

# EDGECAM Production Milling

## Intelligent Machining From Native CAD Data

EDGECAM Milling provides functionality to program wire frame geometry or solid model component parts on a variety of machine tool configurations, from 2.5 Axis milling to complex surface tool paths on 3 to 5 axis milling machines.

EDGECAM offers intuitive dialogues making programming simple for the new user and comprehensive tool path control for advanced requirements.

### Features at a glance:

- ◆ In process stock
- ◆ Rest material removal
- ◆ Easy operation interface
- ◆ Total tool path control
- ◆ Angle Heads
- ◆ Probing
- ◆ Tool Libraries
- ◆ Shop Floor Documentation

The ability to use Solid models and the design data from all the major CAD system maintaining associative links allows quick and efficient programming, especially when a Design modification is made. Edgcam will inform the user of model updates and show what the changes are and where tool paths need to be corrected. A simple update of the tool paths is all that is required rather than a reprogram of the part. Tombstone and multi part fixture is supported, along with extended datum shifts.

Edgcam offers ease of use operational programming with intuitive dialogues making programming simple for the new user and comprehensive tool path control for the more advanced requirements. Edgcam offers range of milling commands for the production engineer which can be used on both milling machines with W axis and Quills plus lathes with driven tools with . Face Milling, Roughing, Profiling, Hole Cycles, Thread Milling,

Chamfering, Slot Milling are some of the standard operations available and recognise the active stock

**Update stock.** Tool paths can be controlled using the current stock which ensures tool path approach is secure and eliminates fresh air cutting. The stock can originate from automatically created stock or from a forging or casting model produced in CAD.

**Face Milling** Creates a series of straight cuts on a horizontal plane. Face milling will recognise the boundary shape and remove air cuts where applicable, links moves for change in cut direction can be controlled to give smooth transition producing an even flow tool path which is kinder to the machine tool and cutter engagement.

**Hole Cycles** Edgecam includes all the standard drilling, tapping and rigid tapping routines with canned cycle and sub routine output. Back boring is also available where conventional machining cannot be achieved on the part. When using solid models the hole size, thread data, depths etc is extracted and the necessary tooling suggested from the tool libraries.

**Rough Milling** Has a variety of tool path control and methods, from face, concentric, spiral and the wave form tool path. Edgecam produces the required tool path and tool entry into the material. Shape recognition allows sub routines to be applied with a simple check box. Edgecam will recognise the change in geometry walls and create sub routines as appropriate.

**Waveform Roughing** Waveform cycle is superior to the traditional roughing cycle where machinable geometry is offset inward or outward by % step over. Traditional tool paths have to run slower feeds and speeds due to the variable widths of cut condition when encountering corners and material entry.

Wave form toolpath has been developed to remove tool load spikes and maintain an even chip thickness and generating a fluid tool path throughout the machinable elements using a flowing motion. Consistent tool loads generated from the waveform tool path offers the user the opportunity to rethink speeds, feeds and depths of cut. The Waveform tool path increases tool life and is also kinder to the machine tool.

**Thread Milling** Thread milling is a popular technique when machining threaded holes on larger components, especially in the oil and gas, power generation and other heavy industries. Edgecam's thread milling cycle will automatically suggest entry points, and lead in lead out paths. Single pass or multi-pass helical move tool paths can be output.

**Automation Strategy Manager Automation** is a flow chart decision making process using to your manufacturing methods/ knowledge. Edgecam recognises manufacturing features and data from a 3D solid model and applies your proven manufacturing tool paths and techniques. This creates tool path cycles automatically with your own tooling to your requirements. This greatly reduces the offline programming time maximising on your investment plus aids learning curves for new engineers.

**Probing** Edgecam supports Renishaw probing part setup cycles. The probing cycles can be integrated by simply adding an additional toolbar to the user interface. This supports all datum offset part positional requirements which can be a pre-requisite to actual machining whether on 3 axis mill or multi-pallet tombstone fixture.

**Indexing and Part Positioning** Support for single or compound indexing with a using A, B or C axis combination using safety clearance zones to give 4 or 5 axis positioning. Datum shifting and extended offsets can be output for each new position.

**Angle Heads** Support for Angled Head attachments is available in Edgecam. The holder and tool can be saved in the comprehensive tool libraries and called up in the tool change along with material feeds and speeds. The use of angle heads will use plane switching if supported by the machine tool control. When used, the holder body along with the tool is collision check in the machine simulator.

**Shop Floor Documentation** Documentation of the operation process is automatically created along with tooling kit/list, operation breakdown and can be stored centrally on a server so all production staff can access the data. Machine tool set up information along with digital images can also be added along with stock and fixture requirements. Documents and pre set tool drawing can be attached. This module is standard with all systems and is very useful solution for pre setting tooling areas.