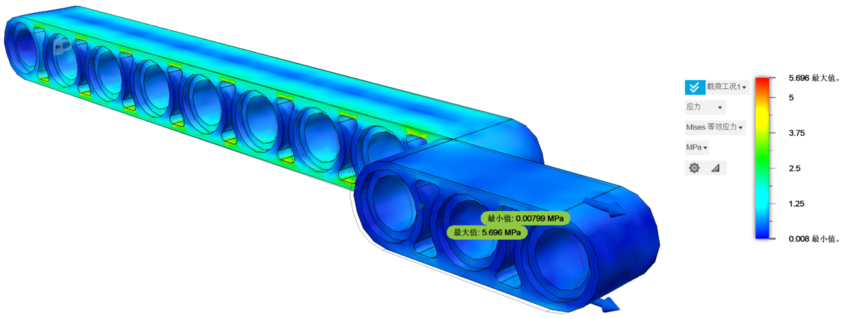
Lab 4 Report ME 371

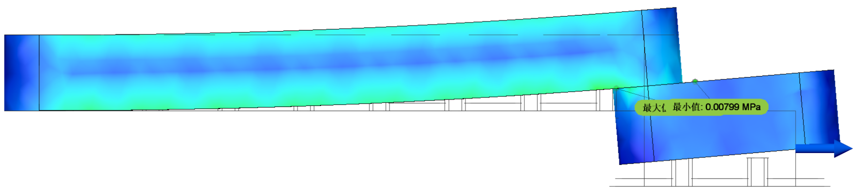
Jiajun Hu
November 8, 2023

Question 1 (Part 1)

Submit a screen shot that shows the von Mises stress distribution on the deformed assembly from the analysis conducted in Part 1. In addition, answer the following questions:

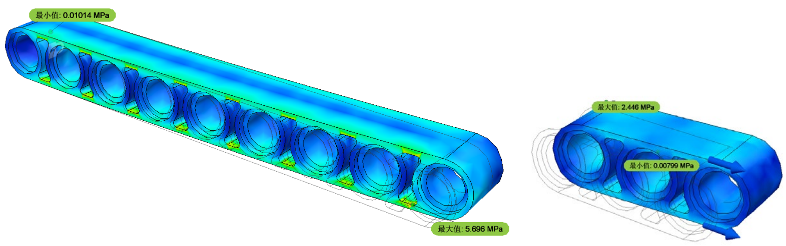
a) ***What appears to be the primary mode of deformation of the assembly (axial stretching/compression, torsion, bending)? Be sure to plot the "Adjusted" deformation of the assembly to help answer this question.***
b) ***What is the maximum von Mises stress in each component? Use the "Tip" box on Page 4 of the tutorial to toggle the visibility of each part on/off in order to clearly identify the maximum von Mises stress in each part (no need to include additional screen shots for this step).***
c) ***Does the maximum von Mises stress occur at a location that you anticipated? Why or why not?***





a) The primary deformation is banding.

b) Max von-Mises stress in longer beam is 5.696 Mpa.
Max von-Mises stress in shorter beam is 2.446 Mpa.



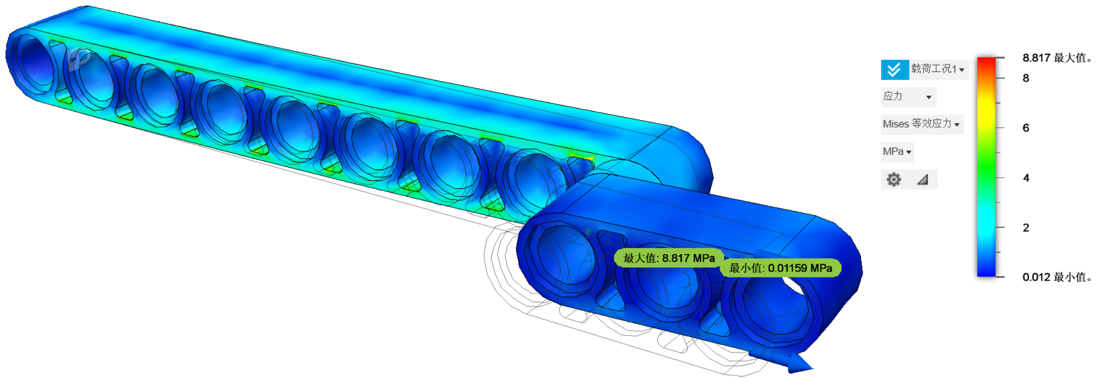
c) Kind of. Not really. The maximum stress appears on the beam connection face on the longer beam.

Question 2 (Part 2)

Submit a screen shot that shows the von Mises stress distribution on the deformed assemblyfor the analysis conducted in Part 2. In addition, answer the following questions:

a) What is the maximum von Mises stress in the assembly? In which part does it occur?

b) How does the maximum von Mises stress compare to the model in Part 1? Offer an explanation for why the maximum value has changed in this assembly despite the magnitude of the applied load being the same



a) The maximum stress is 8.817 Mpa, which appears at the same place like case 1 (the beam connection face on the longer beam)

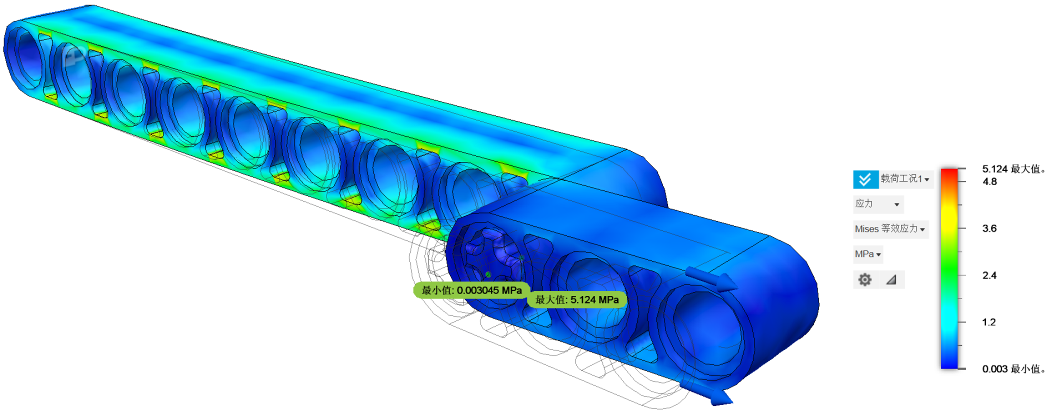
b) $\sigma_{Max,part\ 1} < \sigma_{Max,part\ 2}$. I believe it is because the addition of a connector increases the moment, therefore, increases the stress.

Question 3

Submit a screen shot that shows the von Mises stress distribution on the deformed assembly from Part 3. In addition, answer the following questions:

a) What is the maximum von Mises stress in the assembly? In which part does it occur?

b) How does the location of the maximum von Mises stress change in this model compared to the models in Parts 1 and 2? Does the change in location coincide with your expectations? Why or why not?



a) The maximum stress is 5.124 Mpa on the rightest hole of longer beam.

b) The location is the same as the previous two cases.

Question 4

FEA Labs 3 and 4 represent two different modeling approaches:

1) ***Isolate a single component of interest and apply loads that simulate not only the external loads that act on it but also its connections to other components or external supports, e.g. the "Bearing Load" used in FEA Lab 2 to simulate connection to a bearing which allows rotation (but not translation) at the connecting joint location.***
2) ***Simulate an entire assembly (two or more parts) subjected to external loads, where the transfer of loads between components depends on how the connection***

List at least two specific situations or reasons that would favor both modeling Approach 1 and Approach 2. Consider that Approach 2 is in general more complex to set up and more costly (in terms of computer resources required to solve the model) compared to analyzing a single component as in Approach 1, so modeling an entire assembly should be done only when there is a clear reason to do so.

Approach 1: Isolating a Single Component

1. **Component Stress Analysis:** When you need to perform a detailed stress analysis on a specific component to understand how it behaves under different loads. For example, if you're designing a critical part of a machine and want to assess the stress distribution, deformation, and safety factors of that component.

2. **Validation of a Connection:** When you want to validate the design and performance of a specific connection or interface, like a bolted joint or a welded area. This approach is useful to ensure that the connection can withstand the expected loads and constraints.

Approach 2: Simulating an Entire Assembly

1. **Interaction and Assembly Analysis:** When you need to study how different components interact within an assembly and how loads are transferred between them. This is essential when you want to ensure that the entire product or system functions as intended and that no part fails due to the interconnected forces.

2. **System-Level Optimization:** When you are interested in optimizing the entire system's performance rather than just one component. For instance, if you're designing an automotive suspension system, analyzing the entire assembly allows you to optimize the performance, weight, and cost by considering the interactions between various components.