and planar entry methods provide the operator with the freedom to produce efficient NC code to suit any component. Combined with smooth corner radii and smooth transitions between passes, the tool will maintain the maximum feedrate on the machine tool, and prevent the cutter from dwelling in corners. For subsequent roughing operations, VISI will remember where remaining stock is left on the component and only machine in those areas. Wasted air cutting will be minimized and unnecessary rapid movements will be eliminated, while the cutter will avoid digging into areas where there is excess material, which could result in tool breakage. Where the starting billet is pre-machined, or possibly a casting, VISI will recognize and machine only where material exists, again eliminating wasted movement, and keep cycle times to a minimum.

Toolpaths inside VISI are tailored towards high speed machining and hard metal cutting. Smooth corners, smooth stepovers and arc fitting are used to minimize sudden direction changes. The elimination of tool retracts, maintaining a constant tool load and optimized NC code make it easy to successfully program high speed machine tools with VISI.







Adaptive clearance.

Adaptive clearance toolpaths allow the tool to rough the part in a unique way by roughing out from bottom to top. The principle behind this method is to machine large steps utilizing the full flute length of the tool with a small stepover and then machine the intermediate levels back up the part. Continually repeating the process until the entire component is fully machined. The tool remains on the part as much as possible and the toolpath automatically switches to a trochoidal type motion when the shape of the part requires. This toolpath ensures that there are never any full width cuts and guarantees a constant tool load. Tool wear is spread evenly across the cutting surfaces and the center of force is half-way up the tool, reducing deflection and the potential for vibration Using adaptive roughing, the cycle automatically adjusts the toolpath for efficient and safe machining, improving cutting conditions and allowing higher machining speeds to be maintained. The result is savings of up to 40% in actual cutting time.

ISO-machining.

ISO-machining is based on single or multiple surfaces and machines the surface directly instead of creating a triangulated mesh. This strategy is ideal for machining groups of surfaces that make up radii as the contact point of the tool machines to the full edge of the geometry. This flexible strategy is also extremely useful for picking out small areas without having to machine the entire component. All toolpaths are fully gouge protected against neighboring surfaces with multiple collision detection options available.

Planar surface machining.

For areas of the part which are flat, VISI will automatically detect these areas and machine them using a flat bottomed tool. The machining time for these areas will be significantly reduced and the surface finish will be greatly improved by using a flat tool.

3D step over finishing.

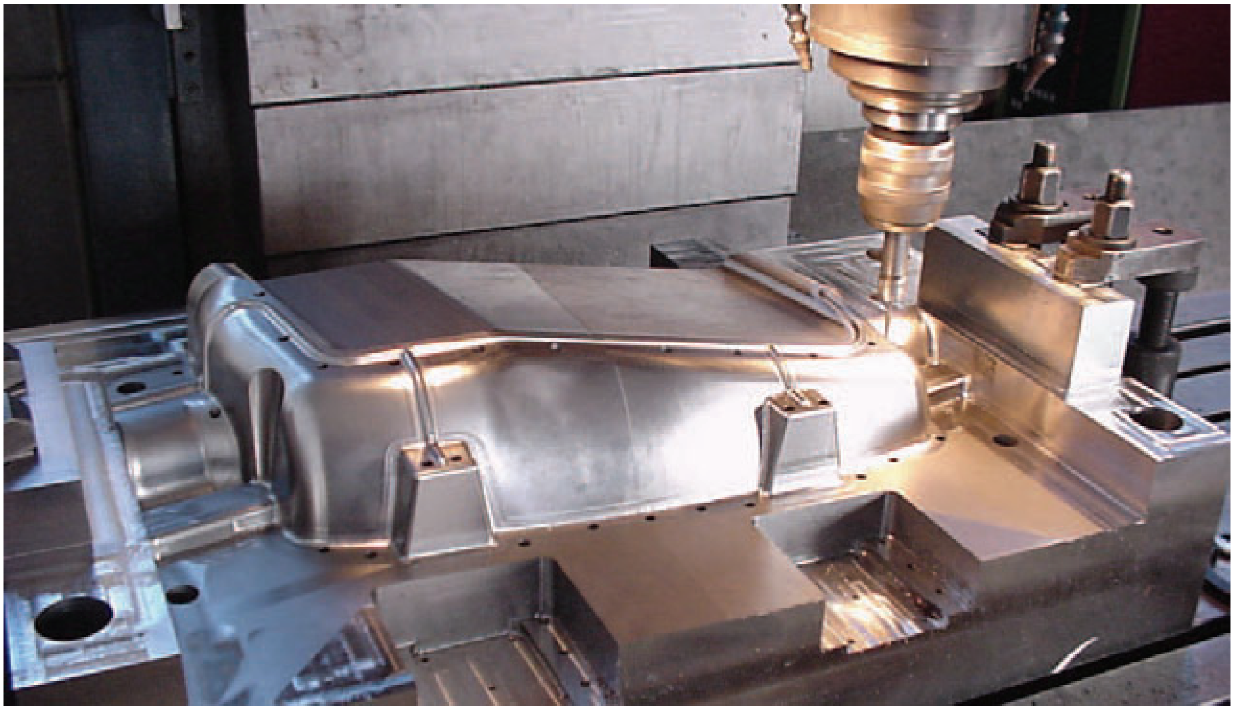
The 3D stepover toolpath provides a constant surface finish irrespective of the component shape. By morphing the toolpath across the surface of the component, one toolpath will finish the entire job, keeping the tool on the surface, minimizing retract movements and eliminating duplicate cutter paths. As the toolpath step over is smoothly adapted to the shape of the part, shock loading of the tool will be minimized, enabling the machine tool to run at the optimum feedrate.

True spiral / radial finishing.

Both toolpaths make an ideal finishing strategy for circular components as they are based on an inner and outer circular limit. The spiral toolpath has only one start and one finish point ensuring the tool remains on the component eliminating any redundant moves or sharp direction changes. This toolpath will enable the machine tool to run at very high feederates as it eliminates the acceleration and deceleration caused by sudden changes in direction. The radial toolpath allows upwards only, downwards only or zigzag machining parameters providing complete strategy control.

Parallel plane finishing.

Unidirectional and zigzag toolpaths can be set at any angle. Angle limits can be set for steep and shallow areas eliminating the need for complex geometry boundaries. Optimized cross-machining can be applied to steep areas within one toolpath. This automatically creates additional toolpaths at 90 degrees to the original toolpaths, machining the areas only where necessary to produce a constant surface finish across the entire component. Roughing mode within the Parallel plane toolpath can be used to rough and finish the part in one operation. Smooth stepovers and tangential extensions to the passes can be used to produce a better surface finish and smoother running of the NC file on the machine tool.



Constant Z / combined finishing.

For components with steep walls, cutting in Z slices provides a good surface finish. VISI provides many options within the constant Z strategy to produce the best performance from this toolpath. Where the angle of the walls change, VISI can automatically adapt the slice heights of each level for shallow areas. Wireframe geometry can also be used to control the slice height and angle limitations can be used to eliminate passes in shallow areas. A helical option allows one continuous toolpath to be created which eliminates witness lines on the part and improves surface finish. Additionally a combined constant Z toolpath is available for finishing steep and shallow areas in one toolpath. This strategy allows steep areas to be machined using a constant Z method and shallow areas to be machined using a 3D constant stepover method. This strategy operates as a one stop finishing toolpath.

Leading curve & 3D curve machining.

The operator can control the cutting area by machining between two driving curves across a model. Parallel machining will morph between the curve geometry using the curve shape as a toolpath guide. Perpendicular machining will run normal to the guide curves giving a choice of cutting directions, allowing more control of the machining method. 3D curve machining forces the cutter to run along the 3D curve in open space (without model geometry) making the strategy ideal for scribe lines and engraving onto the surface of the model.

Rest machining of fine details.

Small features on a model will usually require rest machining with a smaller tool to completely finish the detail. The rest machining command will reliably detect areas left by previous tools, so that they can be re-machined. For very fine details, this process can be repeated as many times as required to make it possible to successfully machine with very small cutters. The toolpath can work from the outside to the center or from the center to the outside of small blends. For features, which are very close together, the toolpath will morph and blend together around obstacles to provide a smooth and flowing toolpath without any sudden direction changes minimizing the number of retract movements and helping to eliminate shock loading on the tool and keep feedrates as high as possible.

Full gouge protection.

All 3D toolpaths are gouge checked against neighboring surfaces to eliminate the possibility of a tool collision. In addition small smoothing radii are automatically added to internal corners. These movements stop the tool from dwelling in the internal corners, which can cause the tool to pull into the job creating an unexpected gouge, which would not be detected by toolpath verification.

Tool holder collision checking.

Checking the tool and the holder against the model provides a warning of a potential collision, together with relevant information about the tool length required to complete the job. By limiting the Z cutting envelope for the tool, it is possible to use several tools to machine a cavity, taking advantage of the rigidity of shorter tools to remove most of the material.

Tapered tool support.

Where models have no draft, it is possible to use tapered tools to machine draft directly onto the model. Straight tooling will require modification of the model to add the correct draft angle before machining can start. Adding draft to imported geometry can often be a very difficult and time consuming task.

2D machining & feature recognition.

Toolmaking applications often contain features which require 2D machining. Due to the integrated nature of VISI, manufacturing of individual plates can be completed using feature recognition. Drilled hole features and apertures are automatically selected with the correct drilling cycles and 2D milling routines applied creating practical CNC code for the most complex plates.