Creating a “Nozzle” from the Loft Tutorial
(Draft 1, 5/12/06)

We want a slightly larger cylindrical nozzle to have the interior shape of the original SolidWorks loft tutorial. Assuming the nozzle will be as long as the lofter fluid, add a temporary base handle for (visibility of the) Boolean operation:

Now build the larger cylinder centered on this loft by inserting a new sketch in the front plane, view it normal to that plane, insert the larger circle.
The outer nozzle will be larger, say 100 mm:

![Figure 3](image)

Extrude the cylinder **Up to surface** of inner loft, turn off "merge result" to form two overlapping centered bodies:

![Figure 4](image)
The result looks like one body, but its two overlapping ones:

![Figure 5](image)

Now do the Boolean (combine) cut to remove the original loft from the cylinder:

![Figure 6](image)

Check the preview, and accept the new “nozzle”:
Now let's double check some sections:

Figure 7

Figure 8

Figure 9
Note that it possible to get the same final result in a reverse order. First you would establish the same planes, extrude the solid cylinder, and complete the same four loft plane sketches. Now you want to form the loft as a cut out.

![Figure 10](image)

**Insert⇒Cut⇒Loft**, create the loft (with or without the extra base handle) and select the cylinder as the part to cut. The same nozzle part appears.