

# Rhino入门手册

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# 第一章 Rhino

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# 第一章Rhino

- Rhino学习目标：
  - 1.了解Mesh与Nurbs、Tsplines手工建模常用命令
  - 2.使用Rhino制作一个笔筒，并输出加工图。
  - 3.使用Rhino手工建模一个莫比乌斯戒指，并在表面附上渐变3D肌理。
  - 4.使用mesh设计并建模一个科大60周年纪念茶杯，并用tsplines转换为Nurbs,并在表面刻字或添加纹理。
  - 5.使用Rhino的Neon出渲染图。

# 第一章Rhino

## • 1.1 什么是Rhino?

- Rhino 可以创建、编辑、分析、提供、渲染、动画与转换 NURBS\* 线条、曲面、实体与多边形网格。不受精度、复杂、阶数或是尺寸的限制。
- **不受约束的自由造形3D建模工具**。以往您只能在二十至五十倍价格的同类型软件中找到这些工具。让您可以建立任何想象的造形。
- **精准性**，完全符合设计、快速成形、工程、分析和制造从飞机到珠宝所需的精确度。
- **兼容性**，兼容于其它设计、制图、CAM、工程、分析、着色、动画以及插画软件。
- 读取与修补难以处理的IGES文档。
- **容易使用**. 非常容易学习使用，让您可以专注于设计与想象而不必分心于软件的操作上.
- **高效率**，不需要特别的硬设备，即使在一般的笔记本计算机上也可以执行。
- **开发平台**，上百套定制化建模软件的开发平台（仅windows）。
- **经济实惠**. 普通的硬件设备，容易上手，价格相当于一般的Windows软件，并且不需额外的维护费用。
- (摘自官网)

# 1.1 什么是Rhino?

- 1.1.1 Rhino+GH+Plug-ins
- ->设计自由造型，利用NURBS曲面参数化设计幕墙，设计复杂建筑节点

Rhino and Grasshopper are heavily used for **freeform roofs, parametric facades, repetitive components, versatile building shapes or complex structures** in architecture and engineering.



**Rhinoceros®**

+



**Grasshopper**

+



**Plug-ins**

**food 4 Rhino**  
APPS FOR RHINO AND GRASSHOPPER

# 1.1 什么是Rhino?

- 1.1.2 与BIM系统无缝对接，支持IFC格式

## BIM & Documentation



VisualARQ, FlexibleBIM® Plug-in  
for Rhino and Grasshopper

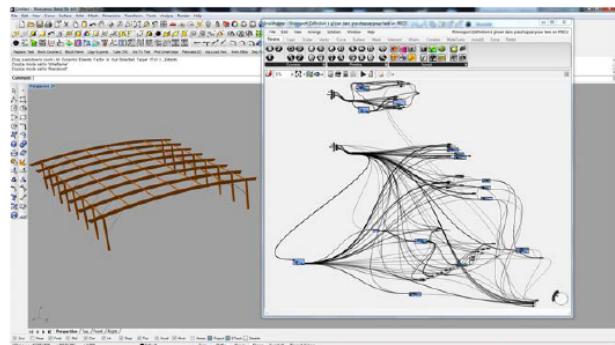


**FlexibleBIM® (Building Information Modeling) features** link geometry and object data. They support import and export from and to the **IFC file format** to exchange Rhino models with other **AEC applications**.

# 1.1 什么是Rhino?

- 1.1.3 利用Grasshopper设计关联图案或者模型(2D/3D)(NURBS 曲面、网格、实体)

## Design & Drafting



Rhino/ Grasshopper UI

Rhinoceros in combination with Grasshopper is an ideal addition for all 2D and 3D applications in architecture and engineering in order to generate **complex associated geometries such as NURBS surfaces, meshes and solid models.**

# 1.1 什么是Rhino？

- 1.1.4 支持各种为Rhino量身定制的渲染插件，例如Vray， Maxwell等

## Visualization & Animation



Many popular rendering and animation plug-ins work in Rhino. They help generating **high-quality images** of unparalleled realism and **animating objects**.

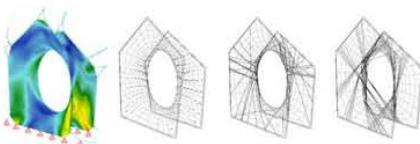
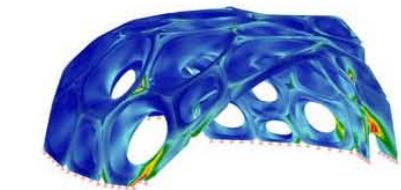


Render, V-Ray Plug-in

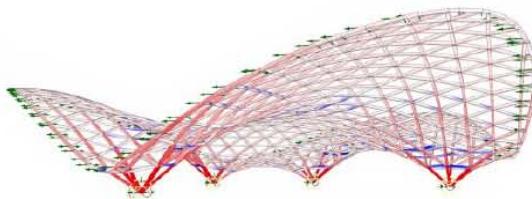
# 1.1 什么是Rhino?

- 1.1.5 各种分析插件：有限元插件Karamba，环境分析插件Ladybug Tools，物理引擎Kangaroo (shapeop)，力学分析插件Millipede

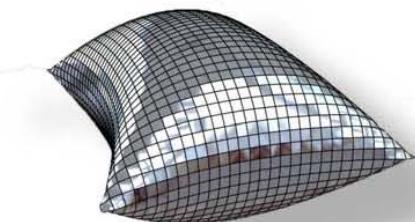
## Analysis & Simulations



ICD/ITKE Research Pavilion  
Stuttgart 2014



Karamba



Kangaroo Physics

Karamba

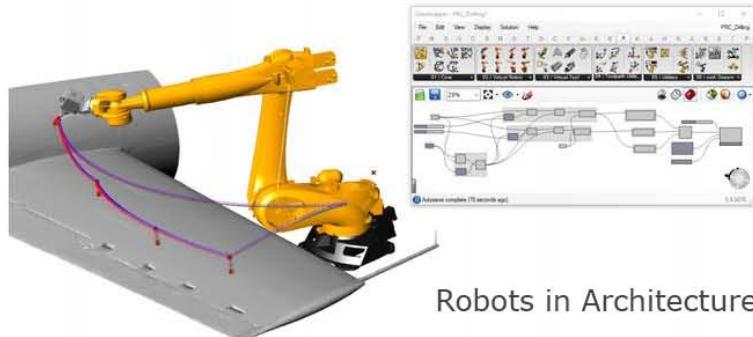


Rhino and Grasshopper have a rich ecosystem of **plug-ins** for **structural analysis, simulation, visualization and fabrication** of your design.

# 1.1 什么是Rhino?

- 1.1.6 与电子硬件、各种数字制造设备衔接。直接输出obj、dwg、step、stl到激光切割机、铣床、3D打印机。firefly插件：Kinect接口，arduino接口等。KukaPRC可实时输出到KUKA，HAL输出到各种机械臂设备。

## Fabrication & Construction



Robots in Architecture

Rhino and Grasshopper can be tightly integrated in any step of the manufacturing process.

**Robotic and Digital Fabrication, 3-D Printing, and Rapid Prototyping**



HAL, GH Add-On

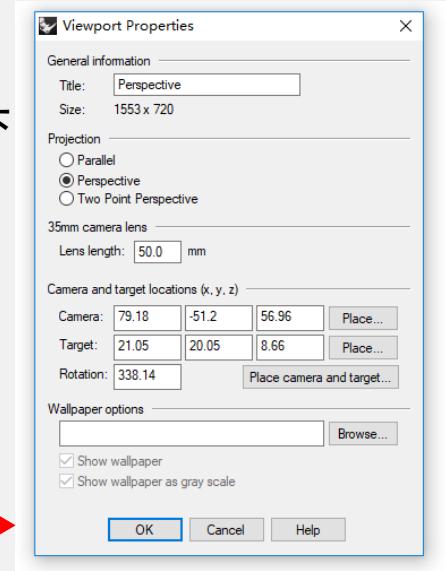
# 1.2 Rhino相关网站

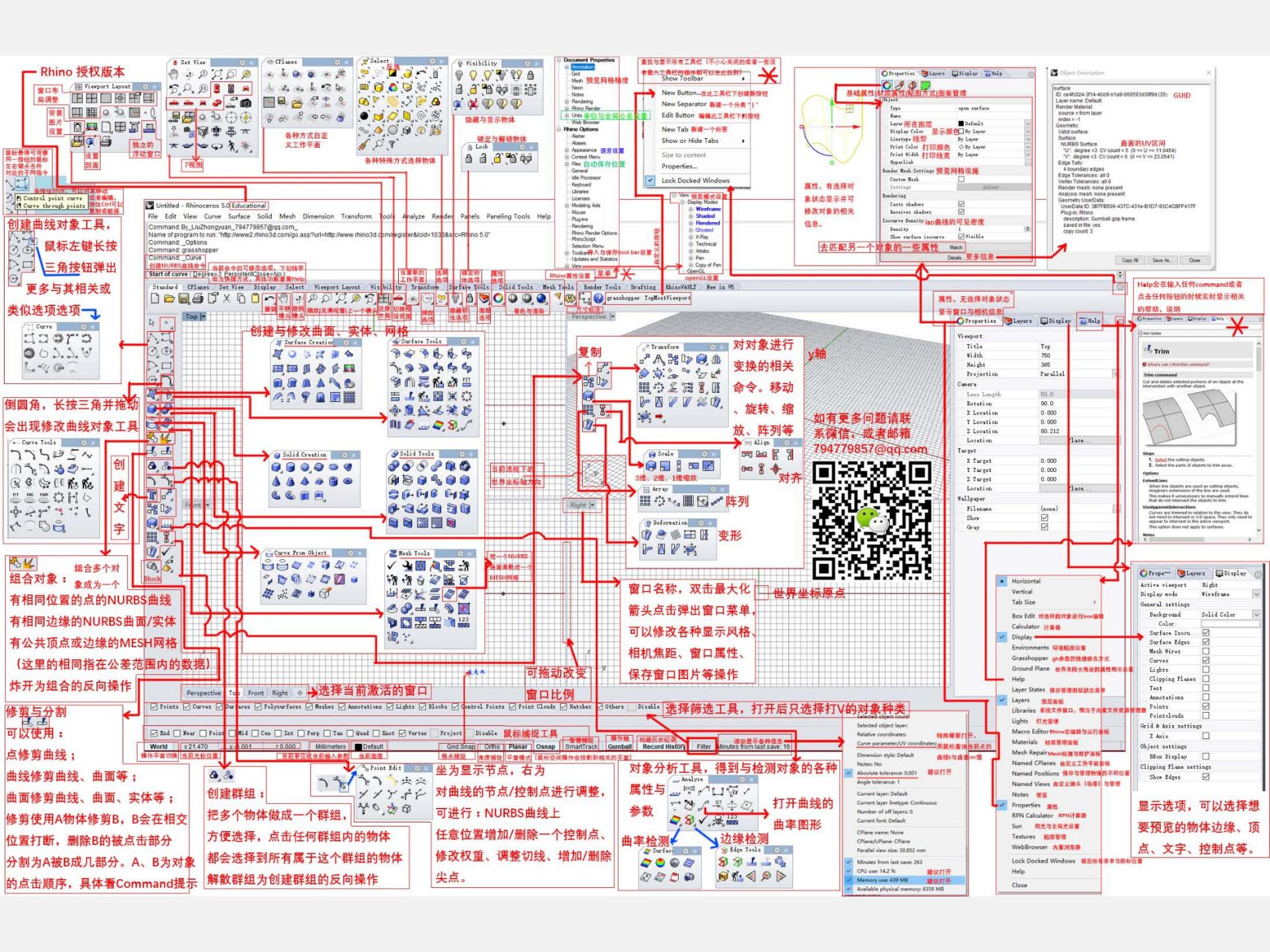
- **Rhino官方介绍:**  
[http://v.youku.com/v\\_show/id\\_XNTIxMzQyNjk2.html](http://v.youku.com/v_show/id_XNTIxMzQyNjk2.html)
- Rhino官网: <http://www.rhino3d.com/>
- Rhino开发者平台: <http://developer.rhino3d.com/>
- Rhino官网下载: <http://www.rhino3d.com/download>
- Rhino官方第三方插件平台: <http://www.food4rhino.com/>
- Grasshopper官方论坛: <http://www.grasshopper3d.com/>
- RhinoFab: <http://www.rhinofabstudio.com/>
- **Rhino与图形学:**
- Tsplines(T样条/曲面插件): <http://www.tsplines.com/>
- Evolute(网格优化插件): <http://www.evolute.at/>
- Rhinovault (受压网格Thrust Network Analysis优化) :  
<http://www.block.arch.ethz.ch/brg/tools/rhinovault>
- Vray写实渲染插件: <https://www.chaosgroup.com/vray/rhino>



# 1.4 Rhino界面

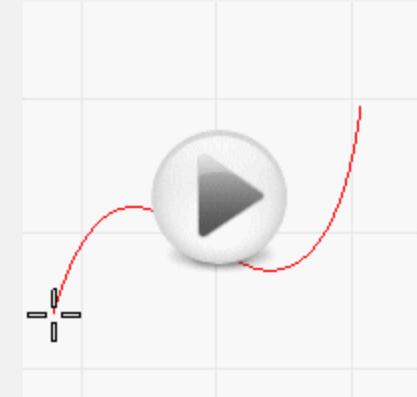
- Rhino窗口的各项说明与操作详见：  
[rhino窗口1.jpg](#)
- 鼠标操作：
- 在透视图窗口：
- 鼠标右键为以Target位置为中心水平旋转镜头
- Ctrl+Alt+鼠标右键为以Camera位置为中心水平旋转镜头
- Shift+Alt+鼠标右键为平行于镜头平面旋转镜头
- Shift+鼠标右键为平移镜头
- Ctrl+鼠标右键(鼠标滚轮)为缩放镜头
- Alt+鼠标右键为向前移动镜头
- Command命令：\_ViewportProperties
- 弹出窗口(Camera)属性 





# 1.5 NURBS Curve

- 1.5.1 创建不同degree的NURBS控制点曲线 |
- Command: \_Curve
- Start of curve ( Degree=3 PersistentClose=No ):
- Next point. Press Enter when done ( Degree=3 PersistentClose=No Close Sharp=No Undo ):
- -----



## Steps

- Pick the start of the curve.
- Pick the next points.
- Press Enter to end the curve.
- Note: The Curve command makes a curve of degree <number of points> -1 as long as the number of control points is less than or equal to the degree setting.

# Bezier Curve

The equation of a Bézier curve is similar to the equation for the center of mass of a set of point masses. Consider the four masses  $m_0, m_1, m_2$ , and  $m_3$  in Figure 2.3.a located at points  $\mathbf{P}_0, \mathbf{P}_1, \mathbf{P}_2$ ,  $\mathbf{P}_3$ .

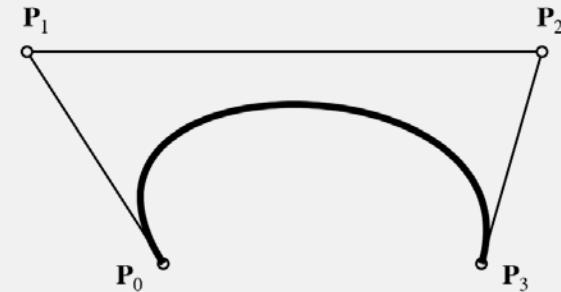
$\mathbf{P}_1$   
•

$\mathbf{P}_0$  •

$\mathbf{P}_2$   
•

•  $\mathbf{P}_3$

(a) Center of mass of four points.



(b) Cubic Bézier Curve.

Figure 2.3: Bézier Curves in Terms of Center of Mass.

The equation for the center of mass is

$$\mathbf{P} = \frac{m_0\mathbf{P}_0 + m_1\mathbf{P}_1 + m_2\mathbf{P}_2 + m_3\mathbf{P}_3}{m_0 + m_1 + m_2 + m_3}.$$

Next, imagine that instead of being fixed, constant values, each mass varies as a function of a parameter  $t$ . Specifically, let

$$m_0(t) = (1-t)^3, \quad m_1(t) = 3t(1-t)^2, \quad m_2(t) = 3t^2(1-t), \quad m_3(t) = t^3. \quad (2.1)$$

Now, for each value of  $t$ , the masses assume different weights and their center of mass changes continuously. As  $t$  varies between 0 and 1, a curve is swept out by the moving center of mass.

# Bezier Curve

Figure 2.3.b shows the Bézier curve that results when the point masses in Figure 2.3.a are taken as control points. This curve is a cubic Bézier curve — *cubic* because the mass equations are degree three polynomials in  $t$ .

Notice that the mass equations in (2.1) sum identically to one:

$$(1-t)^3 + 3t(1-t)^2 + 3t^2(1-t) + t^3 = [(1-t) + t]^3 = 1^3 \equiv 1,$$

and so we can write the equation of this Bézier curve as  $\mathbf{P}(t) = m_0(t)\mathbf{P}_0 + m_1(t)\mathbf{P}_1 + m_2(t)\mathbf{P}_2 + m_3(t)\mathbf{P}_3$ .

The mass functions are plotted in the graph in Figure 2.4. Note that when  $t = 0$ ,  $m_0 = 1$  and

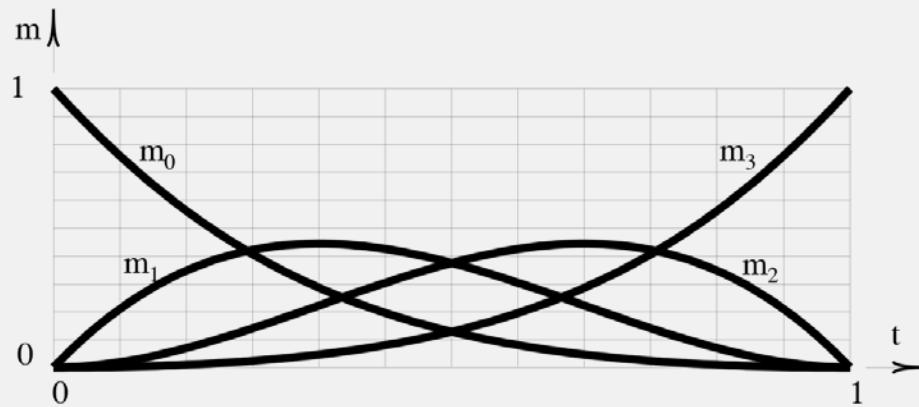


Figure 2.4: Cubic Bézier blending functions.

# Bezier Curve

$m_1 = m_2 = m_3 = 0$ . This explains why the curve passes through  $\mathbf{P}_0$ . When  $t = 1$ ,  $m_3 = 1$  and  $m_0 = m_1 = m_2 = 0$ , and the curve passes through point  $\mathbf{P}_3$ .

The variable masses  $m_i(t)$  are usually called *blending functions* and, as noted before, the locations  $\mathbf{P}_i$  are known as *control points*. The blending functions, in the case of Bézier curves, are known as *Bernstein polynomials*. We will later look at other curves formed with different blending functions.

Bézier curves of any degree can be defined. Figure 2.5 shows sample curves of degree one through four. A degree  $n$  Bézier curve has  $n + 1$  control points whose blending functions are denoted  $B_i^n(t)$ ,

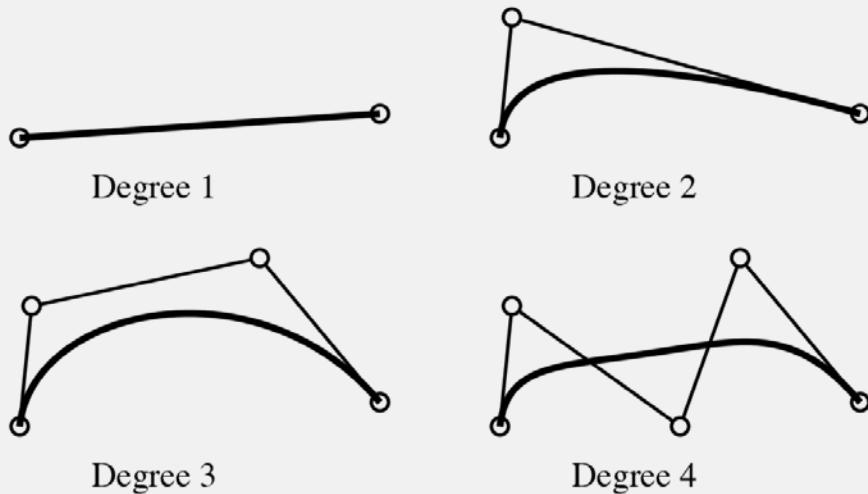


Figure 2.5: Bézier curves of various degree.

# Bezier Curve

where

$$B_i^n(t) = \binom{n}{i} (1-t)^{n-i} t^i, \quad i = 0, 1, \dots, n.$$

Recall that

$$\binom{n}{i} = \frac{n!}{i!(n-i)!}.$$

$\binom{n}{i}$  is spoken “ $n - \text{choose} - i$ ” and is called a *binomial coefficient* because it arises in the binomial expansion

$$(a+b)^n = \sum_{i=0}^n \binom{n}{i} a^i b^{n-i}.$$

In the degree three case,  $n = 3$  and  $B_0^3 = (1-t)^3$ ,  $B_1^3 = 3t(1-t)^2$ ,  $B_2^3 = 3t^2(1-t)$  and  $B_3^3 = t^3$ .  $B_i^n(t)$  is also referred to as the *i*th Bernstein polynomial of degree  $n$ . The equation of a Bézier curve is thus:

$$\mathbf{P}(t) = \sum_{i=0}^n \binom{n}{i} (1-t)^{n-i} t^i \mathbf{P}_i. \quad (2.2)$$

B-Spline Curves、NURBS (Non-Uniform Rational B-Splines ) ...

# 1.5 NURBS Curve

- 1.5.2 创建不同degree的NURBS插值点曲线 

- Command: \_InterpCrv
- Start of curve ( Degree=3 Knots=Chord PersistentClose>No StartTangent ):
- Next point ( Degree=3 Knots=Chord PersistentClose>No EndTangent Undo ):
- Next point. Press Enter when done ( Degree=3 Knots=Chord PersistentClose=No EndTangent Undo ):
- Next point. Press Enter when done ( Degree=3 Knots=Chord PersistentClose=No EndTangent Close Sharp=No Undo ):



- **Steps**

- Pick the start of the curve.
- Pick the next points.
- Press Enter to end the curve.

# 1.5 NURBS Curve

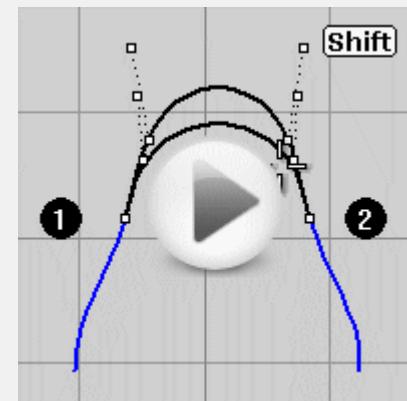
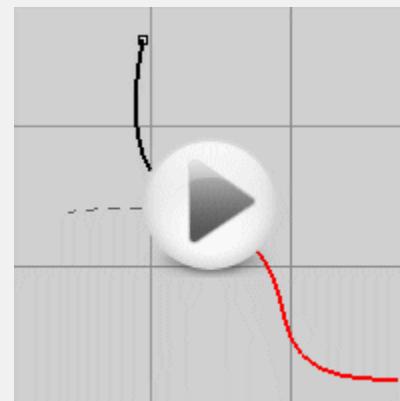
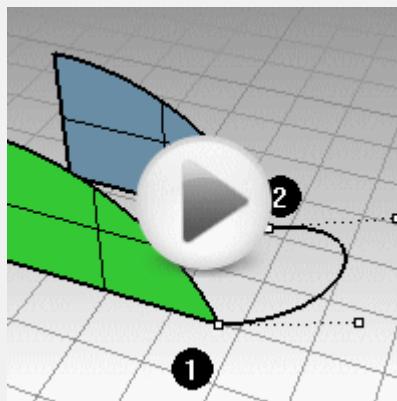
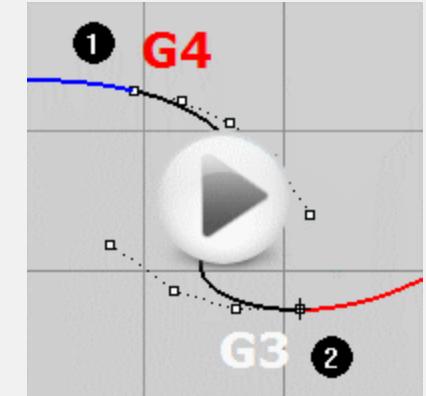
- 1.5.3 对两条不相连的曲线进行混接进行混接 

- Command: \_BlendCrv
- Select curve to blend ( Edges Point ):
- Select curve to blend ( Edges Point ):
- Select control point to adjust. Press SHIFT and select for symmetry.:

• -----

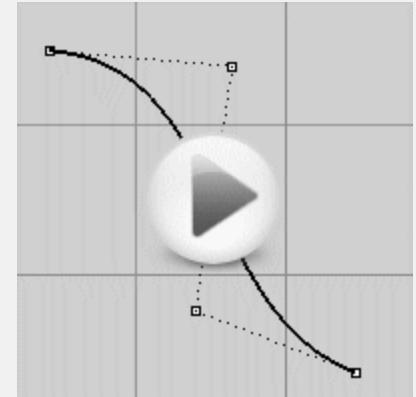
## Steps

- Select the curves.
- Select a control point to adjust.
- Press the Shift key and select for symmetry.
- A preview of the curves displays that allows adjustment of the curve before it is made.



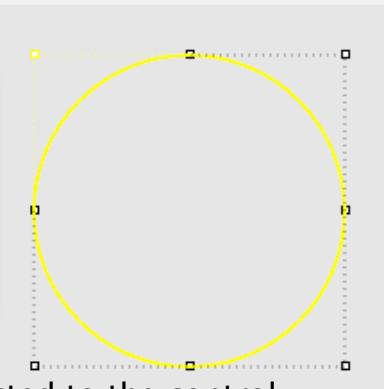
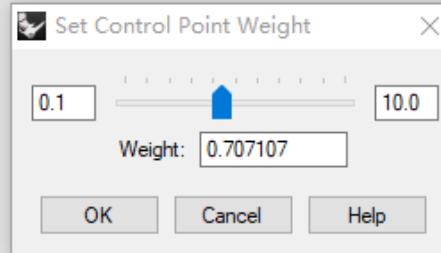
# 1.5 NURBS Curve

- 1.5.4 插入一个尖点 
- Command: \_InsertKink
- Select curve or surface for kink insertion:
- Point on curve to add a kink:
- -----
- **Steps**
  - Select the curve or surface.
  - Pick locations where you want to insert a kink.
  - Press Enter to end the command.
  - A curve is split into multiple joined curves.



# 1.5 NURBS Curve

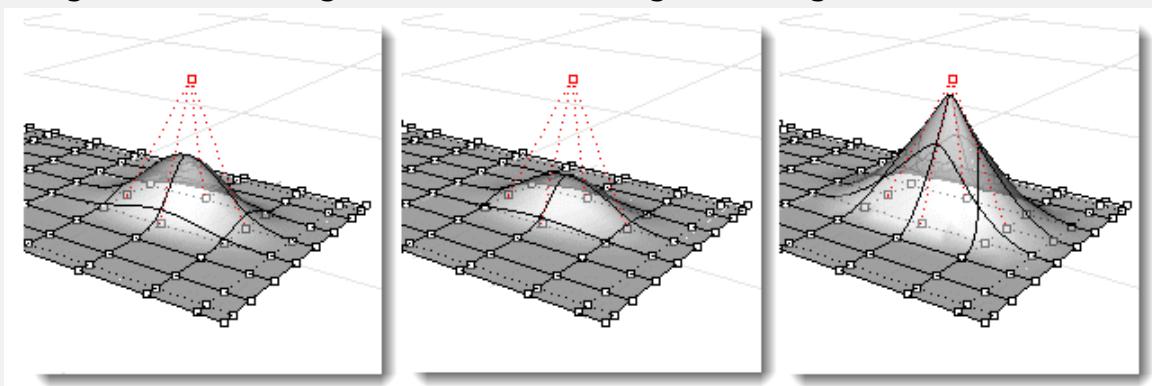
- 1.5.5 编辑点的权重 
- Command: \_Weight
- Select control points for weight editing:



- The weight of a control point determines how much the curve or surface is attracted to the control point. The higher the value of the weight, the closer the curve or surface is to the control point.

## Steps

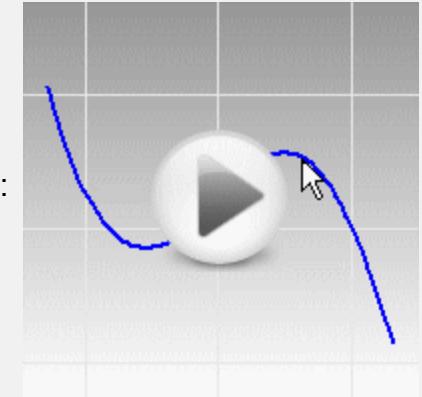
- Select the curve or surface.
- Pick locations where you want to insert a kink.
- Press Enter to end the command.
- A curve is split into multiple joined curves.
- **Warning: If you are exporting to another application, it is best to leave all control point weights at 1.**
- Weight=1 (left), Weight=0.5 (center), Weight=10 (right).



# 1.5 NURBS Curve

- **1.5.6 编辑切线** 

- Command: `_HBar`
- Select curve or surface to adjust:
- Click and drag to edit curve. Press Enter when done ( `FixedHandleLength=No` ):
- New location ( `FixedHandleLength=No` ):
- -----



- **Steps**

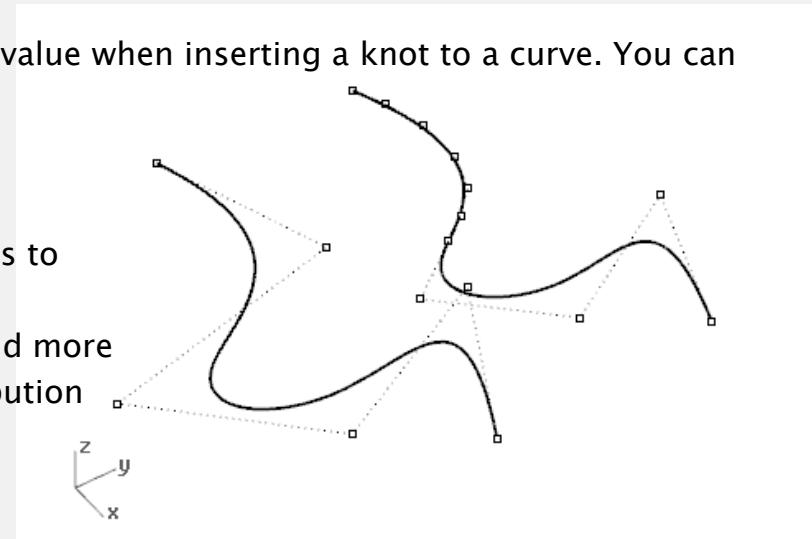
- Select a curve.
- Click and drag the handlebar grips to move it around.
- Click near the object to move the handlebar to the marker position.
- Press Enter to end the command.

- **Options**

- `FixedHandleLength`
- Prevents changing the shape of the curve by stretching the handles.

# 1.5 NURBS Curve

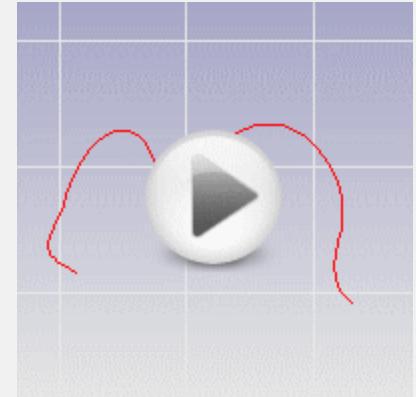
- 1.5.7 插入Knot ✎
  - Command: `_InsertKnot`
  - Select curve or surface for knot insertion:
  - Point on curve to add knot. Press Enter when done ( Automatic Midpoints=No Symmetrical=No ):  
-----
- Steps
  - Select the object.
  - Pick a location for the knot.
  - Note: Inserting a knot in a surface also displays an isoparametric curve at the knot location if isoparametric curves are being displayed.
  - The status bar distance pane displays the parameter value when inserting a knot to a curve. You can also type a parameter value to add a new knot.
- Options
  - Automatic
  - Adds a knot/knot line halfway between existing knots to maintain as uniform a structure as possible.
  - Increases the knot density of a curve or surface to add more control points while maintaining an even knot distribution
  - for better point pulling behavior.



# 1.5 NURBS Curve

- 1.5.8根据公差重建曲线 

- Command: \_Rebuild
- Select curves, extrusions or surfaces to rebuild:
- Select curves, extrusions or surfaces to rebuild. Press Enter when done:
- Maximum deviation = 1.05247e-12
- -----

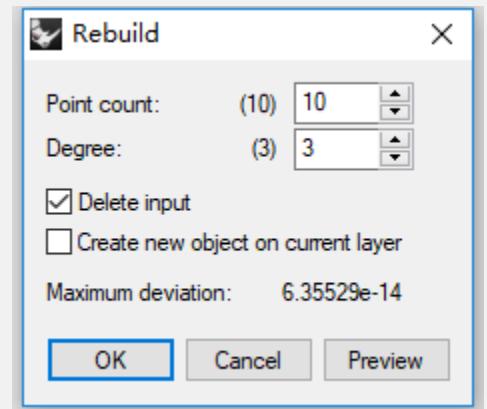


- To rebuild curves

- Click Preview to see what the rebuilt curves will look like.
- When you are satisfied with the results, click OK.

- Notes

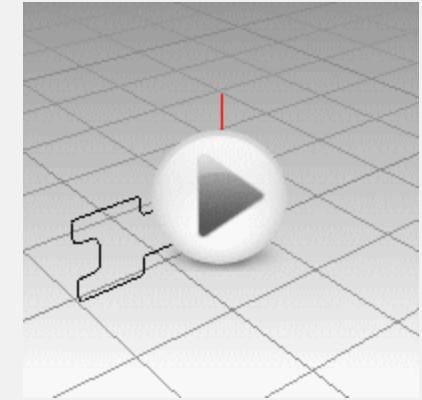
- Turn on control points and curvature graph to see the details of the curve structure.
- A group of curves will be rebuilt with curves of a specified degree and specified number of control points. The knot of the resulting curve more evenly spaced. To rebuild a curve with uneven knot spacing but a more accurate fit, use the FitCrv command.



# 1.6 NURBS Surface

## • 1.6.1 创建旋转扫掠曲面

- Command: \_Revolve
- Select curves to revolve:
- Select curves to revolve. Press Enter when done:
- Start of revolve axis:
- End of revolve axis (Press Enter to use CPlane z-axis direction):
- Start angle <0> ( DeleteInput=No Deformable=No FullCircle AskForStartAngle=Yes SplitAtTangents=No ):
- Revolution angle <360> ( DeleteInput=No Deformable=No FullCircle SplitAtTangents=No ):
- -----



## • Steps

- Select curves.
- Pick the start of the revolve axis.
- Pick the end of the revolve axis.
- Specify options.

# 1.6 NURBS Surface

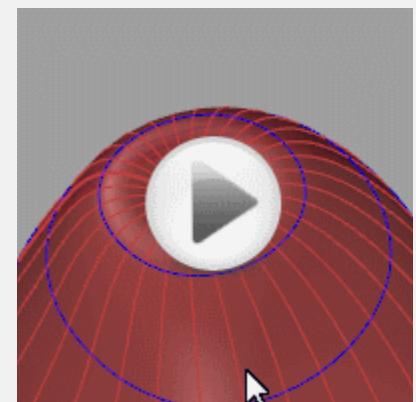
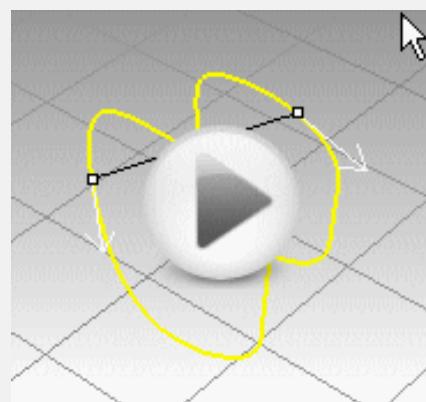
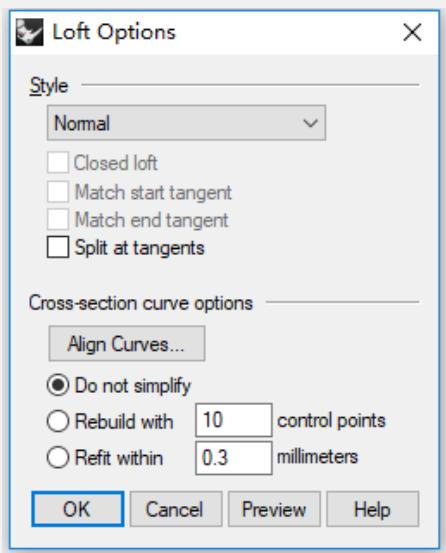
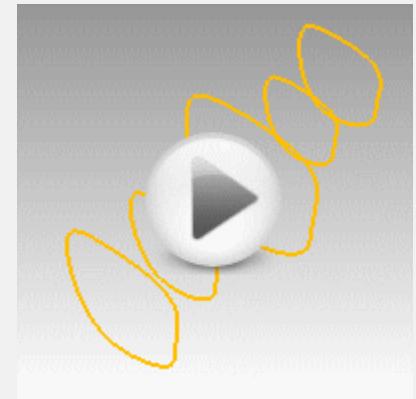
- 1.6.2 放样创建曲面  (最经典命令之一)

- Command: \_Loft
- Select curves to loft ( Point ):
- Select curves to loft ( Point ):
- Select curves to loft. Press Enter when done ( Point ):

• -----

## Steps

- Select the curves in the order in which the surface should pass through them.
- Select open curves near the same ends. For closed curves, adjust the curve seams.



# 1.6 NURBS Surface

- 1.6.3 使用4条边界创建曲面 

- Command: \_EdgeSrf
- Select 2, 3, or 4 open curves:

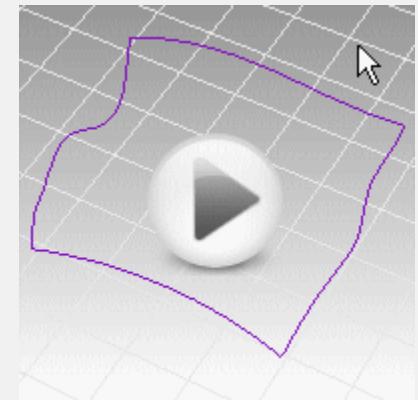
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- **Steps**

- Select the curves, and press Enter.

- **Notes**

- You can use surface edges as the curves for a new surface.
- For a surface from closed planar curves, use the PlanarSrf command.

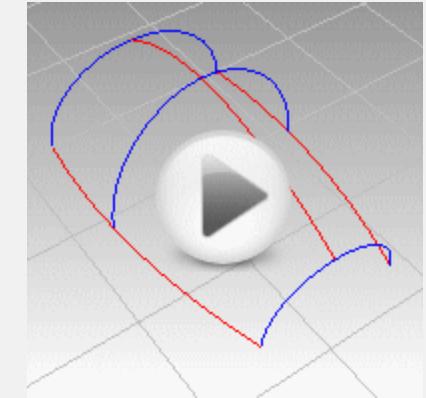


# 1.6 NURBS Surface

## 1.6.4 使用曲线网创建曲面



- Command: \_NetworkSrf
- Select curves in network ( NoAutoSort ):
- Select curves in network. Press Enter when done ( NoAutoSort ):
- Sorting curves...
- Creating network surface... Press Esc to cancel
- 



### Steps

- Select the curves, and press Enter.

### Notes

- Note: All curves in one direction have to cross all curves in the other direction and cannot cross each other.

#### Edge curves

- Sets the tolerance for the edge curves. The edges of the surface will be within this value from the edge curves.

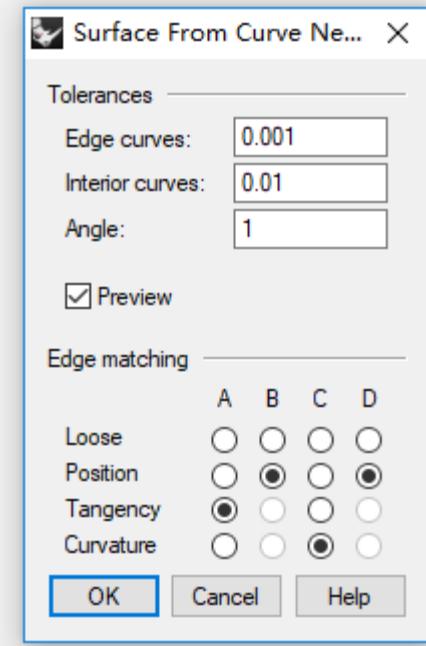
#### Interior curves

- Sets the tolerance for the interior curves. The interior of the curve's surface will be within this value.

- If the curves themselves are farther apart from each other than the tolerance values, the best guess is made at the surface.

#### Angle

- If the edge curves are surface edges, and you want the surface matching the adjacent surfaces with tangency or curvature continuity, this is the accuracy used to match the surface normals.



# 1.6 NURBS Surface

## 1.6.5 使用曲面去拟合一些点、线与网格

- Command: `_Patch` 
- Select curves, points, point clouds and meshes to fit surface through:

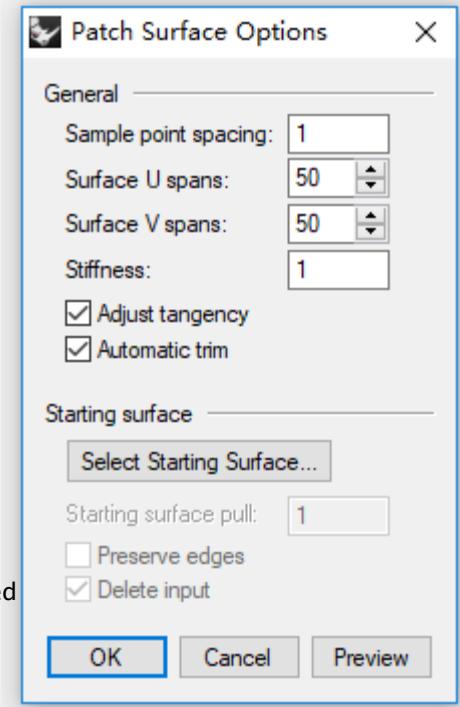
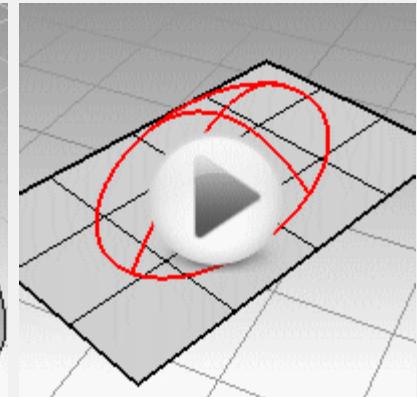
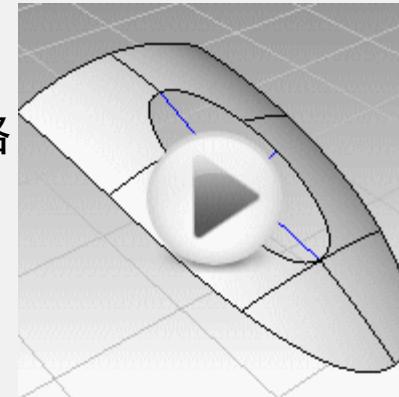
-----

### Steps

- Select point objects, curves, and edges to base the patch on.
- Specify the options.

### Notes

- Sample point spacing  
The nominal 3-D distance between points sampled from input curves. Minimum of 8 points per curve.
- Surface U spans  
The u-direction span count for the automatically generated surface. Also used if the starting surface is a 1x1 span plane.
- Surface V spans  
The v-direction span count for the automatically generated surface. Also used if the starting surface is a 1x1 span plane.
- Stiffness  
Rhino builds the patch surface by first finding the best fit plane (`PlaneThroughPt`) through the selected and sampled points along curves. Then the surface deforms to match the points and sampled points. The Stiffness setting tells how much you allow the best fit plane to deform. The bigger the number, the "stiffer" and more rectangular and planar the resulting surface will be. You can test this setting with small or even very big values (>1000). Use Preview to check the result.



# 1.6 NURBS Surface

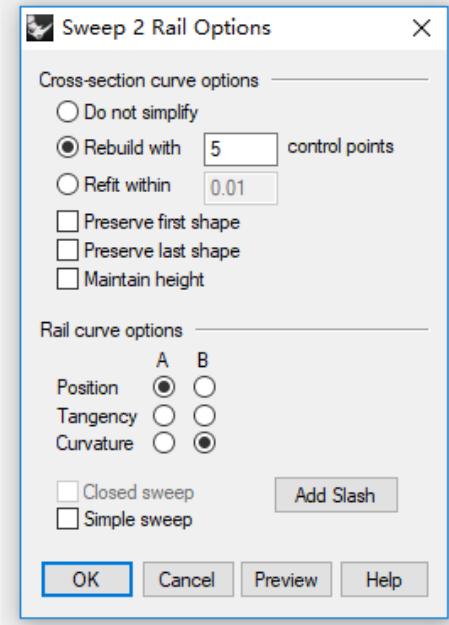
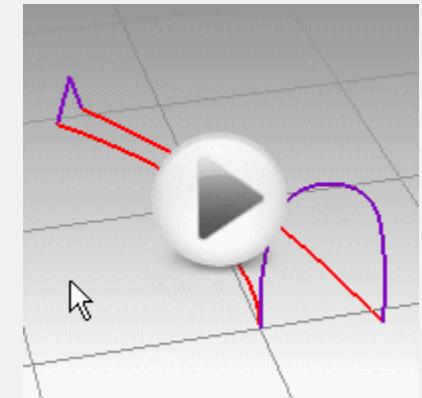
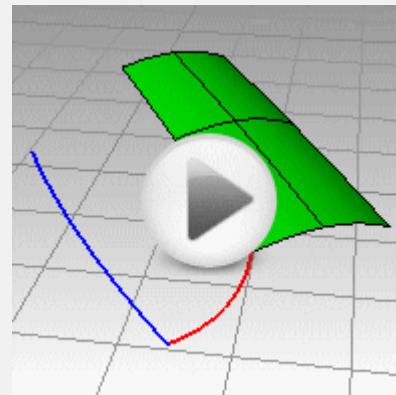
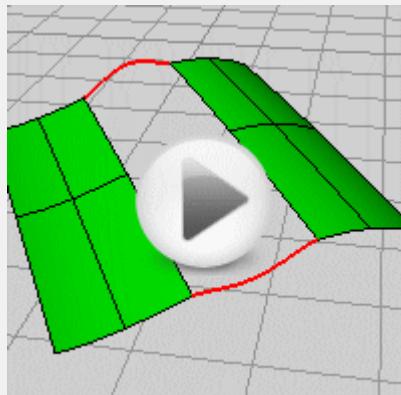
## • 1.6.6 使用sweep2创建曲面 (最经典命令之一)

- Command: \_Sweep2
- Select first rail ( ChainEdges ):
- Select second rail:
- Select cross section curves ( Point ):
- Select cross section curves. Press Enter when done ( Point ):

-----

## • Steps

- Select two curves as the rails.
- Select cross-section curves in the order in which the surface will pass through them.
- **Note:** Select open curves near the same ends. For closed curves, adjust the curve seams.



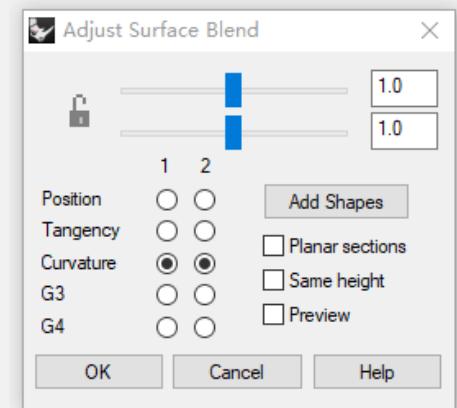
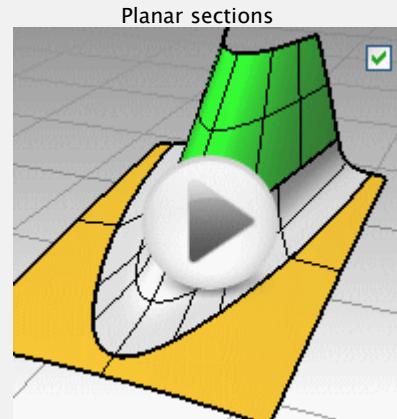
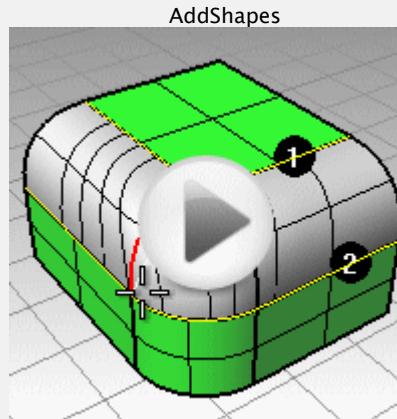
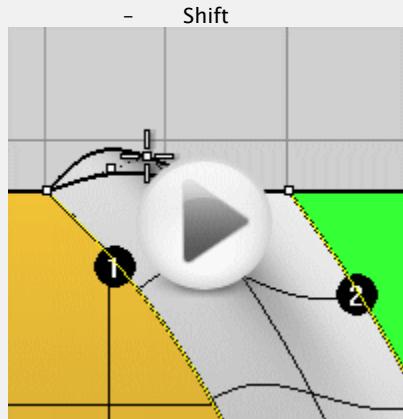
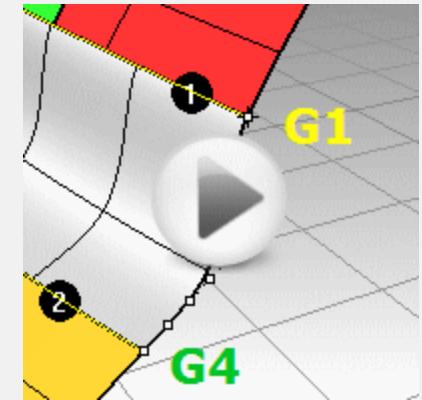
# 1.6 NURBS Surface

- 1.6.7混接曲面Blend Surface ⚙ (最经典命令之一)
- Command: \_BlendSrf
- Select segment for first edge ( AutoChain=No ChainContinuity=Tangency Direction=Both GapTolerance=0.001 AngleTolerance=1 ):

-----

## Steps

- Select a surface edge.
- Select adjacent edges or press Enter.
- Select the edges to blend to and press Enter.
- Select adjacent edges or press Enter.
- Select and adjust control points or press Enter.

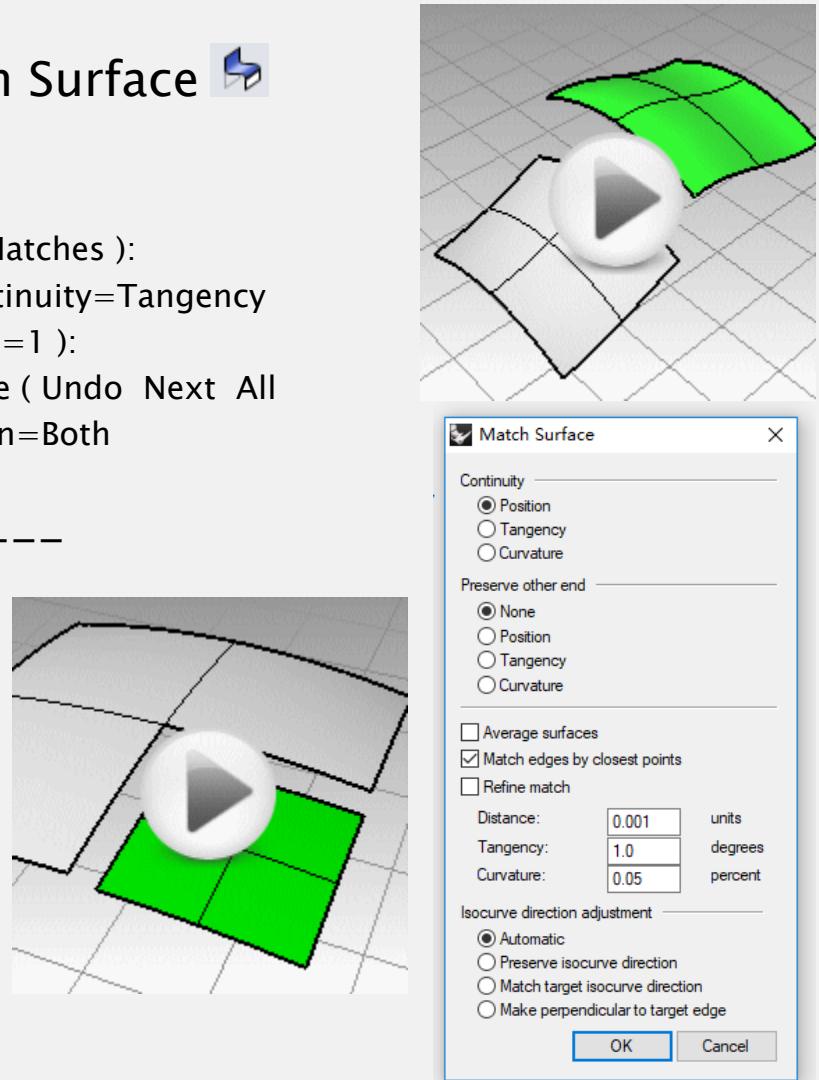


# 1.6 NURBS Surface

## • 1.6.8.1 混接曲面(相互混接匹配) Match Surface

(最经典命令之一)

- Command: \_MatchSrf
- Select untrimmed surface edge to change ( MultipleMatches ):
- Select segment to match ( AutoChain=No ChainContinuity=Tangency irection=Both GapTolerance=0.001 AngleTolerance=1 ):
- Select next segment to match. Press Enter when done ( Undo Next All )
- AutoChain=No ChainContinuity=Tangency Direction=Both GapTolerance=0.001 AngleTolerance=1 ):
- -----



## • Steps

- Select an untrimmed surface edge.
- Select a surface edge or curve to match.
- Pick the two surfaces near same ends. This surface can be either trimmed or untrimmed.
- Choose options.
- **MultipleMatche.**----->
- Allows selection of more than one edge to match.

# 1.6 NURBS Surface

- 1.6.8.2 混接曲面(相互混接匹配) Match Surface 

## Match Surface Options

- Continuity----->

- Sets the continuity for the match.

- Position

- Location only.

- Tangency

- Position and direction.

- Curvature

- Position, direction, and radius of curvature.

- Preserve other end-----\

- Changes the surface structure to prevent modification

- of the curvature at the edge opposite the match.

- None

- No constraint.

- Position

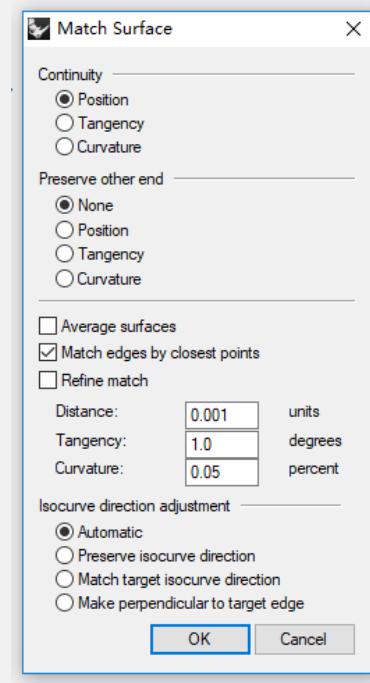
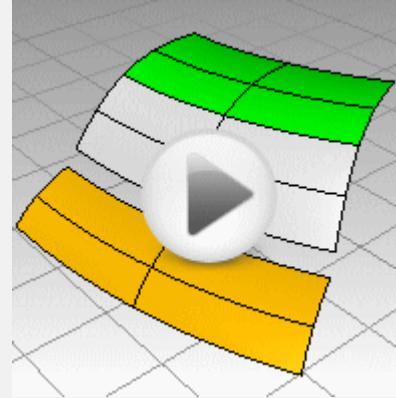
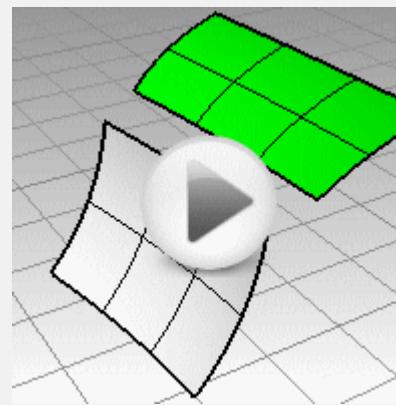
- Location only.

- Tangency

- Position and curve direction.

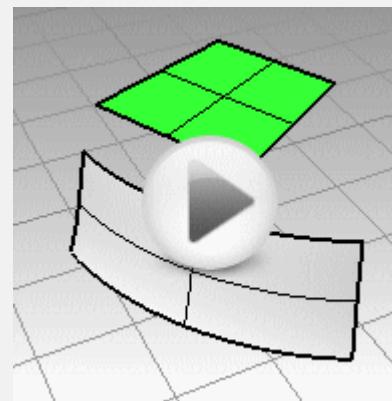
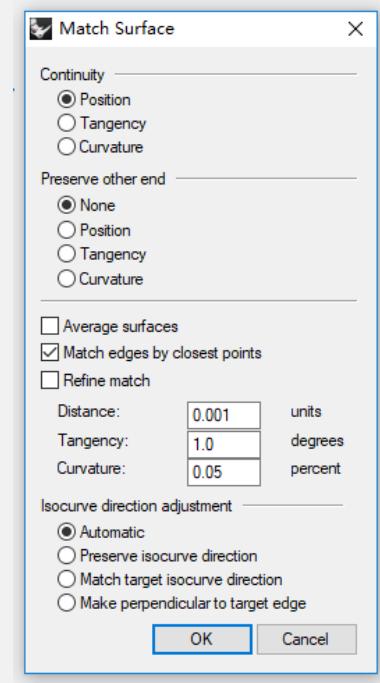
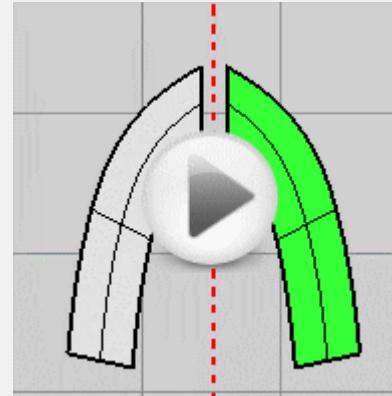
- Curvature

- Position, direction, and radius of curvature.



# 1.6 NURBS Surface

- 1.6.8.3混接曲面(相互混接匹配) Match Surface 
- Average surfaces----->
- Both surfaces are modified to an intermediate shape. If the target surface is also untrimmed, the surfaces match by averaging the two.



- Match edges by closest points----->
- Aligns the surface being changed to the target edge in two ways:
- Stretches or compresses the surface to match the entire edge end to end, or pulls each point object on the edge to the closest point object on the other edge.

# 1.6 NURBS Surface

## 1.6.8.4 混接曲面(相互混接匹配) Match Surface

### Refine match----->

Determines if the match results should be tested for accuracy and refined so the faces match within tolerance. If necessary, Rhino adds knot lines to the modified surface or surfaces until the surfaces are within tolerance.

### Distance \_\_ units

Position matching in model units.

### Tangency \_\_ degrees

Tangency matching.

### Curvature \_\_ percent

Curvature matching, in percent of the radius of curvature.

Specifies the way the parameterization of the matched surfaces is determined.

### Automatic----->

If the target edge is not trimmed, it works like the Match target isocurve direction option.

If the target edge is trimmed, it works like the Make perpendicular to the target edge option.

### Preserve isocurve direction

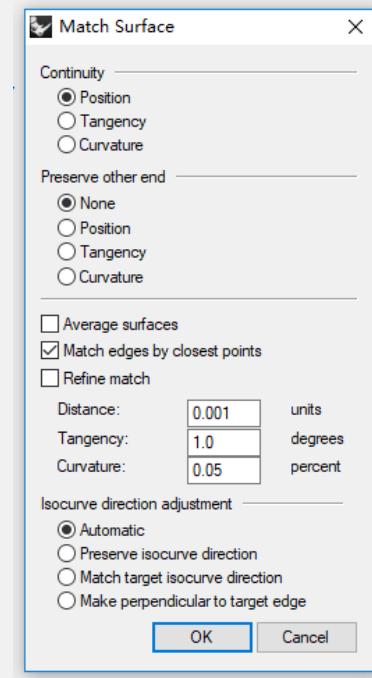
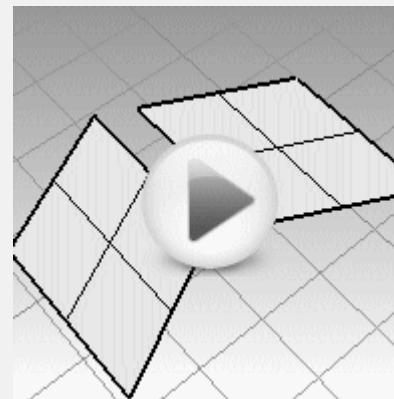
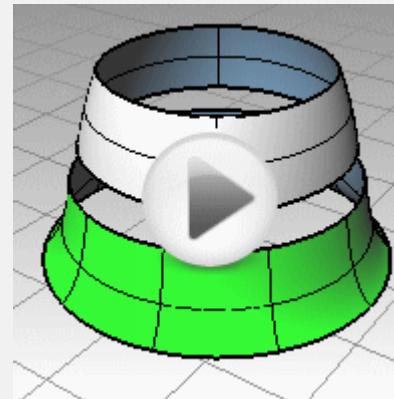
Does not change the existing isoparametric curves directions.

### Match target isocurve direction

The isoparametric curves of the surface will be parallel to those of the target surface.

### Make perpendicular to target edge

The isoparametric curves of the surface will be perpendicular to the target surface edge.



# 1.6 NURBS Surface

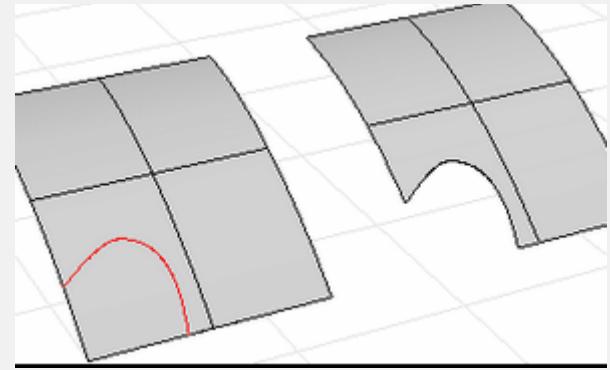
- 1.6.8.5混接曲面(相互混接匹配) Match Surface 
- Notes
  - The edge of a surface being modified must be an untrimmed edge.
  - A closed edge cannot be matched to an open edge.
  - Only single complete **edge** curves can be matched. If you need to match to part of an edge, trim the surface or split the edge (using the **SplitEdge** command).
  - MatchSrf is the most reliable when the surfaces are nearly matched already and require only a small amount of movement to get an accurate match.
  - MatchSrf can be useful for matches that are more similar to geometry creation than to fine-tuning. It is possible to move edges a long way and change their shape drastically, but it might take some experimenting to get what you want.
  - You can sometimes change the results of MatchSrf by adding or removing knots manually before you do the match using the **InsertKnot** and **RemoveKnot** commands.

# 1.6 NURBS Surface

## 1.6.9 裁剪曲面



- Command: \_Trim
  - Select cutting objects ( ExtendLines=No ApparentIntersections=No ):
  - Select cutting objects. Press Enter when done ( ExtendLines=No ApparentIntersections=No ):
  - Select object to trim ( ExtendLines=No ApparentIntersections=No ):
  - Splitting... Press Esc to cancel
- 

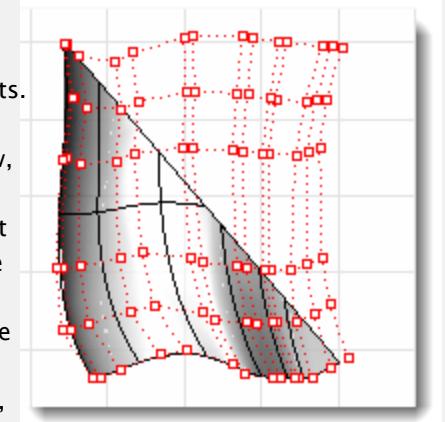


### Steps

- Select the cutting objects.
- Select the parts of objects to trim away.

### Notes

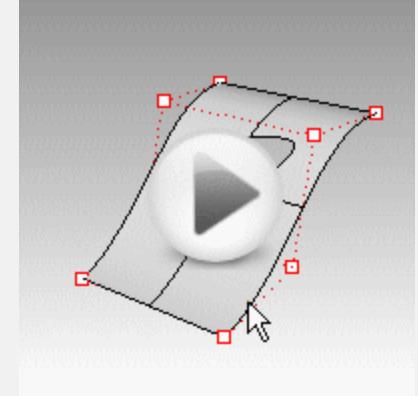
- If it is difficult to select the parts to trim off, use the Split command and then delete the unwanted parts.
- Use the Untrim command to remove a trimming boundary from a surface.
- When trimming a surface with a curve in a plan parallel view like the default Top, Front, and Right view, the cutting curve is projected on the surface in the view direction.
- When trimming a surface with a planar curve in an angled parallel or a perspective view like the default Perspective view, the cutting curve is projected on the surface in a direction perpendicular to the curve plane.
- When trimming a surface with a 3-D curve in an angled parallel or a perspective view, the cutting curve is pulled on the surface by closest points.
- A trimmed surface has two parts: a surface that underlies everything that defines the geometric shape, and trimming curves that either trim away the outside portion of the surface or cut holes in its interior.
- Those trimming curves exist on the underlying surface. The underlying surface may be larger than the trim curves, but you won't see the underlying surface because Rhino doesn't draw anything for the part of the surface that is outside the trim curves.
- Only the underlying surface defines the actual geometry of the shape. The trim curves do not define a surface -- they only mark which part of the surface is to be considered trimmed away.
- If you have a trim curve that runs diagonally across a surface, the trim curve itself doesn't have any real relationship to the control points structure of the surface. You can see this if you select such a trimmed surface and turn its control points on. You'll see the control points for the whole underlying surface, which doesn't necessarily have any connection with the trim curves.



# 1.6 NURBS Surface

- 1.6.10 为曲面/曲线增加控制点 

- Command: \_InsertControlPoint
- Select curve or surface for control point insertion:
- Point on surface to add control points ( Direction=U Toggle Extend Midpoint=No ):
- -----



- Steps

- Select a curve or surface.
- Pick a location for the control point or control point row.
- Note: Inserting control points affects the shape of the curve or surface. The InsertKnot command does not change the curve or surface shape.
- Press Enter to complete the command.

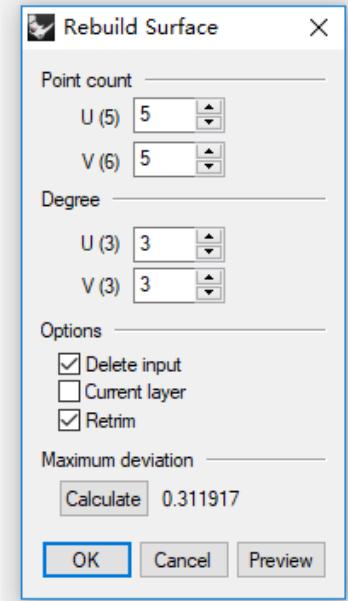
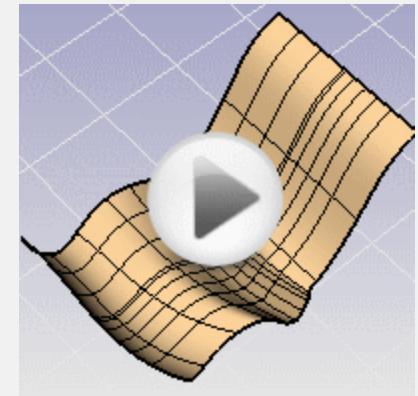
# 1.6 NURBS Surface

## 1.6.11 重建曲面

- Command: \_Rebuild
- Select curves, extrusions or surfaces to rebuild:
- Select curves, extrusions or surfaces to rebuild. Press Enter when done:
- 

### Steps

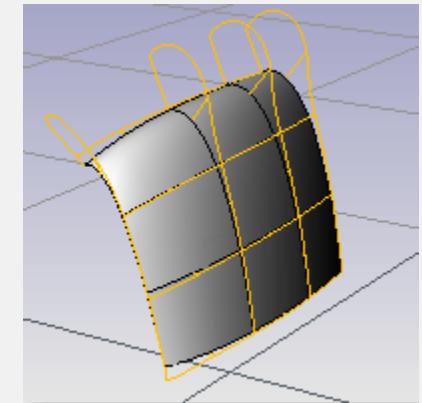
- Click Preview to see what the rebuilt surfaces will look like.
- When you are satisfied with the results, click OK.
- Calculate
- The calculation tests how far away the new surface is at knot line intersections and half-way between knot lines. Conducts tests at knot line intersections and halfway between knot lines.
- The display color indicates how far away the new surface is from the original.
- Points are green if the surface is within absolute tolerance, yellow if it is between tolerance and 10 times tolerance, and red if it is farther away than that.
- The lines are 10 times longer than the measured deviation in the direction of the deviation.



# 1.6 NURBS Surface

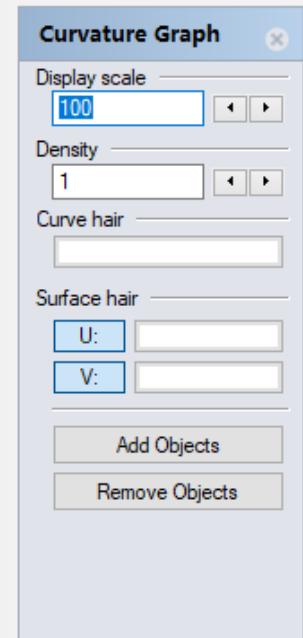
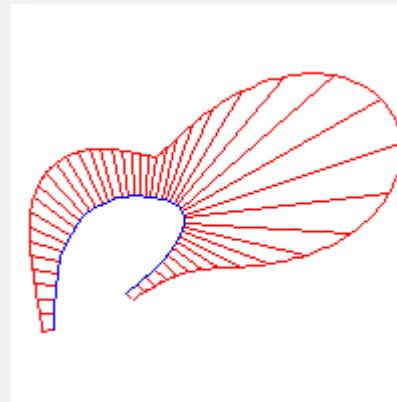
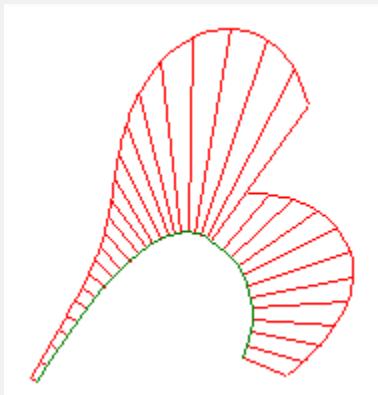
- 1.6.12 显示曲率图形 

- Command: \_CurvatureGraph
- Select objects for curvature graph display:
- Select objects for curvature graph display. Press Enter when done:



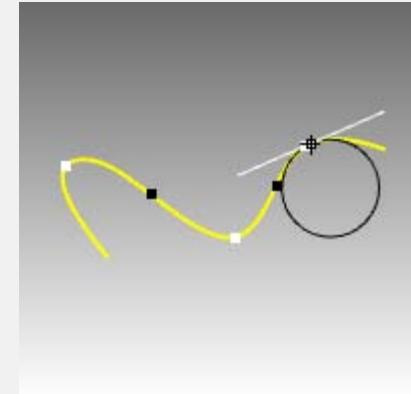
- Steps

- Select curves or surfaces.
- A graph showing curvature appears on the selected curves, and the Curvature Graph dialog box appears.
- Adjust the length, frequency, color, u- and v-direction display of the curvature indicators using the controls in the Curvature Graph dialog box.
- Even when other commands are started, the curvature graphs and the Curvature Graph dialog box persist until you turn them off.



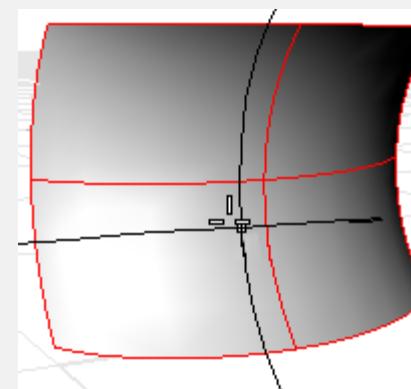
# 1.6 NURBS Surface

- 1.6.13 显示某点最大/最小相切圆  (右键)
  - Command: Curvature
  - Select curve or surface for curvature measurement:
  - Select point on surface for curvature measurement (MarkCurvature=No):
  - Surface curvature evaluation at parameter (63.3151, 24.0892):
    - 3-D Point: (-46.6978, 2.82284, 3.01725)
    - 3-D Normal: (0.0396095, -0.332032, 0.942436)
    - Maximum principal curvature: 0.0150292 (-0.312719, 0.891676, 0.327292)
    - Minimum principal curvature: -0.000658602 (0.949019, 0.307682, 0.068514)
    - Gaussian curvature: -9.89826e-06
    - Mean curvature: 0.00718529
- 



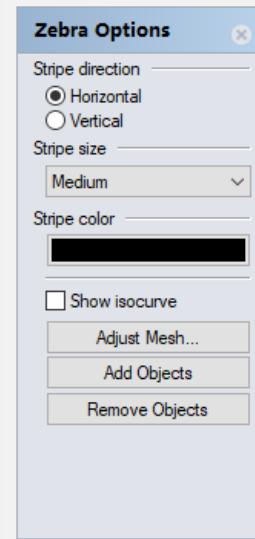
## Steps

- Select a curve.
- The curvature radius of the curve at the marker will display in the status bar, and a black circle of that radius will display tangent to the curve at the marker. A white line tangent to the curve will also display.
- White points mark the maximum curvature points in a portion on the curve where the curvature starts to decrease in both directions from the points.
- Black points mark the minimum curvature points where the curvature circle jumps from one side of the curve to the other side. The curvature at the black points is always 0.
- Pick to mark the curvature with a circle, or press Esc to end the command.

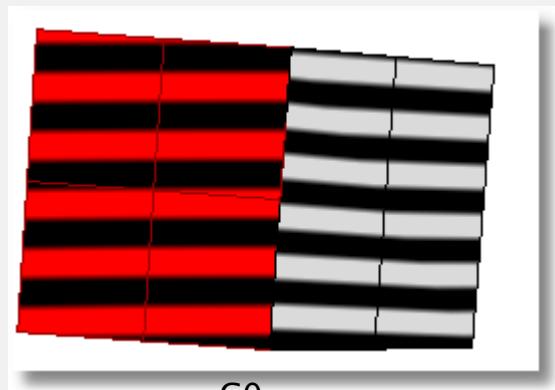


# 1.6 NURBS Surface

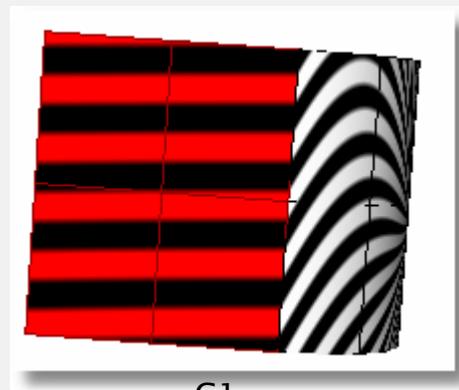
- 1.6.14 等高线环境贴图/斑马纹 Zebra 
- Command: \_Zebra
- Select objects for zebra stripe analysis:
- Select objects for zebra stripe analysis. Press Enter when done:
- Creating meshes... Press Esc to cancel
- -----



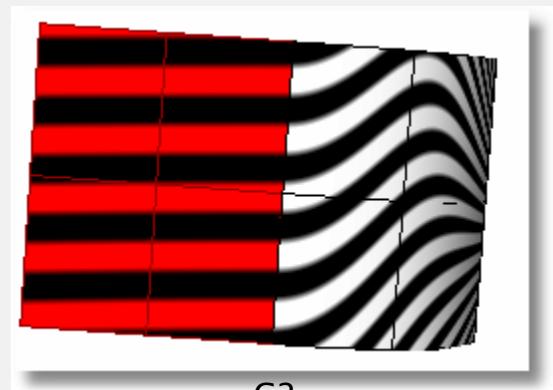
- Steps
  - Set the stripe direction, size, and color.
  - Set the stripe color to contrast with the base color of the object to see the zebra stripes.
  - The first stage is to set the detail level for the analysis mesh. You can increase the density of the mesh if the level of detail is not fine enough.



G0



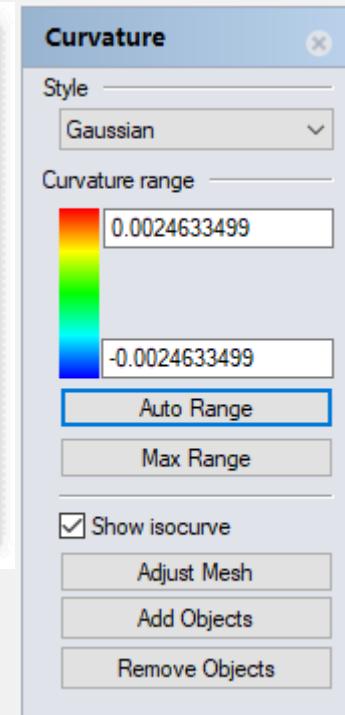
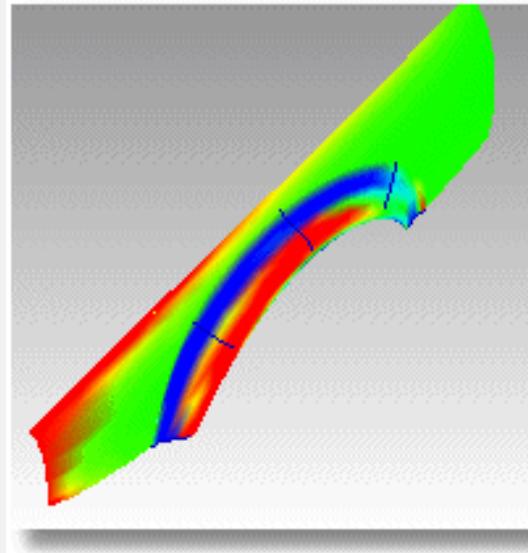
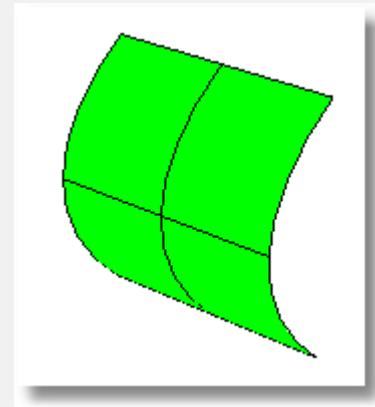
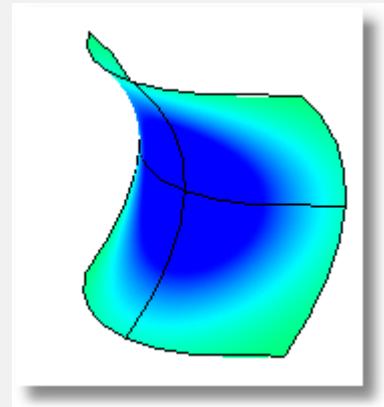
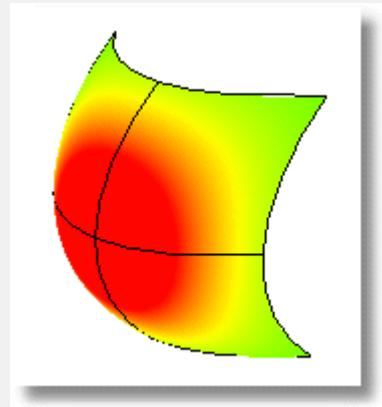
G1



G2

# 1.6 NURBS Surface

- 1.6.15 显示曲率颜色贴图 
- Command: \_CurvatureAnalysis
- Select objects for curvature analysis:
- Select objects for curvature analysis. Press Enter when done:
- Surface Curvature Analysis Summary:
  - Gaussian curvature: -0.014298 to 0.0027492
  - Unsigned mean curvature: 7.4116e-05 to 0.14077
  - Unsigned maximum radius of curvature: 24.045 to +infinity
  - Unsigned minimum radius of curvature: 3.5923 to 157.32
- -----
- Gaussian/Mean/Max radius
- /Min radius
- 



# 1.6 NURBS Surface

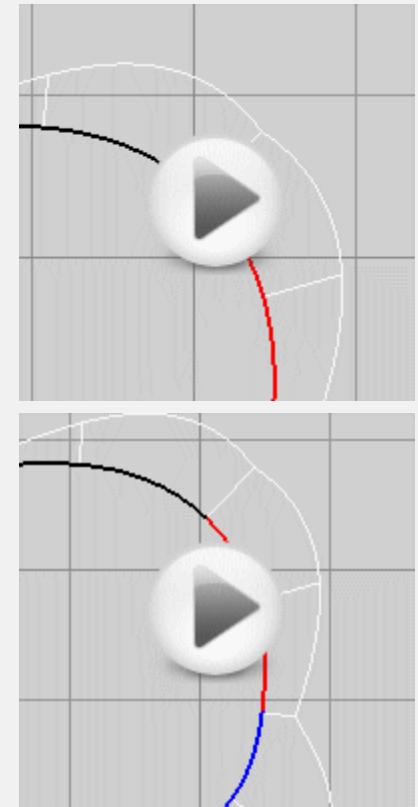
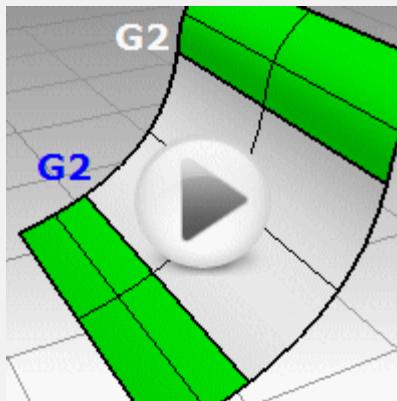
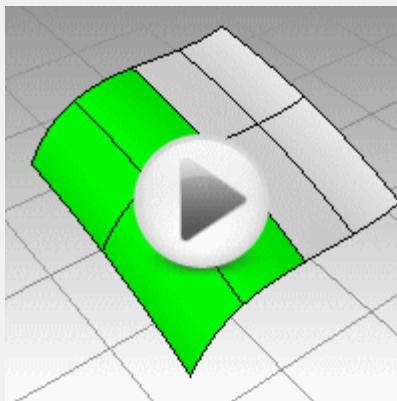
## • 1.6.16 调整切线



- Command: \_EndBulge
  - Select curve or surface edge to adjust:
  - Select point to move. Press Enter when done
  - (Continuity1=Curvature Continuity2=Curvature):
- 

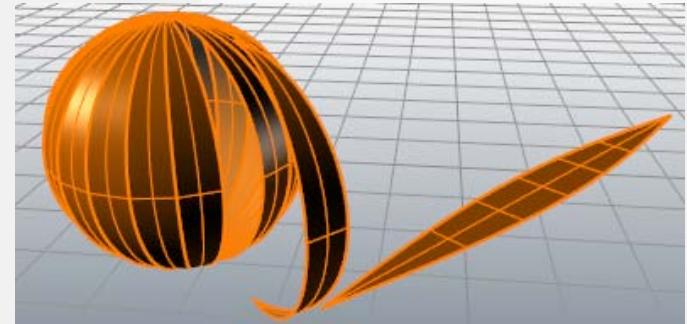
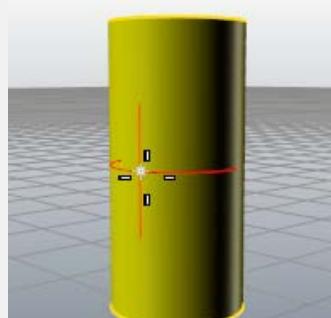
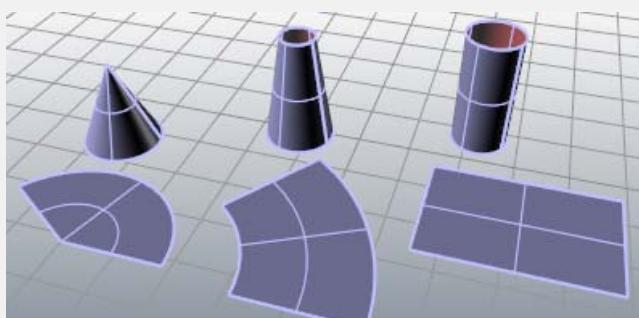
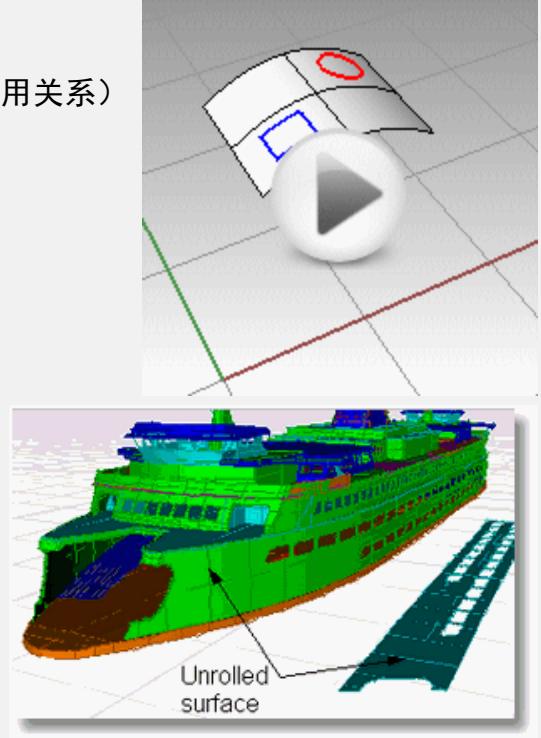
## • Steps

- The EndBulge command lets you edit the shape of a curve without changing the tangent direction and the curvature of the curve. This is especially useful with curves that conform to other geometry, as with the Blend command.
- The curve's control points are constrained along a path that keeps the direction and curvature from changing.
- Turn on the CurvatureGraph when using EndBulge to watch the graph change while the curve is being adjusted.



# 1.6 NURBS Surface

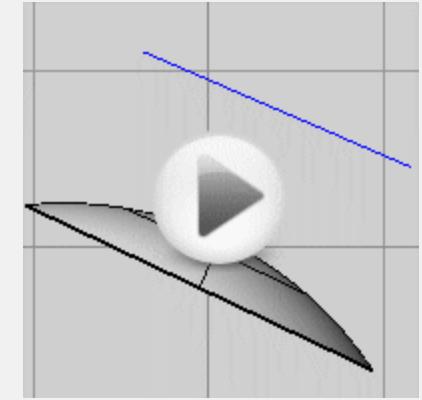
- 1.6.17 展平曲面 
- Command: \_UnrollSrf (后面有利用UnrollSrf解释Rhino命令与底层算法的调用关系)
- Calculating starting area... Press Esc to cancel
- Select curves on surface to unroll ( Explode=Yes Labels=No KeepProperties=No ):
- Calculating starting area... Press Esc to cancel
- Area is 1.2125 sq millimeters (0.05 %) bigger after unrolling
- One surface unrolled.
- -----



# 1.6 NURBS Surface

## • 1.6.18 曲线或点向曲面法线方向投影

- Command: \_Pull
- Select curves and points to pull ( Loose=No DeleteInput=No OutputLayer=Current ):
- Select curves and points to pull. Press Enter when done ( Loose=No DeleteInput=No OutputLayer=Current ):
- Select surfaces and meshes that pull ( Loose=No DeleteInput=No OutputLayer=Current ):
- Select surfaces and meshes that pull. Press Enter when done ( Loose=No DeleteInput=No OutputLayer=Current ):
- Pulling.
- ... Press Esc to cancel
- -----



## • Steps

- Select curves.
- Select the surface to pull the curves back to.

## • Notes

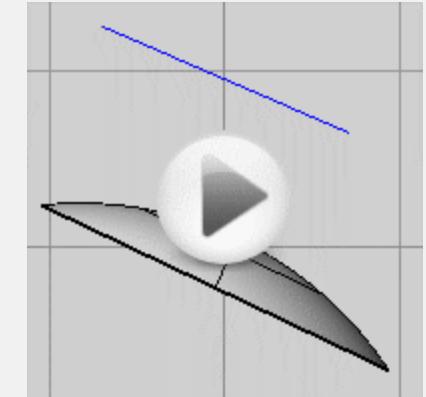
- When exiting the command, the newly created curves will be always be selected without regard for pre-selection.
- Use the Pull command to create complex trim curves, such as a curve that goes most of the way around a cylinder.
- Use the Project command if you know what the trim curve looks like from a single viewport.
- Use Pull command if you know where on the surface (in 3 D) the trim path should be. Use curve commands to draw the curve, drag the control points or edit points to move the curve near the surface. Then, use the Pull command to pull the curve onto the surface. You can also use the InterpCrvOnSrf command to create a curve on a surface.
- When drawing the curves, use the fewest control points possible. This guarantees the smoothest possible trim curve.

# 1.6 NURBS Surface

## • 1.6.19 曲线或点向曲面垂直投影



- Command: \_Project
- Select curves and points to project ( Loose=No DeleteInput=No OutputLayer=Current ):
- Select curves and points to project. Press Enter when done ( Loose=No DeleteInput=No OutputLayer=Current ):
- Select surfaces, polysurfaces and meshes to project onto ( Loose=No DeleteInput=No OutputLayer=Current ):
- Select surfaces, polysurfaces and meshes to project onto. Press Enter when done ( Loose=No DeleteInput=No OutputLayer=Current ):
- Projecting... Press Esc to cancel
- -----



## • Steps

- Select curves and points to project.
- Select surfaces and polysurfaces.

## • Notes

- You can select all the projection objects and target surfaces before starting the command.
- The curves are projected vertical to the construction plane that is active when the surface selection is completed.
- If the projection misses the selected surfaces and polysurfaces, a curve will not be created. Make sure the correct construction plane is active when you select the surfaces.
- The Pull command will suck the curve back toward the surface by closest points. The Project command will not work in situations where you want to pull a curve onto a cylinder when the curve goes most of the way around the cylinder. Use the Pull command in this case.
- The Project command creates complex curves that can be simplified with the Rebuild command. You will need to be careful with the Rebuild command and use enough points to keep the curve trimmable.
- The Project command can be faster than Extrude followed by Trim or Split.
- Smooth projection curves create smooth trim curves. Basic shapes like ellipses, circles, lines, and free-form curves work well for projection curves.

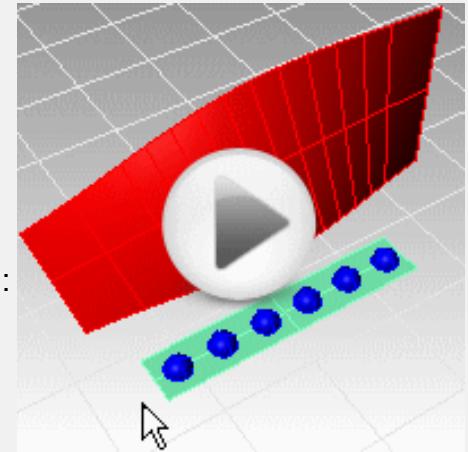
# 1.7 变形工具

## • 1.7.1 把物体沿曲面流动

- Command: \_FlowAlongSrf
- Select objects to flow along a surface:
- Select objects to flow along a surface. Press Enter when done:
- Base surface – select near a corner ( Copy=Yes Rigid=No Plane ):
- Target surface – select near matching corner ( Copy=Yes Rigid=No Plane ):
- Flowing objects along a surface... Press Esc to cancel
- -----

## • Steps

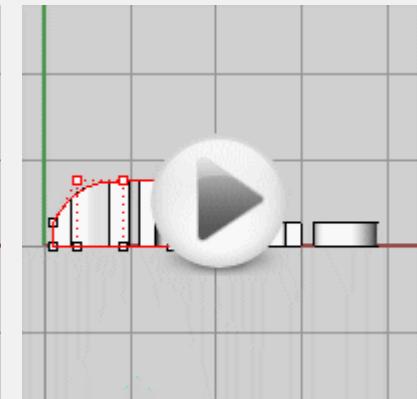
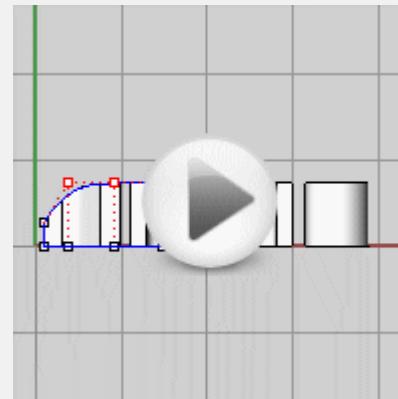
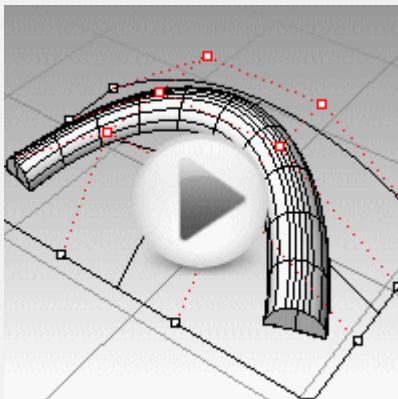
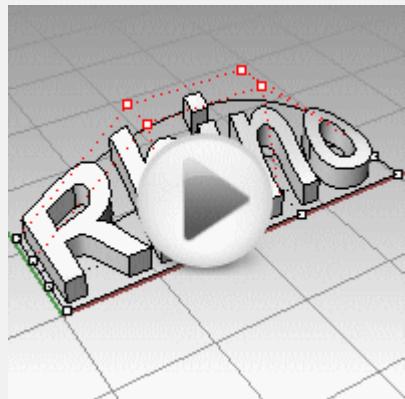
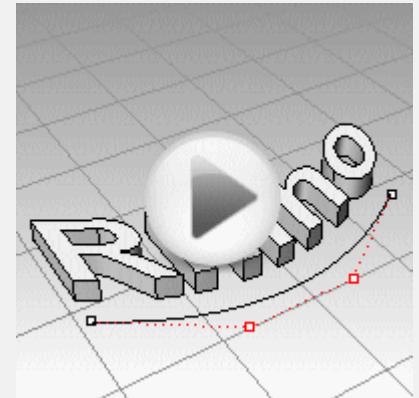
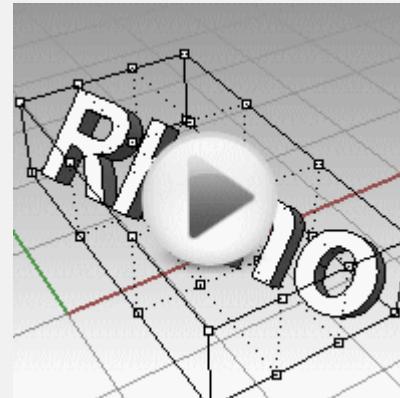
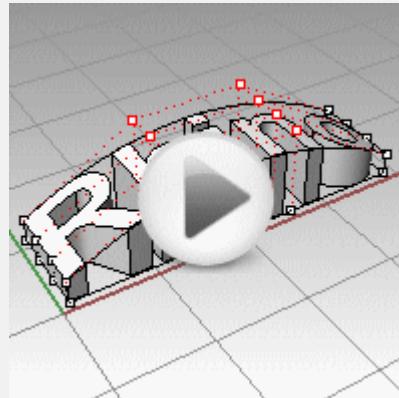
- Select objects.
- Select a base surface near a corner.
- Select the target surface near a matching corner.



# 1.7 变形工具

## • 1.7.2 控制点变形编辑 CageEdit

- Command: \_CageEdit
- Select captive objects:
- Select captive objects.
- Press Enter when done:
- Select control object (
- BoundingBox Line
- Rectangle Box
- Deformation=Accurate ):
- Region to edit <Global>
- ( Global Local Other ):
- -----



Rectangle

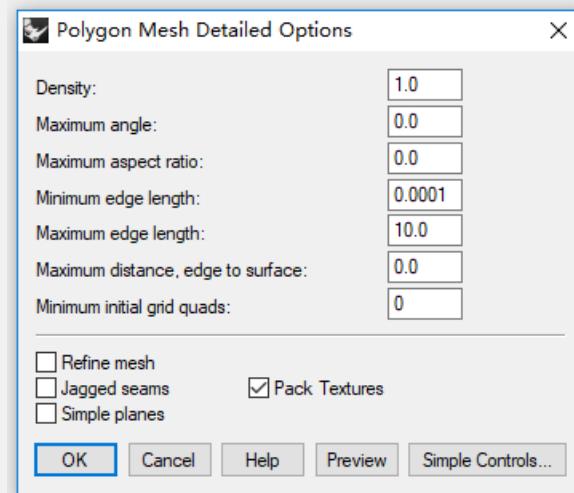
Accurate

Global

Local

# 1.8 网格工具

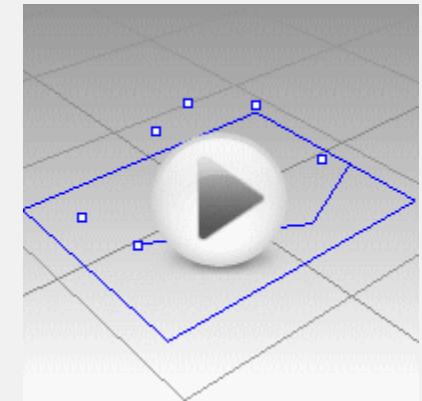
- **1.8.1 把NURBS曲面转换为网格（最常用之一）** 
- Command: `_Mesh`
- Select surfaces, polysurfaces and extrusions to mesh:
- Select surfaces, polysurfaces and extrusions to mesh.
- Press Enter when done:
- -----
- A mesh is a collection of vertices and polygons that define the shape of an polyhedral object. Meshes in Rhino consist of triangles and quadrilaterals.
- Rhino creates triangles and quadrilaterals meshes for export into various file formats. If a mesh is generated from a solid, the mesh will be seamless/watertight.
- **Notes**
  - The meshes created by the Mesh command are visible and editable, and separate from the NURBS objects they were created from.
  - The meshes created by any shaded viewport display mode on NURBS surfaces and polysurfaces are invisible. They can be updated from the NURBS object with the RefreshShade command, and can be extracted with the ExtractRenderMesh command.
- **Seamless (watertight or closed) mesh**
  - Rhino creates triangles and quadrilaterals meshes for export into various file formats. When surfaces are joined together in Rhino, the meshes along the joined edge have coincident vertices. If a mesh is generated from a solid, there will be no holes in the mesh. This is valuable for export to STL rapid prototyping files.
- **Mesh vertex**
  - The location where the edges of the mesh faces meet. The mesh vertex (plural vertices) contains x-, y-, and z-coordinates and may contain a vector normal, a color value, and texture coordinates.



# 1.8网格工具

## • 1.8.2 Mesh Patch网格插值

- Command: \_MeshPatch
- Select curves and points. ( AngleTolerance=15 StartingSurface ):
- Select holes. ( AngleTolerance=15 StartingSurface ):
- Creating mesh... Press Esc to cancel
- -----



## • Steps

- Select curves and points.
- Select closed interior curves to be used to create holes in the mesh, or press Enter to create a mesh with no holes.
- Closed curves used as the inner boundary are treated as holes.

## • Command-line options

### • AngleTolerance

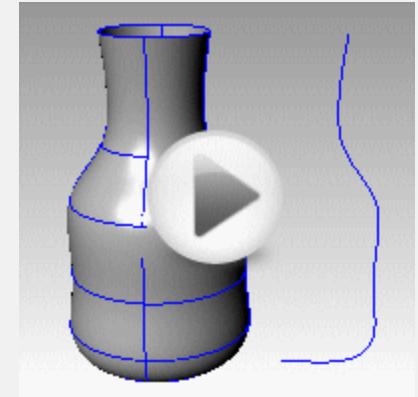
- Used to make polyline approximations of curves. If only polylines are selected, this setting has no effect on the final output.

### • StartingSurface

- Use a reference surface that is similar in shape to the mesh you are trying to create. The surface will influence the mesh shape.

# 1.8网格工具

- 1.8.3 Extract mesh from NURBS control polygon
- 得到NURBS surface的控制点网格 
- Command: `_ExtractControlPolygon`
- Select curves or surfaces to extract control polygon ( `OutputLayer=Current` ):
- Select curves or surfaces to extract control polygon. Press Enter when done ( `OutputLayer=Current` ):
- -----
- **Steps**
  - Select curves or surfaces.
  - This creates polylines and/or meshes that have vertices at the original objects' control points.
- **Options**
- **OutputLayer**
  - Specifies the layer for the results of the command.
- **Current**
  - Places the results on the current layer.
- **Input**
  - Places the results on the same layer as the input curve.
- **TargetObject**
  - Places the results on the same layer as the target surface.

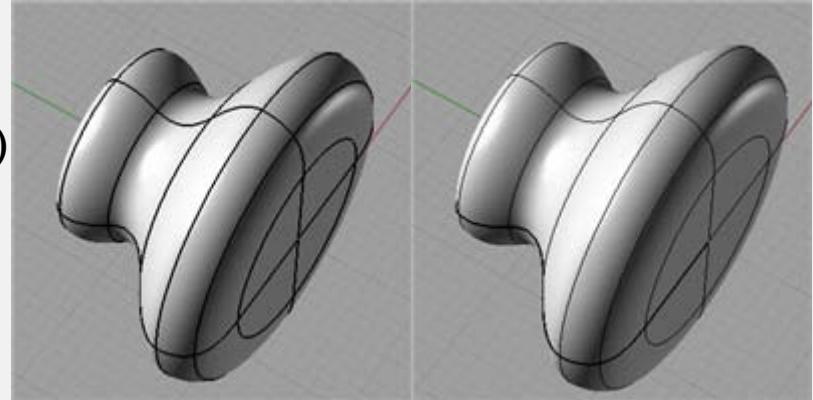


# 1.8网格工具

- 1.8.4其他网格工具简述

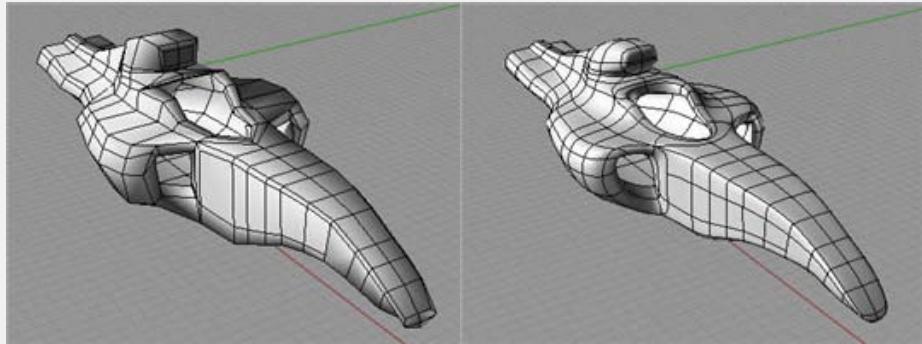
# 1.9 T-Splines T样条曲面

- 1.9.1 Ts工具栏最重要的工具1:  (左键)
- **tsConvert:** 曲面、网格转换为ts面
  - Converts an untrimmed NURBS surface,
  - Rhino mesh, or .obj file to a T-spline.
- **Converting NURBS to T-splines:**
  - Trimmed NURBS surfaces will lose the trimming data when converting to T-Splines. T-Splines surfaces don't support trimming information.
  - Any untrimmed, degree 3 NURBS surface can be converted directly to a T-spline, since T-spline surfaces in Rhino are all degree 3.
  - NURBS of degree 4 or higher can be converted to T-Splines, and will be rebuilt to degree 3 during conversion.
  - NURBS of degree 1 or 2 can also be converted to T-Splines. In the tsOptions, there is an option to "Rebuild when increasing degree" when converting degree 1 or 2 NURBS to T-Splines. If this option is checked, the surface will be rebuilt, which may change the surface slightly. If this option is not checked, the surface will be degree elevated. This will ensure that the surface shape does not change; however, there will be creased control points in the model which will make it difficult to smoothly manipulate the T-spline surface.



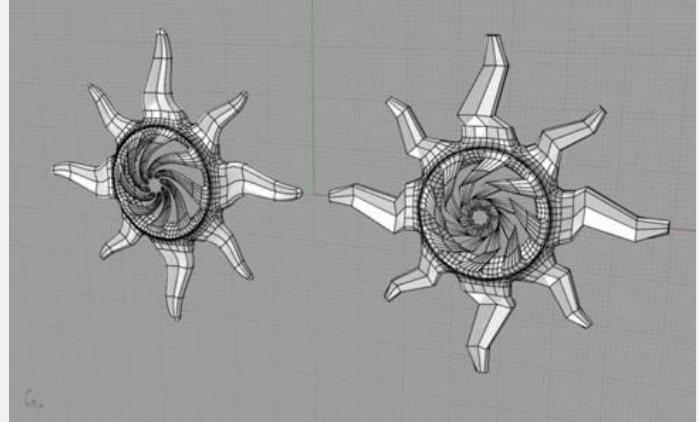
# 1.9 T-Splines T样条曲面

- 1.9.1 Ts工具栏最重要的工具1: 
- **tsConvert:** (左键)
  - Converts an untrimmed NURBS surface, Rhino mesh, or .obj file to a T-spline.
- **Converting meshes to T-splines:**
  - Conversion of meshes to T-splines is easy. Each mesh control point becomes a T-spline control point. The T-spline surface won't pass through the points, but will be a smooth surface derived from the points. The more points there are on the mesh, the tighter the T-Splines surface will be hugged to the control points.
  - Warning: .stl meshes
    - Do not attempt to directly convert .stl files to T-Splines. The resulting surface will be too large and Rhino will crash. Instead, you could try reverse engineering the mesh by using the tsAppend command with RetopoSnap on.
  - Warning: triangle meshes
    - Do not attempt to convert triangle meshes to T-Splines. The resulting surface will be too large and Rhino will crash. It is okay to have a few triangle faces in a mesh that has mostly four-sided faces, but if you have more than 20–50 triangle faces your computer will likely run out of memory.



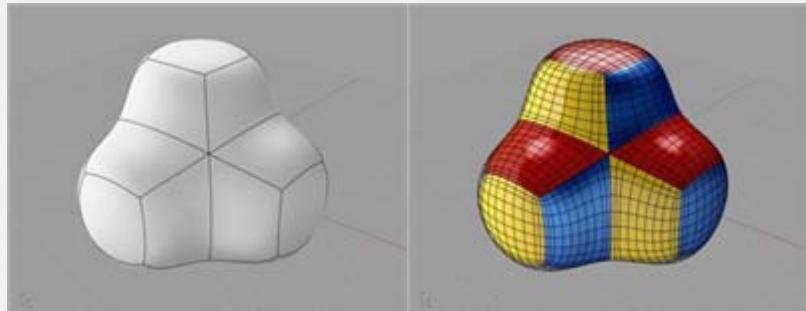
# 1.9 T-Splines T样条曲面

- 1.9.2 Ts工具栏最重要的工具2：
- **\_tsSmoothToggle:**
- Toggles the T-Splines surface between smooth mode and box mode. A
- T-spline can be viewed as a smooth
- surface or as a boxy mesh. On complex
- models, manipulating in box mode can be much faster, while smooth mode can be used to check the aesthetics and dimensions of the model. To toggle between box mode and smooth mode, select the object you would like to toggle, and press the tsSmoothToggle icon. If you are in edit mode, you can also press the TAB key to toggle all T-splines in the scene.
- 切换是smooth状态，就是在tsmesh与tssurface之间进行切换，也决定了这个ts在转换成rhino物体的时候是曲面(或多重曲面)还是网格



# 1.9 T-Splines T样条曲面

- 1.9.3 Ts工具栏最重要的工具3:  (右键)
- **tsConvertToRhinosurf:**
- Converts any T-Splines surface to a Rhino surface or mesh.
- The conversion is precise, will not deform the surface, and maintains continuity. If there are no star points in your surface, the T-Splines surface will convert to a single Rhino NURBS surface. If your model contains star points (and is therefore not rectangular), it will be converted to a polysurf.
- 可以把ts网格转换为rhino网格，或者把ts曲面转换为rhino曲面（多重曲面）
- 与\_tsSmoothToggle组合使用可以转换Rhino网格为Rhino曲面（多重曲面）
- 流程为选择Rhino网格->  (左键) ->  ->  (右键) /或\_Explode

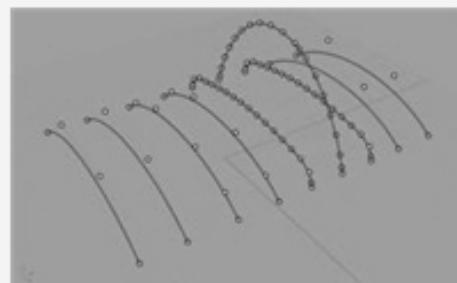
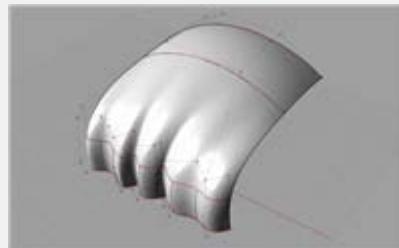


# 1.9 T-Splines T样条曲面

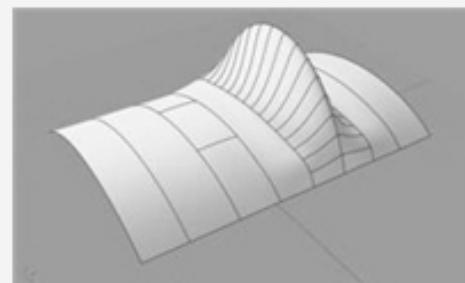
- 1.9.4 Ts工具栏最重要的工具4:

**\_tsLoft:**

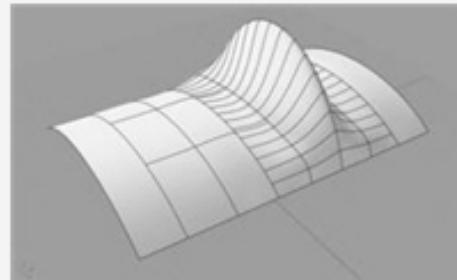
- Creates a T-spline surface that can contain varying amounts of detail.
- tsLoft keeps control points out of areas where they are not necessary, and can make for easier editing. The number of control points in the T-spline lofted surface is determined by each curve.



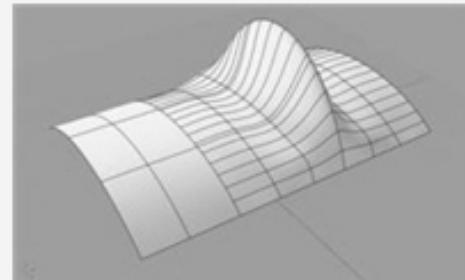
Curves



Minimal T-spline loft (few control points)



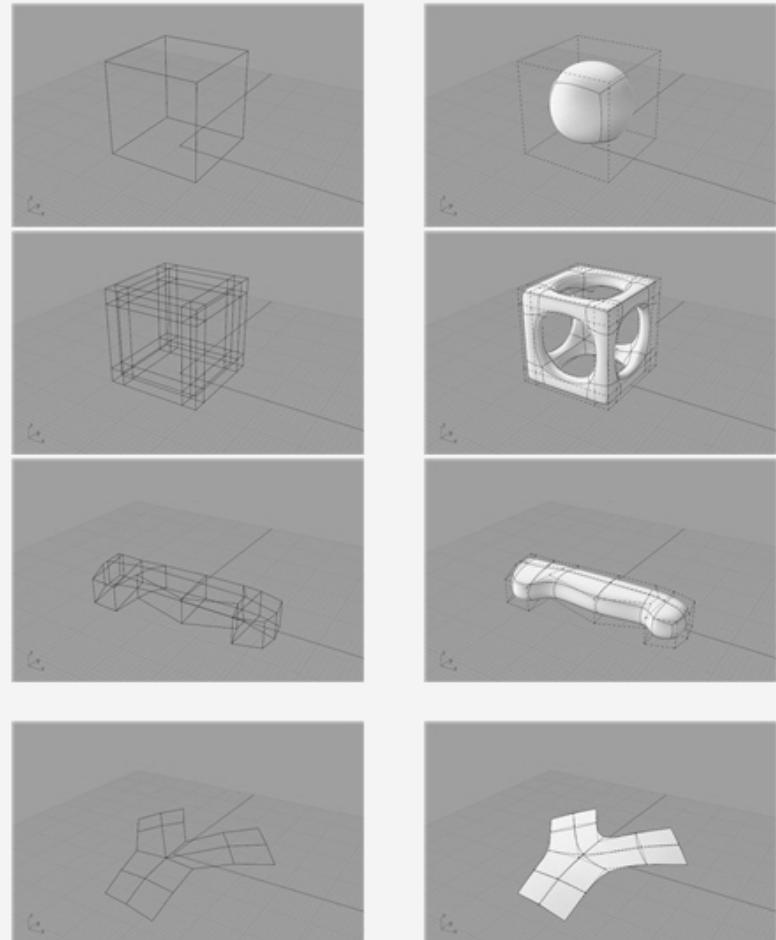
Moderate T-spline loft (more points)



Exact T-spline loft (most points)

# 1.9 T-Splines T样条曲面

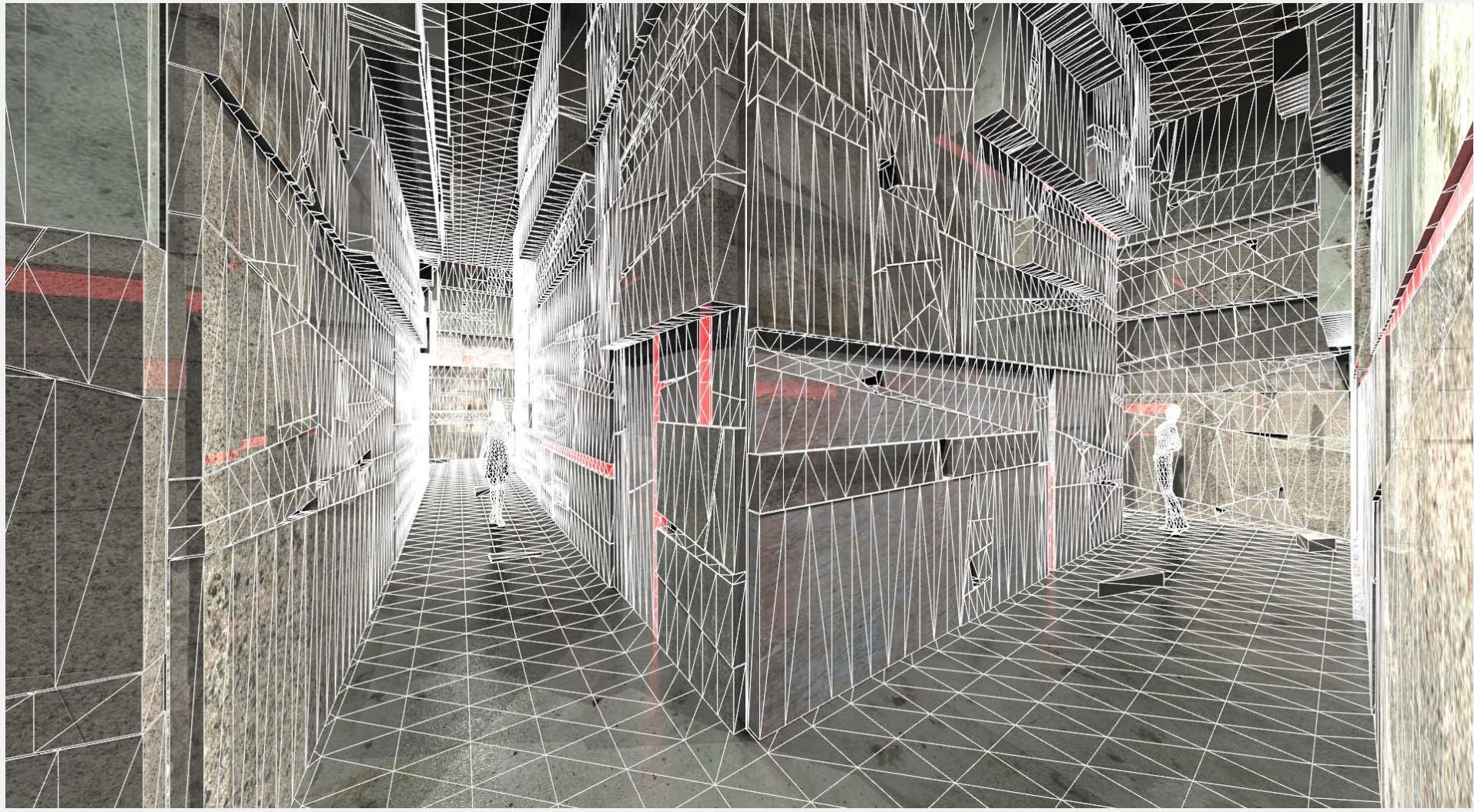
- 1.9.5 Ts工具栏最重要的工具5：
- **tsFromLines :**
  - tsFromLines provides a way to make complex surfaces with predictable results. It can also be used to create custom primitives that are closer to the desired final shape than the default primitives. It can generate a closed or an open single surface, rectangular or not, with holes and all kinds of different topologies.
  - To use tsFromLines:
    - Select one or more networks of lines or polylines and hit enter to create a preview mesh of faces. This preview is the best guess of the command at what your desired layout is, and is determined according to the MaxAutoFace option.
    - If the preview includes faces that you do not want to include in your final model, select (click on) faces to remove them from the preview mesh.
    - Click on edges to add faces to the preview mesh. A popup box of possible faces will appear; just select the desired face. You can control how many faces will display in the popup box using the MaxManualFace option.
    - Edges which are not connected to any faces are highlighted in red. Vertices which will result in a non-manifold mesh are also highlighted in red. Non-manifold vertices are automatically repaired, but may not give you the result you're expecting.



- 其他TS命令查看TS帮助手册
- 或RhinoHELP面板

# 1.10 渲染器

- 1.10.1 简易的带材质反射与支持Bump（凹凸贴图）贴图的渲染插件Neon



- 1.1



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# 1.10 渲染器

- 1.10.3 业界最受欢迎的渲染引擎：Vray



# 1.11 Rhino建模实例

- 制作笔筒

