ROBOTICS AND MECHATRONICS

Modeling of a Piezoelectric Beam Spring

X. (Xinran) Liu

BSc Report

Committee:
Dr. R. Carloni
Dr. H. Hemmes
Dr.ir. H. Wormeester

August 2016

030RAM2016
Robotics and Mechatronics
EE-Math-CS
University of Twente
P.O. Box 217
7500 AE Enschede
The Netherlands

UNIVERSITY OF TWENTE.

MIRA BIOMEDICAL TECHNOLOGY AND TECHNICAL MEDICINE

CTIT

Blank page

Abstract:
This report contains a brief study of the piezo material, an analytical calculation of the beam bending and a dynamic simulation of the changing of stiffness. Finite Element Method is used in statics simulation in SolidWorks. Feedback PI controller and spring-damper system is used in Simulink to simulate the piezo stiffening. The outcomes of this assignment are a SolidWorks assembly that can be used in an upper level system to simulate the artificial muscle and robot joint, and a Simulink voltage controller for the robot to have faster react. It also gives theoretical proof, evaluation and suggestion on the sample.

Blank page

Table of Contents:

1 Introduction

To make an artificial muscle and joint, the springs should be able to change its stiffness. A way to do so is using a movable pivot to change the lever length. It is called variable stiffness actuator. However this setup uses two motors and is too heavy and too big for an artificial muscle. The piezo Variable Stiffness Joint (VSJ) is the alternative for it. It is made of organic composite material mostly. Therefore it is much lighter and smaller.

For example, in the case of the VSJ used in a robot, the actuator should be able to change the stiffness from 0.7 Nm/rad to 948 Nm/rad in 0.6s.[1] Therefore to know dynamic behavior of the material is crucial for later on designing control function for the material to achieve the goal of changing stiffness.

1.1 Problem definition

This bachelor assignment will focus on the piezo leaf spring in the VSJ. A leaf spring with length 59.8mm, width 19.8mm and height 0.19mm is composed by a pair of plastic covering layers, a pair of aluminum layer to provide voltage difference, an active piezo layer and an insulating nylon layer. These springs will be put radially on the VSJ in Figure 4. The first part of the report will focus on the material properties of this sample.

This piezo element can change its stiffness when a voltage difference is applied. In the second part of the report, a SolidWorks model will be built to try simulating the bending under different stiffness (=different voltage), since the robot is designed in SolidWorks.

Finally, the report will focus on the delay of the stiffness changes and give a physical model describing the delay. To compensate this delay, a feedback PI controller is also designed.

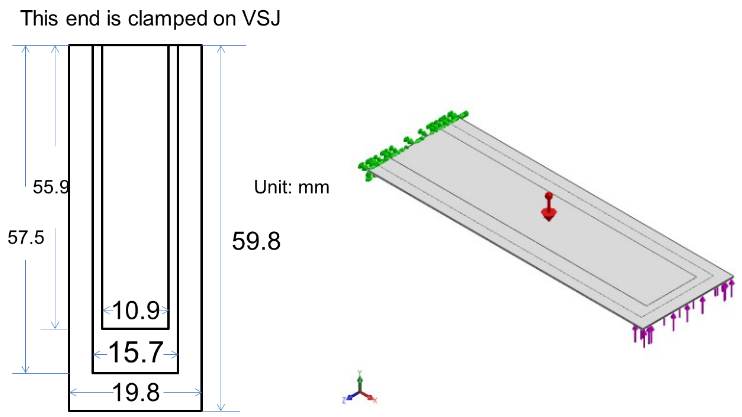


Figure 1: The geometric configurations of the leaf spring.

2 Materials

2.1 Introduction

In this chapter the principle and mechanism of the variable stiffness material are introduced. The materials those constructing the actuator and its leaf spring are described. An experiment about the mechanical properties is given.

2.2 Principle of variable stiffness material

The active piezo layer of the leaf spring is mainly PVdF (polymer of copolymer of vinylidene fluoride and trifluoroethylene $(C_2F_2H_2)_n$). The theory of piezoelectric is that electric charge is accumulated in response to stress applied on the piezo material and reacts back to the material. In the case of the application of VSJ, by adding or removing the charges, the mechanical behavior, the internal stress, is expected to change.

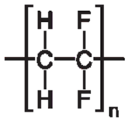
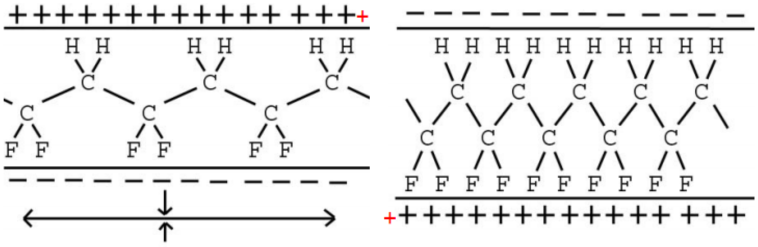


Figure 2: The formula for PVdF[1]



(a) Stretched PVDF polymer under a parallel electric field

(b) Compressed PVDF polymer under an anti-parallel electric field

Figure 3: The piezo effect when the voltage alters its direction [3]

PVdF is a highly polarized organic material. The electric potential center is not in the center of the geometry. This is called the polarization of molecular. The hydrogen is repelled from positive charge as shown in Figure 3 left. The hydrogen is attracted by the electrons as shown in Figure 3 right. When there is a voltage applied in the direction of the polarization, the atom spacing or more precisely bond length will change. This means that the bond energy will change. Along the carbon chain direction, the carbon atoms are closer, and the attraction force is prevalent. The elastic modulus on this direction is larger.

2.3 Applications of PVdF and piezo materials

Organic material became more and more favorable for robot making, because of its low-cost, material variety and mechanical flexibility. [4] In general there are 2 types of piezo material, natural ceramic and polymer piezo. Polymer piezo such as PVdF are famous of its highest piezoelectricity. It can be up to 25pC/N. [4] Due to the pressure sensitivity, piezo materials are widely used in microphones and speakers. Polymer piezo is better over ceramic piezo mainly because of its small size and flexibility. [4]

[1] https://en.wikipedia.org/wiki/PolyVinyliDene_Fluoride

2.4 Actuator construction

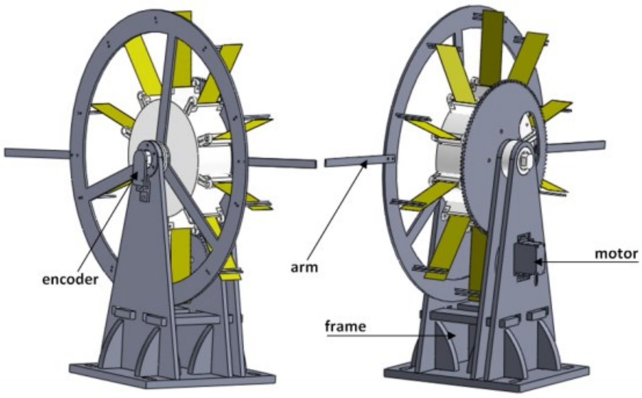


Figure 4: Design of VSJ [2]

Figure 4 is the VSJ (Variable Stiffness Joint) designed by A. Cremonese. The yellow radial placed parts are the piezo springs. The rotation of the outer arm results in a twist effect between inner and outer circles. The leaf spring suffers a bending.
The actuator in Figure 4 is made of motor, laser-cut plastic board and leaf springs. The leaf spring is made of multiple layers, which are:

<u>Aluminum:</u>
This layer is used to provide voltage field for the piezo layer. The two layer of aluminum act as a capacitor.

<u>Piezo:</u>
In between the aluminum, there is the active layer. It becomes stiff when high voltage electric field is present. Without voltage, it is a thin and soft layer like a Teflon tape.

<u>Plastic:</u>
The covering PET plastics fix and seal all the layers. The thickness of the plastic includes also the glue, therefore the combined stiffness is much lower than PET.

<u>Nylon:</u>
The piezo layer does not have a resistance high enough to prevent short circuit. Therefore a layer of nylon is also added in between the two aluminum layers as electric isolation layer to prevent the high-voltage breakdown.

2.5 Experiment result

Considering all layers as a whole, the overall bending constant of the leaf spring is tested by a balance scale experiment. Results obtained from the scale experiment are the force as a function of displacement, and then it is converted to

$$scalar\ spring\ constant \coloneqq\ point\ load/end\ point\ displacement.$$

In the end of this section, using the analytical method, this scalar spring constant is converted to the Young’s modulus of the material.

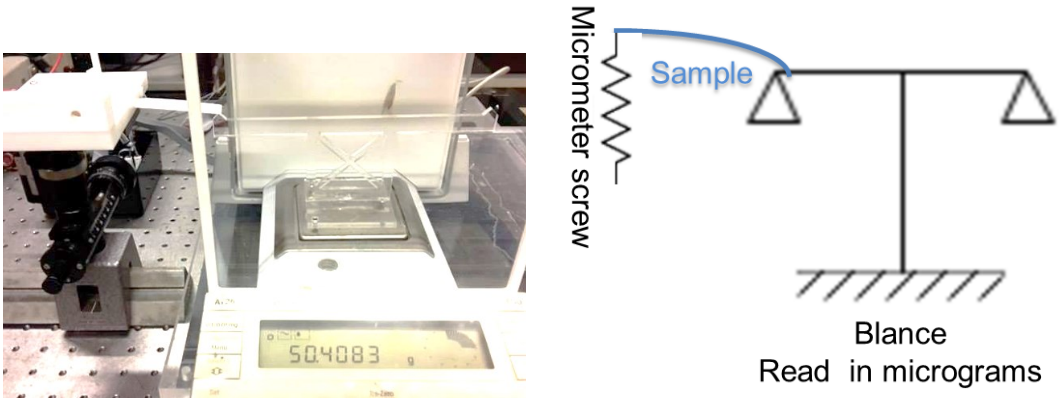


Figure 5: Beam experiment [2] and its schematic diagram

The experiment is operated in the following steps:

- Turn the screw down to lower the clamp, this equivalent to pushing the other end up.
- The difference on the scale read is the equivalent force which bends the spring.

Since, as shown in Figure 4, the spring is placed in all directions, therefore the gravity can be ignored.

When the beam is treated as a whole, the dimensions are: 59.8mm, 19.8mm, and 190μm.

Table 1 the force measured on the balance, as a function of micrometer screw reads and voltage.[2]

Supporting force [μN]	End point displacement = 1.70 mm	2.00 mm	2.40 mm	2.60 mm	3.20 mm	4.00 mm
Voltage = 0 [V]	331	413	489	535	659	822
Voltage = 400 [V]	465	551	658	717	880	1107
Voltage = 700 [V]	549	642	770	844	1033	1290

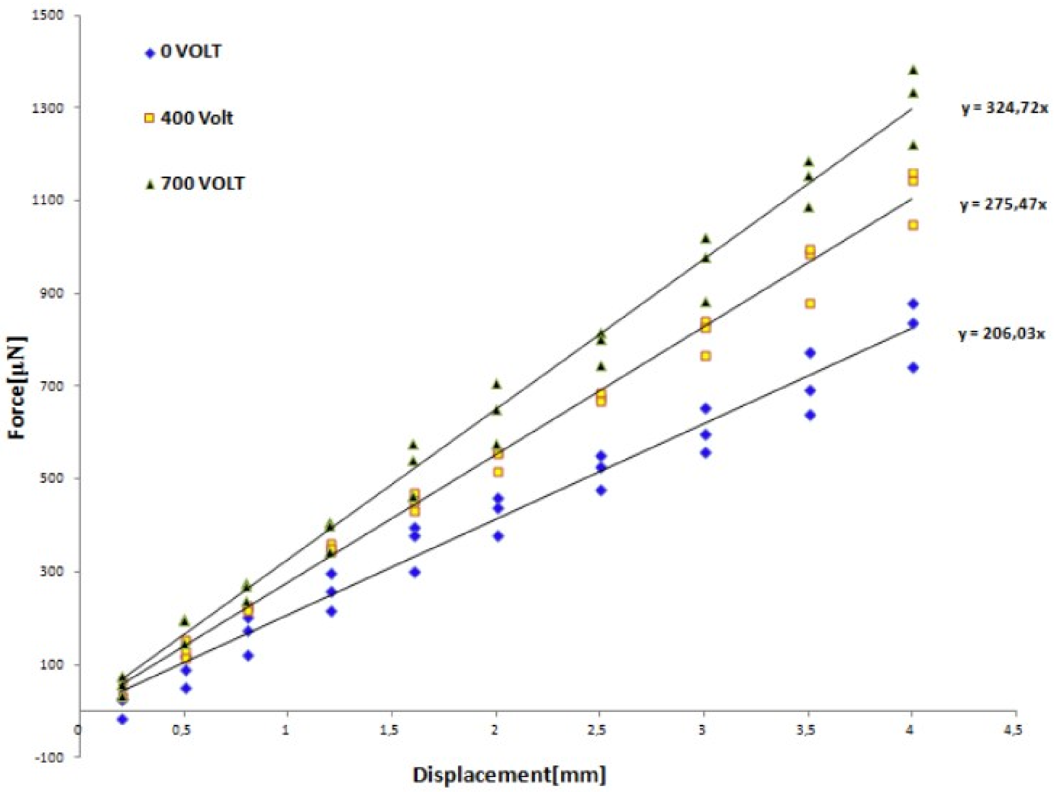


Figure 6: Force measurement results approximated by linear regression. X axis is end point displacement. Y axis is supporting force at end point. [2]

In Figure 6, 'y=206.03x' means:

force[μN]=206.03*end point displacement[mm] when voltage=0.

Similarly,

force[μN]=275.47 *end point displacement[mm] when voltage=400.

force[μN]=324.72 *end point displacement[mm] when voltage=700.

In other words, y is the force in micro Newton; x is the displacement in mm.

Take 0v as an example, by using the statics formula, the end point displacement for an analytical beam is: [6]

$$v_{max} = \frac{FL^3}{3EI}$$

So the 'y=206.03x' in Figure 6 can now be written as

$$x = \frac{1}{206.03}y$$

Let v_{max} replace x and F replace y,

$$v_{max} = \frac{L^3}{3EI} * F$$

For the coefficient part '1/206.03', it has:

$$\frac{1}{206.03} = 0.001\frac{L^3}{3EI}$$

, where the I is the moment of inertia of the cross-section area. For rectangle, it follows: [6]

$$I = \frac{bh^3}{12}$$

Fill in I=1.1317e-14, L=59.8e-3:

$$E = k * 6.2987e6$$

, since k=206.03, 275.47, 324.72 for 0v, 400v, 700v respectively.
So, it gives E=1.298e9, 1.735e9, 2.045e9 Pa respectively.
After a linear fit using code in Appendix 8.1.3, the following voltage-stiffness relation is found.

$$Young's\ module\ E = 1.301e9 + 0.1069e6 * voltage$$

3 Model

3.1 Statics model

3.1.1 Analytical statics model

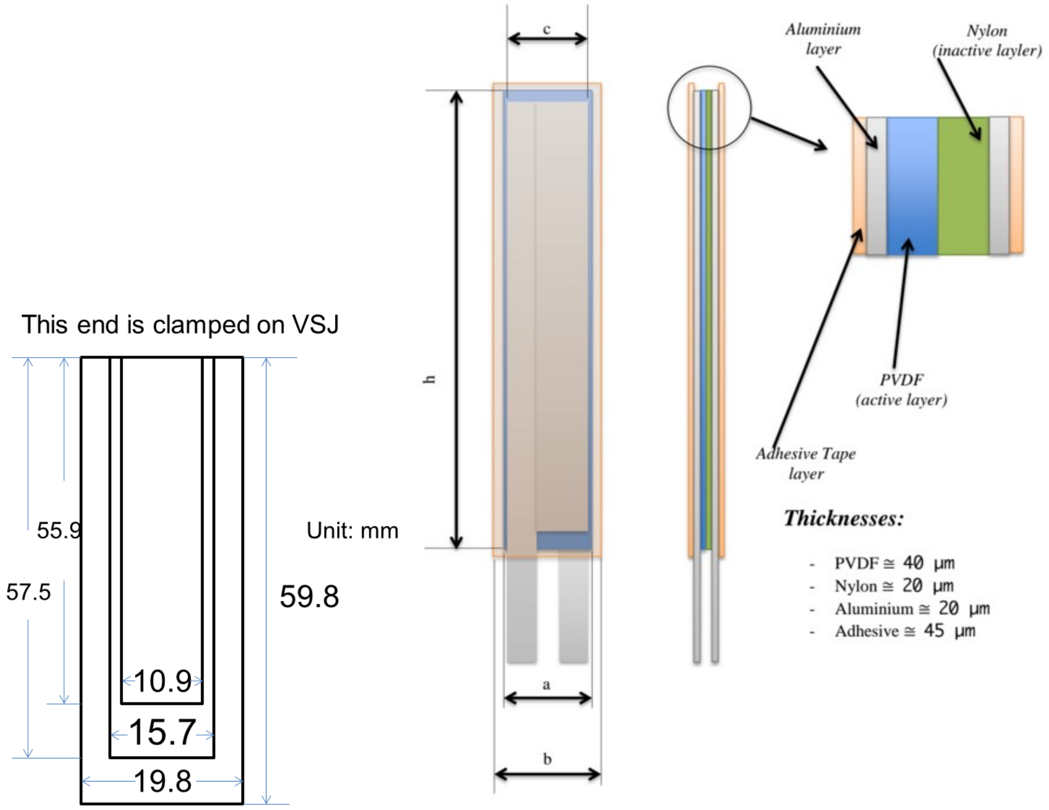


Figure 7: The geometric configurations [2]

Table 2 Material properties and layer dimensions

Material name	Young's modulus, (GPa) [5]	Dimensions (L*W*H) (each layer) (mm)
Aluminum	69	55.9*10.9*0.020
Nylon	1.38	57.5*15.7*0.020
PET plastic	2.96	59.8*19.8*0.045
PVdF	0.4	57.5*15.7*0.040

The first part of this section treats each separate material layer as a beam and applies analytical statics in the following steps:
Since it is called 'Analytical', the Young's Modulus of the material E is used. One layer of the material is a cuboid. The material is homogeneously distributed on this

cuboid. The 3D model can be simplified to 2D model with only the longitude and the thickness direction left. [6] Besides, the cuboid beam can be seen as a linear system. [6] The effective external force loaded on the beam can be separated into two components ---- the supporting force and the body weight. The supporting force in the 2D model is a point load at the end, which makes the beam curves as the following function: [6]

$$v = \frac{Px^2}{6EI}(3L - x) \tag{1}$$

, where P is the force, v is the vertical displacement at point x.

The body weight, the homogenous gravity will make the beam bend as the following function: [6]

$$v = -\frac{wx^2}{24EI}(x^2 + 6L^2 - 4Lx) \tag{2}$$

, where w is the force density along the beam length direction x.

The superposition of the two is the final shape:

$$v = \frac{Px^2}{6EI}(3L - x) - \frac{wx^2}{24EI}(x^2 + 6L^2 - 4Lx) \tag{3}$$

, where the I is the moment of inertia of the cross-section area. For rectangle, it follows: [6]

$$I = \frac{bh^3}{12} \tag{4}$$

Thus the displacement as a function of position x is described as Equation 3. The simulation results of this model are in section 4.1.2.

3.1.2 Finite element model

If the leaf spring is needed to model in more details, i.e. as a stack of multiple components, it can only be modeled with finite element method, since the shape become complicated. The different size and position of different layers will make the analytical equation difficult to derive.

In this section, the leaf spring is modeled in SolidWorks to be able to run the finite element method for the displacement field.

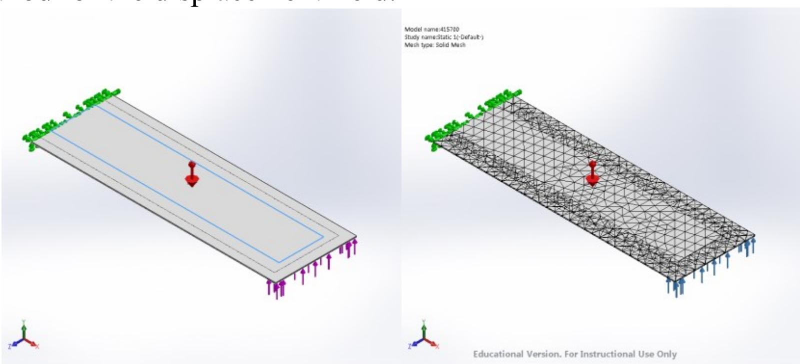


Figure 8: The multi-layer assembly and its meshing

The result is in Section 4.1.3.

3.2 Dynamic model

3.2.1 Introduction to dynamic model

This chapter mainly describes the delay of changing stiffness, when voltage changed. The delay can be found numerically from the piezoelectric strain experiment [3].

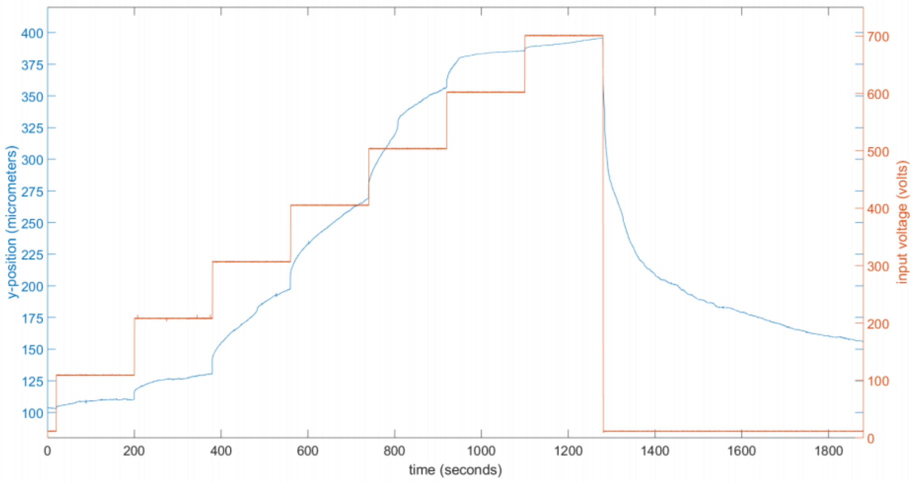


Figure 9: Example plots of the piezoelectric strain experiment [3]. Keep the external load constant; input a stair voltage; observe the displacement.

In summary, the material get fully ready to the new stiffness on its 30 seconds no matter how much the voltage changes is, and get half of its transition within 2.5 seconds. But the requirement is to get almost fully (97%) ready in 0.6 seconds. Since redesign the material is out of the scope of this assignment, the PID controller for the voltage input is designed (in Section 4.2.3) to accelerate the stiffness changing. And in this case, PI is enough precise and stable in describing lagging. [8]

To describe the delay, there are 3 methods in the following section.

3.2.2 Introduction to creep effect

Creep effect is known in polymer science. When a polymer is exposed to a pulling force, it will immediately get an elongation A, and then elongates to B slowly, where A<B. These polymers are usually represented with springs and dampers. [7] In Figure 10, the force will compress the spring K_1 instantly. And then the system is continuously compressed till the spring K_2 reaches its equilibrium. For polymer, the damping coefficient D in the damper is very large, so the second step takes very long. This model is called Zener model.

3.2.3 Zener model

The stiffness changing process in the piezo beam is not an instant process. It takes time to gradually change its stiffness as well as displacement just like the creep effect. So it can be modeled in the Zener model. [7]

Assume spring K_1 changes its stiffness. Since spring K_1 does not have any damping or mass, as the stiffness changed, the displacement changes immediately. So spring K_1 cannot be the piezo part.

Now assume spring K_2 changes its stiffness. The displacement has to change gradually due to the presence of the damper. From the observer point of view, it is the stiffness changes slowly, because the stiffness is measured as the ratio of force over displacement.

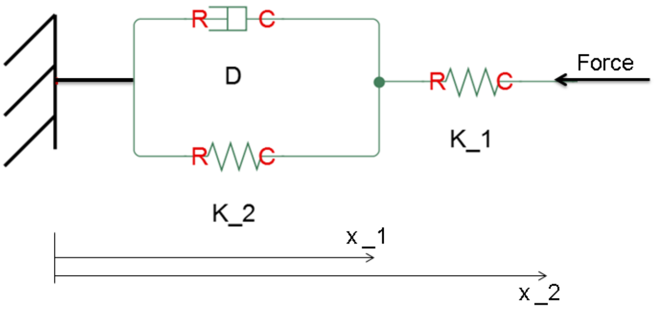


Figure 10: The Zener model [8] [2]

Figure 9 (blue line) shows an instantaneous response to a stiffness change, which can only come from K1, and then shows an gradual change, which can only come from K2-damper system. This means when changing the stiffness, both K1 and K2 changes instantaneously. As a suggestion to future experiment, the ratio of the change of stiffness between K1 and K2 should be found. For this moment, it is assumed the change of stiffness only happens gradually. In other words, only stiffness of K2 contributes to the dynamic behavior. To simplify this model, the spring K1 will be removed.

Besides, the K1 does not change its displacement during any change on the spring-damper part. So removing K1 does not affect K2 and Damper part. This is also proved in its electric analog in Appendix 8.1.7.

Therefore the model now becomes:

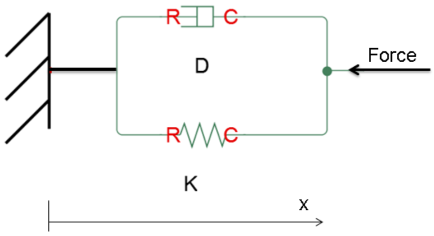


Figure 11: The spring-damper system to simulate the change of stiffness in the piezo.[3]

$$F_D + F_K = F$$
$$F_K = K \cdot (l - x)$$
$$F_D = -D \cdot \frac{dx}{dt}$$
$$K \cdot (l - x) - D \cdot \frac{dx}{dt} = F$$

Note that K and x are independent time domain variables. The set of differential equations will be solvable if there are two independent equations. Since the K is the input, K is a known function. The system becomes solvable in the Simulink. See Section 4.2.2.

The feasibility of removing K1 is the main confusion. And this confusion is now focusing on the definition of 'instantaneous change'. Change in 0 second is called instantaneous change, but it is not realistic. A practical approach is to take it as change

[2] The model is built in Simulink, see Appendix 8.1.5.
[3] Simulink model is in Appendix 8.1.6, result is in Appendix 8.2.1.

in a very short time, shorter than the sampling time. In the simulation results for all 3 of the Zener, Simplified-spring-damper system and the PI models (in Section 4.2.2), they show a rapid change, which are all faster than the sample time. So they can be taken as instantaneous change.

In Simulink for Zener model, not only the parameters but also the initial values in the integrator are dependent on the material and assembly. The PI model (in Section 4.2.2), however, only has two parameters depend on the material and assembly. The integrator initials are always zero, independent from the material and assembly.

So this report tries using PI model to realize the properties of Zener model. And the steps are in Section 4.2.2. PI is just one of the many ways to (partly) describe Zener model.

4 Result

4.1 Result of statics models

4.1.1 Results of statics experiment

The beam bending experiment has been described in section 2.5. In that section, the bending experiment is operated on the whole sample, the obtained Young's modulus is equivalent to the combination of multiple material properties.

4.1.2 Results of analytical statics simulations

Using Equation 3, the displacement field can be plotted in Matlab.

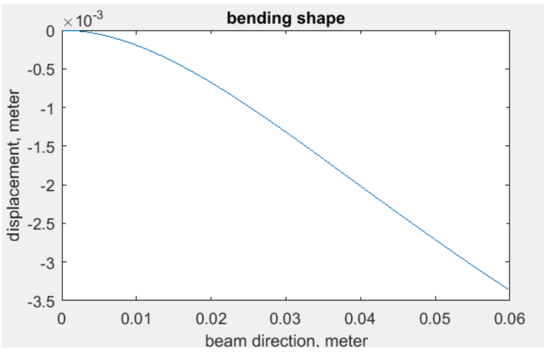


Figure 12: The Matlab plot for one layer of aluminum with a thickness of 40 μm, supporting force is 500μN. The result (displacement) is 3.4 mm down. The code is in Appendix 8.1.1.

For other combinations of stiffness, thickness and point force, figures are omitted, but the results are available in the following chart:

<table>
<tr><td>Disp(mm) (down)</td><td>PET, E=2.76 (GPa)
Thick=90(μm)</td><td>Al, E=69
Thick=40</td><td>PVDF, V=0, E=1
Thick=40</td><td>PVDF, V=400, E=1.7
Thick=40</td><td>PVDF, V=700, E=3.3
Thick=40</td><td>Nylon, E=6
Thick=20</td></tr>
<tr><td>Force=300 uN (up)</td><td rowspan="3">Droop</td><td>3.50</td><td rowspan="3" colspan="3">Droop</td><td rowspan="3">Droop</td></tr>
<tr><td>Force=500 uN</td><td>3.40</td></tr>
<tr><td>Force=700 uN</td><td>3.20</td></tr>
</table>

Force=900 uN		3.00		

In the table above, droop means the material is too soft and it completely bends down. In other words, it is out of the linear deformation range. Therefore in the per layer analysis, only the results for the aluminum layer is useful.

Its 3D model is simulated in SolidWorks.

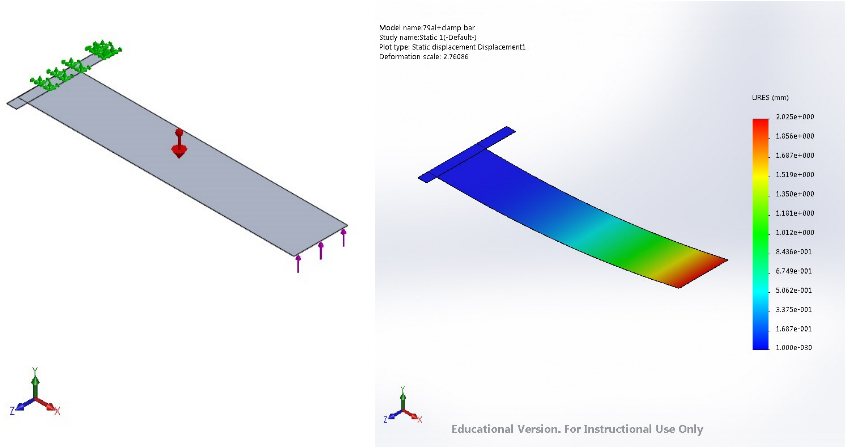


Figure 13 Left: The green arrows are the clamp restrain; the red arrow is the gravity attributed on the whole body; the pink arrows are the point load at the end of the beam.
Right: The displacement plot, max=2.025 mm, direction=up. See also Appendix 8.2.2.
(conditions are: thick=20μm, force=300μN)

For the steps of building the SolidWorks model, see Appendix 8.1.4.

4.1.3 Finite element model

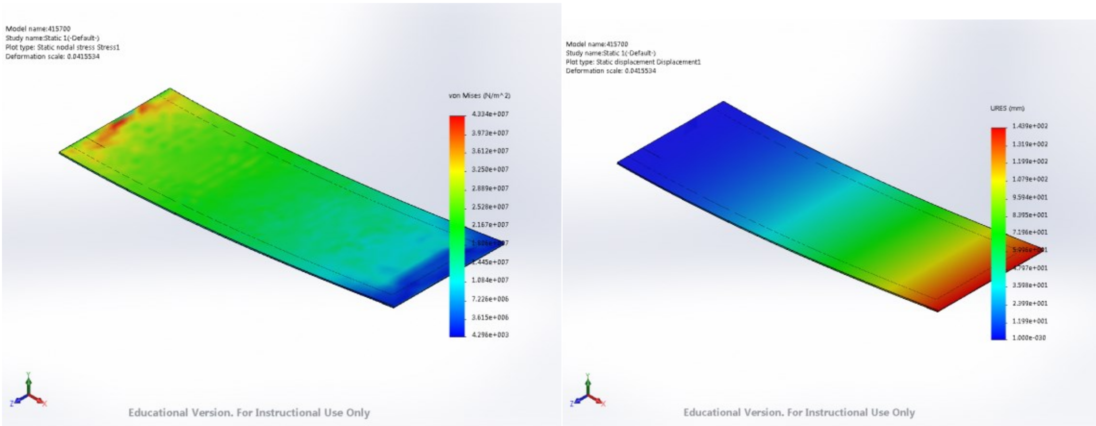


Figure 14 Left: the nodal stress plot, max=4.334e7 N/m^2. Right: the displacement plot, max=1.439 mm, direction=up. (Conditions are: force=500μN, voltage=700v)

For a complete SolidWorks report, see Appendix 8.2.3. For the steps of building the SolidWorks model, see Appendix 8.1.4.

4.1.4 Comparison

Condition	(Average) Experimental [2]	Analytical	Finite element
20µm of AL Force=3e-4N(up) With Gravity.	1.70 mm (up)	1.70 mm (up)	
Multi-layer(FEM) Force=3e-4N(up) Voltage=0	1.58 mm (up)		1.759 mm (up)
Multi-layer(FEM) Force=5e-4N(up) Voltage=0	2.74 mm (up)		2.985 mm (up)
Multi-layer(FEM) Force=3e-4N(up) Voltage=400	1.26 mm (up)		1.450 mm (up)
Multi-layer(FEM) Force=5e-4N(up) Voltage=400	1.78 mm (up)		2.023 mm (up)
Multi-layer(FEM) Force=3e-4N(up) Voltage=700	0.93 mm (up)		1.012 mm (up)
Multi-layer(FEM) Force=5e-4N(up) Voltage=700	1.56 mm (up)		1.784 mm (up)

4.1.5 Conclusion

The material property filled in to SolidWorks assembly is the standard material data. The analytical model for separate layer also uses standard material data.

But the analytical model for beam as a whole uses the effective Young's modulus. It means this model will have little error from the experiment.

The SolidWorks assembly model has a larger systematic error from the experiment, because, due to the glue, the actual material is softer than in the standard data library.

The FEM is useful when the distortion is not observable. For example, a bridge is loaded with a truck or a desk is loaded with mass. The stress will accumulate in the sharp curves or narrow connections. FEM is not suitable for large displacement calculation because of two problems. The first problem is the computation error by discretize a continuous object into discrete linear objects. The second but more important problem is, in FEM, the deformation direction is assumed remain linear with the original direction. When large displacement occurred, this rule will be violated. The FEM usually analyze problems where the inside or the detailed deformation is the main concerns. When the deformation already largely changes the shape and configuration of the outside or the overall of the object, the internal detailed stress and strain are not of concern any more. Therefore the FEM works for both situations but only optimal for small displacement situation. However since the FEM is a requirement of this assignment, the FEM model is not deleted for this report.

When FEM is going to be used in the future reports, the element size should be redefined to avoid the problem of aspect ratio problem in the current model. The average ratio of length, width and height of the 3D element after meshing is called aspect ratio. In the per layer simulation, the aspect ratio is about 110. In the multi-layer simulation, the aspect ratio is about 700. In Finite Element Theory, an acceptable aspect ratio has to be less than 3. The per-layer model and the multi-layer model have aspect ratio out of this range, the calculating error is uncontrollable. This is because, for this sample, length:height=3000:1 approximately. However if the optimal aspect ratio 1.5 is chosen, about 10 million elements will be created. This is out of the range of the memory of a personal computer.

The whole-beam model, which treats all layers as a whole, has an aspect ratio about 10, which makes the error in a reasonable range comparing to the other two models.

In addition to the aspect ratio, there is a problem about the vague definition of gluing the sample. In the SolidWorks model, parts are connected with locking. In practice, for the available samples, the connection may not be very good. But measuring the sliding coefficient is of no use, since these samples are made by hand and may vary a lot.

4.2 Results of dynamic models

4.2.1 Results of dynamic experiment

The test of impulse response and frequency response of the sample is mainly described in section 3.2.1.

4.2.2 Results of dynamic simulation

The creep analog and the electrical analog use differential equation model. It is sensitive to the initial condition and the parameters. Therefore the model is not easy to adapt the parameter changes. The previous report by Visschers had given the impulse response by voltage-strain measurements. On a modeling and controller designing propose, this impulse response model will be enough. It is

$$K = c_1 - c_2 e^{-t}$$

, where c_1, c_2 are constants. [3]
It is a solution to the first order differential equation given in Section 3.2.3.
And the time constant is found to be 5s for -3dB and 30s for settle point. Using this system transfer function, it is found by trial that the following equations have such impulse response as well:

$$H(s) = K_p + \frac{1}{T_i s} \tag{5}$$

$$u(t) = 0.1e(t) + 5\int_0^t e(\tau)\, d\,\tau \tag{6}$$

, where $u(t)$ is the system output, $e(\tau)$ is the error on the feedback loop.

A proportion and integration feedback loop can describe equation (5) and (6). In Figure 15, the plant contains the PI feedback loop. The piezo plant first compute the normalized stiffness and output to port2 for observation, and then compute the stiffness.

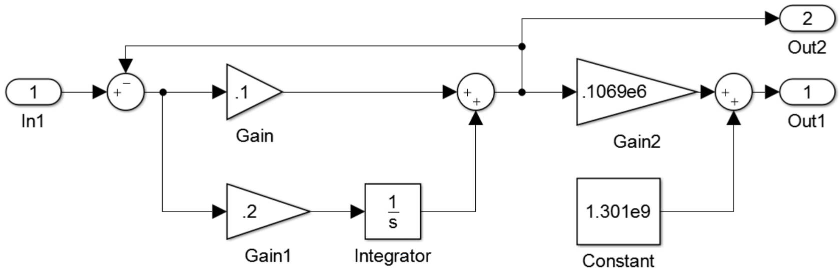


Figure 15: The exploded view of the group ‘plant’ in the Simulink model.

Increasing the gain will make the system react faster, but will also introduce overshoot. The gain Ki 0.1 and gain Ti 0.2 are found by trial. They can make the reaction curve accurate enough to describe the material dynamic.
For control propose, the out1 is the stiffness, meanwhile the out2 is the normalized stiffness which follows the equation below.

$$Young's\ module\ E = 1.301e9 + 0.1069e6 * normalized\ stiffness$$

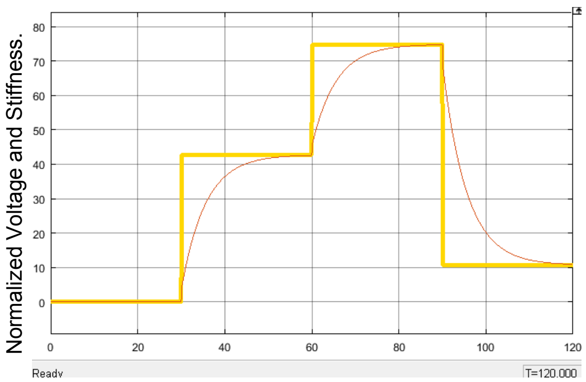


Figure 16: The scope read of PI model, the yellow line is the normalized voltage input; the red line is the normalized Young’s modulus of the whole beam.

This model is independent with the initial conditions and has fewer parameters. It is more stable compare to the differential model, especially when connected to a controller in front of it, as shown in Section 4.2.3.

4.2.3 Controller design for the dynamic model

As shown in Figure 16, the normalized stiffness follows the trend of the voltage with a significant delay.
The method to minimize this delay is called feedback control. A feedback controller should be able to reduce the delay. Therefore a PI controller is added in front of the plant. The plant can be built with any one of the models mentioned in Section 3.2. For the sake of simplicity, PI model is used for the plant to describe the delay of the stiffness change. It can be regarded as a PI cascaded by a PI.
For example, when the voltage rises, the controller will output a higher voltage to let the stiffness change faster.

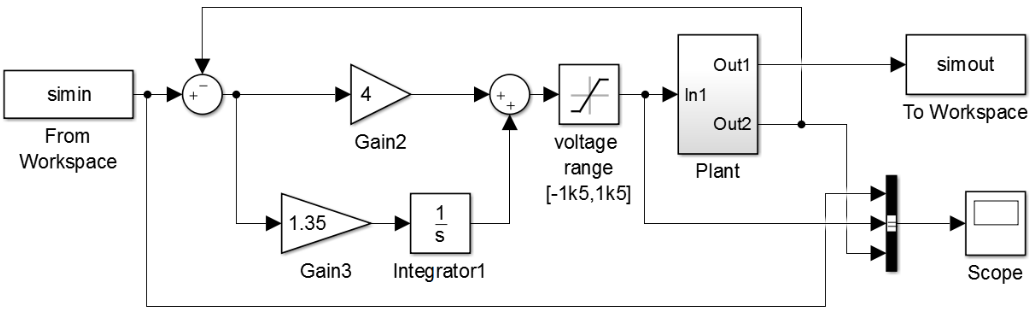


Figure 17 the PI controller is added in front of the PI piezo plant.

In Figure 18, the plant is modeled in PI. So it is a PI plant cascaded by a PI controller. There are two useful outputs from this model. The first one is the one connected to the second port of the scope. It is the controlled voltage (red dash line in Figure 20). This voltage should be deployed to the robot actuator. The other one is a data stream. It is the Out1, which is connected to the 'To Workspace simout'. It is the simulated stiffness of the beam as a time series. This is the Young's modulus for the beam as a whole that can be put in the material library of any mechanical software like SolidWorks.[4] In the application of robot joint, the robot computer calculates the desired stiffness; converts to voltage and feeds to the input port 'From Workspace'.

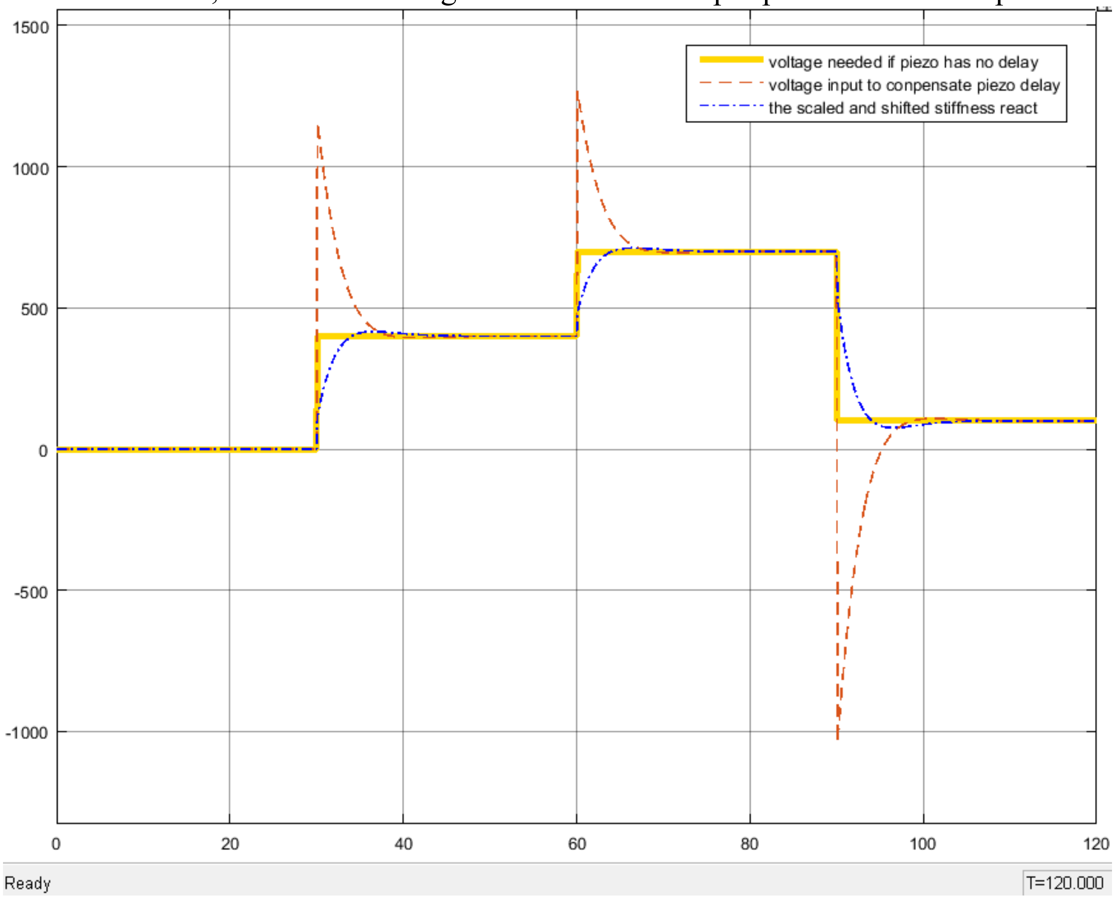


Figure 18: The delay is significantly reduced comparing with Figure 16.

[4] Currently the SolidWorks does not support using a variable in its 'customized material library'.

The third method finally outputs the elasticity Young's modulus that can be read by other simulation software. For example use a variable in the Elastic Modulus of the customized material.

4.2.4 Comparison

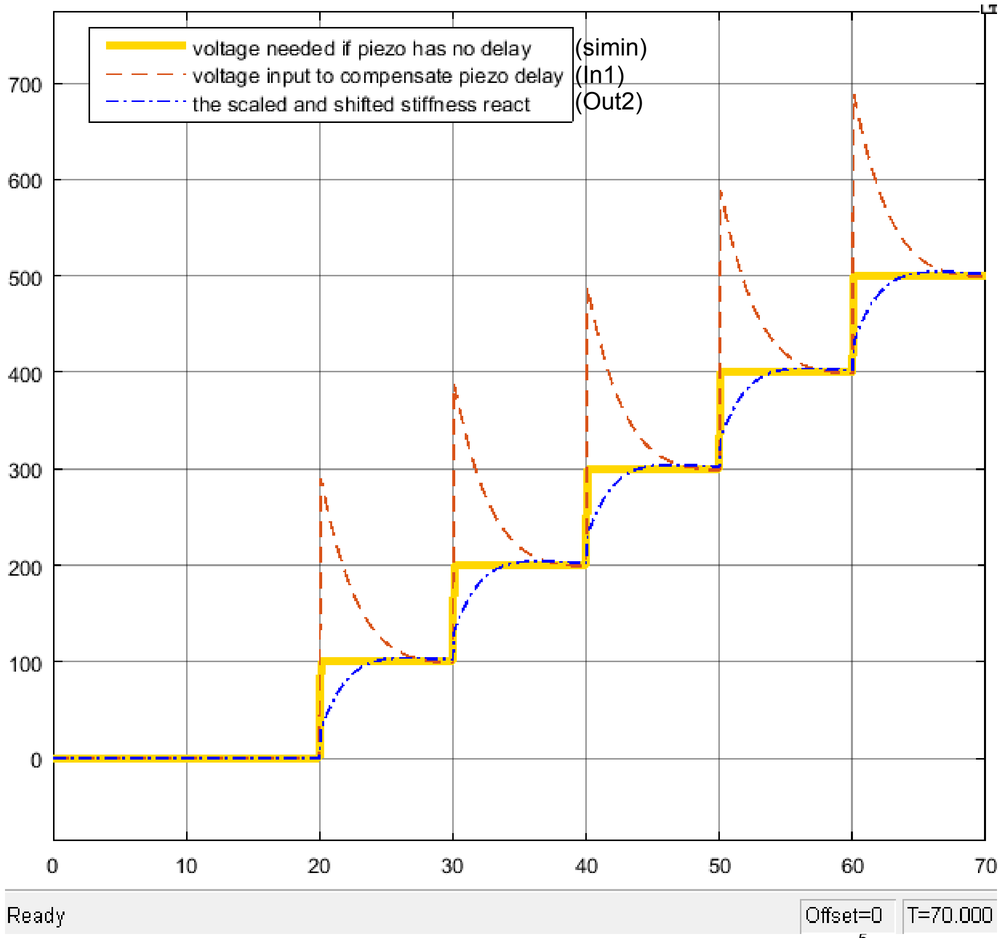


Figure 19: The simulated response (blue line) with controller. [5]

[5] To make the stair voltage input as 'simin', use code in Appendix 8.1.2.

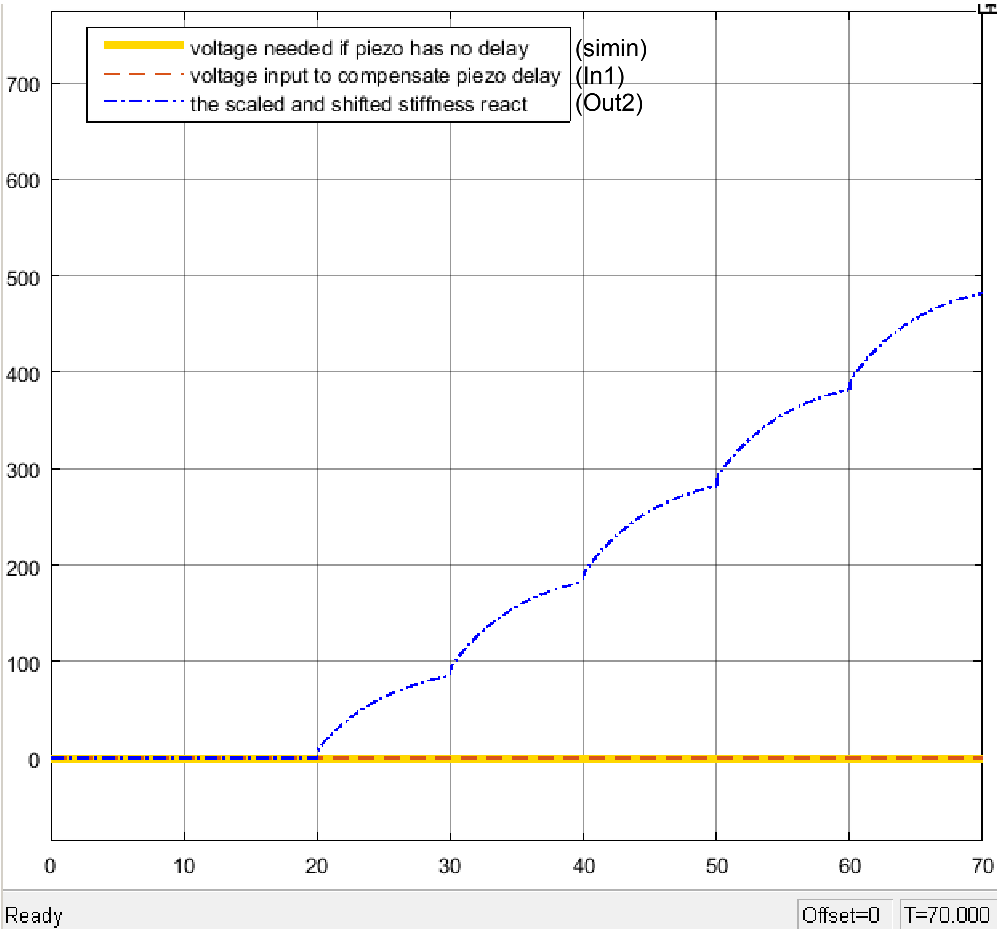


Figure 20: The simulated response (blue line) without controller. Since the controller part is removed, the scope cannot read the other two values. The 'simin' is the same as in Figure 19.

Table 3: Result comparison

Time	Voltage	Measured shifted and scaled stiffness [2] (see also Figure 9 blue line)	Simulated shifted and scaled stiffness PI model **with** controller (see also Figure 19 blue line)	Simulated shifted and scaled stiffness PI model **without** controller (see also Figure 20 blue line)
0	0	0	0	0
20.1	100	43	28.74	9.91
21	100	45	59.15	23.51
30.1	200	67	130.88	95.28
31	200	70	160.77	111.09
40.1	300	127	230.89	192.91
41	300	134	260.76	209.08
43	300	155	293.68	236.79
45	300	167	303.14	256.06
50.1	400	321	330.88	292.53
51	400	330	360.76	308.75

53	400	343	393.68	336.57
55	400	367	430.14	355.90
60.1	500	375	430.88	392.46
61	500	408	460.76	408.69
63	500	443	493.68	436.53
65	500	479	503.14	455.88

The error between experiment and simulation is not small. It may be caused by the fact that the sample in the experiment is an old sample with some electrical shorts and charge leakage.
With the controller, it takes 3 seconds to get more than 90% of the change of stiffness. Without the controller, it takes 5 seconds and only gets 50% of the change of stiffness. So the controller makes a better performance on the application requirement.

5 Conclusion

This report gives the analytical equation for whole leaf spring bending when the spring is treated as a whole; the result of it matches the experiment with little error.

This report gives the Finite Element Model for leaf spring bending when the spring is treated as an assembly of multiple layers; the result of it matches the experiment with maximum of 45% error. FEM is not suitable for large displacement model. As a side product, the SolidWorks assembly can be put in the VSJ model to complete the details.

This report gives the dynamic model developed from the Zener model of Creep. The result matches the experiment only qualitatively maybe because the sample is of low quality. There are two side products. One is the simulated stiffness data as a function of time. This considers the delay of the change of stiffness. This variable can be filled into the material library of the simulation software to run the simulation with a variable stiffness. The other one is the PI controller which let the spring change the stiffness faster by applying a voltage overshoot. The controller achieved to shorten the half point time by more than 80%. (With controller, the half point time is less than 1s. Without controller the half point time is more than 5s on both experiment and simulation.)

6 Suggestion

The sample is now cut and made by hands. There are a lot of variant in size, way of glue and connections. It is advisable to improve the process of manufacturing.

7 References

[1] S.S. Groothuis et al. "The vsaUT-II: A novel rotational variable stiffness actuator." Robotics and Automation (ICRA), 2012 IEEE International Conference on 14 May. 2012: 3355-3360.
[2] A. Cremonese, “Variable Stiffness Joint”, Master thesis, Robotics and Mechatronics, University of Twente, APR 2016.
[3] S.H.J. Visschers, “The characterization of an electrospun PVdF terpolymer for the design of a piezoelectric variable stiffness actuator”, University of Twente, BSc report 026RAM2015, Aug. 2015
[4] P.L. Regtien, Piezoelectric Sensors, In Sensors for Mechatronics, Elsevier, Oxford, 2012, Pages 219-239, ISBN 9780123914972, http://dx.doi.org/10.1016/B978-0-12-391497-2.00008-X.

[5] W.D. Callister and D.G. Rethwisch, Fundamentals of Materials Science and Engineering, 4th edition, John Wiley & Sons, 2012, ISBN 9781118061602.
[6] R.C. Hibbeler, Statics and Mechanics of Material, 12th edition, Pearson Education, 2011, ISBN 9789810686321.
[7] N.G. McCrum, Principles of Polymer Engineering, 2nd edition, Oxford University Press, 1997, ISBN 9780198565260.
[8] H. Meng, The Principle of Automatic Control, 2nd edition, China Machine Press, 2013, ISBN 9787111448273.

8 Appendix

8.1 Models

8.1.1 Matlab code for analytical models

```
E=69e9;%material
L=59.8e-3;
P=900e-6;%force
x=1e-3*59.8;%for multiple point, say x=1e-3*[.1:.1:59.8];
%for end-point only, say x=1e-3*59.8
b=19.8e-3;
h=20e-6;%thick
I=(b*h^3)/12;
mcl=2700*1*b*h;%density e.g. 2.7g/cm3, fill in mcl=2700*1*b*h= how many kg a
1 meter long such beam
w=mcl*9.8/1;
v=(x.^2*P)/(6*E*I).*(3*L-x)-(w*x.^2)/(24*E*I).*(x.^2+6*L^2-4*L*x) %positive
direction up, so gravity down
```

8.1.2 Matlab code for voltage input

```
t=[.1:.1:70];
v(1:200)=0;
v(201:300)=100;
v(301:400)=200;
v(401:500)=300;
v(501:600)=400;
v(601:700)=500;
simin=[t;v]';
```

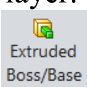
8.1.3 Matlab code for linear fit

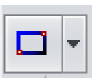
```
>> st=1e9*[1.298,1.735,2.045]';vt=1e2*[0,4,7]';
>> f0=fit(vt,st,'poly1')
f0 = Linear model Poly1:
      f0(x) = p1*x + p2
       Coefficients (with 95% confidence bounds):
        p1 =    1.069e+06  (8.574e+05, 1.28e+06)
        p2 =    1.301e+09  (1.203e+09, 1.399e+09)
```

8.1.4 Building SolidWorks models

Build parts and assembly:

Create one layer:

- Select 

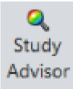
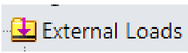
(Boss extruded)
- Click Top Plane
- Select sketch tool:  (corner rectangle) to create board of the sheet part
- Enter dimensions of the two edges
- Exit sketch mode and then enter thickness for the extrusion

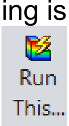
Assembly multiple layers:

- Select  to enter mate selection mode
- Select an edge and a face, by default it will make them coincident, which means in same plane
- Repeat this for an edge that is orthogonal to the previous one. Now two layers are completely defined/constrained.
- Repeat these for the other contacts

Apply material properties, mesh and run Statics simulation:

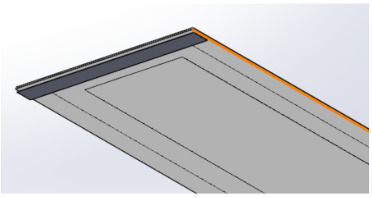
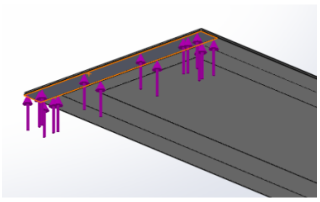
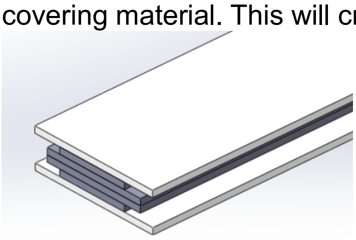
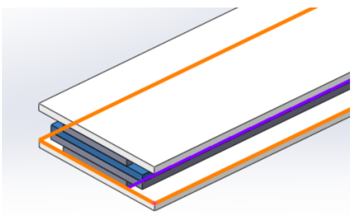
Prepare a simulation study:

- Click  to start a new study, save it as "Statics 1"
- Fill in material's elasticity, shearing modulus, Poisson ratio and density to a customized material library.
- Right click  and then select fix geometry
- Right click  to add force and gravity
- Right click  to create mesh, For the first round of meshing, the coarsest meshing is chosen. There will be parts failed for this time because they are small. And then a finer meshing is applied to them by using mesh control per part.

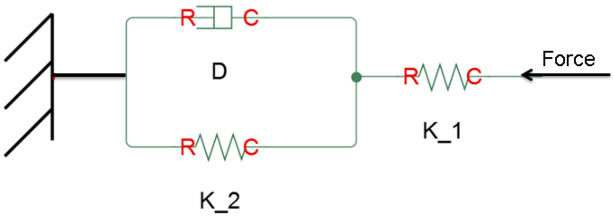
Run the simulation: click  to run the simulation, common errors and their solutions are listed below:

- "fixture transfer failed" error can be solved by remesh with a finer mesh, especially for the face that selected as fixed geometry.
- "database not found" error or "share violation" error may have something to do with using dropbox to save temporary files.

Other tips:

- If SolidWorks files are located in Dropbox, please quit the Dropbox to avoid 'database not found' error.
-  Attach a small piece of the covering material. This will create a face that force is act on.
-  Make the part exaggeratedly thick before assembly. This will allow you to click the face and edges and mate them. And then change the thickness by clicking edit extrusion. And then click "rebuild".

8.1.5 Equation and Simulink model of the full Zener dynamic models



The summation of force at the left end of the spring 1 is 0,

$$-F_1 + F_2 + F_D = 0$$

Express the forces in terms of variables,

$$F_1 = k_1(x_1 - l_1)$$
$$F_2 = k_2(x_2 - x_1 - l_1)$$
$$F_D = -D\frac{d(x_2 - x_1)}{dt}$$

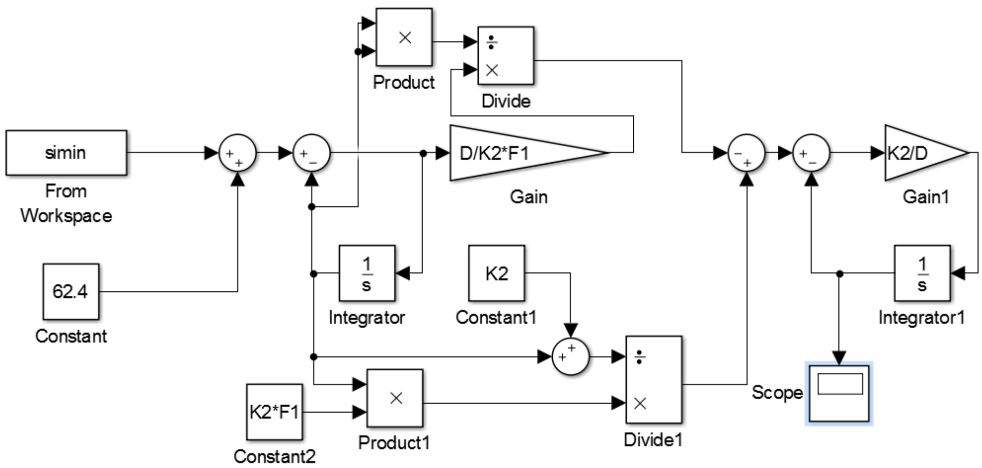
So, [8]

$$x_2 - \frac{F_1}{K_1} = \frac{F_1}{K_2} - \frac{\frac{D}{K_2}dx_2}{dt} + \frac{\frac{D}{K_2}d\left(\frac{F_1}{K_1}\right)}{dt}$$

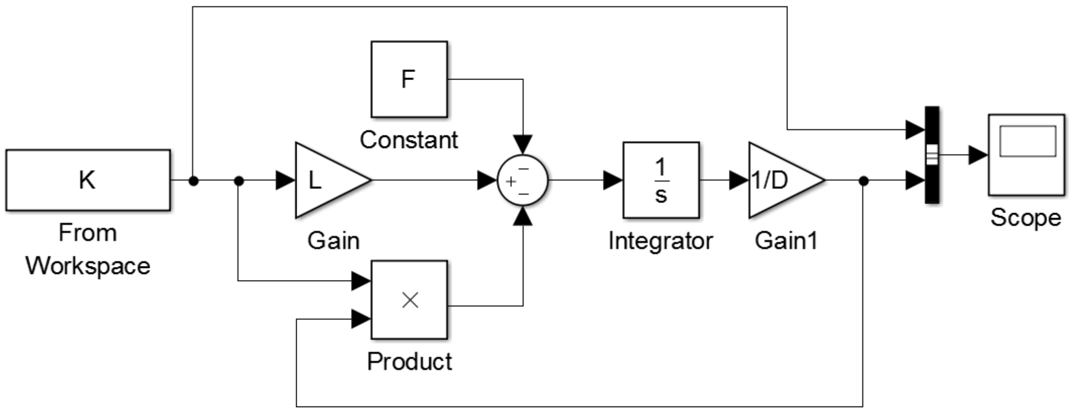
Since the force does not change, the term dF_1/dt =0

$$x_2 + \frac{D}{K_2}\frac{dx_2}{dt} = \frac{F_1(K_1 + K_2)}{K_1K_2} - \frac{D}{K_2}\frac{F_1}{K_1^2}\frac{dK_1}{dt}$$

It is built in Simulink as the following graph. x_2 is the output of the simulation, K_1 is the input (simin).



8.1.6 Simulink model of the simplified spring-damper system



Result of it is in Appendix 8.2.1

8.1.7 Electrical analog

Applying the bond graph theory, the following analog can be used:

- The external force is the voltage. When springs are in series, they share same force. So the corresponding capacitors will be connected in parallel to share the same voltage.
- The spring is the capacitor. A spring with larger spring constant compresses less, therefore stores less energy. Therefore the large spring constant actually corresponds to small capacitance.
- The damper is the resistor. The damper restricts the speed of spring compression just like the resistor limiting the current which charges the capacitor.

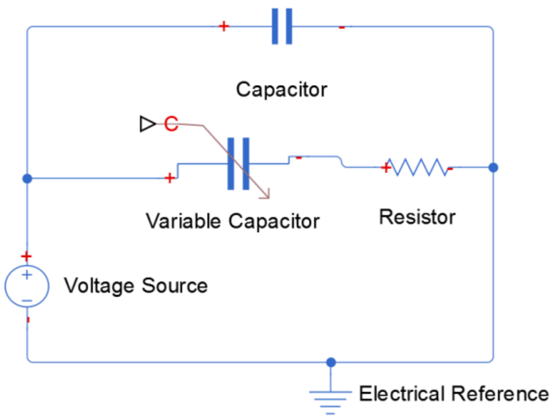
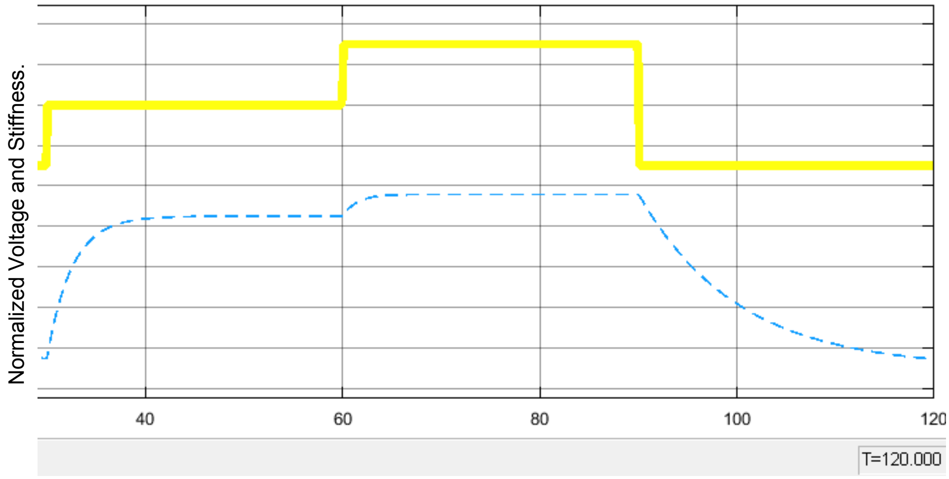


Figure 21: The electric analog of the spring-damper system

As shown in Figure 16, the capacitor loop is independent from the variable capacitor loop, since the voltage on the capacitor is always the source voltage. Remove the capacitor loop. The rest of the system will have the same dynamic as in Section 3.2.3.

8.2 Results

8.2.1 Simulink results for spring-damper model



, where the parameters are:

```
t=.1:.1:120;
vol(1:300)=1;
vol(301:600)=4;
vol(601:900)=7;
vol(901:1200)=1;
K=[t;vol]';
D=10;
L=1;
F=10;
```

8.2.2 SolidWorks result for single layer

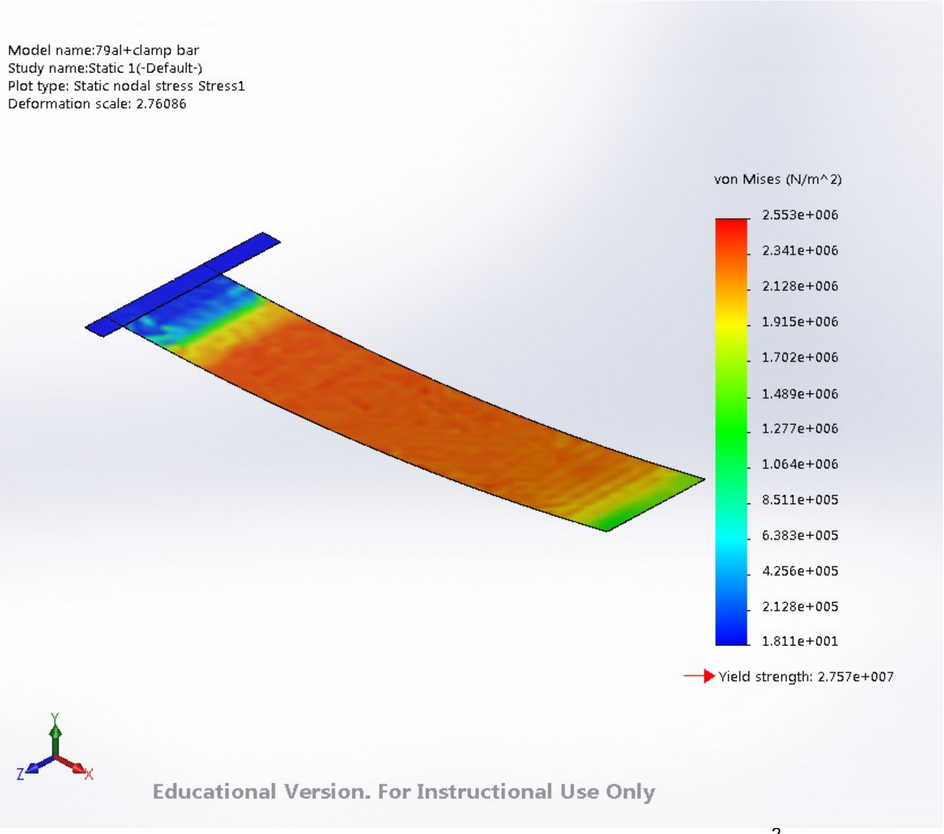


Figure 22: The nodal stress plot, max=2.553e6 N/m^2

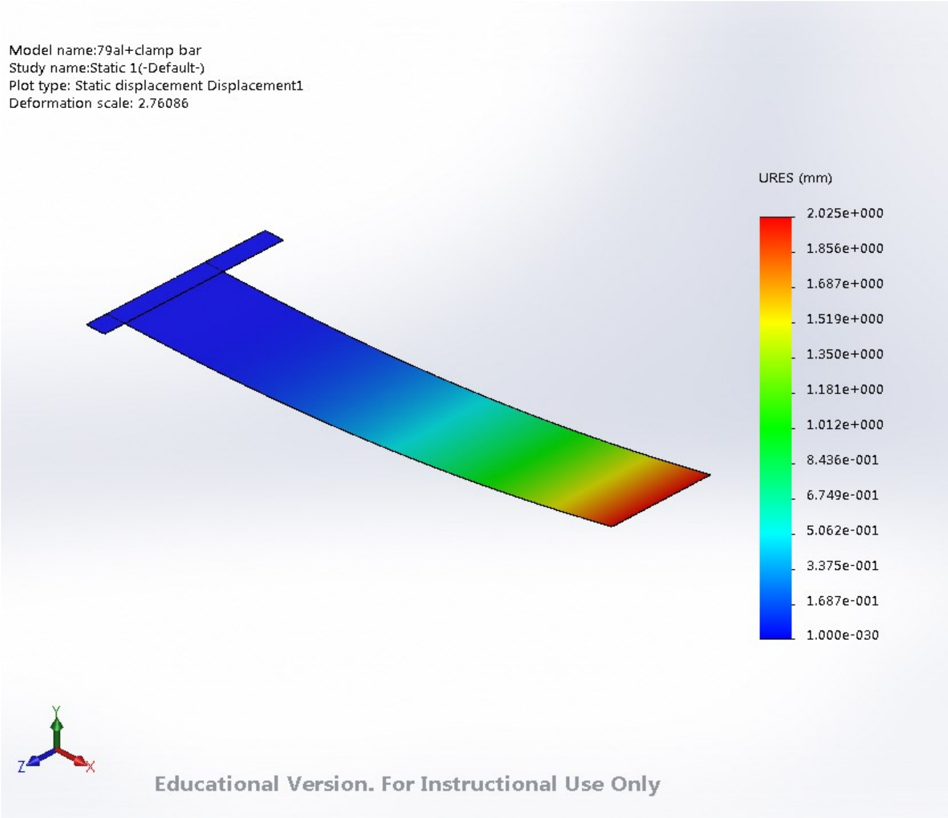
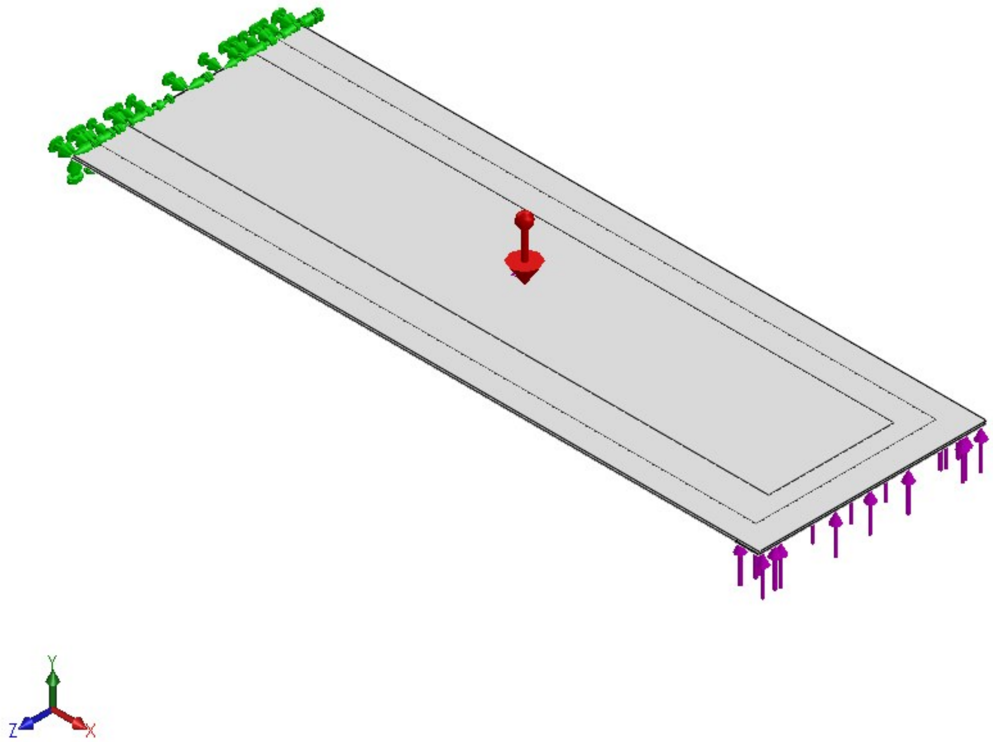


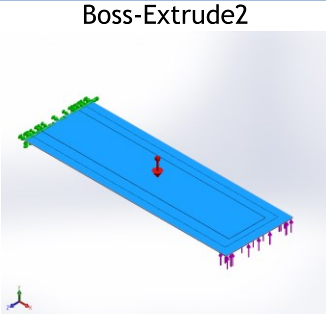
Figure 23: The displacement plot, max=2.025 mm, direction=up.

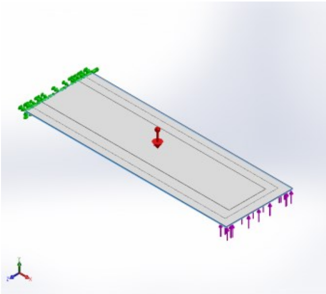
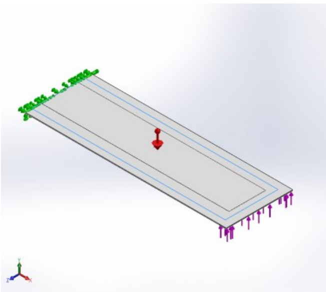
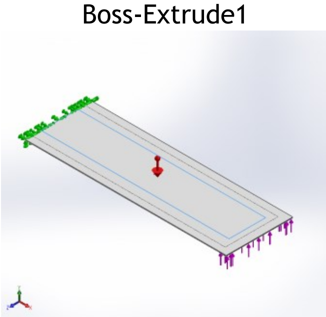
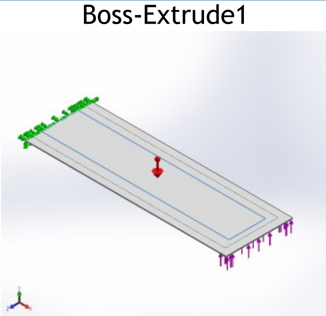
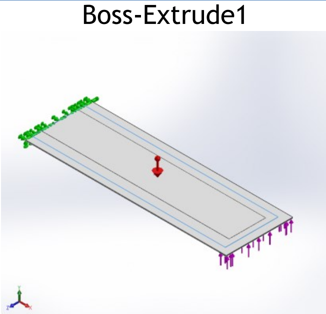
8.2.3 Information on SolidWorks multilayer assembly model and its full result report.

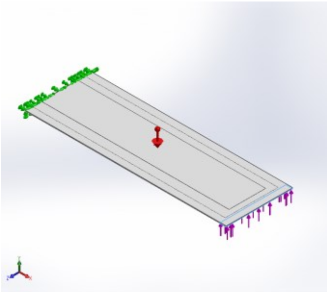
Model Information



Model name: 415700
Current Configuration: Default

Solid Bodies			
Document Name and Reference	**Treated As**	**Volumetric Properties**	**Document Path/Date Modified**
Boss-Extrude2 	Solid Body	**Mass:7.56602e-005 kg** **Volume:5.32818e-008 m^3** **Density:1420 kg/m^3** **Weight:0.00074147 N**	**C:\Users\Administrator\supporting files\323PET.SLDPRT** Apr 10 18:52:11 2016

Boss-Extrude2 	Solid Body	**Mass:7.56602e-005 kg Volume:5.32818e-008 m^3 Density:1420 kg/m^3 Weight:0.00074147 N**	**C:\Users\Administrator\supporting files\323PET.SLDPRT** Apr 10 18:52:11 2016
Boss-Extrude1 	Solid Body	**Mass:7.222e-005 kg Volume:3.611e-008 m^3 Density:2000 kg/m^3 Weight:0.000707756 N**	**C:\Users\Administrator\supporting files\408act.SLDPRT** Apr 08 13:13:27 2016
Boss-Extrude1 	Solid Body	**Mass:1.24299e-005 kg Volume:1.21862e-008 m^3 Density:1020 kg/m^3 Weight:0.000121813 N**	**C:\Users\Administrator\supporting files\408al.SLDPRT** Apr 10 18:52:11 2016
Boss-Extrude1 	Solid Body	**Mass:1.24299e-005 kg Volume:1.21862e-008 m^3 Density:1020 kg/m^3 Weight:0.000121813 N**	**C:\Users\Administrator\supporting files\408al.SLDPRT** Apr 10 18:52:11 2016
Boss-Extrude1 	Solid Body	**Mass:2.07632e-005 kg Volume:1.8055e-008 m^3 Density:1150 kg/m^3 Weight:0.00020348 N**	**C:\Users\Administrator\supporting files\408nyl.SLDPRT** Apr 08 13:13:27 2016

Boss-Extrude1 	Solid Body	**Mass:2.8116e-006 kg** **Volume:1.98e-009 m^3** **Density:1420 kg/m^3** **Weight:2.75537e-005 N**	**C:\Users\Administrator\supporting files\forcebar329.SLDPRT** Mar 29 19:41:13 2016

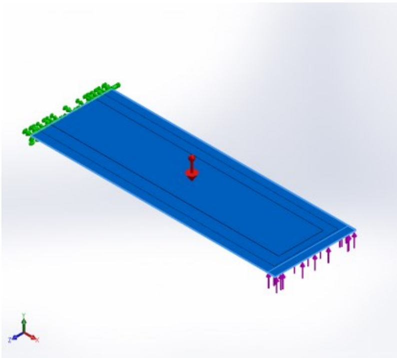
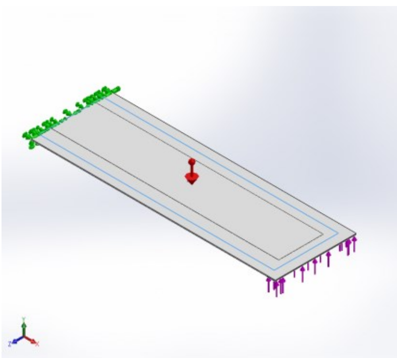
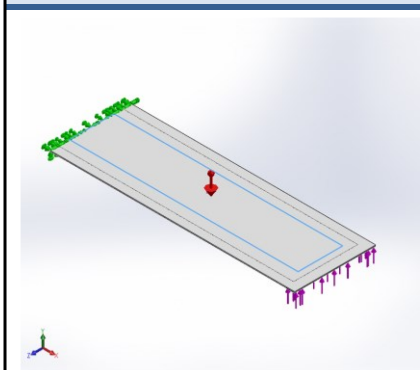
Study Properties

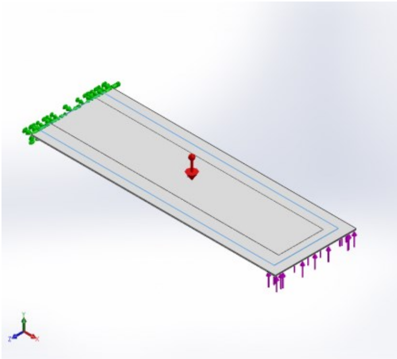
Study name	Static 1
Analysis type	Static
Mesh type	Solid Mesh
Thermal Effect:	On
Thermal option	Include temperature loads
Zero strain temperature	298 Kelvin
Include fluid pressure effects from SOLIDWORKS Flow Simulation	Off
Solver type	FFEPlus
Inplane Effect:	Off
Soft Spring:	Off
Inertial Relief:	Off
Incompatible bonding options	Automatic
Large displacement	Off
Compute free body forces	On
Friction	Off
Use Adaptive Method:	Off
Result folder	SOLIDWORKS document (C:\Users\Administrator\supporting files)

Units

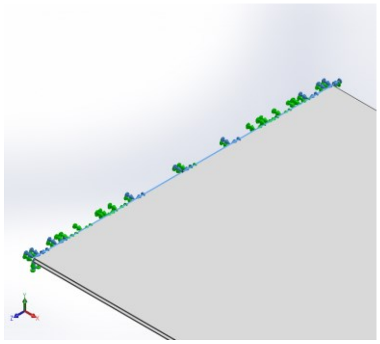
Unit system:	SI (MKS)
Length/Displacement	mm
Temperature	Kelvin
Angular velocity	Rad/sec
Pressure/Stress	N/m^2

Material Properties

Model Reference	Properties	Components
	**Name:** PET **Model type:** Linear Elastic Isotropic **Default failure criterion:** Unknown **Tensile strength:** 5.73e+007 N/m^2 **Compressive strength:** 9.29e+007 N/m^2 **Elastic modulus:** 2.96e+009 N/m^2 **Poisson's ratio:** 0.37 **Mass density:** 1420 kg/m^3	SolidBody 1(Boss-Extrude2)(323PET-1), SolidBody 1(Boss-Extrude2)(323PET-2), SolidBody 1(Boss-Extrude1)(forcebar329-1)
Curve Data:N/A		
	**Name:** PTFE (1) **Model type:** Linear Elastic Isotropic **Default failure criterion:** Max von Mises Stress **Yield strength:** 1.4e+008 N/m^2 **Tensile strength:** 3e+007 N/m^2 **Elastic modulus:** 2e+008 N/m^2 **Poisson's ratio:** 0.46 **Mass density:** 2000 kg/m^3	SolidBody 1(Boss-Extrude1)(408act-1)
Curve Data:N/A		
	**Name:** Default **Model type:** Linear Elastic Isotropic **Default failure criterion:** Max von Mises Stress **Yield strength:** 1e+015 N/m^2 **Tensile strength:** 3e+007 N/m^2 **Elastic modulus:** 2.7e+006 N/m^2 **Poisson's ratio:** 0.394 **Mass density:** 1020 kg/m^3 **Shear modulus:** 3189 N/m^2	SolidBody 1(Boss-Extrude1)(408al-1), SolidBody 1(Boss-Extrude1)(408al-2)
Curve Data:N/A		

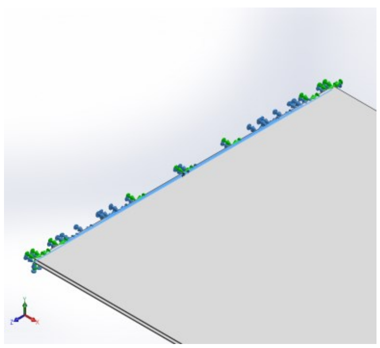
	**Name:** Nylon 101 **Model type:** Linear Elastic Isotropic **Default failure criterion:** Max von Mises Stress **Yield strength:** 6e+007 N/m^2 **Tensile strength:** 7.92897e+007 N/m^2 **Elastic modulus:** 1e+009 N/m^2 **Poisson's ratio:** 0.3 **Mass density:** 1150 kg/m^3 **Thermal expansion coefficient:** 1e-006 /Kelvin	SolidBody 1(Boss-Extrude1)(408nyl-1)
Curve Data:N/A		

Loads and Fixtures

Fixture name	Fixture Image	Fixture Details
Fixed-4		**Entities:** 1 face(s) **Type:** Fixed Geometry

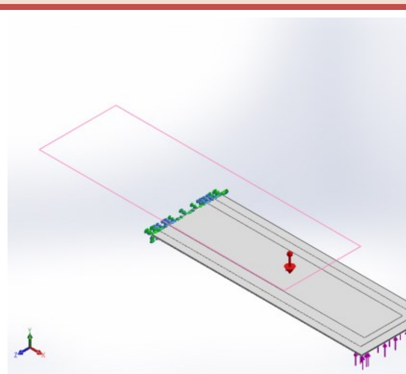
Resultant Forces

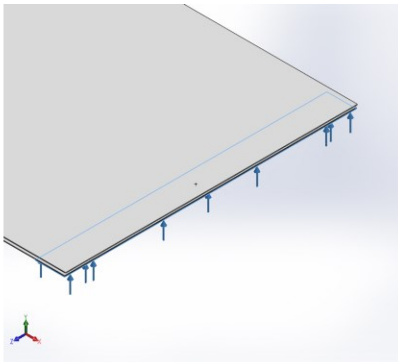
Components	X	Y	Z	Resultant
Reaction force(N)	23.6437	-11.8298	-0.369024	26.4406
Reaction Moment(N.m)	0	0	0	0

Fixture name	Fixture Image	Fixture Details
Fixed-5		**Entities:** 5 face(s) **Type:** Fixed Geometry

Resultant Forces

Components	X	Y	Z	Resultant
Reaction force(N)	-29.1365	48.1737	0.678528	56.3037
Reaction Moment(N.m)	0	0	0	0

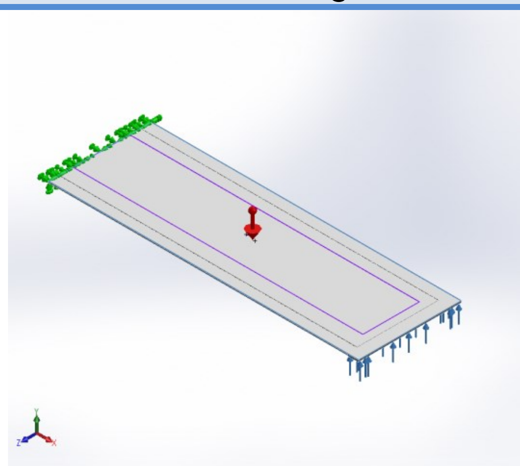
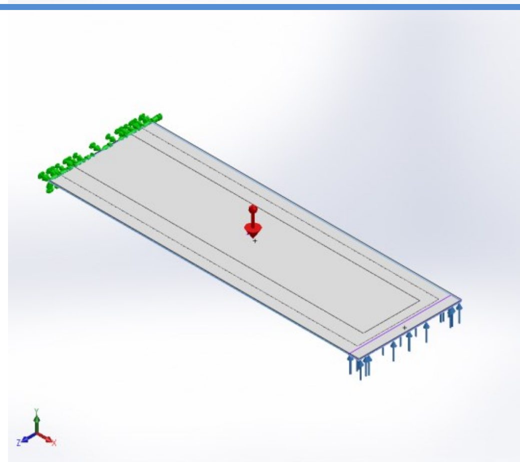
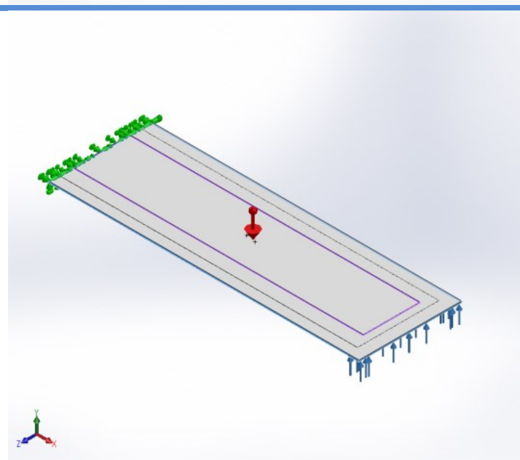
Load name	Load Image	Load Details
Gravity-1		**Reference:** Top Plane **Values:** 0 0 -9.81 **Units:** SI

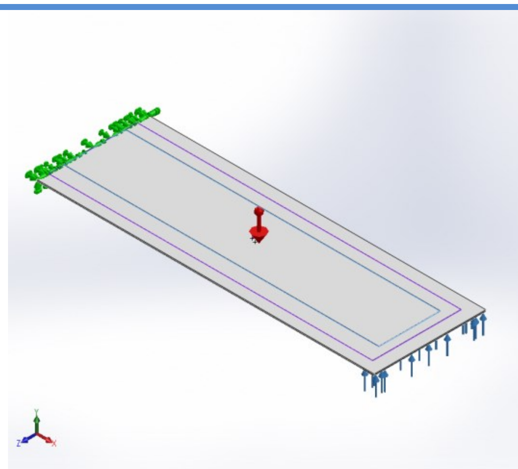
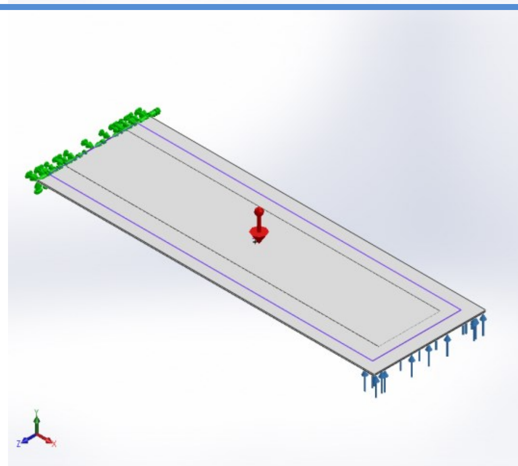
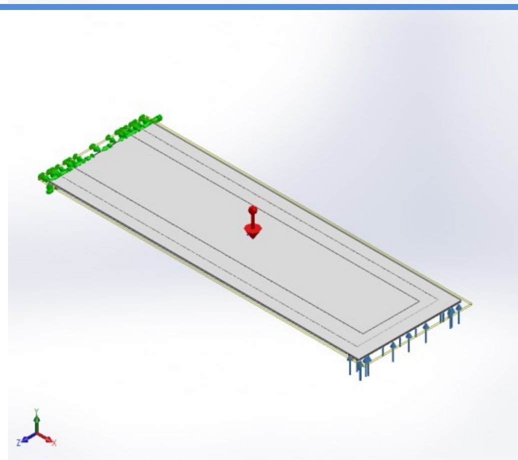
Force-1		**Entities:** 1 face(s) **Type:** Apply normal force **Value:** 0.15 N

Connector Definitions

No Data

Contact Information

Contact	Contact Image	Contact Properties
Contact Set-1		**Type:** Bonded contact pair **Entites:** 2 face(s)
Contact Set-2		**Type:** Bonded contact pair **Entites:** 2 face(s)
Contact Set-3		**Type:** Bonded contact pair **Entites:** 2 face(s)

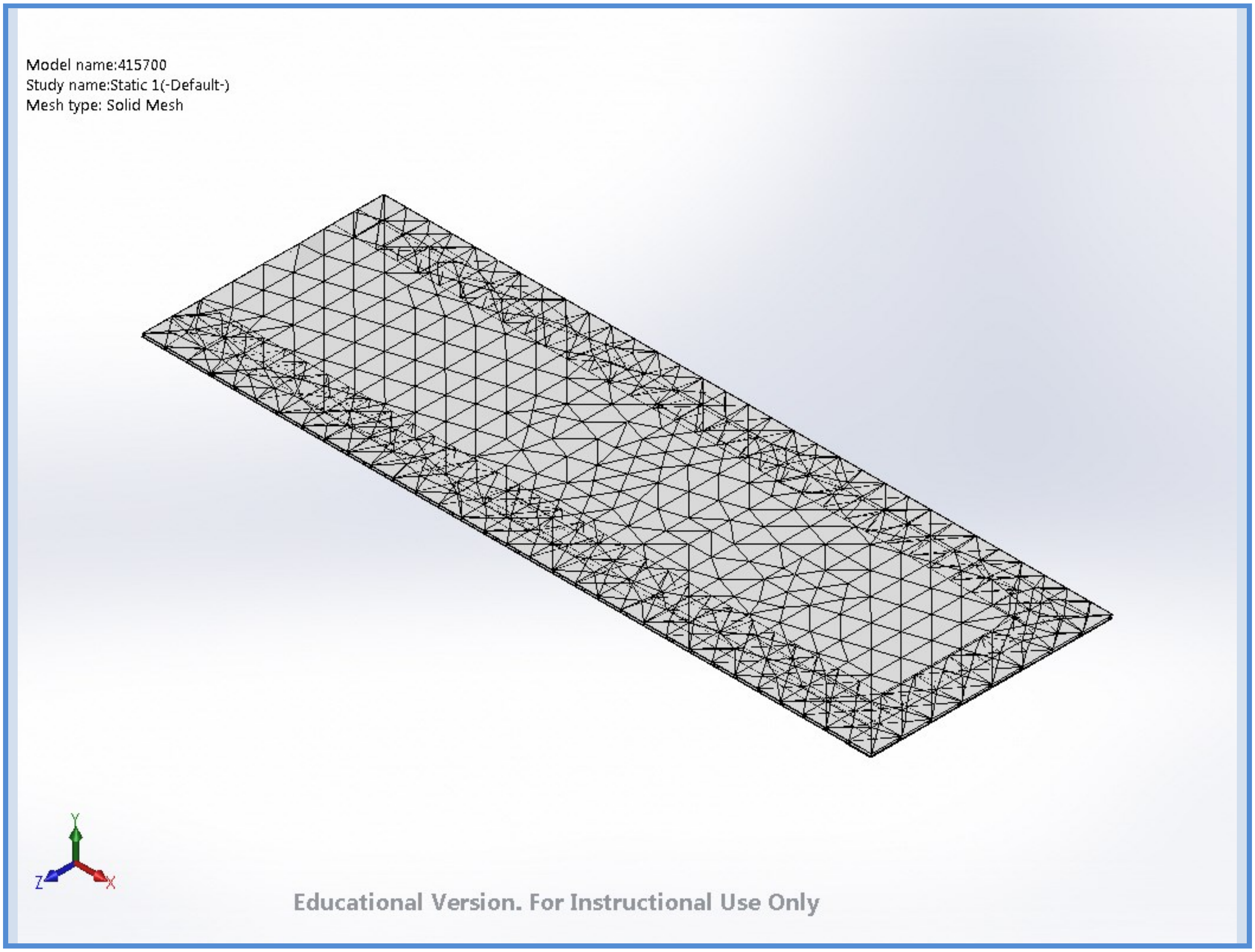
Contact Set-4		**Type:** Bonded contact pair **Entites:** 2 face(s)
Contact Set-5		**Type:** Bonded contact pair **Entites:** 2 face(s)
Global Contact		**Type:** Bonded **Components:** 1 component(s) **Options:** Compatible mesh

Mesh information

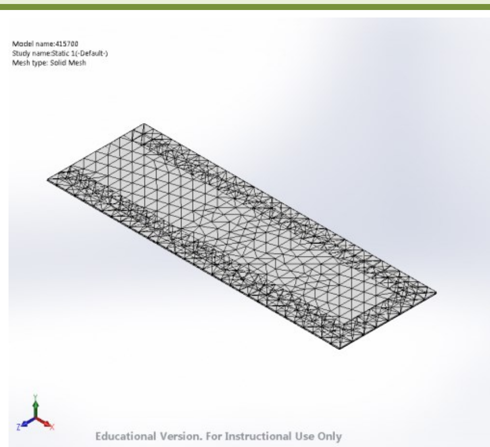
Mesh type	Solid Mesh
Mesher Used:	Curvature based mesh
Jacobian points	4 Points
Maximum element size	2.08751 mm
Minimum element size	2.08751 mm
Mesh Quality	High
Remesh failed parts with incompatible mesh	Off

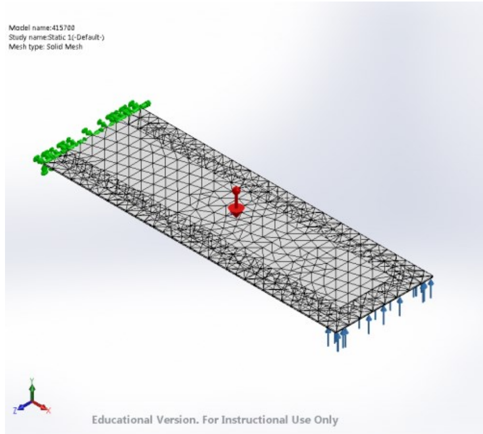
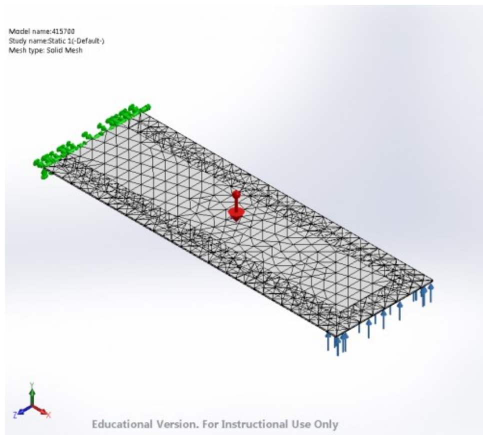
Mesh information - Details

Total Nodes	20500
Total Elements	11257
Maximum Aspect Ratio	7682.2
% of elements with Aspect Ratio < 3	0
% of elements with Aspect Ratio > 10	100
% of distorted elements(Jacobian)	0
Time to complete mesh(hh;mm;ss):	00:00:04
Computer name:	WIN7X200S



Mesh Control Information:

Mesh Control Name	Mesh Control Image	Mesh Control Details
Control-1		**Entities:** 5 Solid Body (s) **Units:** mm **Size:** 1.93095 **Ratio:** 1.5

Control-2		**Entities:** 2 Solid Body (s) **Units:** mm **Size:** 1.6961 **Ratio:** 1.5
Control-3		**Entities:** 1 Solid Body (s) **Units:** mm **Size:** 1.93095 **Ratio:** 1.5

Sensor Details

No Data

Resultant Forces

Reaction forces

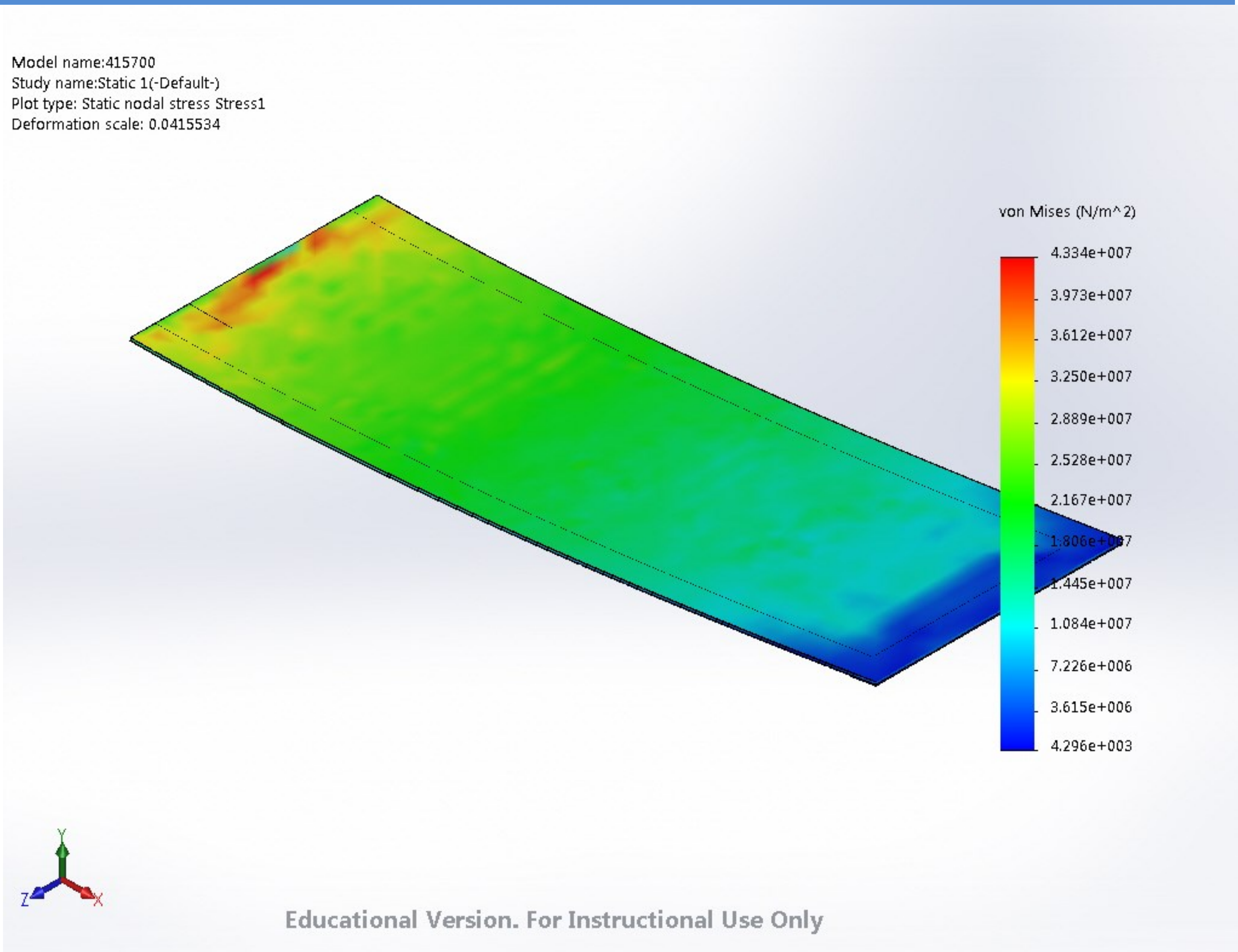
Selection set	Units	Sum X	Sum Y	Sum Z	Resultant
Entire Model	N	-7.31358e-005	-0.148445	0.000742228	0.148447

Reaction Moments

Selection set	Units	Sum X	Sum Y	Sum Z	Resultant
Entire Model	N.m	0	0	0	0

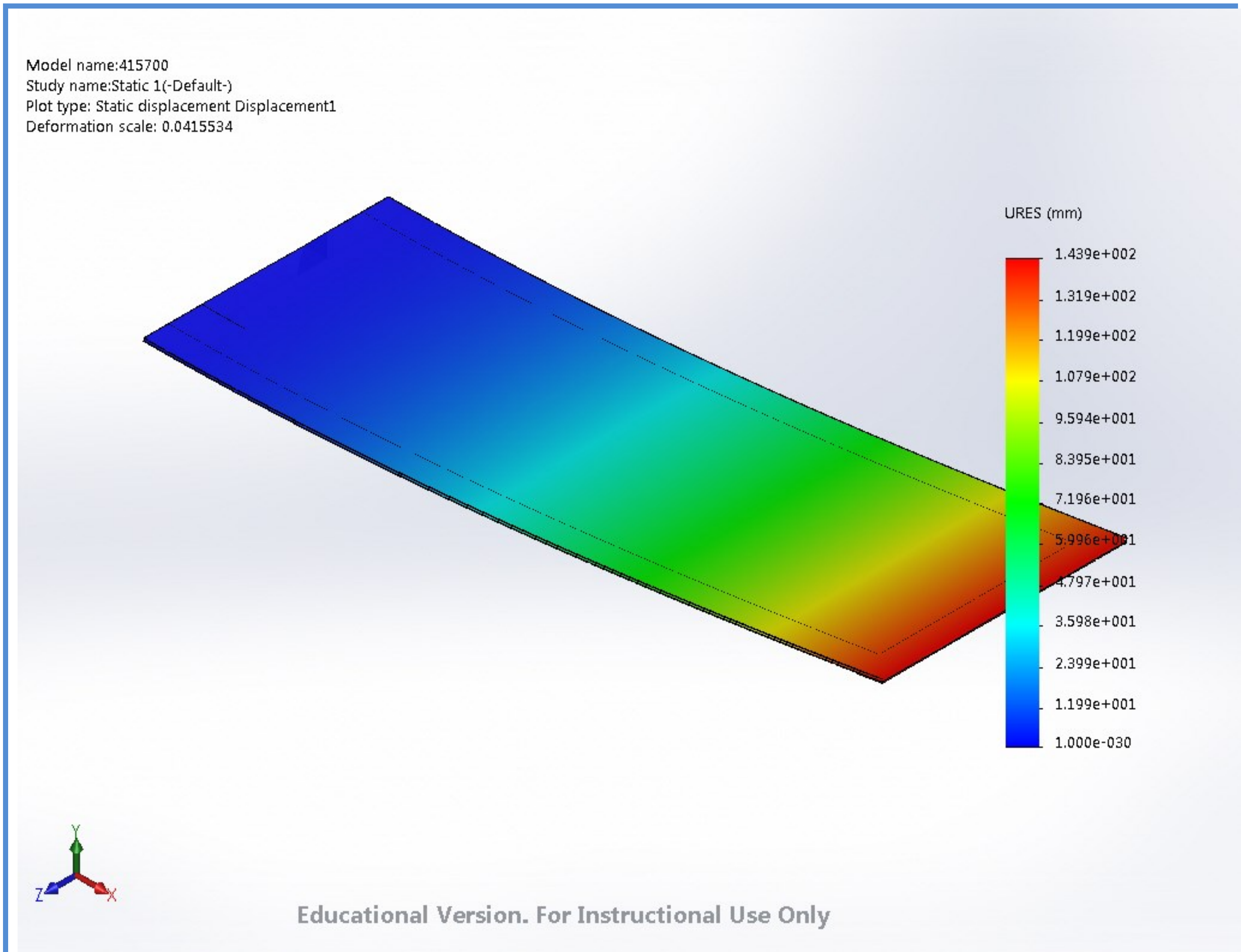
Study Results

Name	Type	Min	Max
Stress1	VON: von Mises Stress	4295.55 N/m^2 Node: 18618	4.33374e+007 N/m^2 Node: 1849



415700-Static 1-Stress-Stress1

Name	Type	Min	Max
Displacement1	URES: Resultant Displacement	0 mm Node: 3	143.912 mm Node: 17144



415700-Static 1-Displacement-Displacement1

Name	Type	Min	Max
Strain1	ESTRN: Equivalent Strain	5.71244e-005 Element: 9747	1.72311 Element: 11079