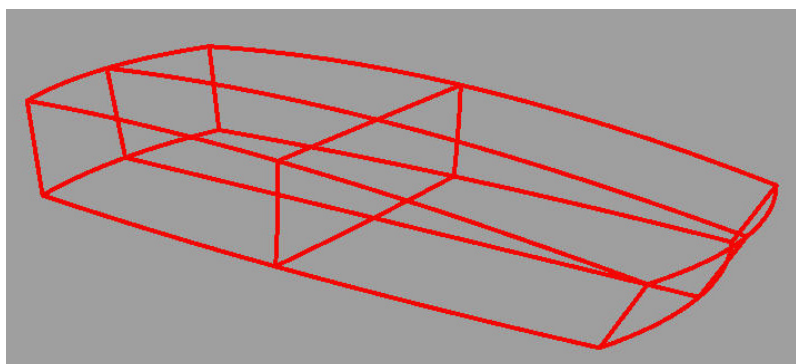




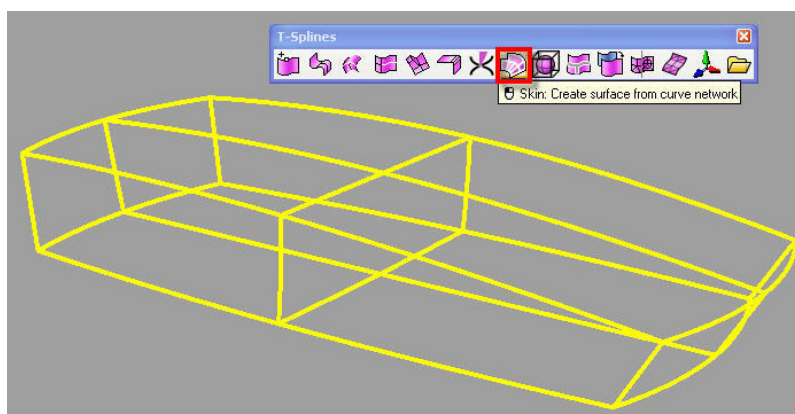
Surface from curves beginner tutorial (tsSkin)

The T-Splines for Rhino plugin brings industrial designers new ways to easily create and modify organic designs. One T-Splines workflow is to create entire, editable, NURBS compatible surfaces from input curves using the tsSkin command. This beginner tutorial will show you how to use the tsSkin command as you create a handheld device. Special thanks to Gustavo Fontana for the curve network.

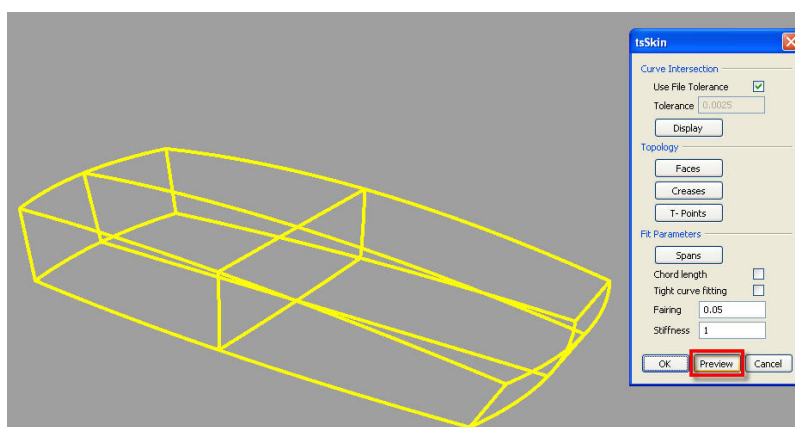
First, lay out curves to define the surface.



Select all the curves and enter the tsSkin command.

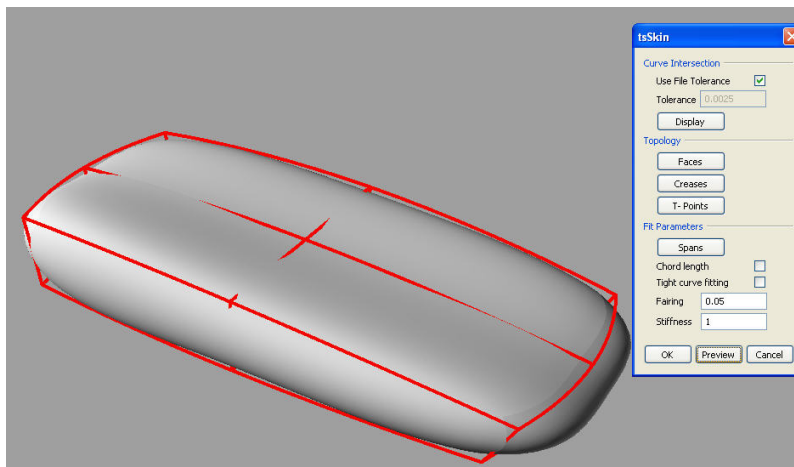


There are many options here to help resolve the different ways the selected lines can be used to form a surface. To see if we need to use these options, hit "Preview."

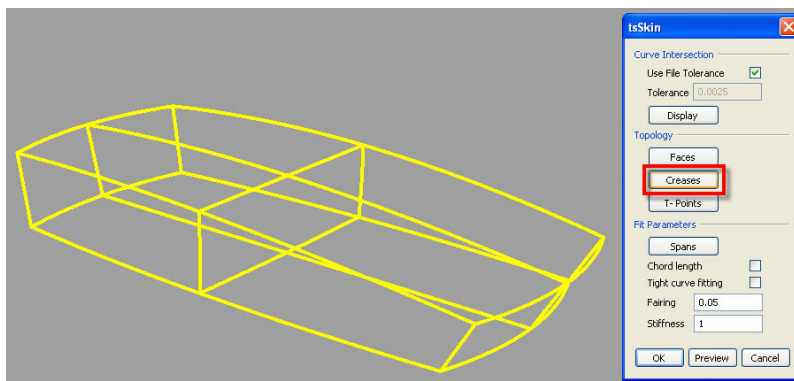




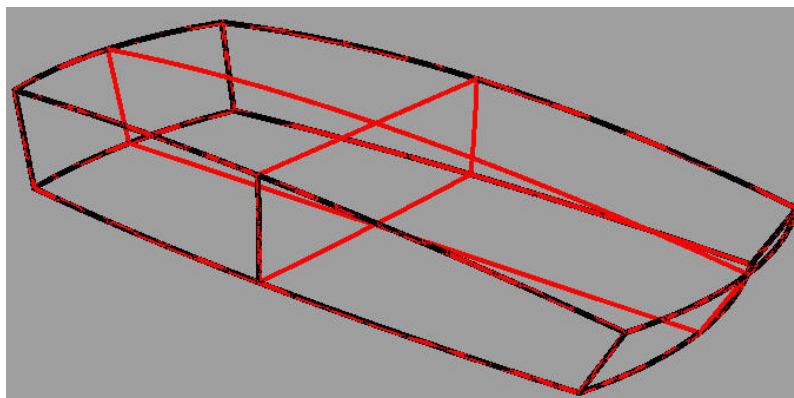
The generated surface looks similar to what we want, except that we would like to have the edges creased.



To crease edges, hit the "Crease" button.

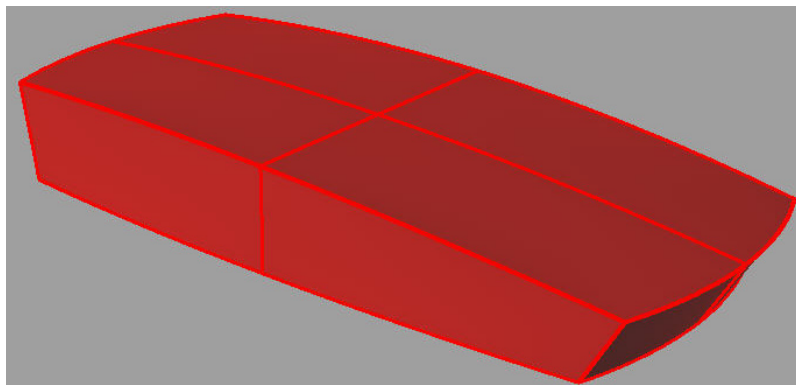


Click on edges to designate them as creased edges.

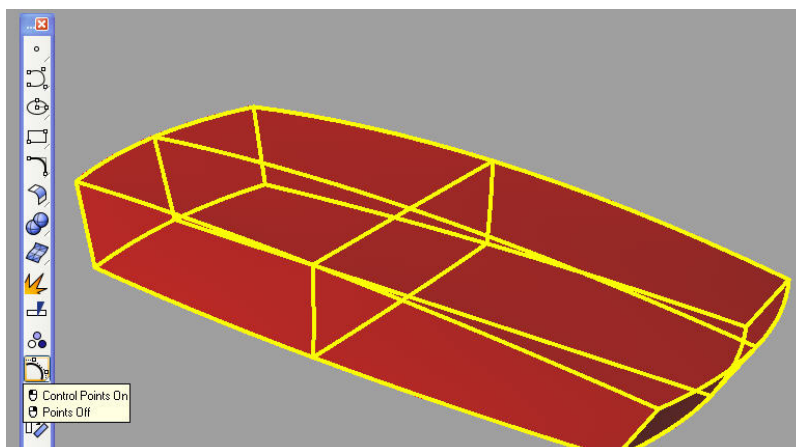




Now hit “Preview” again. This is the desired surface, so we will not use the other options. More advanced tutorials cover how to use these options to fix issues like curves that don’t intersect, or to remove faces to add holes to the object. Hit OK to generate the surface and complete the command.



Because this is a single T-Splines surface, not a polysurface, control points can be turned on.



The surface can be edited using tools like the T-Splines manipulator.

