

CONTACT

Bursa, Türkiye

alihanalayoglu@gmail.com

linkedin.com/in/alihan-alayoğlu

PROFILE

Durability CAE engineer with 5 years of FEA experience, passionate about solid mechanics, fatigue and workflow automation through superintelligence.

Durability CAE

ABAQUS · ANSA · nCode · HyperMesh · HyperWorks · Nastran

CAE automation

CAE tools built with Claude Code.

Claude Code

Python · Tcl/Tk · Bash · Fortran

Instructor

TOFAŞ Virtual Analyses course; mentor within the team and to other teams; FEA handbook and FEA document archive for the TOFAŞ CAE Academy.

INTERESTS

Bağlama & guitar

Mixed martial arts (MMA)

Calisthenics

Swimming

Mountain camping

Muhammet Alihan Alayoğlu

CAE Technical Specialist

2017–22 B.Sc. Mech. Eng. · 2018 Intern · 2021 CAD & CAE Lead · 2021–22 CAE Intern · 2022 CAE Engineer · 2022–23 DSM Engineer · 2023 CAE & Test Engineer · 2023–25 DSM Engineer · 2025– CAE Technical Specialist

EXPERIENCE

TOFAŞ
Sep 2023 – Present

Stellantis JV · Bursa

CAE Technical Specialist – Moving Parts Durability
Sep 2025 – Present

CAE Digitalization Lead (concurrent role) Apr 2024 – Dec 2025

Virtual Analyses Engineer – Durability Sep 2023 – Sep 2025

- **Modeling:** hex/quad mapped meshes of plastic, rubber and metal parts (HyperMesh, ANSA); ply/zone-based composites for Nastran and Abaqus; spot/seam weld, adhesive and bushing connectors; Nastran, Abaqus, LS-DYNA and Radioss model conversion; Tcl/Tk automation.
- **Analyses: Nastran** linear static, inertia relief, modal and modal transient; **Abaqus/Standard** nonlinear static, quasi-static, implicit dynamic, thermo-mechanical and buckling; **Abaqus/Explicit** slam, overslam, overcheck and misuse; contact convergence fixes.
- **Fatigue:** HCF/LCF of spot welds, plastics and metals in FALANCS (static and dynamic); closure fatigue in Virtual.Lab and nCode.
- **Design support:** orphan-mesh alternatives that raise low eigenvalues and cut stress and deformation; post-processing in META, HyperView, HyperGraph and ODB Viewer.
- **From vans to super sports race cars:** Maserati · Alfa Romeo · Jeep · Fiat (BEV, HEV, MHEV, PHEV, ICE, HFCV).

Coşkunöz Ar-Ge Bursa
Jul 2023 – Aug 2023

Advanced Analysis & Test Engineer

TOFAŞ Contract · Bursa
May 2022 – Jul 2023

Virtual Analyses Engineer – Durability Team

PROJECTS

ANSA Workspace – team toolbox inside ANSA TOFAŞ 2026

Six Python tools inside ANSA for two CAE teams, from model setup to connection and material checks.

Abaqus Notepad – Abaqus-aware .inp editor Side project 2026

Keyword completion, **51** live checks with one-click fixes and a model tree; opens **2M** lines in a second. **Praised by SIMULIA's R&D Director.**

ODB Viewer – Abaqus results as a web page Side project 2026

Turns ODB files into one shareable HTML page holding every history output, with time, X–Y and contact charts, CFC filters and templates; uses no Abaqus license.

Bolt Calculator – preloaded bolted joints Side project 2026

User-friendly bolted-joint design, from bolt sizing and preload to a safety check under the loads from simulation (Shigley Ch. 8 with VDI 2230 additions).

Alfa Romeo badge, cut from a photo: Ivan Radic, CC BY 2.0

CERTIFICATIONS

ANSA for Durability
ANSA & META Scripting
BETA CAE Systems

Education of the Educator
Certified Internal Trainer
Tofaş Akademi

Data Science, Stages 2 & 3
CNR AI HUB

Generative AI, Stage 2
Kaykayoglu Innovation Group

Generative AI, Stage 3
Rise Technology,
Consulting & Academy

ABAQUS COURSES

10-Week Abaqus Training Program
Grup Otomasyon, 2021

- Introduction to Abaqus/CAE
- Linear Dynamics
- Explicit Dynamics
- Heat Transfer and Thermal-Stress Analysis
- Buckling, Postbuckling and Collapse Analysis
- Obtaining a Converged Solution
- Metal Inelasticity and Metal Forming
- Analysis of Composite Materials
- Modeling Rubber and Viscoelasticity
- Substructures and Submodeling

SIMULIA webinars
Grup Otomasyon, 2020–2021

- Element Types & Mesh Structure
- Mesh Modeling Techniques
- Connections of Assembled Parts
- Fundamentals of Nonlinearity
- Contact BCs in Static Analyses
- Getting a Quicker Solution
- Material Data in FE Software
- Heat Transfer Analyses
- Metal Forming Analyses
- Interference-Fit Parts
- Pretensioned Bolted Joints
- Welded Parts
- Form-Connected Parts
- Riveted Parts

LANGUAGES

Turkish	Native
English	Professional working proficiency

ADDITIONAL

Volunteering	Board Member, BİSPADER
Driver's license	Class B

AWARDS & TALKS

2025 **TOFAŞ R&D Technical Excellence Award**
"Simulation of Air Entrapment Effects in Door Gap Seal using Fluid Cavity Modeling Technique"

2025 **TAKT 2025 – SIMULIA Abaqus Türkiye Users Meeting**
Speaker: "Door OverSlam Analysis via Abaqus/Explicit"

2022 **Altair Community Leader of the Month**

EARLIER EXPERIENCE

Positive Design Bursa — Mar 2022 – Apr 2022
CAE Engineer

- FE models in Simcenter 3D and Apex, nonlinear contact in Marc Mentat and welding simulations in Simufact; GISSMO damage model research.

Hexagon Studio CAE team · Gebze, Kocaeli — Sep 2021 – Feb 2022
Structural Analysis Engineer Intern

- Built HyperMesh models and ran static, inertia relief, modal, frequency/transient response, complex eigenvalue and explicit analyses in OptiStruct and Radioss.
- Sized bolted joints for ECE R110 (hand calculations, Excel, 1D/3D FE); prepared MNF flexible bodies of suspension parts for vehicle dynamics.

Zümrüd-ü Anka UAV Team Teknofest · Bursa — Feb 2021 – May 2021
Mechanical Design & Structural Analysis Lead

- Designed a multipurpose quadcopter in Fusion 360 and simulated drop impacts with explicit dynamics (HyperWorks, SimLab, Radioss).
- Topology- and lattice-optimized the frame arms at a 1.25 reserve factor (Inspire, OptiStruct); redesigned in PolyNURBS for 3D printing.

HUSAN ARMS Beyşehir, Konya — Jun 2018 – Aug 2018
Mechanical Engineering Intern

- Design, production, assembly and firing tests of AR-15-style shotguns.

EDUCATION

B.Sc. Mechanical Engineering Selçuk University · Konya — 2017–2022

Graduation project: design and analysis of a 5+1-speed manual gearbox – gear, shaft and bearing calculations, gear-contact FEA (HyperMesh, Radioss) and shaft rotordynamics with a Campbell diagram.

TRAINING

Abaqus – Advanced Topics Dassault Systèmes — 4 days – 2024

- Defining an Abaqus Model
- Linear Static Analysis
- Element Selection Criteria
- Multistep Analysis in Abaqus – Restart & Import
- Constraints & Contacts
- General Contact & Contact Pairs
- Introduction to Nonlinear FEA
- Nonlinear FEA with Abaqus/Standard
- Why Abaqus Fails to Converge
- Modeling Tips for Contact & Convergence
- Introduction to Dynamics
- Overview of Abaqus/Explicit
- Explicit Dynamic Quasi-Static Analyses
- Mechanisms and Multibodies in Abaqus
- Introduction to Composites
- Macroscopic Modeling
- Laminate Modeling
- Modeling Damage and Failure in Composites
- Cohesive Elements

BETA CAE logo, cropped: BETA CAE Systems, CC BY-SA 4.0