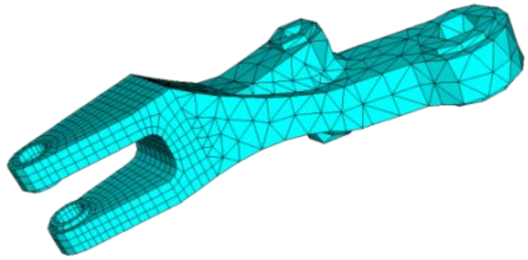
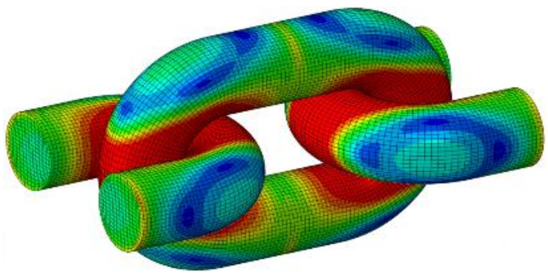
Abaqus software training course

- **Preliminary introduction**
  - Introduction to Finite Element Method (FEM).
  - Introduction to Abaqus software package such as Abaqus CAE, Abaqus standard, Abaqus explicit.
  - Defining and analyzing steps in Abaqus software such as determination of the material model, element section properties, loads and boundary conditions.
  - Introducing failure criterions such as maximum normal stress, Tresca and Von-mises.
- **Introducing different types of elements and their properties.**
  - Solid elements such as C3D8, C3D8R and C3D8I.
  - Shell elements such as S4, S4R and STRI3.
  - Beam elements such as B21, B22 and B32OS.
- **Meshing**
  - Presenting different methods of meshing such as structured method, sweep method and free meshing.
  - Partitioning the model to improve the mesh.
- **Step-by-step approach for modeling and analyzing parts.**
  - Modeling and analyzing parts using solid elements.
  - Modeling and analyzing parts using shell elements in a wide variety of loads and boundary conditions.
  - Modeling and analyzing parts using beam elements and dynamic loading.





Email address: Hamid.moeenfard@gmail.com **Instructor:** Dr. Hamid Moeenfard