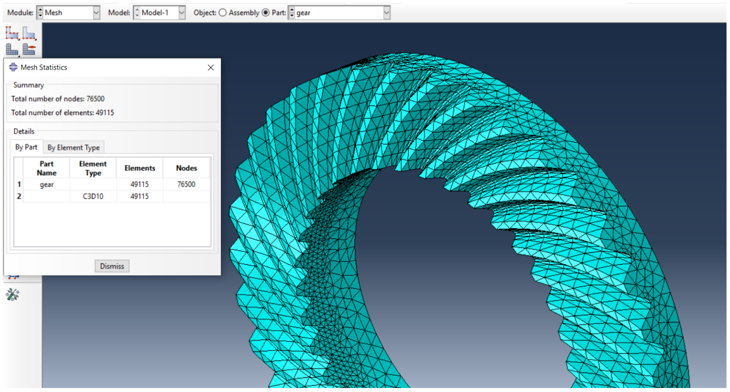
FEA Services LLC
Inspired Solutions... Impeccable Service

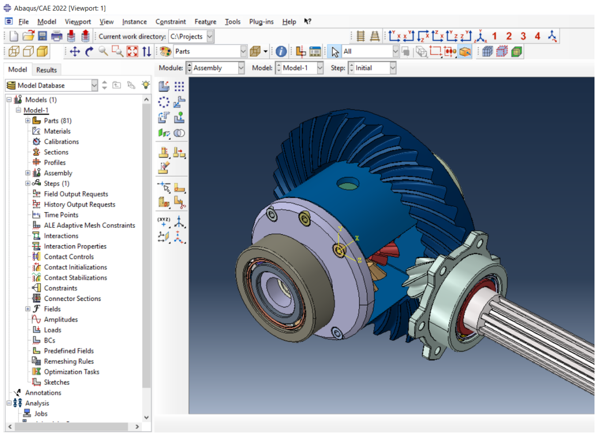
Abaqus FEA Software Training
SYLLABUS

ABNSA - Startup Nonlinear Structural Analysis with Abaqus



Geometry Operations
- Importing
- De-featuring
- Building Assemblies

Basic Material Definitions
- Linear Elastic
- Elastic-Plastic (Metals)
- Hyper-Elastic (Rubber)



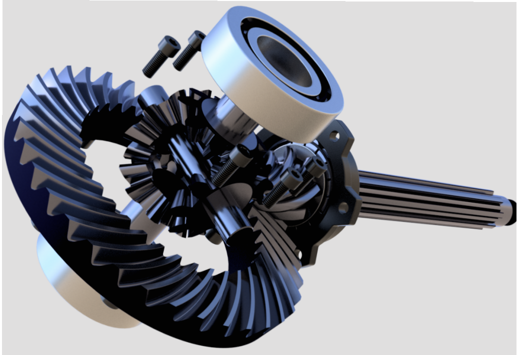
Choosing Appropriate Analysis Procedures
- Linear v. Nonlinear
- Static v. Dynamic

Meshing Practices,3D and 2D

Applying Loads, Boundary Conditions and Constraints

Multi-Body Contact and Friction Modeling Processes

Defining Complex Multi-Step Simulations



Understanding the Input Deck

Output Processing
- Binary Output/Contour Plots
- Simple Text Output

FEA Services LLC
15720 Brixham Hill Ave., Suite 300
Charlotte, NC 28277

phone: (704) 594-1113
email: info@feaservices.net
www.feaconsulting.net