



## Edgecam Milling Cycles Group

This is a collection of three Edgecam training courses that explore the core toolpath commands used for prismatic milling, 3D contour milling, and 5-axis simultaneous machining. Each class covers a specific industry segment and can be taken independently of the other courses in this group. This approach provides users with the flexibility to take the training most applicable to their current needs.

Users that have used Edgecam for several years should consider taking one or more of these courses as refresher training. These courses focus on how the current software technology for toolpath creation should be used. Users that were trained several years ago and feel uncomfortable with the latest Edgecam software technology will come away with many useful ideas that translate directly to improved productivity.

### Courses

- Edgecam Prismatic Milling Applications
- Edgecam 3D Contour Milling Applications
- Edgecam 5 Axis Simultaneous Machining Applications

### Duration

Each course in this group is 1 day. Students can attend one or more segments, and pay only for the number of training courses taken.



## Edgecam 5 Axis Simultaneous Machining Applications

This 1-day training course is designed for the Edgecam user seeking to build simultaneous machining toolpath. The course is applicable to 4 and 5 axis machining, for both milling and mill/turn CNC machines. Students learn the basics of creating toolpath using Edgecam's 5-Axis Simultaneous Machining capability, and are provided with a wide range of useful application examples showcasing the machining technology. This is an essential course for customers that want to fully leverage the capability of modern 4/5 axis CNC milling centers with ability to perform simultaneous machining. The course is also applicable to users with live tool machining capabilities on a CNC mill/turn.

### Objectives

After completing the course, students should be able to:

- Build functional toolpath using Edgecam's 5-Axis Simultaneous Machining license.
- Differentiate between the basic inputs required to build toolpath and the specialty options that provide flexibility to adapt to a wide variety of machining requirements.

### Prerequisites

- Complete the **Edgecam Workflow** course, or have similar basic software knowledge.
- Experience using Edgecam, with at least 6 completed programs that have run in CNC milling machines recommended.
- Knowledge of CNC milling applications.
- Users are strongly encouraged to complete both the **Edgecam Prismatic Milling Applications** and **Edgecam Multi-Axis Milling** courses before taking this one.
- Proficient with Microsoft Windows. Unless otherwise specified, workstations currently run on Windows 7 operating system.

### License Requirements

This course is applicable to any of the following Edgecam licenses:

- 4-Axis Simultaneous Milling
- 5-Axis Simultaneous Milling
- Advanced 5-Axis Simultaneous Milling

### Topics Covered

Understanding 5-axis Simultaneous Machining

- Toolpath Types
- Controlling Tool-Workpiece Tilt
- How Drive Curves, Drive Surfaces & Check Surfaces influence toolpath

Code Wizard Configuration (specific to 4 & 5-axis machining)

Simulation

Machining Application Examples

Creating Toolpath

M2 Technologies is the leading manufacturing design solutions provider in the Northeast. We specialize in providing solutions that deliver on the business value of digital prototyping to all size companies. For more information on M2 or to speak with our Training Coordinator, call 877.311.6284.