

BILAL ABDUL HALIM

phone: (xxx) · xxx · xxxx * email: bilal.utoledo@gmail.com

SUMMARY

Mechanical Engineer with a Master's degree, specializing in Finite Element Analysis using Abaqus and Python scripting. Experienced in structural mechanics, modeling contact, and customer support. Excellent communication skills with a passion for innovative product design, seeking to contribute to a dynamic product structural analysis team in a fast-paced environment.

EDUCATION

University of Toledo, Toledo, Ohio *August 2020 – May 2022*
Master's of Science in Mechanical Engineering
Concentration in Fluid Mechanics
Overall GPA: 4.0

University of Toledo, Toledo, Ohio *August 2015 – May 2020*
Bachelor's of Science in Mechanical Engineering
Concentration in Applied Mathematics
Overall GPA: 3.9

EXPERIENCE

GoEngineer *May 2022 – Present*
Advanced Simulation Specialist *Ann Arbor, Michigan*

- Expertly utilize Abaqus for finite element analysis, focusing on nonlinear material behavior, contact mechanics, and metallic material behavior.
- Conduct static and dynamic structural FEA analysis using Abaqus/Standard and Abaqus/Explicit.
- Provide insightful customer support by diagnosing and resolving model setup and convergence issues, ensuring optimal mesh quality and proper load application in Abaqus.
- Design and deliver engaging, in-depth Abaqus training sessions tailored to the client's applications, encompassing fundamentals to advanced simulation techniques.

University of Toledo *May 2020 – May 2022*
Graduate Researcher *Toledo, Ohio*

- Utilized Abaqus to create finite element models of the ice adhesion interface with aluminum, focusing on studying the stress development at the interface.
- Conducted simulations to measure the minimum force required to remove ice and investigated the effects of different surface topologies on ice adhesion.
- Employed Particle Image Velocimetry to analyze fluid flow and extracted dominant frequencies using Python.
- Automated data extraction and streamlined report generation with Python and \LaTeX .

Plastic Technologies Inc *January 2019 – August 2019*
Finite Element Analysis Engineer *Holland, Ohio*

- Developed and optimized finite element models in Abaqus to run structural, thermal (uncoupled and coupled), and buckling (pre and post) analysis.
- Collaborated with design teams, providing data-driven FEA recommendations for design optimization and material selection.
- Conducted material characterization, including uniaxial and equibiaxial testing of HDPE, to develop a hyperelastic material model for simulations.

TECHNICAL SKILLS

FEA & CAD	Abaqus, Isight, XFlow, SOLIDWORKS Simulation, SOLIDWORKS
Software & OS	Microsoft Office Suite, Linux, Vim
Computer Languages	Python, MATLAB, C, AWK, Bash, \LaTeX