Tzu-Yu Wu
Opti 521 – Fall 2011
Introductory to opto-mechanical engineering

December 4th, 2011

Introductory Tutorial:

SolidWorks Motion and Finite Element Analysis (FEA) Simulation

1. Introduction: usage and motivation

After a design is built, there might be many questions that a designer might need to answer:

- Will the part fit into the system?
- Can the part withstand the force before breaking?
- How would it deform?
- How does the part move with certain external loads and constrains?

These "what if?" questions derive the motivation of simulation. We can bring these questions into an infinite workspace to virtually test new ideas, develop and optimize designs which would greatly reduce the expenses and time consumed from product development cycles.
FEA simulation can be applied to problems involving vibrations, heat transfer, fluid flow, and many other areas. This introductory tutorial will focus on motion and stress simulation. Two examples will be given in Motion simulation. One will be given on FEA. The first example on Motion study highlights the key capabilities of Solidworks using simple structures. This tutorial will walk through the steps for the first example. The other two examples of each topic demonstrates the simulations that I apply on my current research.

2A. Motion simulation

Example1 : 4 bar linkage

Here we will model a 4 bar linkage. The technical drawing is shown in Figure 1.1 with unit in inch. Create each part separately and assemble them as shown in Figure 1.1 by selecting "Make Assembly from part" under "file". The bottom bar numbered 3 should be selected first to make the assembly so that it is the fixed link. Each part is assigned "Alloy Steel" as material. (right-clicking "material" in the FeatureManager for each part, and selecting "Edit Material".)

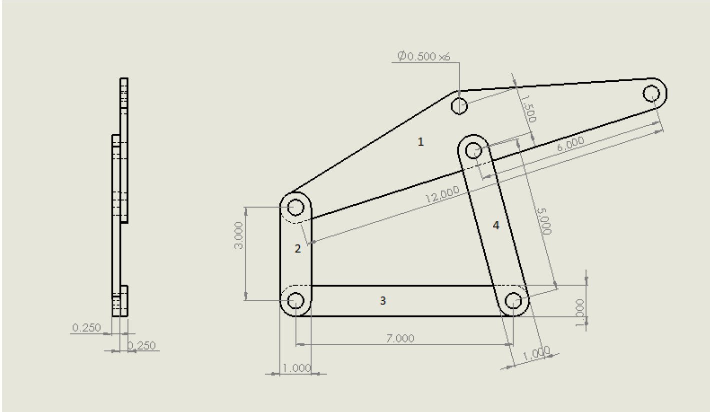
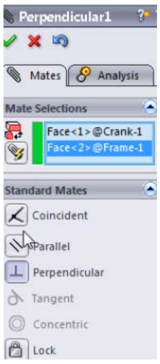
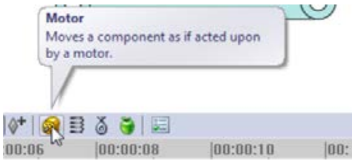


Figure 1.1 Assembly and dimensions of the parts

Before beginning the simulation, we will set the links to a precise orientation. This will allow us to compare our results to hand calculations more easily. Add a perpendicular mate between the two faces shown here. Expand the Mates group of the FeatureManager, and right-click on the perpendicular mate just added. Select Suppress.

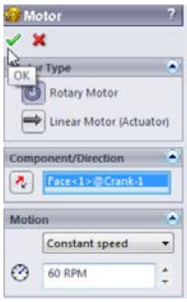




Alignment

The perpendicular mate aligns the bar 2 at a precise location. Suppressing the perpendicular mate allows us to rotation bar 2 freely and unsuppressing it would align it again.

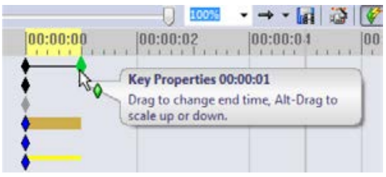
Motor



Activate Solidworks Motion in add-in. Select the Motor icon.
In the PropertyManager, set the velocity to 60 rpm.
Click on the front face of the bar 2 to apply the motor, and click the check mark.
A one-second simulation will include one full revolution of bar 2 due to 60 rpm.

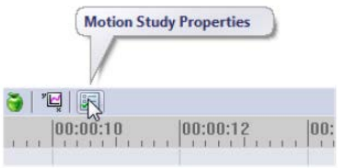
Time setting

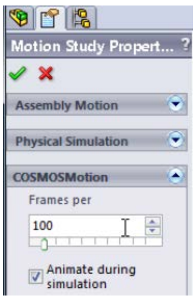
Click and drag the simulation key from the default five seconds to one second (0:00:01).



Frame rate

Click the Motion Study Properties Tool.
Set the number of frames to 100 (frames per second), and click the check mark.
This setting will produce a smooth simulation.
Press the Calculator icon to run the simulation.

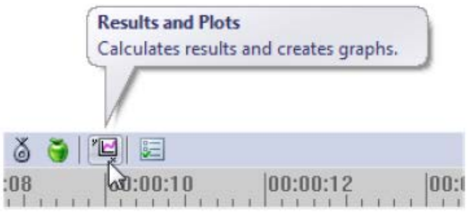


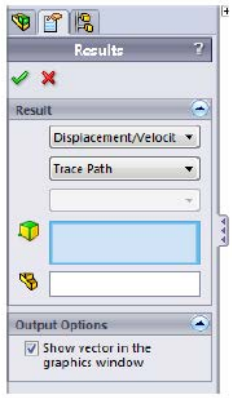


Results and plots

Displacement

Click the Results and Plots Tools. In the PropertyManager, set the type of the result to Displacement/ Velocity/Acceleration: Trace Path. Click on the edge of the top hole of the bar 4.





Play back the simulation to see the open hole's path over the full revolution of bar 2.

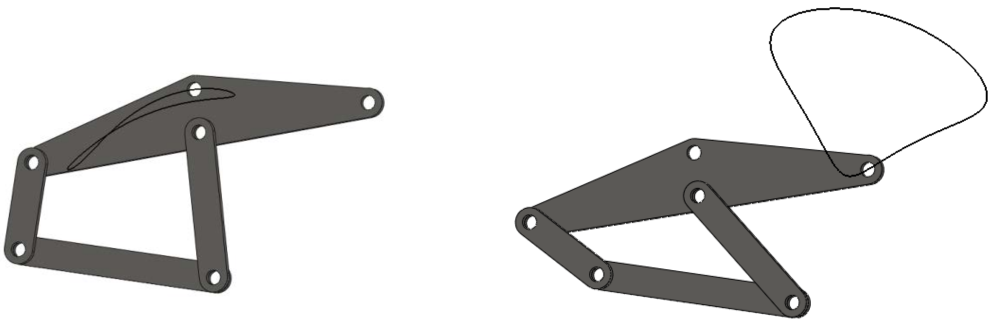
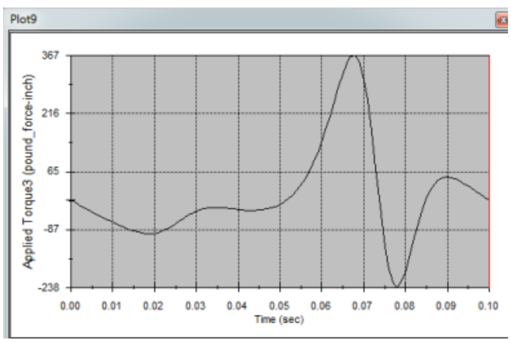
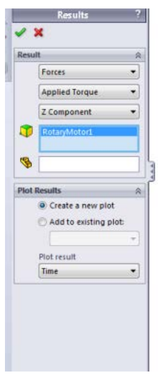


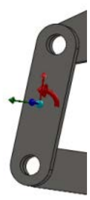
Figure 1.2 Trace path of holes of bar 4

Force and Torque

Select Results and Plots. In the PropertyManager, specify Forces: Applied Torque: Z Component. Click on the RotaryMotor in the MotionManager to select it, and click the check mark in the PropertyManager.







Exporting data

Right-click in the graph and choose Export CSV. Save the file to a convenient location, and open it in Excel.

Plot9	
Time (sec)	Applied Torque3 (pound_force-inch)
0	-9.624859835
0.001	-16.51576128
0.002	-23.17359183
0.003	-29.59304906

Summary of this example

In this example, we simulate the motion of 4 bar linkage with respective to each other.
This tutorial shows the following key figures of Motion Simulation in SolidWorks.

1. Applying motors to simulate the movement.
2. Setting the time flow and frame rate.
3. Getting the results and plots and exporting the data.

Note: The video can be found in the corresponding PowerPoint file.

2B. Motion simulation

Example: Linear and rotational motions of a fiber by a focus motor (actuator)

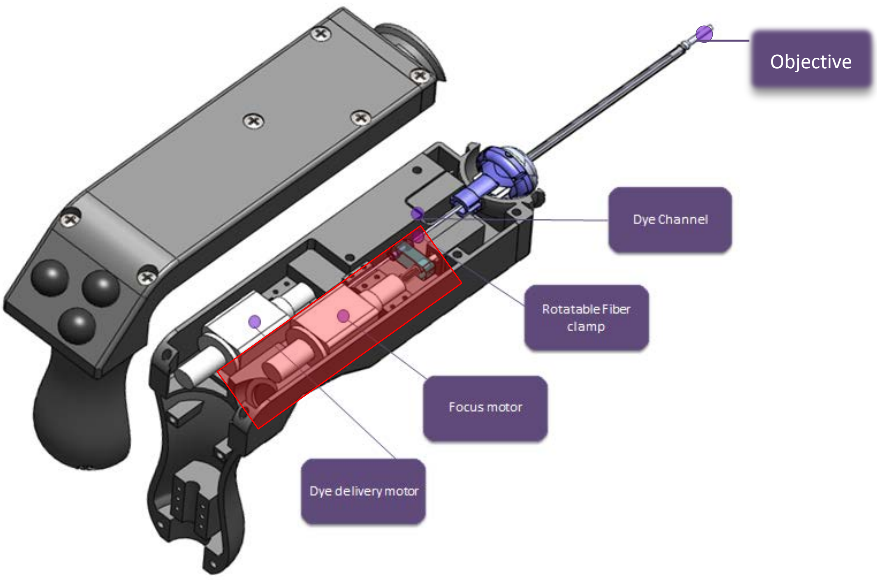


Figure 2.1 Surgical handle for confocal microendoscope. The red highlighted part is our focus of motion simulation.

Figure 2.1 shows a microendoscope handle with two linear actuators. One of the actuators serves as a focus motor, which will be our focus. The actuator serves as a focus motor since it moves the image fiber relative to an objective lens at the distal end. An imaging fiber is connected to the actuator through a rotatable fiber clamp. The rotatable fiber clamp is designed to relieve damaging twist of the fiber during surgery while still allowing axial translation via the focus motor. The details of functionality of this handle are neglected. We will only simulate the motion of the fiber._

Components

This model is composed by 4 components- fiber, dumbbell-like holder, connector between fiber and actuator and the actuator. Fiber will be fixed inside the dumbbell holder by injecting epoxy into two small holes on the dumbbell holder. Materials are assigned to each part- silica with customized Young's modulous 10.4 Msi, plastic, and stainless steel for fiber, dumbbell holder and connector, and actuator, respectively.

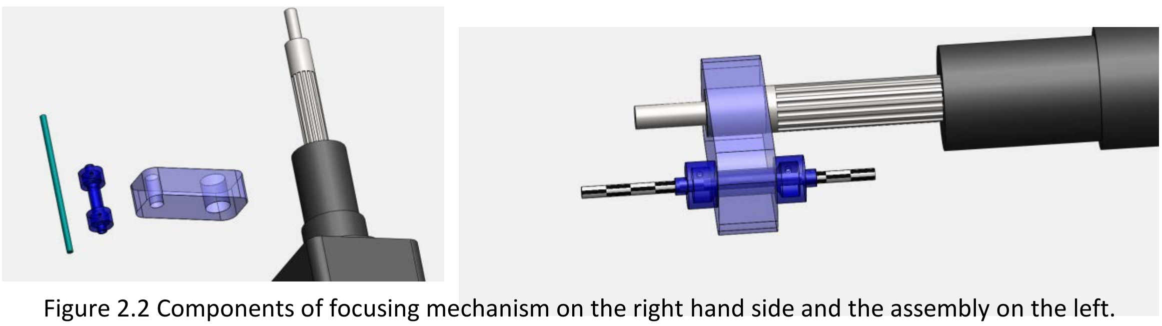


Figure 2.2 Components of focusing mechanism on the right hand side and the assembly on the left.

Simulation

Linear motor is used for simulating the sinusoidal force (4N) from the actuator from 0 sec to 2, and constant force from 4 sec to 6 sec. Damper(0.1N) is constantly applied during the whole simulation. Twisting torque on the fiber is simulated by a rotary motor with 30 RPM constant speed from 2 sed to 4 sec. Figure 2.3 shows the simulated loads and figure 2.4 shows the time flow when different conditions are applied.

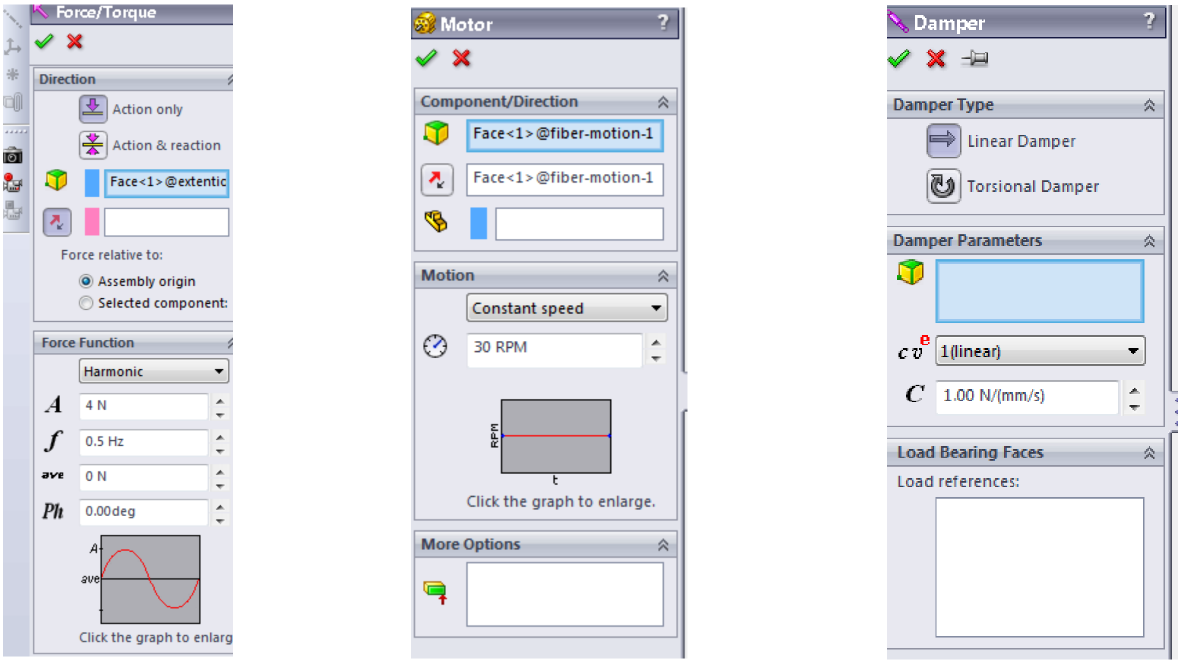


Figure2.3 Forces and torque are applied at different time points.

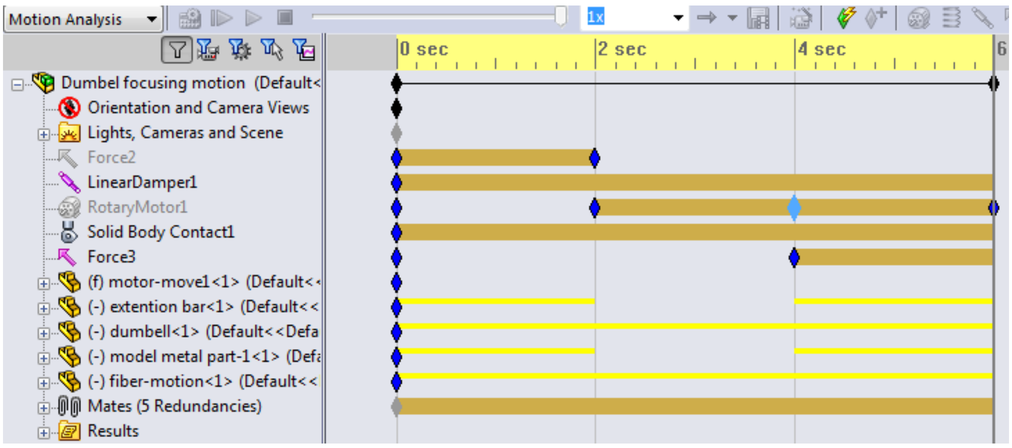


Figure 2.4 Time flow of the simulation

The results of the simulation can be plotted as a graph or exported as an Excell file. Here we plot angular and linear velocity as demonstrations. Linear velocity reflects the sinusoidal forces during the first 2 seconds and 0 force from 2seconds to 4 and constant force for the last two seconds. Negative value refers to the direction of movements.

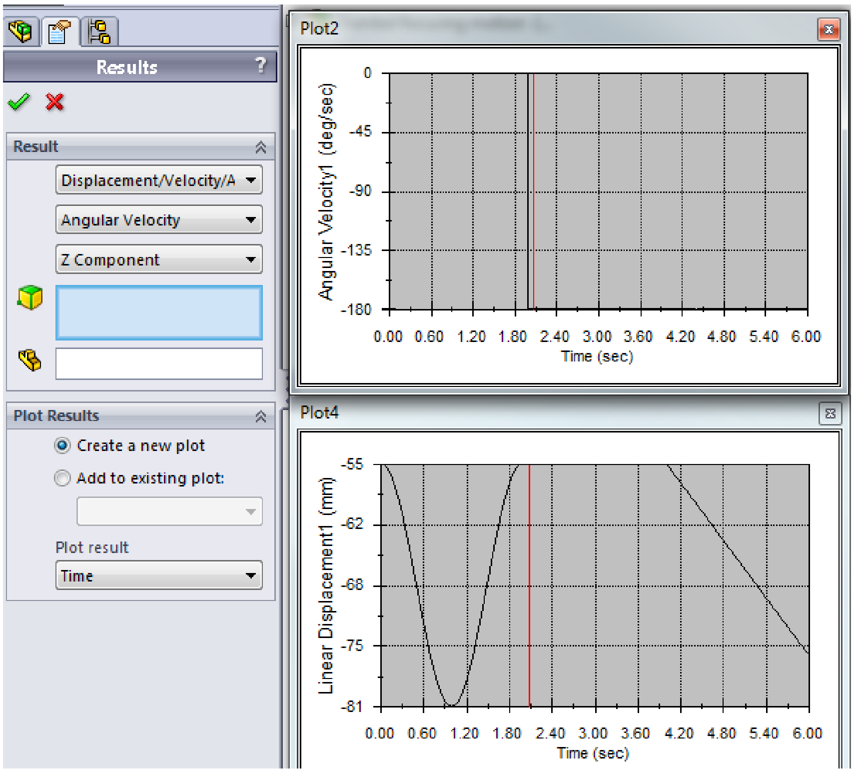


Figure 2.5 Results of the simulation.

Motion simulation is recorded as a video which can be found in the corresponding PowerPoint file.

3.FEA Simulation

Background understanding:

1. **What is FEA?**

FEA performs the stress analysis and also shows critical areas and safety levels at various regions in the faucet. Based on these results, you can strengthen unsafe regions and remove material from overdesigned ones.

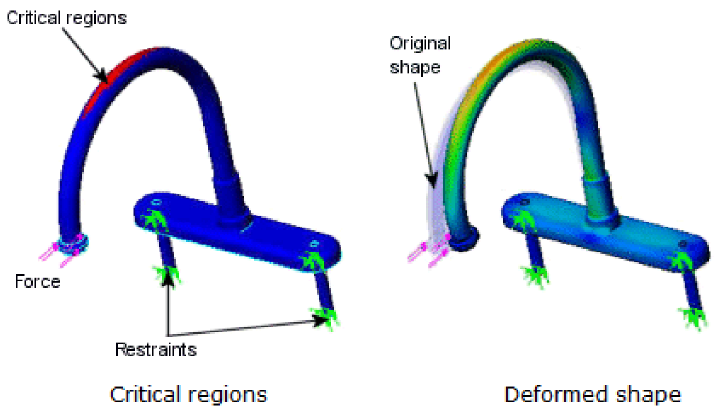


Figure 3.1 Simulation of deformation and critical regions

Parts are simulated by many small elements which share common points called nodes. The behavior of these elements is well-known under all possible support and load scenarios. The motion of each node is fully described by translations in the X, Y, and Z directions, called degrees of freedom (DOFs). Analysis using FEM is called Finite Element Analysis (FEA).

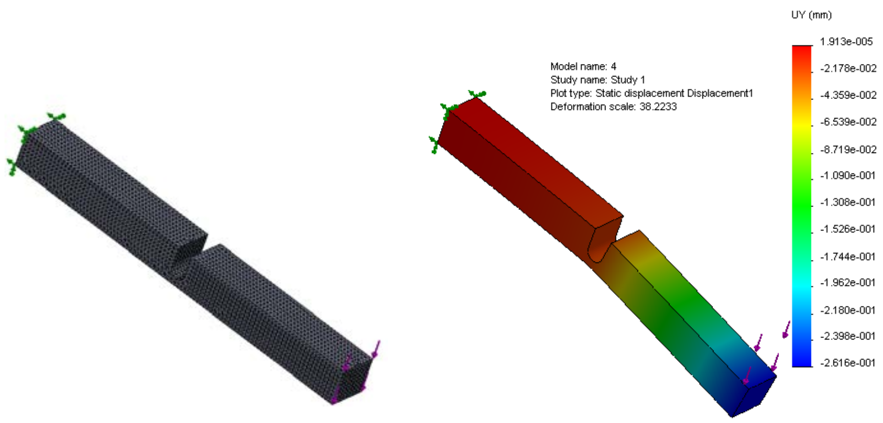


Figure3.2 Model subdivided into small pieces (elements)

2. Stress and strain

$$\sigma = \frac{F}{A} \qquad \varepsilon = \frac{\Delta L}{L} \qquad \varepsilon E = \sigma \qquad \Delta L = \frac{FL}{EA}$$

Stress Strain Young's modulus Deformation

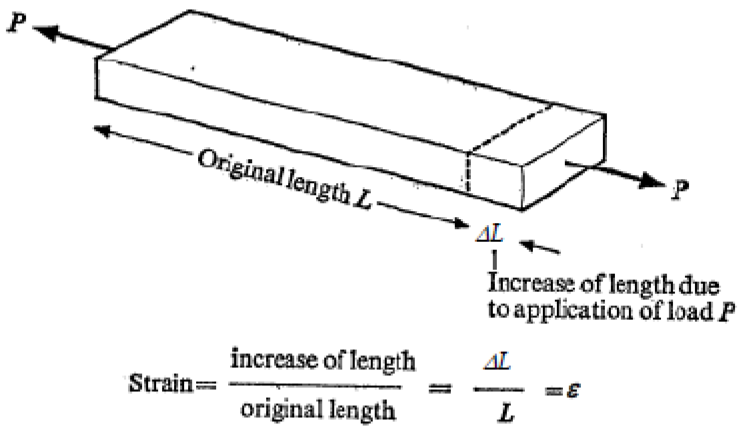
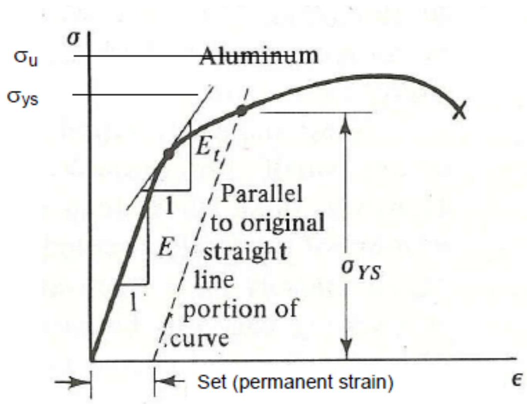


Figure 3.3 Explanation and definition of strain



E=dσ/dε for small loads and small deflections

σ_{YS} ⇒ **Yield Strength** - The maximum stress that can be applied without exceeding a specified value of permanent strain (typically 0.2% = .002 in/in).

σ_{PEL} ⇒ **Precision elastic limit** or micro-yield strength - The maximum stress that can be applied without exceeding a permanent strain of 1 ppm or 0.0001%

σ_{U} ⇒ **Ultimate Strength** - The maximum stress the material can withstand (based on the original area).

Figure 3.4 Plot of Strain v.s. stress

Example: Stress and displacement of a bending fiber

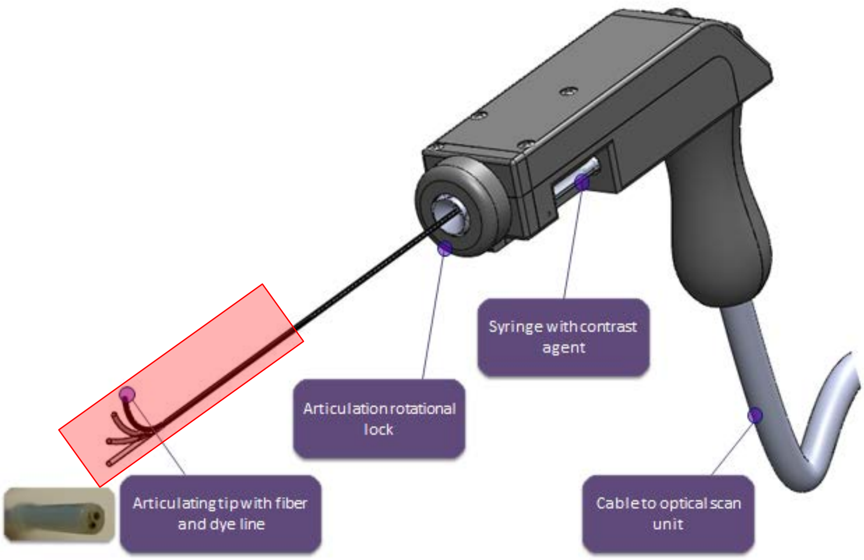


Figure 3.5 An articulating tip of a microendoscope handle with an imaging fiber and hypothermic dye line. The red highlighted region is our simulation focus.

System background and motivation of simulation

This microendoscope handle has a flexible catheter which gains more access to imaging organs such as fallopian tubes during surgeries compared to a rigid device. The articulating distal tip of the instrument consists of a 2.2 mm diameter bare fiber bundle catheter with automated dye delivery channel. This simulation is performed to understand the maximum load that the fiber can withstand before breaking and also the displacement and angles that it can reach within a safe range (safety factor about 3).

Model

Results

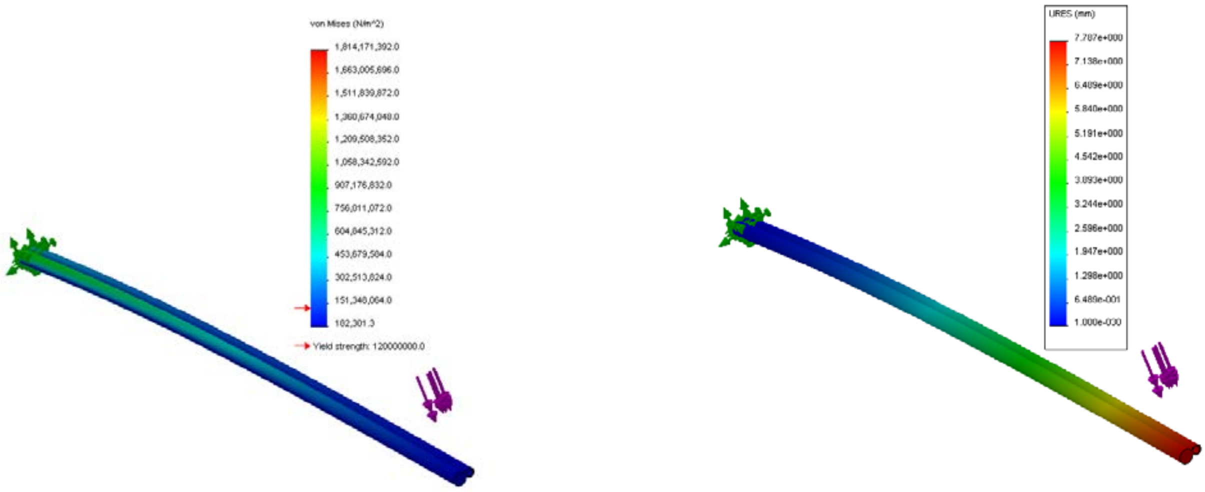


Figure3.6 Stress distribution (on the left) and displacement (on the right)

Discussion:

The fiber and the dye tubing are bonded together by applying face constraint. Notice that the stress distribution is different between fiber and dye tubing. It might be due to higher Young's modulus in the fiber with the same displacement. More cares have to be taken. In this model, many assumptions have been make. For example, fiber and dye tubing are constrained together in some degree but not fixed with each other. However, I bounded them by apply face constraint so that they won't move relatively to each other. The edge fixture isn't true in reality. Material properties of the imaging fiber are critical for the simulation.

The safety factor can be used to understand the maximum load the part can withstand according to the following equation:

Current applied force X Current safetly factor = Maximum affordable force.

Summary:

This tutorial highlights the key functions of SolidWorks Motion and FEA Simulation. The introduction of FEA theory is briefly given. Detailed tutorial on SolidWorks Motion is given by walking through an example. Two examples related to imaging fiber motion driven by a linear actuator and fiber bending are given. There has been much discussion during the past decade over who should be using FEA software. As the software has become easier to use, the potential for misuse has risen. An inexperienced user can quickly obtain results, but the interpretation of the results requires knowledge of the applicable engineering theories. In the FEA example, assumptions have been pointed out that could affect the accuracy of the results.

References:

1. Solidworks tutorial and help
2. OPTI521 Classnote from Dr Jim Burge
3. Tutorials from mcgraw-hill.com

Useful learning resources:

Complex shape
http://www.youtube.com/watch?v=3MoowmIKwYQ&feature=related
Assemble study motion
http://www.youtube.com/watch?v=f4BA86PExAQ&feature=related
http://www.youtube.com/watch?v=cq-kmLC9jkA&feature=related
Spring animation
http://www.youtube.com/watch?v=OYlJT7rhq_g&feature=related
Motion simulation with forces
http://www.youtube.com/watch?v=rMeyaO1Kqe0
Orally explain rotation movement
http://www.youtube.com/watch?v=efATpeRyLlc&feature=related
Gear motion animation
http://www.youtube.com/watch?v=EAGOnKQfJVA&feature=related
Motion study -ball move along a path
http://www.youtube.com/watch?v=ICwlmlwwCqQ&feature=related
Collision detection
http://www.youtube.com/watch?v=I_xFWKi0xGQ&feature=related