Summer of Science Topic: Topology Optimization using Finite Element Methods

Plan of Action

	Planned Activity		Completed Activity
(light gray)		(black)	

Work Activity by weeks	May		June				July	
	3rd	4th	1st	2nd	3rd	4th	1st	2nd
Plan of Action Submission	Planned							
	Completed							
Literature Review	Planned	Planned	Planned					
	Completed							
Weekly meet with mentor for progress tracking (if mentor is available)	Planned	Planned	Planned	Planned	Planned	Planned		
Hands on with Fenics software for FEM computation	Planned	Planned	Planned					
Selection of a particular topology optimization method with selection of suitable application		Planned	Planned					
Development Pseudo-codes showcasing the algorithm implementation			Planned	Planned				
Midterm report preparation and expected submission				Planned				
Final coding and validating the results using ANSYS/ABACUS					Planned	Planned	Planned	
SoS report submission and video submission							Planned	Planned

Description of goals:

1. Literature Review: Before diving down into specific applications, it will be good to have a birds eye view over the work done on the selected topic. Tentative 3 weeks will be sent in gathering the references and studying them

2. Hands on with Fenics software for FEM computation: Fenics and ABACUS are two important tools for Finite Element Methods execution. Even the FEM techniques can be coded in MATLAB. However using the Fenics package in python, FEM can be made a lot easier to apply

3. Selection of a particular topology optimization method with selection of suitable application:
   To select a particular application such as Wind turbine blade, or any other structures, a thorough literature survey will be required, which will be completed within 2-3 weeks

4. Development Pseudo-codes showcasing the algorithm implementation: For the mid-term submissions, a tentative plan will be to showcase some pseudo-algorithms OR if possible some sample codes to explain the method/approach selected.

5. Final coding and validating the results using ANSYS/ABACUS:
   After the midterm submission, effort will be made on completion of the codes. However o validate the results obtained from these codes, we need to check pre-existing FEM tools like Abacus/ANSYS for correctness of the approach