

Aug 12, 2024

Mechanical Model V.1

This protocol is a draft, published without a DOI.

Wolfram Moebius¹

¹University of Exeter



Brandon Tuck

OPEN  ACCESS



Protocol Citation: Wolfram Moebius 2024. Mechanical Model. [protocols.io https://protocols.io/view/mechanical-model-b3zdqp26](https://protocols.io/view/mechanical-model-b3zdqp26)

License: This is an open access protocol distributed under the terms of the [Creative Commons Attribution License](#), which permits unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited

Protocol status: Working

We use this protocol and it's working

Created: January 18, 2022

Last Modified: August 12, 2024

Protocol Integer ID: 57093

Abstract

Mechanical Model



- 1 To simulate the stresses and deformation due to material properties and fluid flow, we create a 2D model representing agar overhanging a channel, with forces applied depending on the scenario.
 1. Create a new **2D** project in COMSOL using the Model Wizard, add the **Solid Mechanics (solid)** module under **Structural Mechanics** and select **Study**
 2. Add the **Stationary** study step. A new window with the created model should open in COMSOL.
- 2 To create the spatial aspect of our model, we use the *Geometry 1* menu to add shapes which represent individual PDMS supports or sheets of agar, and set the dimensions and coordinates to arrange the shapes according to the real life geometry.
 1. In the *Model Builder* on the left of the page, select the *Geometry 1* menu within *Component 1*.
 2. In the Settings, change the *Length unit* to **mm**.
 3. Right click *Geometry 1* and add a **Rectangle** to represent the agar sheet. Add further **Rectangles** for PDMS supports. This will be a **Solid** type whether it is PDMS or agar.
 4. In the *Block Settings*, alter the *Size and Shape* and *Position* of each to correspond to the real life geometry. The geometry is symmetrical across the length of the device, so we bisect the structure through the middle of the channel and the middle of the PDMS support - the channel and support widths therefore only need to be **half the physical value**, with the agar sheet the combined width of both (Values used as example below: support w/h = **3/1 mm**, agar w/h = **3.5/1.5 mm**, agar z = **1 mm**). We apply this symmetry later on.
 5. Select **Build All Objects** to show the geometry in the Graphics window (on the right of the screen). If the whole structure isn't visible, you can zoom using the buttons above the Graphics *window*.
- 3 To set material properties relating to stress and deformation, we use the *Materials* menu to specify the Young's modulus, density and Poisson's ratio for PDMS and agar.
 1. Right click the *Materials* menu and add a **Blank Material** to represent agar. In *Settings*, label the material as 'Agar' and set the **Young's modulus, Poisson's ratio** and **density** (used **52000 Pa, 0.32** and **1000 kg/m³** respectively). In the *Graphics* window click on the PDMS support to deselect it (should turn grey).
 2. Add another Blank Material to represent PDMS, and repeat (used **1000000 Pa, 0.35** and **1190 kg/m³** respectively). In the *Graphics* window click on the PDMS support to select it (should turn blue).
- 4 To set the mechanical loads acting on the device, we use the *Solid Mechanics* menu to specify the channel pressure and weight of the agar sheet.
 1. Select the *Solid Mechanics* menu and in the *Settings*, change the **Thickness** to the length of the channel (in this case **0.07 m**).

2. Right click the *Solid Mechanics* menu and under *Volume Forces*, select **Body Load**. In the *Graphics* window, select the agar sheet. Set the weight of the agar sheet in the negative **y** direction of *Force*, corresponding to **$F = mg$** (as we are specifying Force per unit volume, **$F/v = \rho g$**). Density in COMSOL is **mat1.def.rho** (in this case, used **-9.81[m/s²]*mat1.def.rho**).
 3. Right click the *Solid Mechanics* menu and under *Domain Constraints*, select **Fixed Constraint**. Select the support in the *Graphics* window to keep this from reacting to forces.
 4. Right click the *Solid Mechanics* menu and under *More Constraints*, select **Symmetry**. In the *Graphics* window, select the left and right edge of the agar sheet to simulate a continuous sheet of agar.
 5. Right click the *Solid Mechanics* menu and select **Boundary Load**. In the *Graphics* window, select the overhanging part at the bottom of the agar sheet where it is suspended over the channel. In the *Settings*, change the *Load type* to **Pressure** and input the channel pressure (used **-200 Pa**).
- 5 To simulate our device, COMSOL discretises the model spatially and temporally. The spatial discretisation is defined by a mesh of points, created in the *Mesh 1* menu. COMSOL will automatically design a mesh, but this should be optimised for different parts of the design manually.
1. Select the *Mesh 1* menu. In the settings, change the *Sequence type* to **User controlled mesh**, which will open new submenus.
 2. Select the *Size* submenu, and change the predefined element size to **Finer** - this sets an overall maximum mesh size.
 3. Right click the *Mesh 1* menu and select **Size**. In this new *Size 1* submenu settings, change the *Geometric entity level* to **Point**. In the *Graphics* window, select the point where the agar sheet forms a boundary with both the support and the channel. Change the *Element size* to **Predefined - Extremely fine** as this is a stress point.
 4. Click and drag the **Size 1** submenu to above the *Free Triangular 1* submenu so it takes precedence.
- 6 To set up the study parameters, we set steps in the *Study 1* menu. We then set up visualisations in the *Results* menu to be viewed whilst the study is running. We can then run a simulation.
1. Right click the *Study 1* menu, selecting **Get initial value**. This sets up a solution folder the study will be contained in.
 2. Right click the *Results* menu and add **2D Plot Group** - this will visualise the solution.
 3. To visualise the agar deformation, add a **Surface** condition to the new *2D Plot Group*. Type **v** for vertical displacement into the *Expression* box in *Settings*
 4. Right click on the *Surface* submenu and select **Deformation**. Change the **Scale factor** in *Settings* to **1**.
 5. Select *Step 1: Stationary* in *Study 1* and under *Results While Solving*, select the plot group just created. Click **Compute** to run a simulation.
- 7 To run multiple simulations sequentially, add a **Parametric Sweep** under *Study 1*. Any parameters you have defined in *Parameters 1* under *Global Definitions* can be added with the

plus icon. Separate values with a space. This will now automatically run until you disable or delete the *Sweep*.

