

Rhino and Python

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Rhinoceros is a 3D computer aided design program, available for both Mac and Windows. Rhino was developed for industrial design, but works very well for mathematical models - it has a "ruler and compass" feel to it, allowing the user to, for example, create a circle from a given three points, or a sphere from a given four points, and accurately obtain new points and curves from intersections of curves and/or surfaces. Rhino primarily works with NURBS curves and surfaces, rather than polygonal meshes, which means that it can precisely represent circles, spheres, cones, etc.

In addition to the graphical user interface, Rhino has a Python scripting interface, with which one can automate almost everything that one can do with the graphical user interface.

In this workshop, we will start by briefly exploring the graphical user interface, making by-hand a file ready to send to a 3D printer. Next we will look at a simple script to generate a parametric curve (i.e. a map with 1-dimensional domain), followed by another script to generate a parametric surface patch (i.e. a map with 2-dimensional domain). Finally, we will look at a very useful combination of the graphical user interface with the scripting tools: building geometry by-hand that is then transformed by an arbitrary distortion (i.e. a map with 3-dimensional domain).

A 90-day free trial of Rhino is available from <http://www.rhino3d.com/download>, please have it installed for the workshop.

http://math.okstate.edu/people/segerman/talks/3d_printing_workshop_files.zip