

## Rhino Introduction

Rhino is a 3D computer graphics and computer-aided design (CAD) application software developed by Robert McNeel & Associates. Rhinoceros geometry is based on a NURBS mathematical model, which focuses on producing mathematically precise representation of curves and freeform surfaces in computer graphics. Rhino uses real world dimensions in modeling. While Rhinoceros is developed for Microsoft Windows operating system, a beta version with a reduced toolset is available for OS X.

This class is designed to offer a high-level overview of Rhino, including lessons on navigating the interface and using popular modeling tools and techniques, along with overview of how to manipulate Rhino projects, among other concepts. This grand tour of Rhino will provide students with a good sense of its capabilities and applications to their fields as well as to immerse the student into the software's nomenclature, interface and capabilities to facilitate a quicker ramp-up in usability and comprehension.

This intro class will not be a hands-on workshop

### Upon completion, you will be able to:

- Understand what Rhino is and how it can be used in production
- Get a grand tour of Rhino and get a sense of its capabilities
- Learn about Interoperability and Format Exchange with Rhino
- Understand the nomenclature and comprehension of its tools

### Prerequisites

- There's nothing required for this class

### Software and Hardware Requirements

- The class will be presented on Rhino 5
- To download Rhino 90 day Trial version: Click [HERE](#)
- For reference only:
- Operating system requirements: Mac OS X 10.8.5 or Microsoft Windows 7, 8 or 8.1
- Hardware: 64-bit Intel or AMD multi-core processor, 1 GB of RAM (8GB recommended), 600 MB disk
- free disk space for install and a three button mouse. OpenGL 2 capable video card recommended.

### Questions?

Please call us at 800-336-3375.

### Course Outline

#### Day 1

##### Welcome and Introduction

- What Is Rhino Used For and Why
- Interface Customization

##### Rhino Vocabulary and Terminology

- Curves, Solids and Surfaces
- Bezier curves, B-splines and NURBS
- NURBS surfaces vs. mesh surfaces
- Establishing units
- Adjusting tolerance

##### Navigating the environment

- Delve into the splash screen
- Exploring customization options
- Adding labels and tabs
- Selecting a shading mode
- Pan, Zoom and rotate to navigate
- Built in help options

##### Interface Orientation

- Deconstructing the command philosophy
- Interpreting the workspace
- Utilizing toolbars, panels and popup menu
- Geometry menu overview
- Using the command line and status bar
- Introducing the Gumball manipulator
- Utilizing the Properties panel
- Saving and opening files

## Rhino Introduction

### Productivity workflow

- Adjusting preferences
- Object selection
- Layers for organization
- Deconstructing construction planes
- Establishing grids
- Transformation options
- Utilizing Named Position and View features
- Modeling Techniques
- Material and Lighting overview
- Visualization Options