Electronic Supplementary Material (ESI) for Analytical Methods.

Supplementary Information

3D-printed Microfluidic Devices: Fabrication, Advantages and Limitations—a Mini Review

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In these examples, we used AutoDesk Inventor Professional 2015 for the design of useful structures for microfluidic devices, but any CAD software can be used. Here we present instructions for design of a 3D-printed device with a: (1) a straight microfluidic channel; (2), female threads to accommodate a commercial PEEK fitting (F-120x, IDEX) and a capillary sleeve (F-230, IDEX); (3) a flow splitter; (4) holes for well inserts into a device (a well plate); (5) a mold for PDMS channel casting; and (6) combining multiple designs into a single device such as fitting threads and a microfluidic channel. The .stl (directly 3D-printable) and .ipt (editable in a CAD software) files for the 6 examples are also included as supplementary information.

Example 1. Design of a straight microfluidic channel – These instructions show how to make simple channel designs into an object

The design of this structure is provided as “straight microfluidic channel. ipt” and “straight microfluidic channel. stl” in a separate supplementary information folder.

Below are the steps to design this structure in Autodesk Inventor:

Open AutoDesk Inventor Design Software

1. Start a new file
    a. Select New
        i. Standard (mm).ipt file
2. A base structure must be initially created to cut the channel into
    a. Select Start 2D sketch and pick a work plane
        i. Make 2-dimensional shape of desired dimensions to create one side of the device(Ex. 16mm x 16mm square)
            1. To adjust the dimensions of a shape, select constrain dimension and click on the side to be adjusted
    b. Finish the 2D sketch and select the extrude function
    c. Extrude to 3D structure with desired dimensions (Ex. 16 mm)

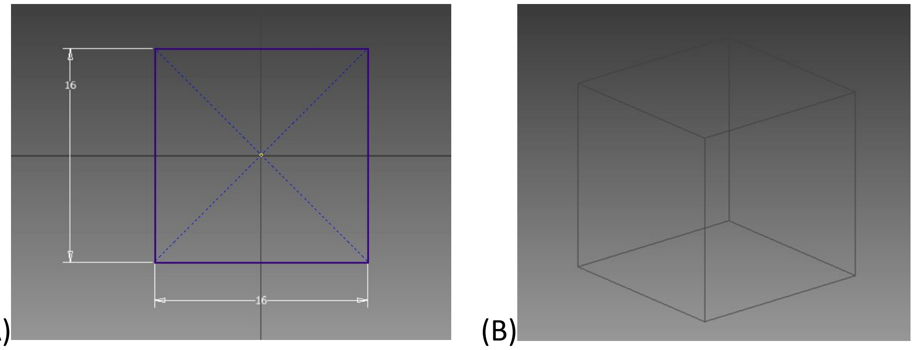
(A) (B)


Figure S1: 2-Dimensional sketch of a square for extrusion into a 3-dimensional cube. (A), 2-D sketch of square (B), Extruded square into cube

3. A 2-Dimensional sketch of the cross section of the channel must now be created
   a. Select the side of the object that the channel will originate and select create 2-dimensional sketch
   b. Draw the size and shape of the desired cross sectional area desired (Ex. Circle, 275 µm diameter)
      i. Various shapes can be used for the channel including square or rectangular
   c. Finish the 2-Dimensional sketch

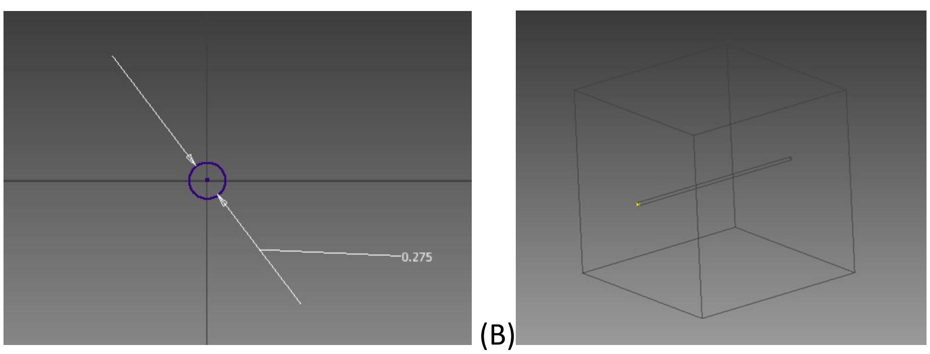
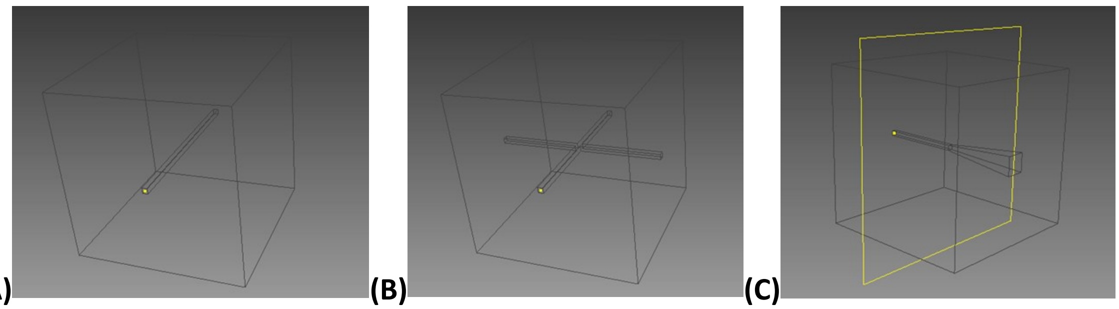
(A) (B)


Figure S2: 2-dimensional sketch of the microchannel and the extrusion through the solid object. (A), Cross section of channel of desired shape (B), Extruded channel through solid object.

4. The channel must now be extruded through the object from the 2-dimensional shape drawn to create the channel
   a. Select the extrude option
   b. If not already selected, select the shape of the channel with the profile option
   c. Choose the direction of the extrude to go into the object
   d. Make sure the extrude is set to cut

e. Input the desired length of the extrude (Ex. 16 mm)

Various shapes can be used for the cross sectional area of the channel along with bends and tapering of structures. Note that bends currently can be an issue due to the need to clean out supporting material.



(A) **(B)** **(C)**

Figure S3: Examples of alternate designs that can be drawn for a straight channel including A.) Square channel B.) Crossed channels and C.) Tapered channel

Example 2. Presented are instructions for design of a 3D-printed device with female threads to accommodate a commercial PEEK fitting (F-120x, IDEX) and a capillary sleeve (F-230, IDEX)

The design of this structure is provided as “threaded adapter. ipt” and “threaded adapter. stl” in a separate supplementary information folder.

Below are the steps to design this structure in Autodesk Inventor:

Create a new file and base object as previously stated in Example 1 (steps 1-2)

Figure S4: 2-Dimensional sketch of a square for extrusion into a 3-dimensional cube

A tapered hole must be created into the object for the threads to be cut into.

1. Select hole from 3D-model Ribbon

2. Select position for extrusion on 3D structure
    a. Set to be a drilled hole
    b. Termination at distance
    c. Hole depth at 8.382 mm
    d. Hole diameter at 4.064 mm
    e. Have an angled drill point with angle of 55 degrees
    f. Choose the simple hole option and linear placement

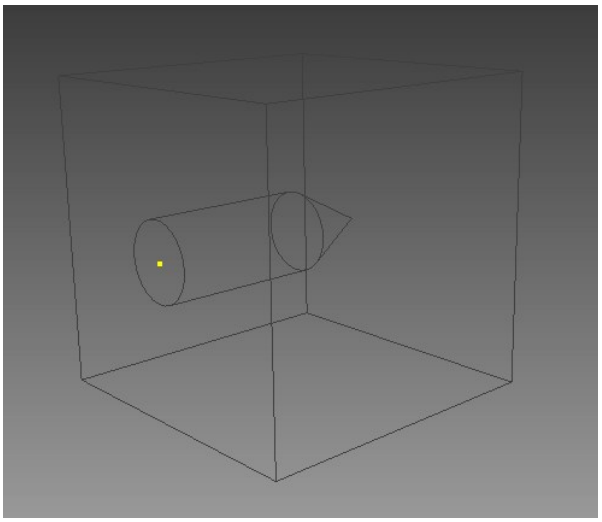


Figure S5: Design showing parameters for the extrusion of a tapered hole cut into a 3-dimensional object

Creating threads into the hole

1. Add axis through center of hole from Work Features Options

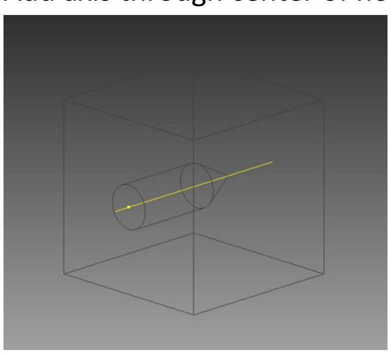


Figure S6: Picture of the location of the work axis function in the software along with the location of the placement on the object

2. Add plane offset from the edge of structure parallel with the axis, that runs through the middle of the hole/axis

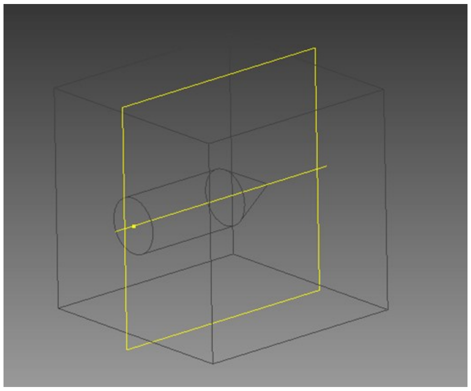
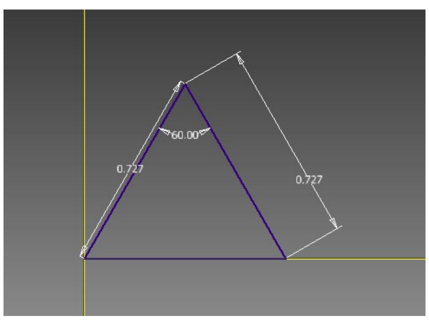


Figure S7: Picture of the location of the work plane function in the software along with the location of the placement on the object

3. Draw an equilateral triangle on work plane on outer edge of the hole by selecting to draw a 2D sketch on the work plane drawn previously
    a. Draw with 3 separate lines with lengths of 0.727 mm on edges

(A) 

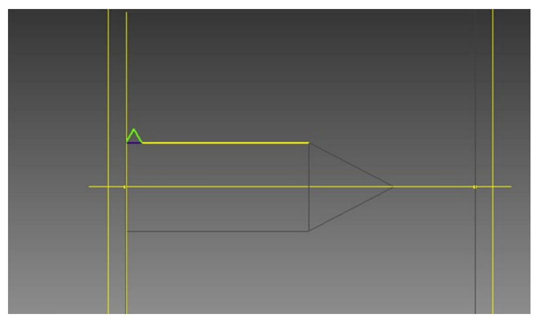
(B) 

Figure S8: Design of the 2-dimensional triangle for coil design on the work axis along with the location of the structure (A), triangle used to cut thread (B), placement of triangle to coil.

4. Select Coil for the Create Options listed under Sweep to create the threads along the drawn extrusion
    a. Coil Shape Tab
        i. Select:
            1. Profile
                a. Select the drawn triangle
            2. Axis
                a. The linear axis through center of hole
                b. Also, select the coil spin direction desired

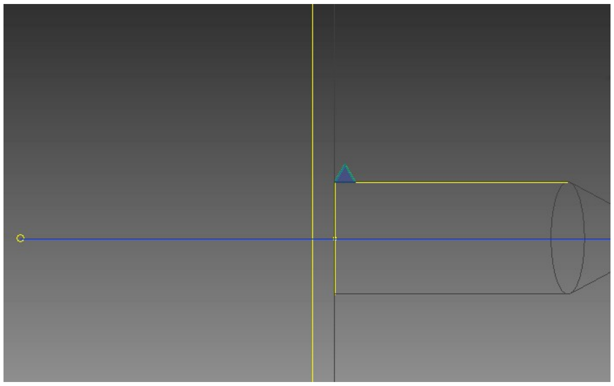


Figure S9: Side view of the selected triangle and the axis for which the threads will be cut into the object using the coil function

3. Select the Cut function
4. Rotation
    a. Up
5. Coil size
    a. Revolution and height
        i. 6.1 mm height
        ii. 8 µL revolution
        iii. 0 Taper

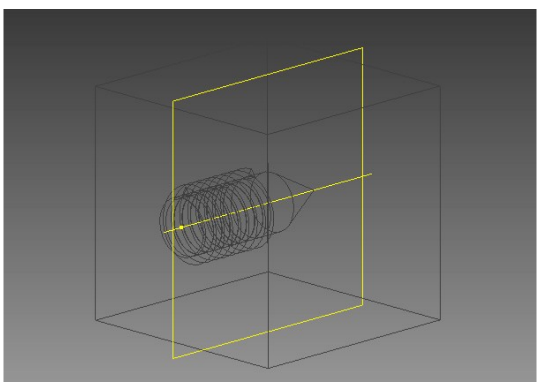


Figure S10: Parameters for the coil size and the drawn coil cut through the design

Creating a fitting hole to accommodate F-230 Capillary Sleeve (IDEX)

1. Create a 2D sketch on side of box parallel to the original hole
    a. Draw a 3 mm diameter hole from the center of extruded hole
    b. Finish the 2D sketch and do an 11 mm extrude (3.6 mm past tip) from the hole
        i. Select cut function

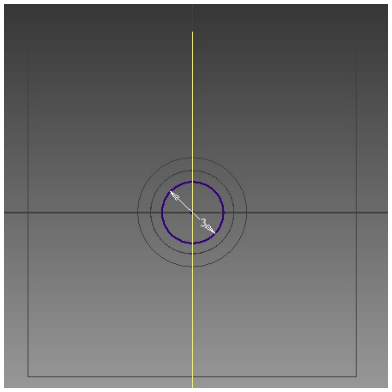
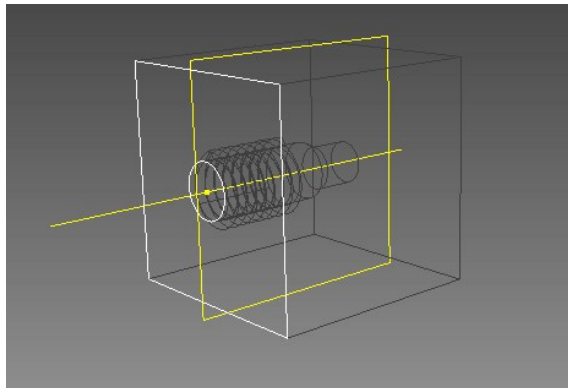
(A)  (B) 

Figure S11: 2-dimensional sketch of the circle and design parameters for the extrusion of a hole through the object to accommodate an F-230 Capillary Sleeve (IDEX) (A), cross-section of channel to fit capillary sleeve (B), Extrusion of channel for capillary sleeve.

Example 3. Design of a flow splitter

The design of this structure is provided as “flow splitter. ipt” and “flow splitter.stl” in a separate supplementary information folder.

Below are the steps to design this structure in Autodesk Inventor:

1. Create a 2-Dimensional sketch for the cylinder portion to be joined with another object
    a. Draw circle
    b. Adjust the diameter either when initially made or by constraining (Ex. 4 mm)
2. Extrude the circle to create the cylinder
    a. (Ex. 8 mm in length)
3. Two lines need to be drawn for the cylinders diverging the fluid flow
    a. Select the XY plane from the origin file and draw two lines from of the end of cylinder (Ex. 8 mm and 30^{o} from the axis)

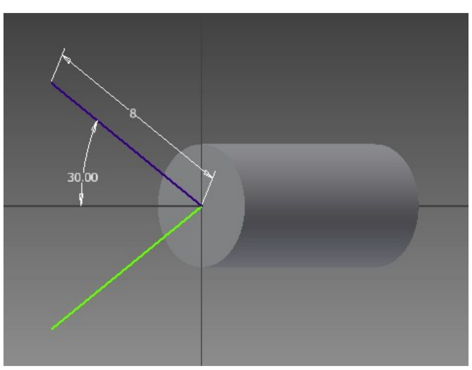


Figure S12: Cylinder portion to be joined with another object and guide lines for diverging cylinder

4. Create two work planes perpendicular to lines to draw cylinders
    a. In the plane menu, select normal to axis through point then select the line and the end of the line as the point.
    b. Repeat to create another plane on the second line.

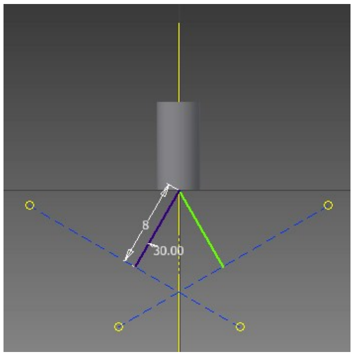


Figure S13: New work planes perpendicular to the guide lines

5. To create the cylinders to split the flow, draw circles on the new work planes
    a. Create a sketch on each work plane
    b. Draw a circle using the end of the line drawn as the center point (Ex. 7 mm)

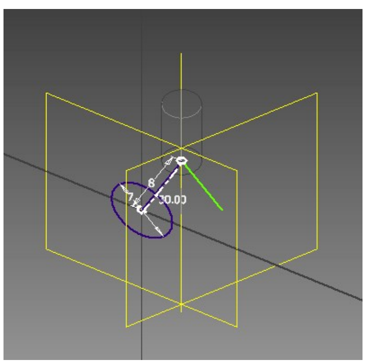


Figure S14: Circle drawn on new plane for diverging cylinders

6. On the YZ plane, create a larger cylinder for connection to the diverging cylinders
    a. Draw a circle, use the axis of the cylinder created in step 1 as the center point. (Ex. 8 mm)
    b. Select the circle and extrude (Ex. 2 mm)

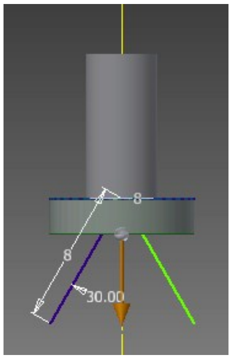


Figure S15: Extrusion of circle for connection of the diverging cylinders

7. Create and join the diverging cylinders to the larger cylinder
    a. Create a new sketch at the bottom of the larger cylinder
    b. Draw a circle of the same diameter (Ex. 8 mm)
    c. Choose the loft option and select the larger circle drawn in step 6 and one of the circles drawn on the new work plane in step 5.

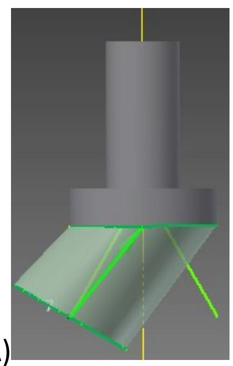
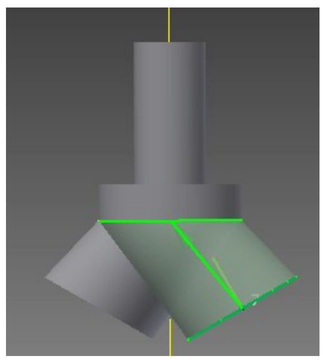
(A)  (B) 

Figure S16: Extrusion of circles drawn on new planes to diverge fluid flow (A), one circle being lofted to create diverging cylinders (B), creation of the second cylinder.

    d. Repeat this for the second circle
8. Draw a triangle to create threads on original cylinder for connection
    a. Draw an equilateral triangle along the edge of the cylinder used for connection where it joins the cylinder from step 6 (Ex. Each side is 0.7 mm)

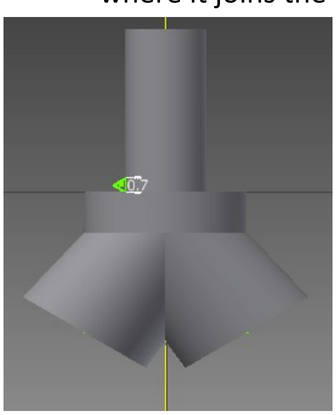


Figure S17: Placement of triangle used to create threads

9. Choose the coil option to make threads from triangle
    a. Select coil, choose the triangle as the profile and the axis of the cylinder for the coil
    b. Select the join option and upward rotation within the coil menu.
    c. Within the coil menu, select the coil size. Set the pitch to 0.800 mm and the height to 7.000 mm with 0.00 deg taper within the pitch and height option in the drop down menu.

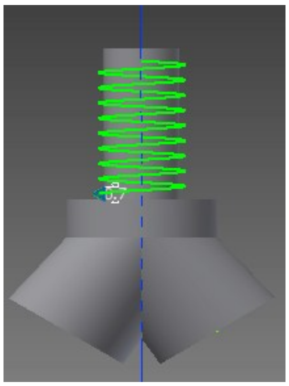


Figure S18: The coil feature is used to fashion the threads using the triangle.

10. Create another work plane to finish the connection tip
    a. In the plane menu, select offset from plane
    b. Select the top of the threaded cylinder and plane and offset it by 4 mm.

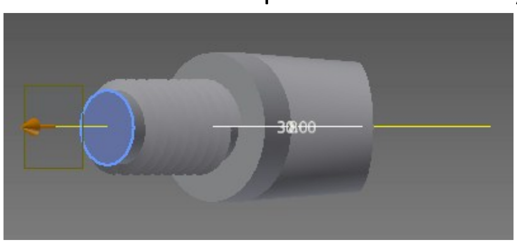


Figure S19: A new plane is offset from the edge of the threaded cylinder.

11. Create the connection tip
    a. On the new work plane, draw a circle with the center point as the axis of the threaded cylinder (Ex. 1 mm)
    b. Choose the loft option and select the new circle and the edge of the threaded cylinder

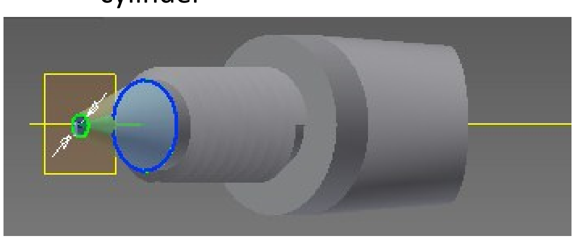


Figure S20: A circle is drawn on the new plane and the loft feature is used to create the tip of the threaded connection.

12. Make a channel for the outlet of the connection tip
    a. Create a circle on the work plane created in step 10 within the circle drawn in step 11. (Ex. 0.9 mm)
    b. Select the circle and extrude it drown through the threaded connection tip (Ex. 12 mm)
    c. Choose the cut option to make an open channel

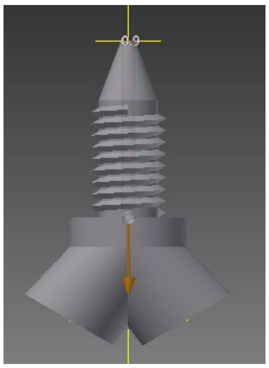


Figure S21: A channel is extruded and cut through the threaded connection tip for fluid flow.

13. Create fluidic channels in diverging cylinders
    a. Draw a circle on the edge of each diverging cylinder (Ex. 1 mm)

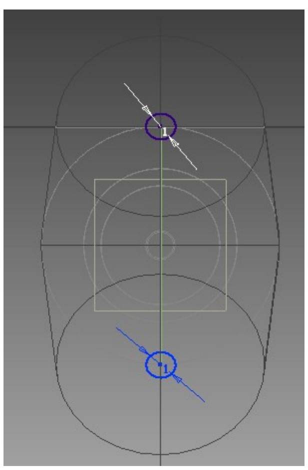


Figure S22: Circles are drawn at the end of the diverging cylinders for channels.

    b. Choose the loft option and select the new circle drawn at the end of the diverging cylinders and the edge of channel within the connection tip. Repeat for the second diverging cylinder.

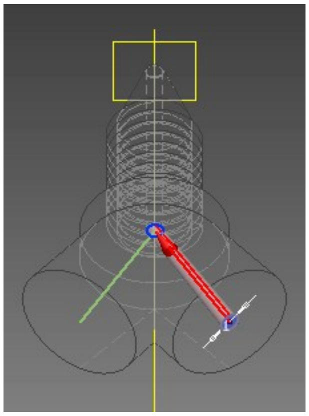


Figure S23: The circles at the end of the diverging cylinders are extruded and cut towards the threaded connection channel.

14. Elongate the diverging cylinders to create inlets for threaded connections
    a. Create a sketch on the end of the diverging cylinder
    b. Draw a circle of the same diameter (Ex. 7 mm)

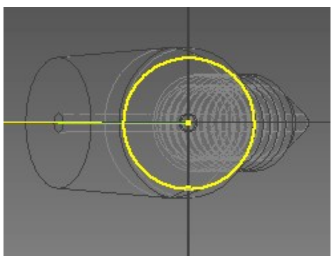


Figure S24: A circle outlining the diameter of the diverging channels is drawn.

    c. Extrude the circle outward (Ex. 4 mm)

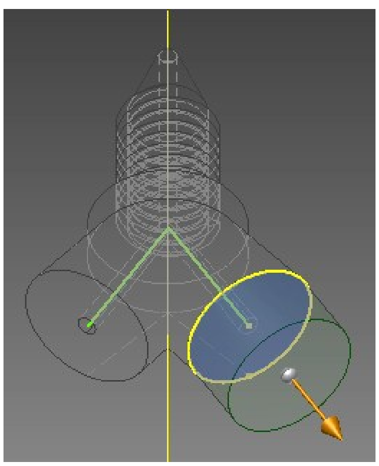


Figure S25: The circle is extruded to elongate the diverging cylinders.

    d. Repeat on other diverging cylinder

15. Create the tip to fit the inserted threaded fitting
    a. Draw a circle on the edge of the extrusion from step 14. (Ex. 4.1 mm)

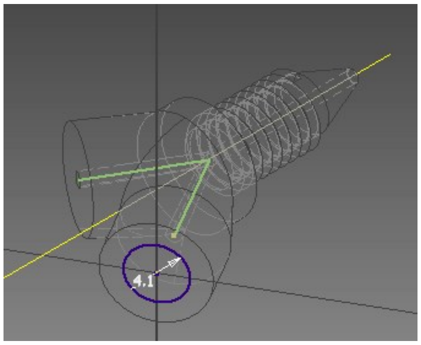


Figure S26: The larger end of the tip is fabricated first by drawing a circle of the appropriate size.

b. Choose the loft option and select the previously drawn circle and the edge of the outlet channel from the diverging cylinder. Choose the cut option within the loft menu.

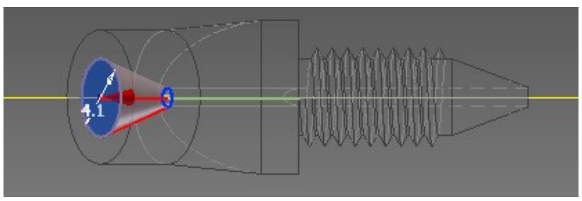


Figure S27. The tip is created within the extended diverging cylinder by using the loft feature.

c. Repeat with other diverging cylinder.

16. Elongate the diverging cylinder for threads
    a. Create a sketch on the end of the diverging cylinder
    b. Draw two circles of identical diameter to the diverging cylinder and the wide edge of the tip (Ex. 7 mm and 4.1 mm)

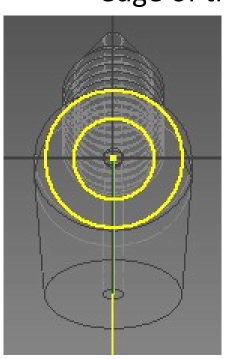


Figure S28: An "O" shaped profile is created by drawing two circles at the end of the elongation of the previously extruded cylinder.

c. Select the "O" shaped profile and extrude to the length of the threads on the fitting (Ex. 7 mm)

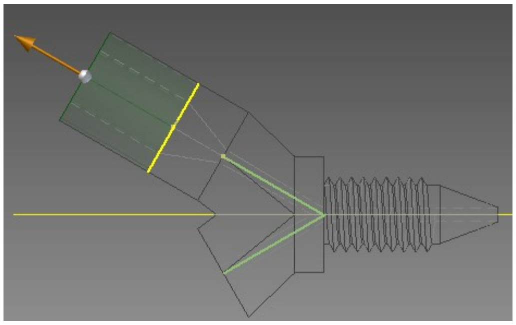


Figure S29: The “O” shaped profile is selected and extruded outwards.

    d. Repeat on the other diverging cylinder

17. Create a triangle within the diverging cylinder for threaded inlet
    a. Draw an equilateral triangle along the inside edge of “O” shaped extrusion

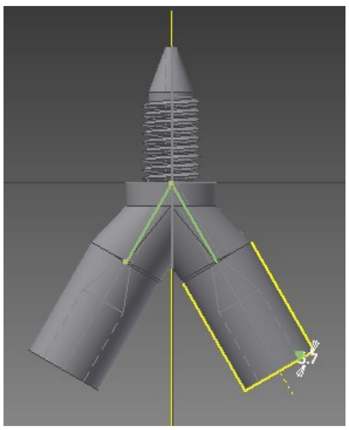
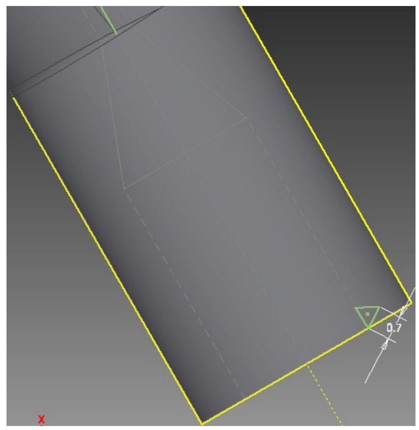
(A)  (B) 

Figure S30: A triangle is drawn at the end of the cylinder in order to cut threads for fittings (A), placement of triangle to coil for threads (B), Zoomed in view of triangle placement

    b. Choose the coil option
    c. Select the triangle and the axis of the extrusion.
    d. Select the cut option within the coil menu and upward rotation.
    e. Within the coil size tab of the coil menu, pitch and height were adjusted (Ex. 0.800 mm pitch and 7.000 mm height)

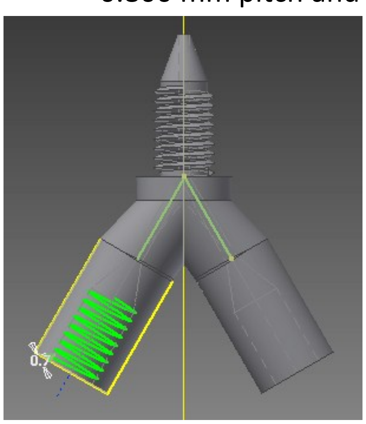


Figure S31: The coil feature is used to create the threads from the previously drawn triangle.

f. Repeat on other diverging cylinder

Example 4. Design of well inserts - These instructions describe a method to create well inserts into an object

The design of this structure is provided as “well plate. ipt” and “well plate. stl” in a separate supplementary information folder.

Below are the steps to design this structure in Autodesk Inventor:

1. Create a new file and base object as previously stated in Design S1 for the base object
    a. The object made for this example is (100mmx6mmx92mm)
2. Create a 2-dimensional sketch on the side the well inserts will go into
    a. Draw a circle with the desired diameter on the surface
    b. Select dimension from the constrain options to set the circle diameter
    c. To add more circles for other wells, highlight the original circle and copy and past
    d. You can constrain the distance between the circles as well
    e. Circles can be copy and pasted to speed up the design process
    f. Finish the 2D sketch when all the circles are drawn

(A) (B) (C)

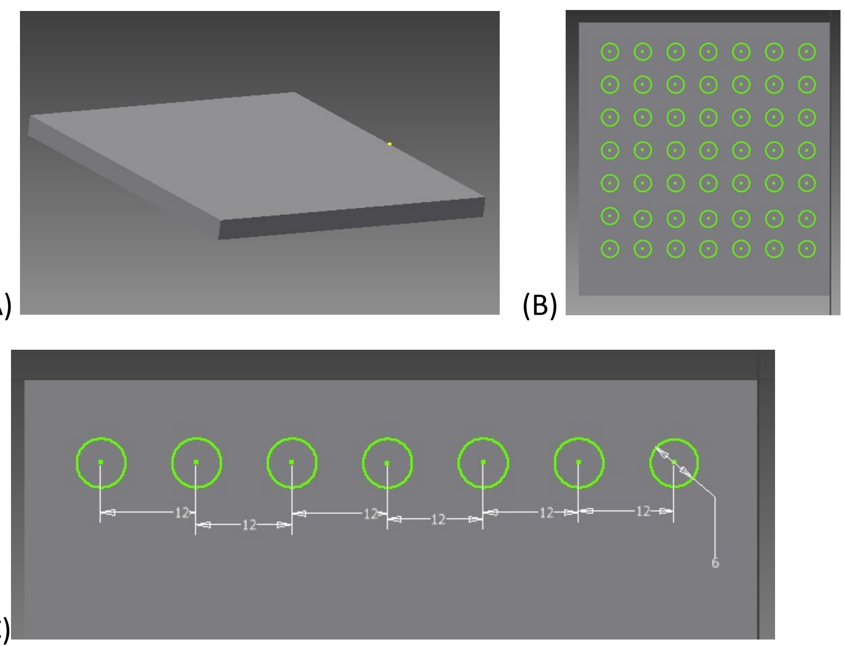


Figure S32: Extruded base and 2-dimensional well design patterns for extrusion into the solid (A), solid base (B), cross-section of circles for wells (C), dimensions and spacings of wells.

3. The circles must now be extruded into the base to create the wells
    a. Select the extrude option
    b. Highlight the circles to be extruded
    c. Choose the direction and length of the extrusion into the wells (ex. 4 mm)

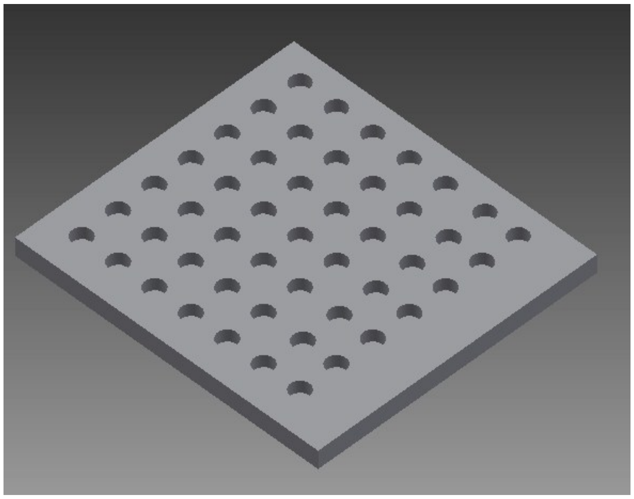


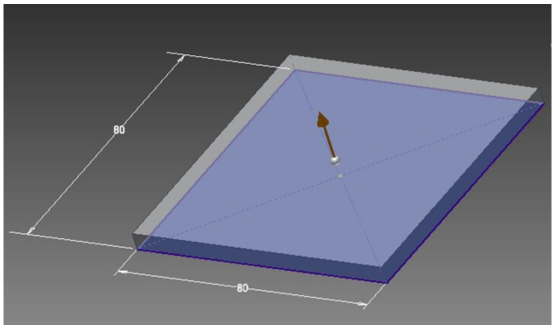
Figure S33: Finished well plate after extrusion into the object

Example 5. 3-D Printed mold for soft lithography

The design of this structure is provided as "mold for PDMS casting. ipt" and "mold for PDMS casting. stl" in a separate supplementary information folder.

Below are the steps to design this structure:

1. Create a base to build the microfluidic channels on
    a. Create a 2-D sketch
    b. Draw the desired shape (Ex. Square, 80 x 80 mm)
    c. Extrude the desired shape to create the flat surface (Ex. 5 mm)
2. Build a border to enclose mold
    a. On the base, draw the same shape and size of the base (Ex. Square)
    b. On the same sketch, create a slightly smaller shape within (Ex. Square, 75 x 75 mm)
    c. Extrude the hollow shape upward to create a border (Ex. 10 mm)

(A) 

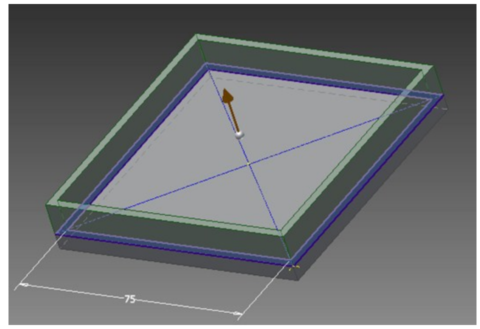
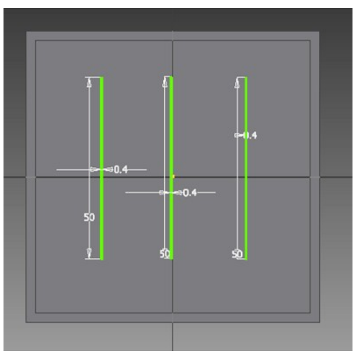
(B) 

Figure S34: Extrusion of base for mold and extrusion of borders around mold (A), extrusion of square for base (B), extrusion of borders joined to base

3. On the base, create protruding channels
    a. On the top of the base within the borders, create a 2-D sketch
    b. Draw the desired shape of channels for the mold (Ex. 0.4 mm wide, 50 mm length)
    c. Select the channel shape and use the extrude feature (Ex. 0.4 mm)

(A) 

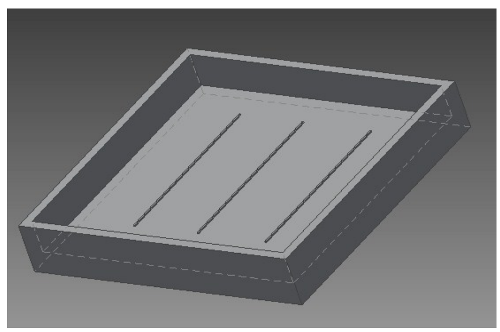
(B) 

Figure S35: Shape of channels for mold and extrusion of channels (A), dimensions of channels (B), extruded channels in mold

Example 6. Combining files - The following instructions show how to combine objects from separate files into one object. This example is combining example 1 and example 2.

The design of this structure is provided as “object assembly. ipt” and “object assembly. stl” in a separate supplementary information folder.

Below are the steps to design this structure in Autodesk Inventor:

1. Open up a new file
    a. Select a standard assembly file (standard(mm).iam)
2. The separate objects have to be placed into the file
    a. In components, select place
    b. Locate and open the file for the first object

c. Place the object into the desired location

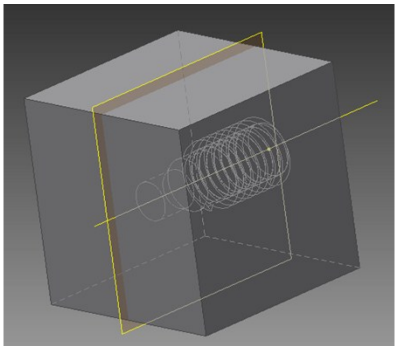
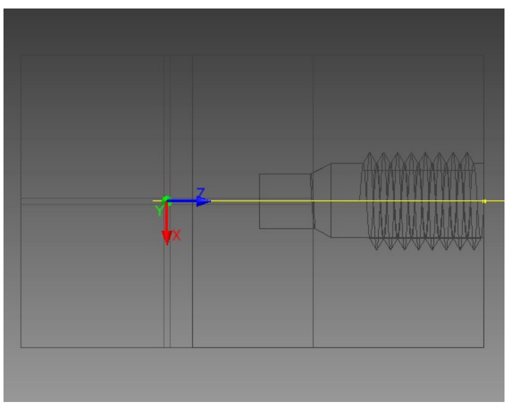


Figure S36: Import of a single file into the assembly file

3. The second object now has to be placed into the file
   a. Again, in components, select place
   b. Locate and open the file for the second object
   c. Place the object into the location in relation to the first object

(A) 

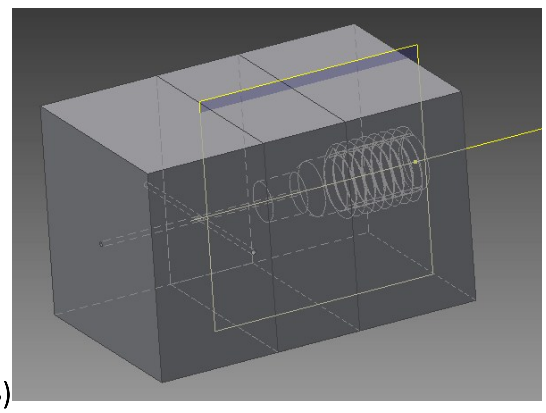
(B) 

Figure S37: Import of a second file into the assembly file. Alignment of the objects is done based on the axis being viewed and the axis that the objects were originally drawn on. (A), alignment of the straight channel object and the threaded fitting object (B), realistic view of the alignment of the straight channel and threaded fitting.