Autodesk Professional in Simulation for Static Stress Analysis

Target Audience

This course is designed for advanced design engineers, simulation analysts, and CAE professionals who want to develop and validate their expertise in linear static stress simulation using Autodesk Fusion 360. It is ideal for professionals in aerospace, automotive, industrial design, mechanical, medical device, and manufacturing industries looking to strengthen their skills in preparing, analyzing, and validating engineering simulations.

Course Objective

This course delivers advanced knowledge and hands-on experience in performing linear static stress simulations using Autodesk Fusion 360. Participants will learn to prepare and simplify models, define loads and boundary conditions, generate and refine meshes, analyze simulation results, and iteratively improve designs for enhanced structural performance. The course is aligned with the competencies required for the **Autodesk Certified Professional in Simulation for Static Stress Analysis** certification, preparing learners to validate their simulation expertise and succeed in certification and real-world engineering applications.

Course Outcome

• **Simulation Preparation Mastery –** Gain expertise in preparing CAD models, simplifying geometry, and defining proper material and boundary conditions.

• **Advanced Setup and Meshing Skills –** Learn to configure simulation studies, assign detailed constraints and loads, and manage global and local mesh refinements.

• **Results Interpretation and Validation –** Master interpreting stress results, evaluating model credibility, and understanding physical behavior under loads.

• **Iterative Study Refinement –** Develop the ability to refine simulations, reuse studies, and adjust designs based on analytical feedback.

Course Outline: The course comprises **40 hours** of advanced **theory and practical labs** and is divided into **6 comprehensive chapters**. Each chapter is followed by hands-on lab exercises to reinforce learning and assess understanding of the topics covered.

Chapter 1. Simulation Preparation

Evaluating and Interpreting Information
- Determining material appropriateness for static stress
- Identifying loads and classifying load types (hydrostatic, remote, bearing)
- Interpreting constraints and assembly connections
- Evaluating sufficiency and prioritizing information for analysis

Simplifying Geometry and Model Prep
- Using simplify workspace and direct modeling tools (split face/body, remove features)
- Cloning and reusing models
- Validating interference and confirming CAD models
- Repairing imported models and cleaning up assembly structures
- Creating manifold bodies from surfaces

Chapter 2. Simulation Setup

Study and Mesh Configuration
- Defining and modifying simulation studies and mesh settings
- Selecting element order, analysis type, and removing rigid body modes

Material Properties Assignment
- Identifying mechanical properties and customizing materials

Defining Loads and Constraints
- Converting diagrams and written scenarios into applied loads and constraints
- Applying vector forces, remote forces, gravity, and pressures
- Applying fixed, pin, frictionless, sliding, bolted, and other constraint types
- Managing symmetry and explaining constraint limitations

Contact Management
- Generating contact sets and assigning tolerances
- Selecting contact types and configuring options

Chapter 3. Solving Simulations

Pre-Check and Error Handling
• Reviewing and addressing simulation pre-check information
• Interpreting errors and warnings, predicting impacts, and taking corrective actions
Mesh Generation and Validation
• Generating and inspecting mesh elements
• Ensuring pre-check alignment with analysis plans

Chapter 4. Result Analysis

Result Extraction
• Obtaining stress data (slice plane, surface probe, point probe)
• Recognizing expected data ranges and interpreting deflection, stress, and deformation
Evaluating Credibility
• Assessing setup correctness and result plausibility
• Identifying misconfigured forces, constraints, contact sets, or geometry simplifications
• Evaluating compliance with linear static stress assumptions

Chapter 5. Study Refinement

Mesh Refinement Techniques
• Applying global and local mesh changes
• Adjusting mesh density, transition, and adaptive mesh configurations
Geometry and Boundary Condition Updates
• Modifying features, suppressing/unsuppressing components
• Selecting workflows for iterative updates
Study Reuse and Iteration
• Cloning load cases and simulation models
• Refining studies with new design changes or boundary conditions

Chapter 6. Documentation and Verification

Documenting and Reporting Results
• Preparing result views and reports for presentation or verification
Verification Techniques
• Validating simulation configurations and comparing to expected physical behavior