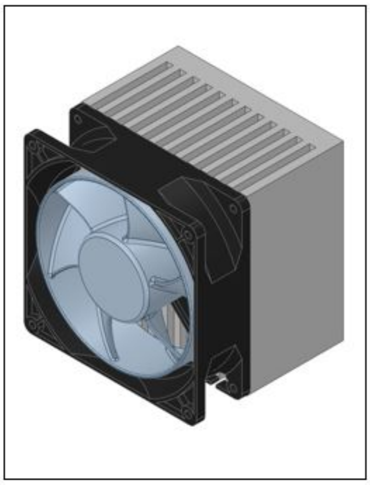
MAE 573 Applications of FEA
Final Project
Thermo-mechanical Analysis of a Computer CPU Cooling System
PowerPoint Presentation Due: 12/16/2021

For this project, you will be be performing thermo-mechanical analysis of a computer CPU cooling system using Fine Element Analysis (FEA). The CAD geometry for the system will be provided to you as .stp file (CPU-Cooling.stp and fan-sector.stp can be downloaded from the backboard).

The geometry of the components used in the system and the material for each component is shown in Fig.1. As it can be seen from the figure, heat sink is made of aluminum, while the fan and the fan-casing are made of glass-filled nylon. The motor that drives the fan is assumed to be integral part of the fan and it is assumed to have same material as the fan. Material properties for Aluminum are presented in Table-1, while material proprieties for glass-filled nylon are shown in Table-2.



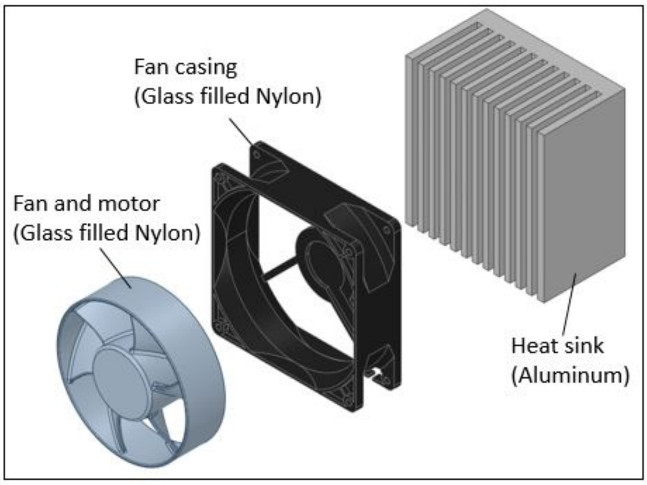


Figure 1: Components and the material of each component for the CPU cooling system

Table-1: Material properties for aluminum	
Density (kg/m^3)	2700
Module of Elasticity (Mpa)	69800
Poisson's ratio	0.33
Coefficient of thermal expansion (1/°C)	23.6E-6
Heat conduction coefficient (W/(m.K)	167
Yield strength (Mpa)	276

Table-2: Material properties for glass-filled nylon	
Density (kg/m^3)	1360
Module of Elasticity (Mpa)	6200
Poisson's ratio	0.35
Coefficient of thermal expansion (1/°C)	25.6E-6
Heat conduction coefficient (W/(m.K)	0.23
Yield strength (Mpa)	55

Part 1: Thermo-structural Analysis of the Whole System

For the first part of the project, you are asked to perform thermo-mechanical simulation of the CPU cooling system, and analyze the thermal and structural response of each component to the heat and to the stresses.

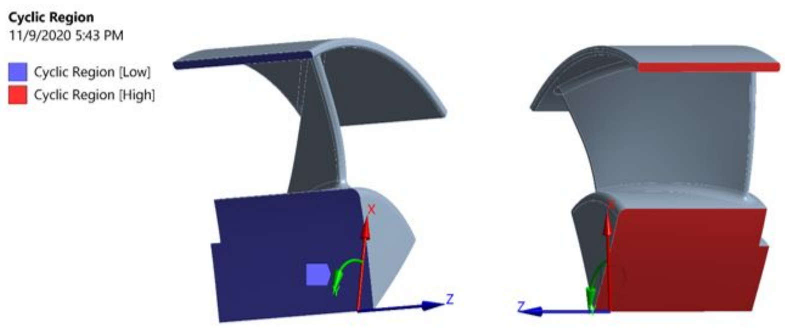
1. Study the steady-state thermal response of the heat sink, the fan and the casing for the case where the heat flux through the base is $\dot{q} = 2000\ W/m^2$. Recall that the fan forces air over all surfaces except for the base of the heat sink, where a heat flux $\dot{q}$ is prescribed. The air forced by the fan over the heat sink is at 28 °C with a heat transfer coefficient of $h = 30W/(m^2°\text{C})$.
   - Plot the contours of the temperature and the heat flux for the whole system
   - Plot the contours of the temperature and the heat flux for each component separately.
   - Do you see a quick dissipation of heat (reduction in heat transfer) between the aluminum heat-sink and the plastic casing/fan? If yes, explain the reasons for this heat dissipation.
2. Suppose the base of the heat sink is fixed, and the fan rotates counterclockwise with an angular velocity of 5000 RPM. Study the thermo-structural response of the components of the CPU system to the the temperature load and the load from rotational velocity of the fan.
   - Plot the Von-Mises stress and the total deformation contours for the whole system
   - For each component plot the Von-Mises, the total deformation, and the safety factor contours. Which component has the highest stress level? Based on your analysis, comment on whether you think there is any risk of failure for this CPU cooling system during the operation.
   - For fan component, also plot the the redial deformation (you will need to generate a cylindrical coordinate system to get the radial deformation). Is the radial deformation (the growth in the diameter of the fan due to loads) is large enough to close the gap between the fan-tip and the casing, and cause the fan tip to hit the casing?

Part 2: Structural Analysis of the Fan Component Only

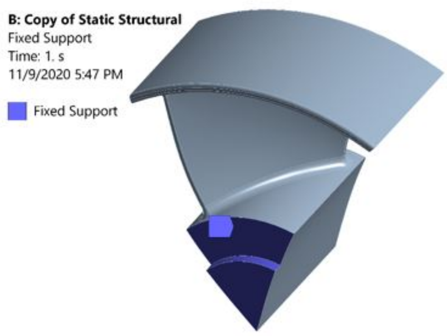
For the second part of the project, you will study the structural and modal analysis of the fan component only (independent of the other components). Since the fan is composed of 7 identical periodic blades, you will work only with one blade (instead of the full fan). By setting up the right boundary conditions, you will let the Ansys software know that you are modeling 1/7 th sector of a fan, and once the simulation is done, Ansys will show you the solution for the complete fan (Although you are only simulating 1/7 th of the fan). Simulating 1/7 th sector of the fan will save you time on the simulation, and you will also be able to build a better quality mesh.

Steps to follow for this part of the project are as following:

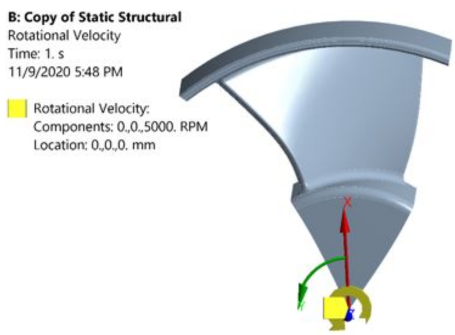
1. Download 1/7 th sector of the fan provided to you as stp file form the blackboard (fan-sector.stp). Run static structural analysis of the model using the boundary conditions shown in Fig.2. Don't forget to use the right material (glass filled nylon) for the analysis. Plot the contours of Von-Mises stress, total deformation, radial deformation and safety factor. Compare your results with the results you obtained in part-1. Do you think the temperature load that was present in Part-1 (and does not exist for this simulation) has any effect on the deformation, the stress level, and the safety factor of the fan?



(a) Defining cyclic region-Periodic boundary conditions



(b) Fixed support boundary conditions



(c) Rotational velocity boundary conditions

Figure 2: The periodic (cyclic) sector of the fan and the boundary conditions

2. Run a parametric model to find the change in total deformation, the stress level (von-Mises), and the safety factor with changing rotational velocity of the fan.

   - Set the rotational velocity of the fan as the input parameter, and the total deformation, the von-Misses stress, and the safety factor as the output parameters. Set table of design points with a range of rotational velocities (You will decide how many design points to run, and which RPM range to select.) Run the design points you select. Generate a table showing your design points and all the results. Include the table in your report.
   - Plot the total deformation and the von-Mises stress as a function of the fan rotational velocity (RPM). What kind of relation do you observe

(Linear, quadratic, etc??). Explain the reason for the relation you see between RPM and deformation/stress using liner elasticity theory.

- if we want to have a safety factor of at least 1.5 everywhere on the fan, what is the maximum rotational velocity that the fan can be run at? What are the total deformation, the radial, and the axial deformation of the fan at that rotational velocity. Do you see any risk of fan hitting the casing if it runs at that rotational velocity?

Part 3: Modal Analysis of the Fan Component Only

You will finally perform modal analysis on the fan component only. Unlike the structural analysis, modal analysis can not be performed on the 1/7 th sector, and you will need to perform modal analysis on the full fan (without the casing and the heat sink)

1. Perform modal analysis on this fan. Include the first 6 natural frequencies, and the first 6 mode shapes from the analysis in to your report.

2. Assume that the natural frequency of the first mode is too high, and we would like to reduce that in half. What would you recommend to change (without changing the design) so we can cut the natural frequency in half?

Grading

You will present your findings in a presentation. You presentation is expected to have the quality of a technical publication. It should include all the important details without being overly lengthy. It is important that you use contour plots that are relevant, clear and readable. You have until 12/08/2020 to submit your report.

Your work will be scored as following:

Setting up material properties correctly: 10 points

Clear demonstration of the boundary conditions: 10 points

Mesh quality and quantity: : 15 points (Remember: your mesh is expected to be fine enough to yield accurate results, but not too fine that it will take you weeks to complete the simulation)

Clear presentation of all the contour plots asked in this document, including correct and detailed answers to the questions asked : 35 points

Quality of the report: 30 points