Arjun V. Narayanan and Lucas Smyk

6.S079 Midpoint Project Writeup

Project Goal

We would like to do an FEM simulation of an object before we 3D print it. First, we will build an FEM simulation for a generic elastic material. Then, we will measure the material properties of the Makerbot plastic and integrate them into our code. Finally, we will see how close our simulation matches the actual behavior of the object after it is printed.

Summary of Technical Approach

We build our FEM simulation using hexahedral elements (cubes). Desai Chen (Arjun's UROP advisor) provided us with some notes containing formulas for these elements. However, we rederived all the results ourselves and did all of the math (we didn't just use the formulas).

For Assignment 4, we did FEM by first calculating the deformation gradient and strain, then converting strain to stress, and finally computing force. In our project, we use an indirect formulation where we first express a total potential energy function and then try to minimize it. The two approaches are equivalent.

For the triangular and tetrahedral elements discussed in class, the deformation gradient is constant across the element. This is not true for the higher order cube element. Thus, we use the 3D 2-point Gaussian Quadrature Rule. We calculate the deformation gradient at each of the eight quadrature points. Then, we use the material model to convert the deformation gradient to strain energy. As of now, we are using the Neohookean material model with parameters mu = 50 and lambda = 50.

Once we calculate the total potential energy, we need to minimize it. For this, we try both gradient descent and Newton's method.

Because this is only the midpoint, we do not include the detailed math and derivations in this report. We will go through all of this in detail in the final report (the midpoint presentation we gave in class does have some of the math though, if you would like to see it).

Results to Date

Currently, we have finished a first draft of the generic FEM simulation. We have implemented all of the physics and rendering code. For the energy minimization, we've coded both gradient descent and Newton's method. However, for Newton's method, we still need to put in a fast sparse matrix solver into our code. We expect to use the GPU assisted solver cusp.

Despite all this, our simulation is not yet fully working. When we only have one element, everything looks good. The cube deforms correctly when we apply external forces. However, with multiple elements, the mesh collapses in on itself and gets tangled up. We suspect that the code is not handling translations properly (translations are inducing strain when they shouldn't).

Remaining Issues to be Solved

We first need to fix the simulation with multiple elements. Next, we need to add the cusp sparse matrix solver and switch our code to use Newton's method. This will speed up the application and allow us to simulate larger meshes. Third, we need to measure the properties of the Makerbot material and integrate them into our code. Finally, we need to test how closely our simulation matches reality.