Surface Modeling Tab

Introduction

The Surface Modeling tab of the modeling palette contains a number of tools for creating surfaces and polyhedral volumes. These tools let you build complex surfaces very simply, using 2D lines and curves as the basis of sections and profiles.

Most of these tools support dynamic geometry, so you can return and edit the original curves (if you are working in full or partial DG) and have your complex surfaces updated in real-time.

Ruled Surface

The ruled surface tool lets you create a surface between several existing curves which will become the sections of that surface.

The most common example is the creation of a face: create a group of vertical sections going from the nose to the outer ear, then with the ruled surface tool, select each curve one after the other and the face will be created.

From left to right: the sections, the surface creation and the final result.

Another example is the hood of a car: draw the lateral profile, the center profile, and maybe one in between if needed, and using this tool, select one after the other to create the hood.
On the left are the sections, and on the right the finished hood of car.

Usage:

- Create the curves to be used.
- Select the ruled surface tool, in the Surface Modeling tab.
- Click on each curve, one after the other, the tool creates the surface as you go.
- Validate the tool when finished.

Options:

- Connect the first and last curve: After selecting the last curve, click this option so the first curve is added, closing the surface.

Remarks:

- If you are using closed curves as your sections, the point that you choose on each curve is taken into account. To avoid getting a crossed surface, try to always select points that are aligned on each curve. A symbol appears on each selected point. If you find that the clicked point is causing problems, use Undo (Edit Menu or Ctrl-Z) and choose another point.
- The minimum curves allowed for this tool is two, and there is no maximum.
- Try to have the same number of points on all the curves. Although not required, it helps avoid crossed surfaces.
- Select the curves in the same order, i.e. from top to bottom or left to right, but not at random.
- Do not mix open and closed curves in the same surface. Though possible, it can give undesired results.
Double Sweep

The double sweep tool lets you create a surface or volume from a section and two profiles connected to it.

An example of this tool is creating a roof gutter, including rounded edges: using an arc for the section, and two lines for the profiles.

On the left, three lines making up the gutter, and on the right, the end result, after having clicked on the section line, and then selecting the double sweep tool and clicking on the two profile lines.

Modeling a rugby ball is also easy with this tool: a circle in the middle for the section, and two profiles connected at their ends and to the circle in the middle. The end result is a volume.
On the left, three lines making up the ball with the section in the middle, and on the right the end result after having clicked on the section line, and then selecting the double sweep tool and clicking on the two profile lines

**Usage:**

- Create the curves to be used.
- Select the curve that will become the section.
- Select the double sweep tool in the Surface Modeling tab.
- Click on the two curves to become the profile. The selecting order is not important.
- The surface is created. If the section is closed (for example, a circle) the tool proposes to close the surface to make a volume. White circles appear on the openings: click on them to close the side of your choice.
- Validate the tool when finished.

**Options:**

- ![Orient the sections](image)  
  Orient the sections: lets you choose how the surface is generated in relation to the section. If the section is not perpendicular to the profiles, the tool can either create the surface perpendicular to the profiles, or keep its original position.

- ![Constant or variable section](image)  
  Constant or variable section: lets you choose the type of section created opposite the first section.

**Remarks:**

- It is best to have the profiles connected to the section.
- The profiles don’t have to start or stop at the section. It is possible to have the profile lines going past the section, as in the case of the rugby ball. However, avoid having only one curve going past, if possible.
- For ease of further editing, try to have the same number of points on all the curves used by the double sweep tool.
- Do not mix open and closed curves in the same surface. Though possible, it can give undesired results.
The Coons surface tool lets you create a surface between a closed curve, or several open curves connected together.

This tool is very useful for the creation of complex shells, for example when two profiles and two sections are necessary. If we use the example of the car hood of the Ruled surface tool, we can see that it could be difficult to create such a complex form quickly.

Using the Coons surface tool, you can define a curve for the lateral part, one for the center, and one for the front, and finally another for the back. Hexagon will then create the surface.
On the left the support curves for the car hood, and on the right the finished surface.

**Usage:**

- Create the curves to be used by the Coons surface tool.
- Select the Coons surface tool in the Surface Modeling tab.
- Click on the curves. The selecting order is not important.
- Once all the connected curves are selected, the surface is automatically created.

**Remarks:**

- If no surface is created, make sure the curves are correctly connected at their endpoints.

**Gordon Surfaces**

The Gordon surfaces tool lets you create a surface defined by a series of latitudinal cross sections and longitudinal profiles. These must be created before the use of this tool.
This tool is used for the creations of complex surfaces often based on very precise curves, like a custom bottle for example.

The bottle could be made using a sweep or double sweep tool, but they may be too limited for a complex project.

Generally this tool is often used in product design.

On the left, the precise curves with connected sections and profiles, and on the right the generated result.

**Usage:**

- Create all the sections to be used.
- Create the profile curves making sure that each point passes through a point of the section curves, to construct a mesh that is connected. Use the Shift key when creating the lines to snap to existing points.
- Select the Gordon surfaces tool in the Surface Modeling tab.
- First click on all the sections and Validate by pressing the Enter key or press the Apply button in the tool properties palette.
- Click on all the profiles and validate again.
- Two things can happen:
  - The curves were correctly connected and the surface is automatically generated.
  - The curves were incorrectly connected, and the surface is generated as best Hexagon can, but due to the missing points, the results may not be satisfactory. You should cancel, verify the connections and start again.

**Remarks:**
- All sections and profiles must be connected.
- All ends of profiles must be connected to the first and last cross sections.
- Unlike the double sweep tool, it is impossible to have a profile go past a section.
- The tool does not have a “close” option. However, you can use the close tool, but this will flatten the dynamic geometry.
- For ease of editing later on, use the same number of points between sections and between profiles, even if it is not mandatory.

Extrude Line

The extrude line tool lets you create a surface or volume, from a section or a profile previously created. The result gives an object which the profile is positioned on each point of the section, in alignment with the center of the section.

Not to be confused with a “lathe” operation, which places the section at each point on the profile, while modifying its radius. With the extrude line tool, it is possible to extrude from a star shape, square, etc.

The extrude line tool is one of the most common and basic tools of surface modeling. A well-known example is a glass of wine: just draw a circle for the section and draw a profile matching the curve of the glass and you will quickly have a perfect result.
Usage:

- First method ? Extrusion from a profile:
  - Create a section and a profile. Be sure the section is perpendicular to the profile.
  - Select the section.
  - Select the extrude line tool, in the Surface Modeling tab.
  - Click on the profile, and the surface is automatically generated.
  - If the section is a closed curve (for example, a circle), the tool shows white circles to indicate the openings. Click on the opening to close it, and it becomes red.
  - Validate the tool to finish the operation.

- Second method ? Extrusion by freehand:
  - Create a section.
  - Select the extrude line tool, in the Surface Modeling tab.
  - Click on the section and move the mouse. The tool creates a second section which follows the mouse, connected to the first section by faces, where the radius and the height vary with the movement of the mouse.
  - Click to create as many sections as necessary.
  - Enter precise measurements for the height and radius for each new section in the tool properties palette if necessary.
  - Validate with the Enter key to fix the last section.
  - If the section is a closed curve (for example, a circle), the tool shows white circles to indicate the openings. Click on the opening to close it, and it becomes red.
  - Validate the tool to finish the operation.

Options:

For the freehand method, there are four modes of extrusion, where the height is the distance of the new section from the previous, and the width is the size of the new section:

- Extrude (default): the movement of the mouse defines the height and the width of the current section.
- Sweep: the movement of the mouse defines the height of the current section and the angle between the new and preceding section. Click the right mouse button to modify the width of the current section.
- Axial: the movement of the mouse defines only the height of the current section, which
stays the same size and is paralleled to the last section.

- 🖫 Radial: the movement of the mouse defines only the size of the current section. The current section stays at the same level as the previous.

Remarks:

- It is highly recommended to have the sections on one plane and the profile on another plane.
- It is best to start the profile on one of the “quarters” of the section: if the section is a circle, start the profile on the top, bottom, right or left, instead of on the diagonals, in order to have a profile in alignment with the center of the circle.
- Although it is not necessary to have a point of contact between the section and the profile, in certain cases like the wine glass, having the curves in contact will get the best results.
The sweep line tool lets you generate a surface or a volume from a section, and eventually a profile curve already drawn and at its center. The result is an object where the section is repeated along the profile, without modification of size.

The sweep line tool can be used for all sorts of tube forms, such as a door handle, or shower head.

On the left the section and profile and on the right the resulting surface.

The tool is most often used in a freehand method, which lets you change the size of each section, for example the shower head.

The creation of a shower head.

**Usage:**

- First method ? sweep from an existing profile:
  - Create a section and a profile. Be sure the section is perpendicular to the profile.
  - Select the section.
  - Select the sweep line tool, in the Surface Modeling tab.
  - Click on the profile, and the surface is automatically generated.
  - If the section is a closed curve (for example, a circle), the tool shows white circles to indicate the openings. Click on the opening to close it, and it becomes red.
  - Validate the tool to finish the operation.
- Second method ? Sweep by freehand:
  - Create a section.
Select the sweep line tool, in the Surface Modeling tab.
Click on the section and move the mouse. The tool creates a second section which follows the mouse, connected to the first section by faces, where the height and angle vary with the movement of the mouse.
Click to create as many sections as necessary.
Enter precise measurements for the height, angle and radius for each new section in the tool properties palette if necessary.
Click the right mouse button to modify the width of the current section if necessary.
Validate with the Enter key to fix the last section.
If the section is a closed curve (for example, a circle), the tool shows white circles to indicate the openings. Click on the opening to close it, and it becomes red.
Validate the tool to finish the operation.

Options:

Section alignment: Three kinds of alignment are available for the sweep tool. These options are available once the surface is generated, at the time Hexagon proposes to close the openings, but are not modifiable once the tool is validated.

- Irregular sweep: (default) each section is the same size, but the appearance may not be regular depending on the angle.
- Regular sweep: each section size may vary by angle in order to give the tube a unified size (parallels contours).
- Sweep with parallel sections: all the sections are parallel to the original.

These options are available for both methods of sweeping.

Modes of sweeping: For the freehand method, there are four modes of extrusion, where the height is the distance of the new section to the previous, and the width is the size of the new section:

- Sweep (default): the movement of the mouse defines the height of the current section and the angle between the new and precedent section. Click the right mouse button to modify the width of the current section.
- Extrude: the movement of the mouse defines the height and the width of the current section.
- Axial: the movement of the mouse defines only the height of the current section, which stays the same size and is paralleled to the last section.
- Radial: the movement of the mouse defines only the size of the current section. The current section stays at the same level as the previous.

Remarks:

- It is strongly recommended to have the profile at the center of the section, to have the most regular sweep possible.
- If the section is not perpendicular to the profile, the section might be automatically repositioned.
depending on the type of sweep chosen.
• Although it is not necessary to have a point of contact between the section and the profile, in
certain cases like the shower head, having the curves in contact will get the best results.

Boolean Operations

The boolean tool generates a surface or volume by using two objects (or group of objects) in an operation such as difference, union, subtraction, addition, (etc.) between the two objects.

The tool works on 2D lines as well as surfaces.

Boolean operations have always been used in 3D modeling, because this operation lets you add and subtract 3D objects easily, therefore make complex forms rapidly.

An easy example is making a hole in a wall for a window: you just make the rectangle for the window,
and subtract it from the wall in your scene.

On the left, the wall form with a polyline for the size of the window, and on the right, the resulting form.

**Usage:**

- Select the A element, which will be the first operator.
- Select the Boolean tool, in the Surface Modeling tab.
- Select the B element which will be the second operator.
- By default, Hexagon subtracts B from the A object.
- Choose the type of operation (union, difference, intersection ...) in the tool properties panel, or look at a preview of all the possibilities with the +/- keys.
- Validate the tool to finish the operation.

Note: you can also choose for element B a curve which can punch a hole in the form of element A.

For example, to create a perforated grid. Make multiple copies of circles over a plane, then use the boolean tool on them, and in a few clicks get the desired results. The circles are automatically transformed into a volume cutting into the first shape. In this case the depth of the cut can be regulated using the stretch tool.

**Options:**

A Boolean operation creates a surface or volume by combining in different ways the two elements. There are 12 possible combinations, accessible in the tool options panel, either by clicking on the chosen icon, or by using the +/- keys to cycle through the choices:

- Remove A from B
- Union of A and B
- Intersection (shared part) of A and B
- Remove B from A
- Intersection of the surface of A from B
- Intersection of the surface of B from A
- Surface of A removed by B
- Surface of B removed by A
- Contour of the intersection (creates a 3D line)
- All surface parts of the cut
- All surface parts of B sliced by A
- All surface parts of A sliced by B

Remarks:

- Boolean operations are not used for finalizing an object. The surface structure is often very dense and triangulated, and you should avoid operations after the boolean such as bevel and smoothing.
- Avoid modeling an organic object using the boolean tool, for example attaching the head of a dog to his body. Choose instead other techniques such as “box modeling” which allows for more control over the number of polygons.
- In some cases (multiple operations, etc...), the calculations can be long.
- An operative should not intersect with itself, like objects in a group that cross with each other.
- Choose to work with composite curves when possible instead of boolean operations between lines.
- In the case of a 2D line cutting into a 3D surface (punch):
  - Verify before using the tool that the 2D curves are well aligned with the object to cut.
  - In some cases (multiple operations, etc.), the calculations can be long.
  - The depth of the cut is done with the stretch tool, and not with the 3D manipulators. A green line appears, representing the depth of the cut, which by default goes past the object. Move the extremities of this line to adjust the depth dynamically.
  - When using the stretch tool to modify the depth of multiple cuts, one manipulator appears for all forms.
  - The cut is always made on a perpendicular plane (XY, XZ, YZ) or perpendicular to the camera plane if you are in a corresponding work mode.

Video
**Thickness**

The thickness tool lets you give a thickness to a surface, volume, or a curve.

In the example of a building, the walls can be extruded from flat planes, and then the thickness tool is used to give volume to the walls. You can also enter precise numbers, among other options.

On the left, a simple plane of the walls, and on the right with the added thickness.

To rapidly create plumbing, electric cables or springs, simply give a thickness to any 2D line.
On the left a 3D spiral, and on the right the spring obtained with the thickness tool.

**Usage:**

- When applying thickness to a surface or 3D volume:
  - Select the surface or 3D volume.
  - Select the thickness tool in the Surface Modeling tab.
  - Define the thickness in the tool properties palette, or use the +/- keys.
  - Use the orientation option to change the side the thickness is created on from interior to exterior, if needed.
  - Validate the tool to terminate the operation.
- When applying thickness to a 2D curve:
  - Select the curve.
  - Select the thickness tool in the Surface Modeling tab.
  - Define the thickness value in the tool properties palette, or use the +/- keys.
  - In the tool properties palette, define the number of sides making up the tube.
  - Validate the tool to terminate the operation.

**Options:**

- The orientation option lets you change the side the thickness is created on from interior to exterior if needed.

**Remarks:**

- The orientation by default is set by the normals of the object. If the normals are not unified, crossed surfaces could appear. Use the inverse normals tool to fix this.
- In the case of a closed volume (a cube for example), if you apply a thickness to the interior, it will not be visible and will increase the amount of polygons for the object. Avoid doing this unless, for example, you are planning to apply texture with transparency and refraction for the illusion of glass later in a rendering program such as DAZ Carrara.
Offset

The offset tool lets you create an offset surface, at a constant distance to the original.

When you need an identical form offset by a certain distance from the original form, the offset tool is very useful. Unlike the thickness tool, this one does not connect the two elements together, which lets you make a custom connection later, if necessary.

On the left is the original surface, and on the right is the offset surface.

Usage:

- Select the surface or 3D volume.
- Select the offset tool in the Surface Modeling tab.
- Define the distance of the offset in the tool properties palette, or use the +/- keys.
- Use the orientation option to change the side of the offset.
- Validate the tool to terminate the operation.
Options:

- The orientation option lets you change the side of the offset from interior to exterior.

Remarks:

- The orientation by default is set by the normals of the object. If the normals are not unified, crossed surfaces could appear. Use the inverse normals tool to fix this.
- In the case of a closed volume (a cube for example), if you apply an offset to the interior, it will not be visible and will increase the amount of polygons for the object. Avoid doing this unless, for example, you are planning to apply texture with transparency and refraction for the illusion of glass later in a rendering program such as DAZ Carrara.
- The created surface does not support dynamic geometry.

Smoothing

The smoothing tool gives a smoother appearance to your polygonal objects, whether a surface or curve. The result increases the number of facets describing the surface, or segments describing a curve.

This tool offers several types of smoothing, results of which vary depending on the chosen type.

In the majority of cases, this tool is applied as a finishing touch to give it a smoother appearance. In this way, a vase made of a section of six points and a profile of four points can be made less angular by applying this tool.

On the left is the original object, and on the right the object with smoothing applied.
The original object and the five kinds of smoothing.

Usage:

- The first method, on an element that has no smoothing:
  - Select the surface or volume.
  - Select the smooth tool in the Surface Modeling tab.
  - Change the type of smoothing in the tool parameters palette:
    - There are five possible choices of smoothing for surfaces and volumes. In most cases the default is the best for polygonal surfaces.
    - There are five possible choices of smoothing for curves. The smoothing by Bézier interpolation as well as subdivision are often the best.
  - Use these options as necessary to change the results of the smoothing.
- The second method, on an element that has already had smoothing applied:
  - Select the surface or volume that has already had smoothing applied.
  - Hexagon shows the parameters of smoothing as they were last applied.
  - You can now modify the tool as above

Options:

- ![Smoothing Range](image) Smoothing Range: the range of smoothing determines the number of facets produced. The higher the value, the finer the smoothing. To increase or decrease the smoothing, use the +/- keys or input the values directly in the tool properties palette.

- ![Smoothing Tension](image) Smoothing Tension: the smoothing have a control over the tension, or the concentration of edges or points when a smoothed object comes in contact with a non smoothed object.

Remarks:

- It is recommended to always work in low resolution or with a low polygon count, to see the model better and not slow down the program.
- Do not flatten the dynamic geometry at the smoothing level, except as a last step. It is
impossible to cancel smoothing if it is no longer dynamic unless you use Undo.

- If at all possible, avoid applying a range above three or four on the object. The higher the range, the slower the editing of the object will be. Choose a lower range (1 or 2) when editing and when necessary or when you have finished, choose a higher range.

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**Chamfer**

The chamfer tool lets you create smooth angles and corners, by rounding chosen edges or vertices of a surface, volume, or vertex of a 2D polyline.

The tool needs input of the radius, which is the circular section that makes the edge or vertex rounded, and the range, which is the quality of the rounded edge, or how many facets/points will be created to describe this curved edge.

Most real objects seldom have truly sharp edges. To get this effect, use the Chamfer tool to round off the angles.

An example is right in front of you: look at your computer screen, whose shell probably has rounded edges, whether for safety or design or both.
On the left, a corner of a computer screen before the chamfer, and on the right the corner after being rounded.

The tool can also be used in making simple or complex curves. The simplest method of making a square with rounded edges is by applying a chamfer to the four corners.
On the left a rectangle, and on the right with rounded corners.

**Usage:**

- The first method, pre-selecting your elements:
  - Select an object.
  - Select the points or edges to apply the chamfer.
  - Select the chamfer tool in the Surface Modeling tab.
  - Change the radius in the tool options palette and adjust the range of the chamfer, if necessary.
  - While the tool is in action, you can select or deselect other entities to be chamfered. Clicking on an entity selects it, and clicking on one already selected deselects it.
  - Validate the tool to finish the operation.

- The second method, selecting elements while in the tool:
  - Select an object.
  - Select the chamfer tool in the Surface Modeling tab.
  - Select the edges or points to be chamfered. Clicking on an entity selects it, and clicking on one already selected deselects it.
  - Change the radius in the tool options palette and adjust the range of the chamfer, if necessary.
  - Validate the tool to finish the operation

**Options:**

- 🛡️ Variable radius: Lets you input a value of a chamfer around the corners.
  - Select the option variable radius.
  - Select the corner which you want to have a different radius.
  - In the tool properties palette, enter a value for the radius

- Select all: Lets you select all the elements to apply a bevel to at once.

**Remarks:**

- Do not use too high of a range on the chamfer, if you are going to later apply a smoothing. This will make the topology too dense.
- If you are beveling corner vertices, making dice corners, the bevel could have strange results:
Weld Objects

The weld objects tool lets you join several objects together permanently, in order to have one object. Not to be confused with the group tool which is only a temporary association of objects. This tool is also located in the Vertex Modeling Tab.

Usage:

- Welding a selected group of objects:
  - Select all the objects to be welded.
  - Select the weld objects tool, in the Vertex Modeling tab.
- Welding objects selected one at a time:
  - Select an object.
  - Select the weld objects tool.
  - Now select the other objects to be joined. The selections are made in inverse mode: clicking on a selected object deselects it, and clicking an unselected object selects it.
  - Validate the tool to finish the operation.

Options:

- Select all: takes all the objects in the scene and welds them together.
- Keep the dynamic geometry: welds the objects together, while keeping the ability to edit further their DG tree.

Remarks:

- Once several objects are welded, the only operation that can take off the weld is Undo, unless
you use the keep the dynamic geometry option.

- This function works also between curves and between surfaces, but you cannot weld a curve to a surface.