Finite Element Analysis of a Smart Nozzle in SolidWorks

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Table Of Contents

I. Executive Summary

The smart nozzle is one component of a soft robot. The growing subfield of robotics, soft robotics, is diverse field that combines cutting edge material science and classical robotics with the goal of actuating highly compliant materials. Some materials, such as elastomers, are implemented due to their large deformations coupled with relatively linearly elastic behavior. The smart nozzle is designed with internal cavities which, when pneumatically actuated, creates uniform bending, extension, and contraction along its body. To quicken the design process, Computer Aided Design when combined with an accurate Finite Element Analysis simulation, tool can predict deformation of a given design. The following document will walk the reader through the process of simulating a smart nozzle design using Finite Element Analysis "simulation" in SolidWorks. For the most accurate results, following each step of the tutorial is very important.

II. Download Model

A. The nozzle design that will be used to complete this tutorial is a simple two chamber design created with thick walls to accept higher pressurization. This model can be downloaded from the DIMLab Drive by following DIMLab → Smart Nozzle → FEA Tutorial: Smart Nozzle and downloading the file titled *Smart Nozzle Tutorial.SLDPRT* (Figure 1).
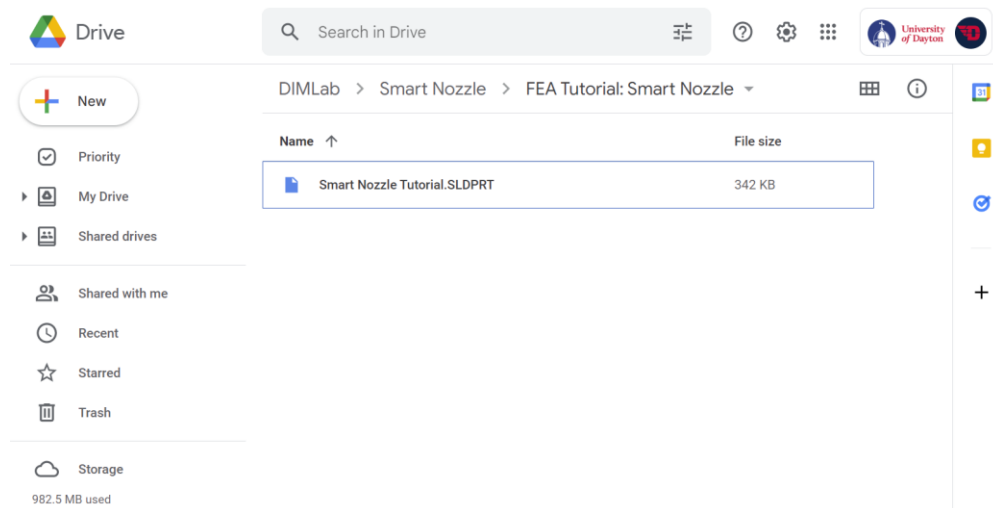

Figure 1: Download Smart Nozzle Part

III. **Creating and Assigning Material Properties**

    A. There are two places in which the material properties of the part can be set. The first of which can be found in the "Part" tab of FeatureManager. Right click on the "Material" menu and click edit material (Figure 2). Once the "Edit Material" menu has successfully been opened, you will see a whole host of material options but, the current material in use, Rubber 65A (R65A), will not be found. Thus, a new material must be created. To do so, when must model the new material off an existing material model. Any material can be chosen but this tutorial will select the "Silicone Rubber" material from the SW library, as it has some similarities to R65A. So, to begin, right click and copy "Silicone Rubber." Once the material is copied, right click on "Custom Materials" and select new category. Name the category "Elastomers." After the category has been created, you can then right click "Elastomers" and paste the "Silicone Rubber"

Figure 2: Finding "Edit Material" Menu

material. Expand the "Elastomer" category and double click "Silicone Rubber" and change its name to "Rubber 65A." At this time, the properties page of the rubber is visible and editable. Change the properties of the material to resemble those of Figure 3. When finished, click save and apply and the close out of the window. You can double check that you applied the material correctly by looking for where you originally clicked to in the FeatureManager and it should now say "Rubber 65A" (Figure 4).
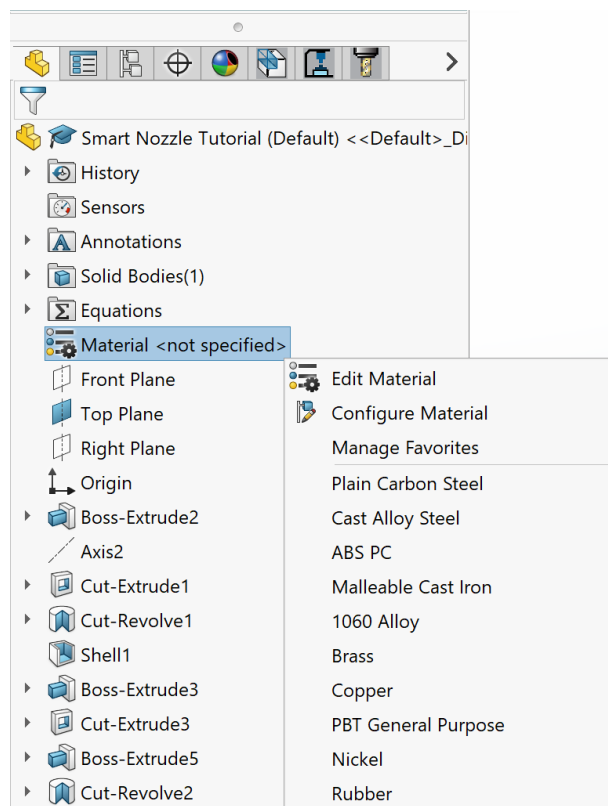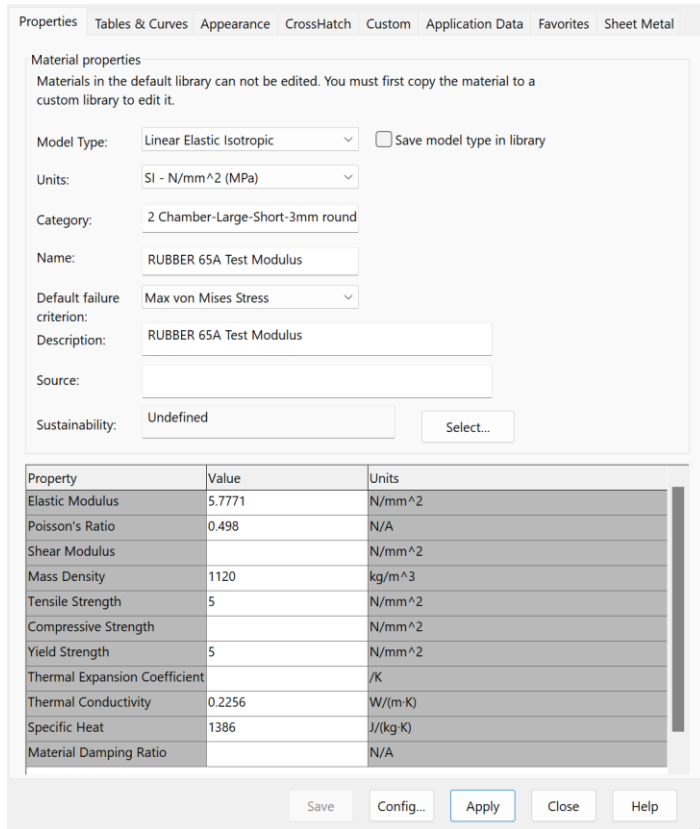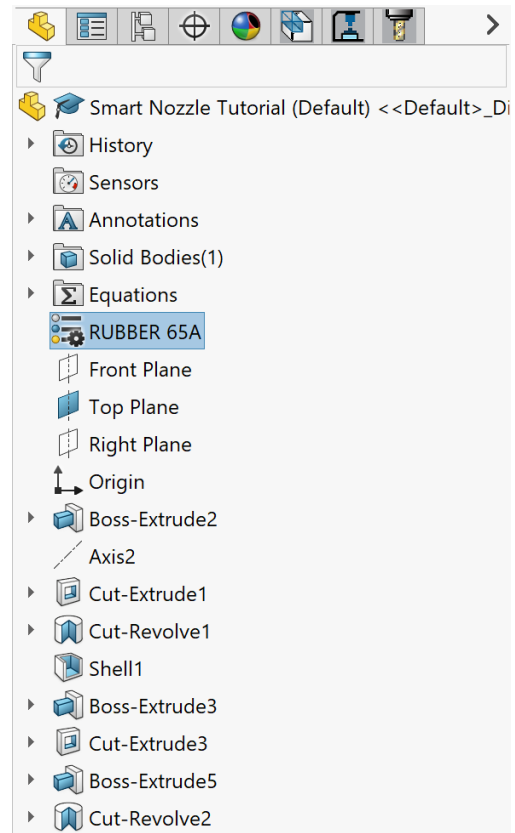
Figure 3: Rubber 65A Material Properties

Figure 4: Application Validation

IV. Creating a New Study

A. With the tutorial model downloaded and the correct material applied the next step is to create a new study using SolidWorks Simulation, SW's FEA solver. To get started, click on the simulation tab are search "Simulation" in the "Search Commands" bar. Once the right menu is selected, click on "New Study" to create a new simulation study (Figure 5).
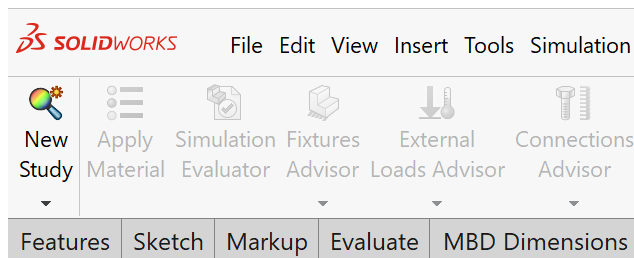

Figure 5: New Study inside Simulation Menu

Once there, select the static, nonlinear option under advanced simulation, name the simulation, and hit the green checkmark to create the study (Figure 6).
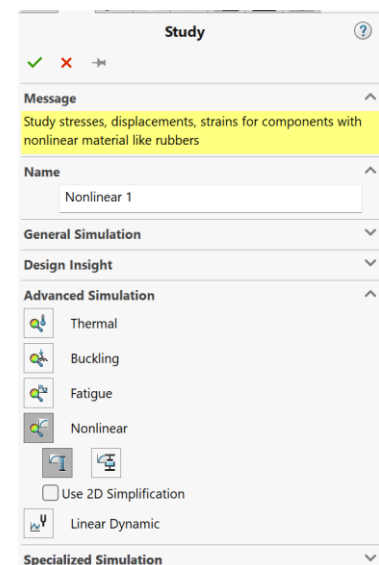

Figure 6: Creating a New

V.   Apply Fixtures

A. To be able to run the study, we must determine which part of the model will be fixed so that the deformation can be actualized. For the use case of the smart nozzle, we will fix one end of the smart nozzle. Right click on "Fixtures" and select fixed geometry (Figure 7). To select the end of the nozzle, simply select the face of the model and click the green checkmark to accept the selection. A preview of your selection should be visible and you can reference this next to Figure 8.
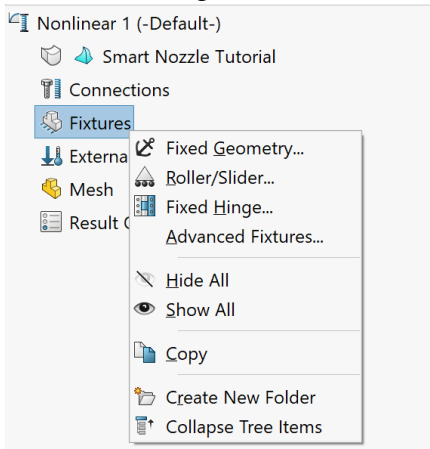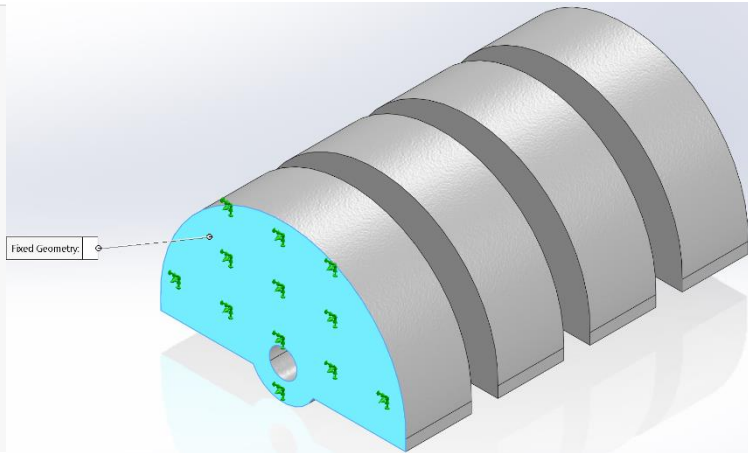

Figure 7: Fixture Menu

Figure 8: Fixture Selection on the Model

VI.   Apply External Loads

A. *Pressure*

The smart nozzle is a part which is dependent on air pressure for actuation. Thus, the external load we must use to simulate the model is a pressure. Before we can select the internal faces upon which the pressure is to be applied, we must be able to see inside the model. Click on the section viewer (⬚) to open the menu and offset the right plane by -5 mm and hit the green check mark to accept the section view (Figure 9). Now that we have access to the internal faces of the nozzle, we can begin to apply the pressure to all internal faces. To do so, right click on "External Loads" and select "Pressure" from the menu (Figure 10). Ensure that the pressure is in units of psi and assign a pressure value of 5. The pressure will be applied only to selected faces so it is important to be diligent. Select each face in the same fashion as in selecting the fixtures. Once you have selected all faces in the current section view, accept the pressure selection, return to the section viewr meu, flip the section view, and then right click on the newly created "Pressure" and hit "Edit Defenition". This will then allow you to select the rest of the remaining feces of the chamber. Note, each chamber should have a total 18 selected faces. Repeat this process for the second chamber and make sure to double check that the unit of pressure is psi and that all faces have been selected (Figure 11)
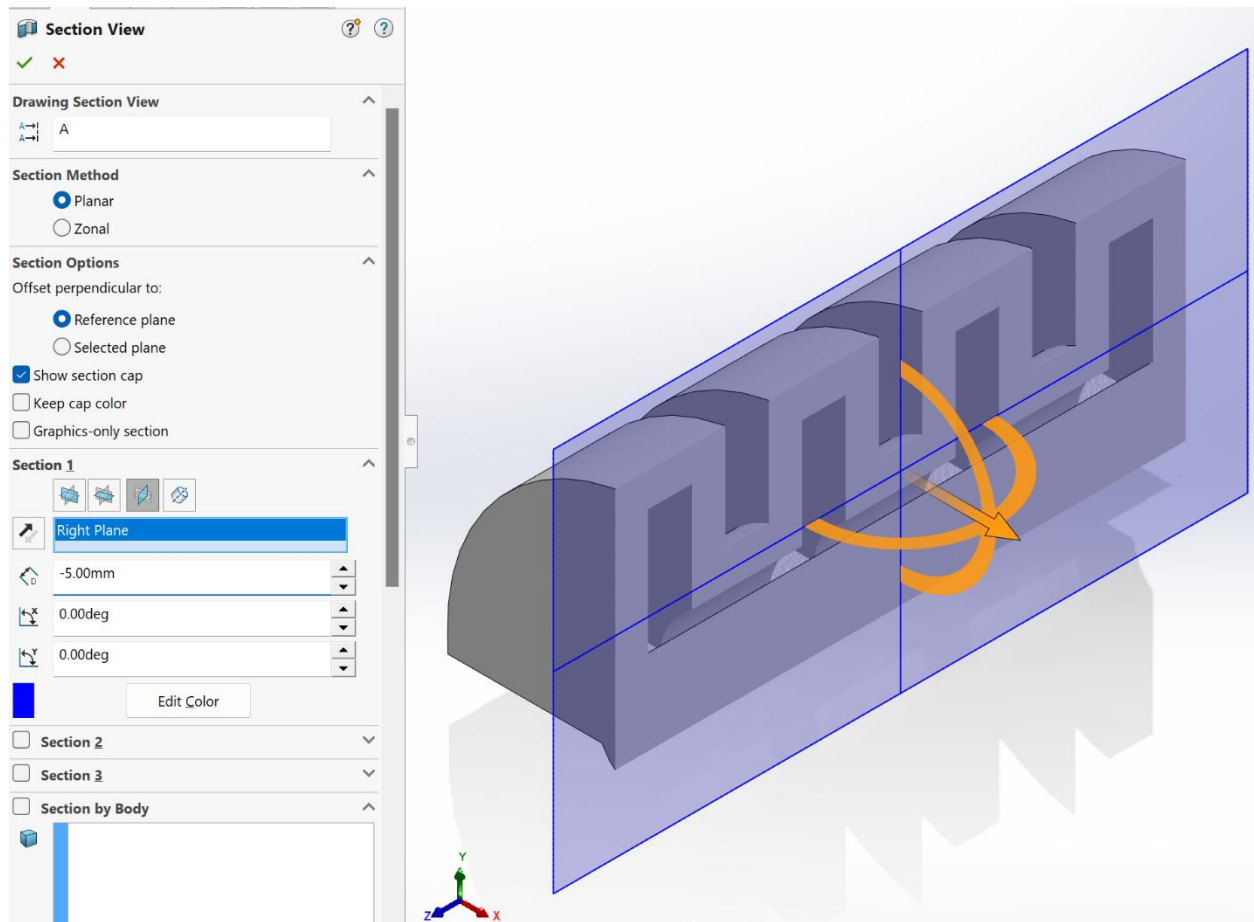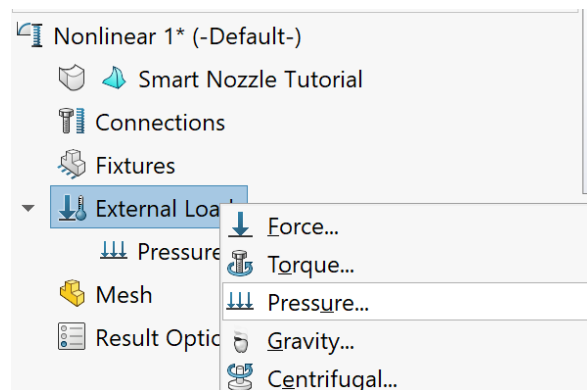
Figure 9: Setting up Cross Section View

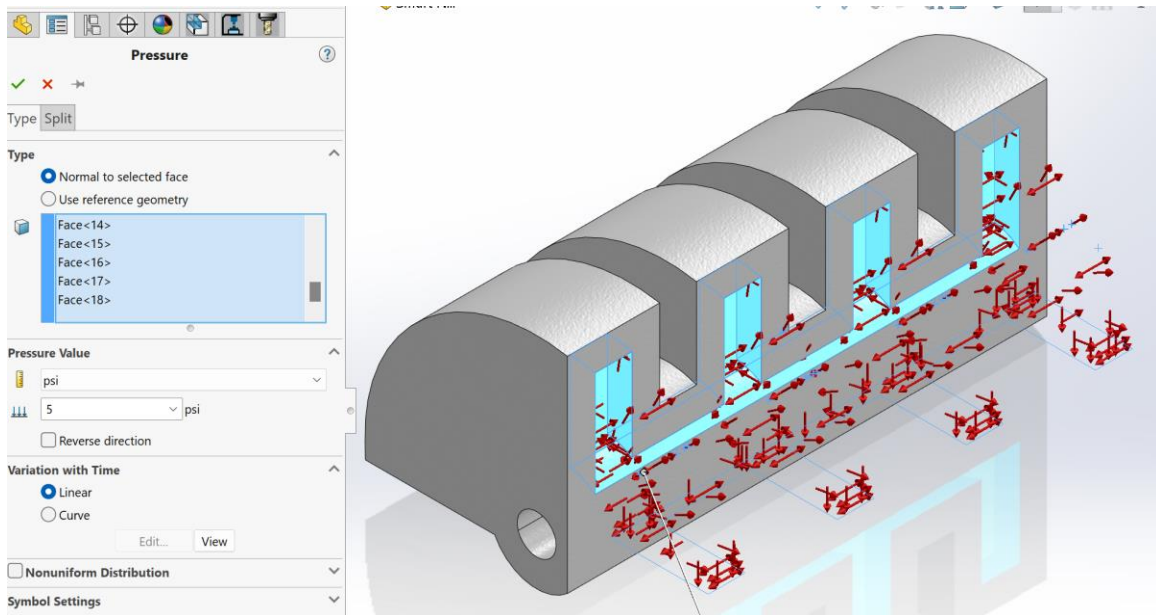*Figure 10: Opening Pressure Load Menu*

Figure 11: Applying Pressure to the Model

B. *Gravity*

The next external load relevant to this study is gravity. Depending on the length, size, and the orientation of the smart nozzle model gravity will have a noticeable effect on the unactuated position of the end effector. Adding the gravitational load is the same process as applying a pressure. Right click on external loads and select gravity. Simply choose which direction and plane, corresponding to the testing orientation, that gravity will act perpendicular to and click he green check mark (Figure 12).
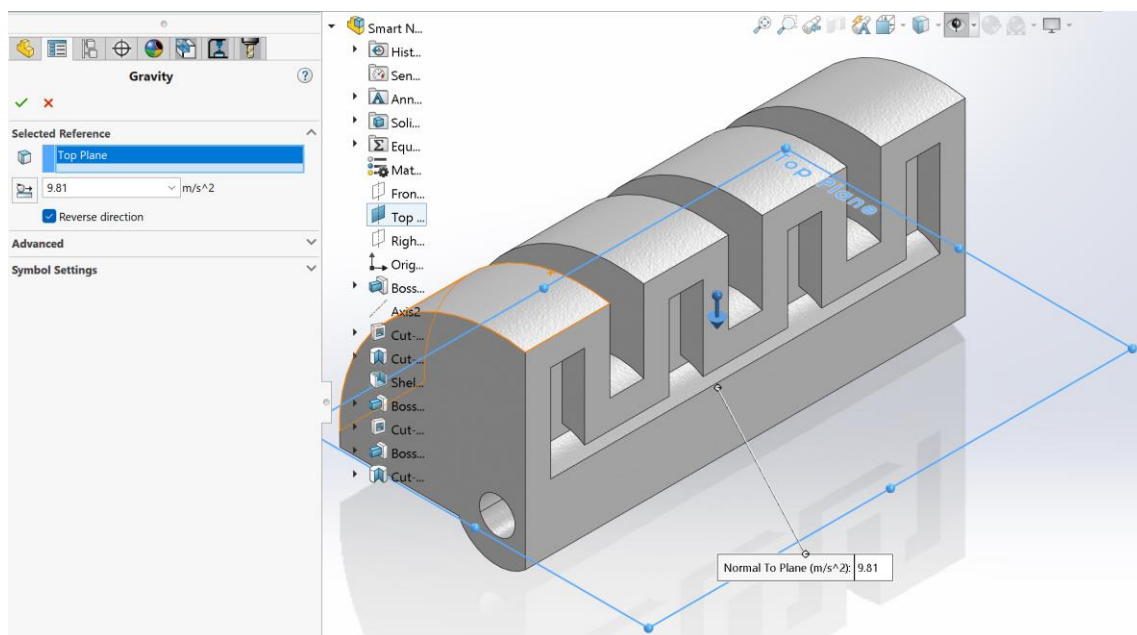

Figure 12: Gravity Application

VII. Create Mesh

    A. One of the last steps we need to take to set up the simulation is creating a mesh. The mesh is where the analysis equations are carried thus the clarity of the mesh will have direct correlation on the accuracy of the results. For research purposes, the finest mesh possible will have the best results in correlation to physical testing. To create the mesh of the model, right click on "Mesh" and click on "Create Mesh…". Once the mesh window has be opened, drag the slider all the way to the right to the fine side to create the finest mesh SolidWorks will allow (Figure 13). Click the green check mark and the mesh will be set up.
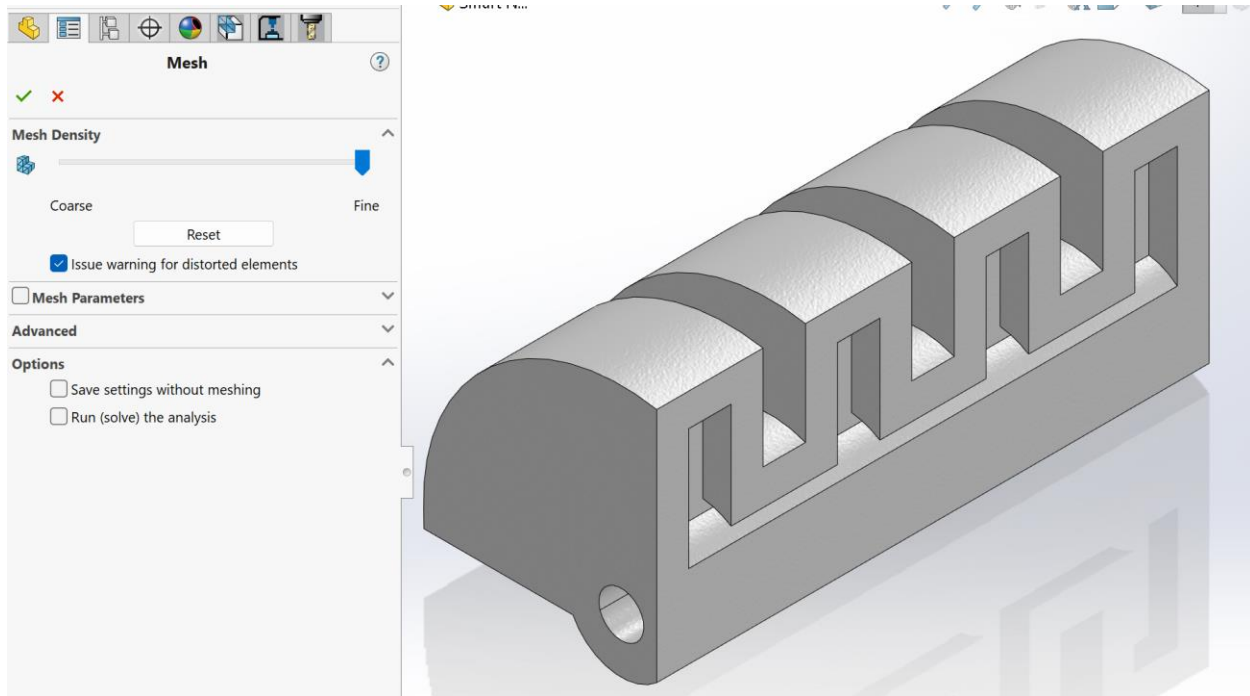

Figure 13: Mesh Menu

VIII. Nonlinear Analysis Properties

    A. After the mesh has successfully been created, the study can be run but it my have some errors in solving problems and can alter the results or cause the study to fail. To optimize a few settings of the nonlinear study to prevent some of these issues, right click on the study name in the part menu (Figure 14). Once in the menu, on the first page, alter the settings to match that of Figure 15. Once complete, click on advanced settings in the original study properties menu and make the corresponding changes from Figure 16.
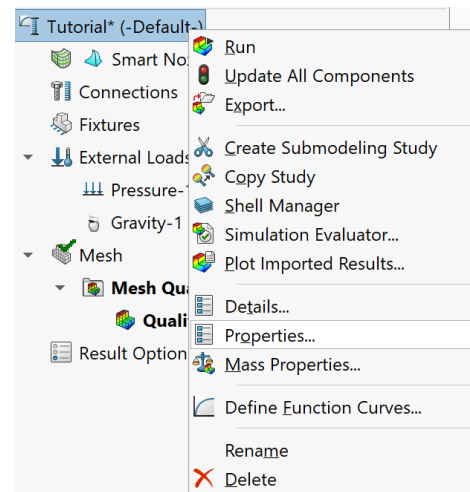

Figure 14: Study Properties

Nonlinear - Static

Solution | In-mold stresses | Flow/Thermal Effects | Notification | Remark

Stepping options

Start time `0`

End time `1`

☐ Restart
☑ Save data for restarting the analysis

Time increment:

● Automatic (autostepping)

Initial time increment `0.0001`

Min `1e-08` Max `0.1` No. of adjustments `10`

○ Fixed `0.1`

Note: For nonlinear static analysis (except time dependent material like creep) pseudo time steps are used to apply loads/fixtures in small increments. For creep, time steps represent real time in seconds to associate loads/fixtures.

Start time and End time are not used by the Arc Length control method defined in Advanced options.

☐ Compute free body forces

Geometry nonlinearity options

☑ Use large displacement formulation
☑ Update load direction with deflection (Applicable only for normal uniform pressure and normal force)
☑ Large strain option
☐ Keep bolt pre-stress

Solver selection

○ Automatic
● Manual

`Large Problem Direct Sparse`

Save Results

☑ Save results to SOLIDWORKS document folder

Results folder `C:\Users\Patrick Hudak\Downloads`

☐ Average stresses at mid-nodes (high-quality solid mesh only)

[Advanced Options...]

[OK] [Cancel] [Help]

Figure 15: Study Properties

Nonlinear - Static

Solution | **Advanced** | In-mold stresses | Flow/Thermal Effects | Notification | Remark

Method

Control `Force`

Iterative technique `NR (Newton-Raphson)`

Integration `Newmark`

Displacement control options

Select a vertex or reference point to control the analysis

Displacement component for the selected location `UX: X Translation` `mm`

Displacement variation with time [Edit...] [Graph]

Arc-Length completion options

Maximum load-pattern multiplier `100000000`

Maximum displacement (for translation DOF) `100` `mm`

Maximum number of arc steps `50`

Initial arc length multiplier `1`

Automatic time increment(autostepping):

Min `1e-08` Max `0.1` No. of adjustments `10`

Step/Tolerance options

Do equilibrium iteration every `10` step(s)

Maximum equilibrium iterations `20`

Convergence tolerance `0.001`

Maximum incremental strain `0.1`

Singularity elimination factor (0-1) `0.001`

Intermediate Results

☑ Show intermediate results up to current iteration (when running)

When enabled, the nonlinear simulation will terminate if you switch to another SOLIDWORKS document or close the active model.

[OK] [Cancel] [Help]

Figure 16: Advanced Study Properties

Once all of the necessary changes have been made, click ok and that will save the changes made to the study properties.

IX.   **Run Study**
   A. If all of the steps of this tutorial have been followed, the last step is to initiate the study. Simulation time is dependent on many things; model complexity, how fine the mesh is, magnitude of loads applied, processing power of the computer running the simulation. To run the study, simply click "Run This Study" and the simulation will begin.

X.   **View Results**
   A. When viewing results, it is important to view the desired data. In the case of the smart nozzle, displacment is of the most interest. To view displacement results, simply double click "Displacement" under the results tab. Solidworks simulation defaults to resultant displacement which isn't much help when looking for coordinate oriented displacement. Luckily, we are able to select planar displacement. To do so, right click on "Displacment" and click edit defenition. Once in the displacement defenition menu, slick on the "Display" drop down menu for displacement type and select "UY: Y Displacement" (Figure 17). Once selected, the plot of the model will change and will present the correct displacment values to the right of the model (Figure 18).
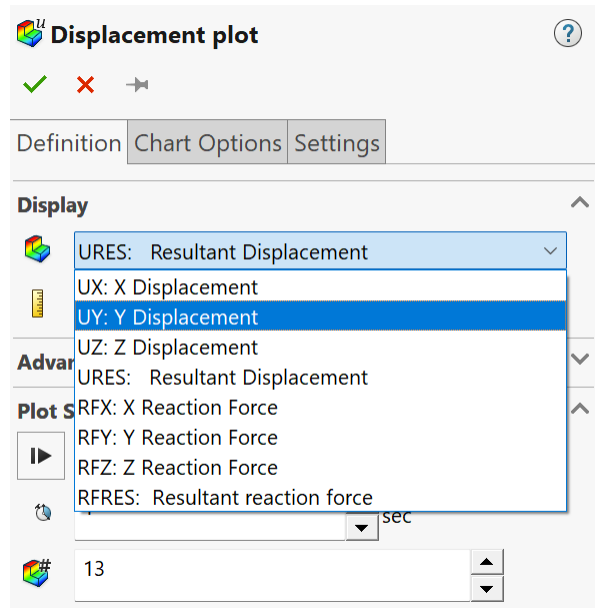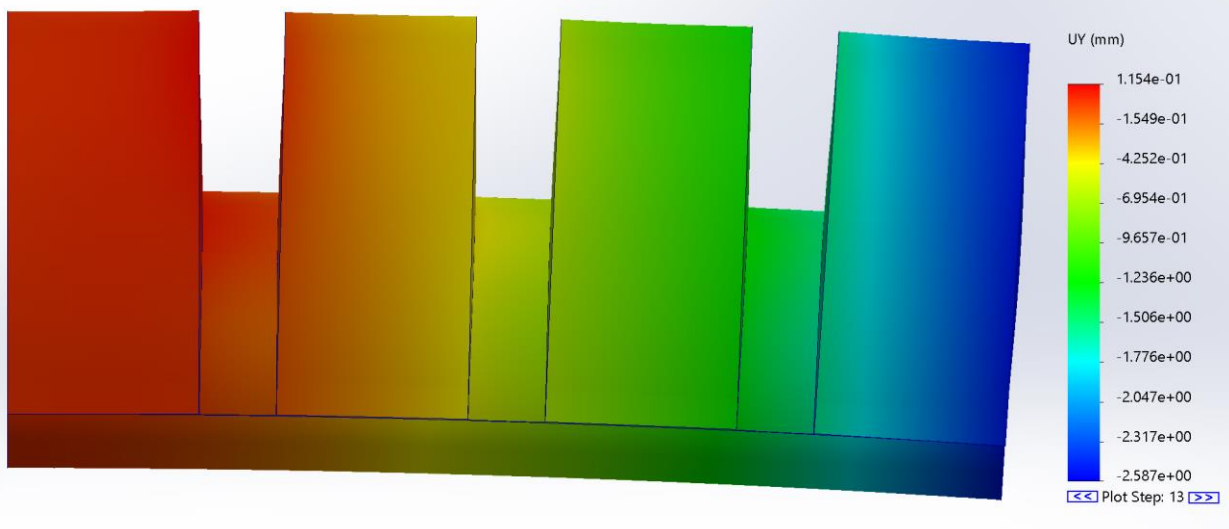
Figure 17: Displacement Selection

Figure 18: Y-Displacement Plot