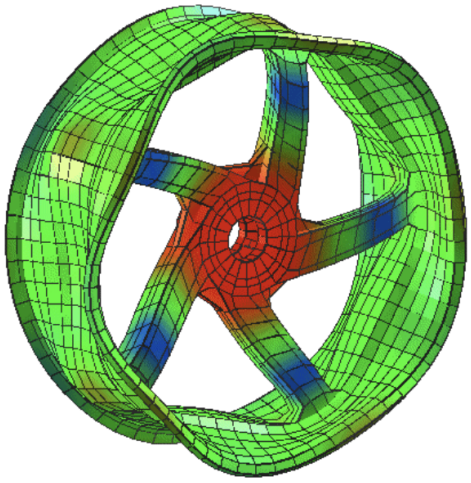
Mini-project 3

Modal and dynamic analysis of a mechanical component or structure

---

Project organization:

- Individual project
- 20% of final grade
- Pdf report: maximum 15 pages
- Programming language: `MATLAB`, `Abaqus`, `Ansys`
- Submission: January 9, 2026
- Total workload: 15h-20h



1. (Re)familiarize yourself with a well-known commercial finite element code such as Abaqus, Ansys or MATLAB PDE Toolbox.
2. Choose one single mechanical component or structure requiring modal and time-domain dynamic analysis, describe the selected example, and justify the relevance of this choice. A connection to a current semester or master project is allowed.
3. Model the chosen mechanical component or structure using solid and/or shell and/or beam finite elements to analyze its dynamic behavior, and discuss the mesh choice (type of finite elements, mesh density, convergence study, etc.).
4. Determine the first five natural frequencies and mode shapes of the mechanical component or structure, and comment on the observed modal behavior.
5. Choose a plausible excitation for the mechanical component or structure, describe and justify the chosen numerical method for time-domain solution, compute the time response to this excitation, and discuss the obtained results.

Deliverables

1. `MATLAB`, `Abaqus`, `Ansys` files including:

   - Mesh definition
   - Material and geometry properties
   - Boundary conditions and loads
   - Modal and dynamic analysis

2. **PDF report:**

   a. Briefly describe the chosen mechanical component or structure, the motivation for its selection, and its relevance to a semester or master project if applicable.

   b. Explain the finite element modeling strategy, including:
      - Type of finite elements used (solid, shell, beam, etc.)
      - Mesh design, density, and any convergence studies performed
      - Material properties and geometric details
      - Boundary conditions and applied loads

   c. Compute the first five natural frequencies and corresponding mode shapes. Comment on the observed modal behavior and its physical significance.

   d. Describe the chosen excitation (e.g., force, displacement, or base motion), justify the selected time-domain solution method, and present the computed response over time. Discuss the results in terms of amplitude, resonance, and practical implications.

   e. Critically analyze the results, including potential sources of error, sensitivity to mesh refinement, and limitations of the model or numerical methods. Summarize the main findings of the modal and dynamic analysis and propose possible extensions or improvements for future studies.

3. **Project defense:** prepare a PowerPoint, or similar, presentation of your project during the oral exam on January 14 (20 minutes presentation followed by a 10 minutes discussion). The presentation should:

   - Summarize the project objectives, modeling approach, and chosen component/structure
   - Highlight key results from the modal and dynamic analysis
   - Include figures of mesh, mode shapes, and dynamic response
   - Discuss conclusions and possible improvements